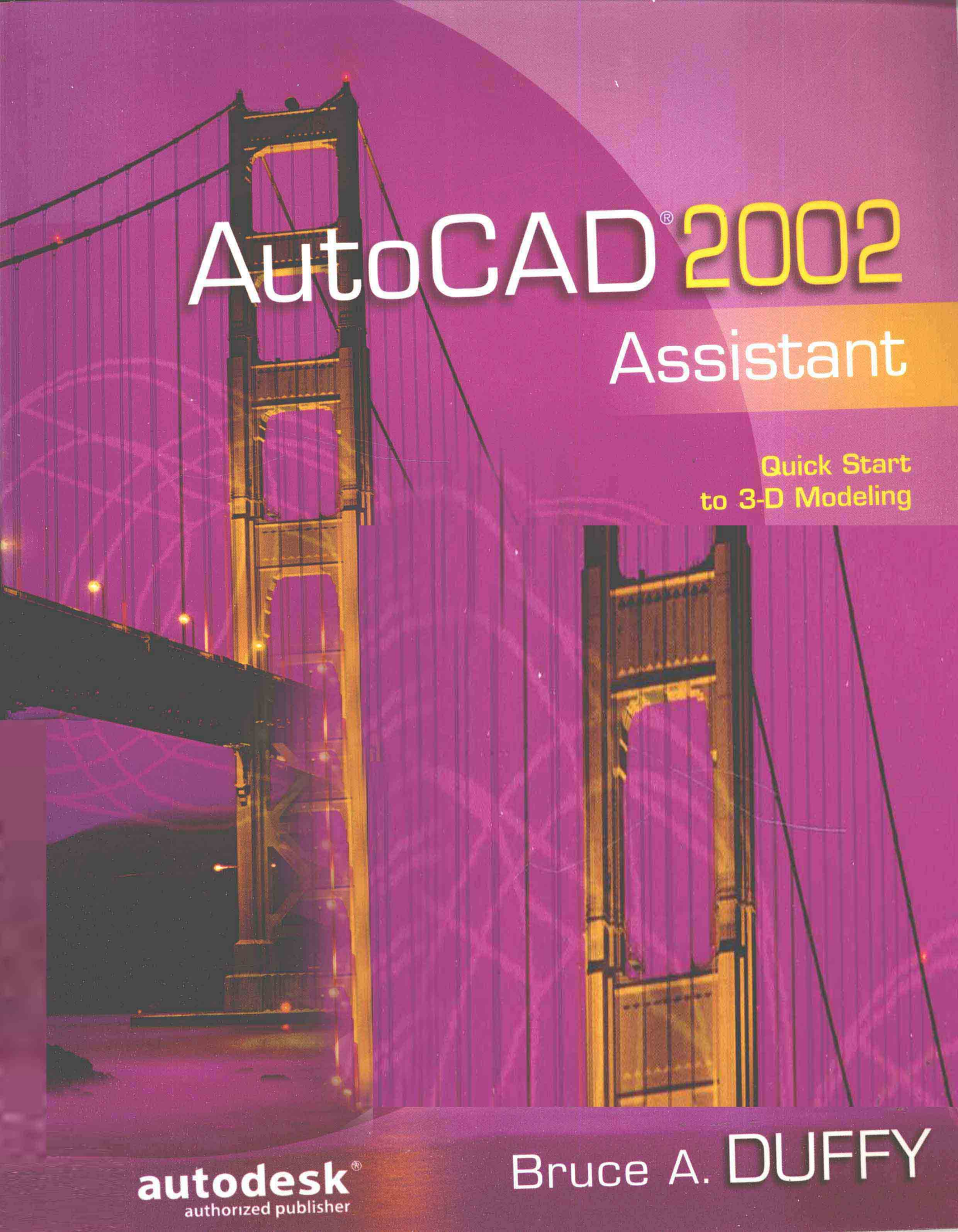

AutoCAD® 2002 Assistant

Quick Start to 3-D Modeling

Bruce A. DUFFY

autodesk®
authorized publisher

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autocad.2002.assistant

Introduction

Welcome to the AutoCAD 2002 *Assistant* – referred to here as the *Assistant*.

Attempting to use an advanced 3D modeling program like AutoCAD with no formal instruction can be a daunting task. Even when formal instruction is available for learning basic AutoCAD, the majority of discussion and practice is often focused on 2-dimensional features of the software. So with little or no instruction, the 3-dimensional design and visualization functions in AutoCAD may seem illusive and confusing.

The *Assistant*, therefore, has been developed to aid students with little or no formal instruction in AutoCAD. The *Assistant* guides the self-learner quickly through 2-dimensional topics into 3-dimensional visualization and design functions of the AutoCAD software.

The *Assistant* is a companion text to Fundamentals of Graphic Communication (FGC), 3rd edition by Bertoline/Weibe. The *Assistant* assumes that you, as a "self-learner," have basic experience with the latest versions of Microsoft Windows. Even if you do not feel confident with Microsoft Windows, using the *Assistant* will help you develop some fundamental skills that are transferable to any type of Microsoft Windows software.

The content of Fundamentals of Graphic Communication text (FGC) is presented by Bertoline and Weibe based upon one fundamental premise: that all forms of engineering graphics communicate, and aid visualization of, 3-dimensional design information. The *Assistant*, therefore, can be used as a supplement for learning the material presented within the FGC text. An additional and more exhaustive AutoCAD reference is available from the same publisher entitled the AutoCAD 2002 Instructor. The self-learner who wishes to delve more completely into the subject of AutoCAD should acquire the AutoCAD 2002 Instructor text after completing all sections of the *Assistant*.

Format of the Assistant

The Assistant provides information and instruction via a simple format. Text describing AutoCAD functions and how to use them is presented on the left pages. Related Figures are presented on the right pages to supplement your exploration and discovery of AutoCAD (see Fig. A).

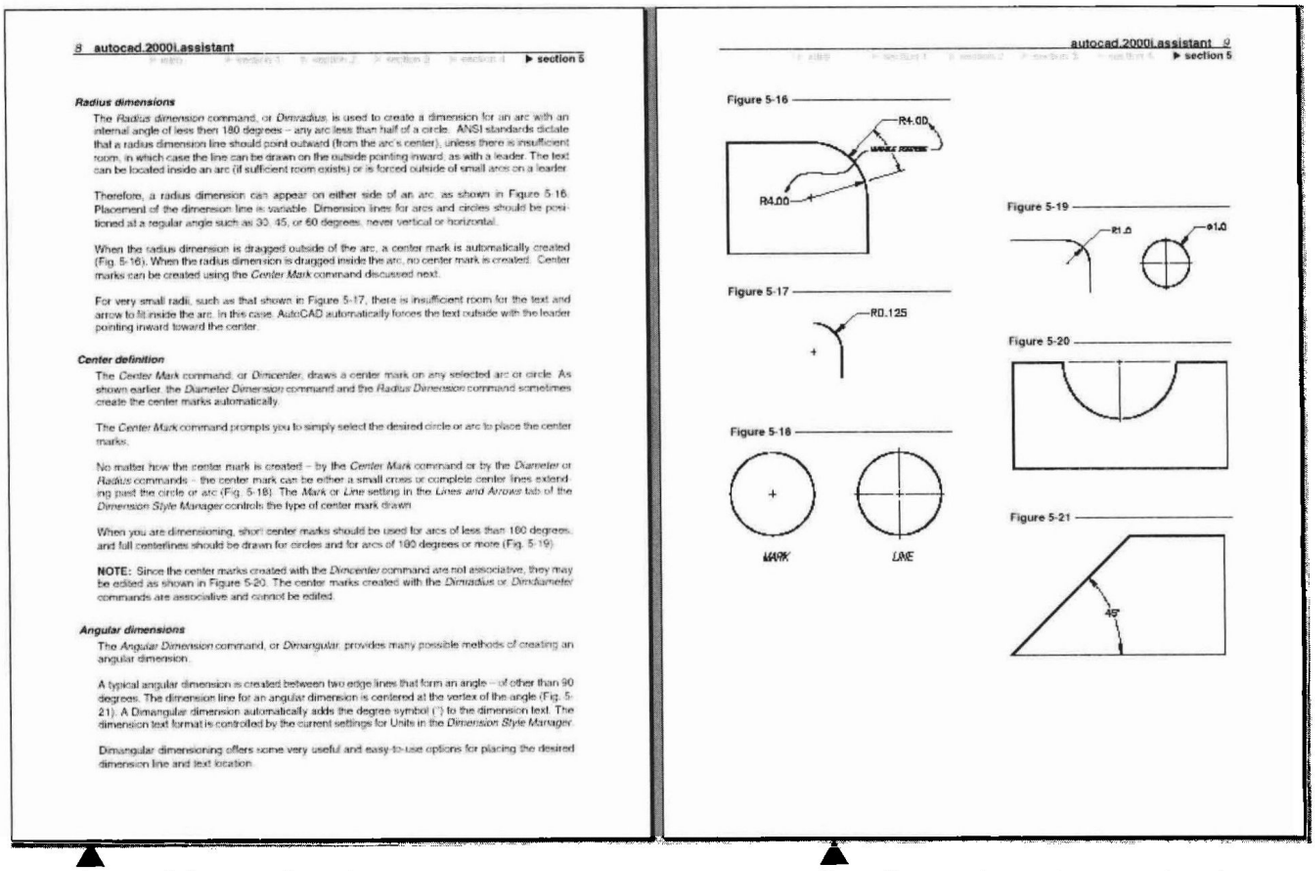
Taking you quickly from basic to advanced utilization of AutoCAD graphic communication functions with no prior formal instruction in AutoCAD may seem a bit extreme. But as you begin to use the AutoCAD software, you will discover that AutoCAD is developed for the Microsoft Windows operating environment. Many of the normal functions available in other Microsoft Windows programs are available to you within the AutoCAD interface.

Sections 1 and 2 have been designed to be highly interactive. As you work through the content in these two sections, you will be practicing the content as it is presented. Drawing example files have been provided for you limiting the amount of setup you must complete in order to discover AutoCAD's 2D and 3D functionality.

Sections 3, 4, and 5 are designed to be more informational and build upon the experience and skills gained from Sections 1 and 2. Practice sessions are included at the end of each of these sections. You will find many of these practice sessions are taken directly from your FGC text.

Once you have completed Sections 1-5 you will be ready to use AutoCAD for experimenting with any exercises within the Fundamentals of Graphic Communication text.

Figure A



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Radius dimensions

The *Radius dimension* command, or *Dimradius*, is used to create a dimension for an arc with an internal angle of less than 180 degrees – any arc less than half of a circle. ANSI standards dictate that a radius dimension line should point outward (from the arc's center), unless there is insufficient room, in which case the line can be drawn on the outside pointing inward, as with a leader. The text can be located inside an arc (if sufficient room exists) or is forced outside of small arcs on a leader.

Therefore, a radius dimension can appear on either side of an arc, as shown in Figure 5-16. Placement of the dimension line is variable. Dimension lines for arcs and circles should be positioned at a regular angle such as 30, 45, or 60 degrees, never vertical or horizontal.

When the radius dimension is dragged outside of the arc, a center mark is automatically created (Fig. 5-16). When the radius dimension is dragged inside the arc, no center mark is created. Center marks can be created using the *Center Mark* command discussed next.

For very small radii, such as that shown in Figure 5-17, there is insufficient room for the text and arrow to fit inside the arc. In this case, AutoCAD automatically forces the text outside with the leader pointing inward toward the center.

Center definition

The *Center Mark* command, or *Dimcenter*, draws a center mark on any selected arc or circle. As shown earlier, the *Diameter Dimension* command and the *Radius Dimension* command sometimes create the center marks automatically.

The *Center Mark* command prompts you to simply select the desired circle or arc to place the center marks.

No matter how the center mark is created – by the *Center Mark* command or by the *Diameter* or *Radius* commands – the center mark can be either a small cross or complete center lines extending past the circle or arc (Fig. 5-18). The *Mark* or *Line* setting in the *Lines and Arrows* tab of the *Dimension Style Manager* controls the type of center mark drawn.

When you are dimensioning, short center marks should be used for arcs of less than 180 degrees, and full centerlines should be drawn for circles and for arcs of 180 degrees or more (Fig. 5-19).

NOTE: Since the center marks created with the *Dimcenter* command are not associative, they may be edited as shown in Figure 5-20. The center marks created with the *Dimradius* or *Dimdiameter* commands are associative and cannot be edited.

Angular dimensions

The *Angular Dimension* command, or *Dimangular*, provides many possible methods of creating an angular dimension.

A typical angular dimension is created between two edge lines that form an angle – of other than 90 degrees. The dimension line for an angular dimension is centered at the vertex of the angle (Fig. 5-21). A Dimangular dimension automatically adds the degree symbol (°) to the dimension text. The dimension text format is controlled by the current settings for Units in the *Dimension Style Manager*.

Dimangular dimensioning offers some very useful and easy-to-use options for placing the desired dimension line and text location.

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Figure 5-16
R4.00
R4.00

Figure 5-17
R0.125

Figure 5-18
MARK
LINE

Figure 5-19
R1.0
ø1.0

Figure 5-20

Figure 5-21
45°

Text on left page describes AutoCAD functions.

Save Time and speed up your learning process by using the right hand page effectively! See note below.

Time Saving Note:

As a self-guided learner, our learning process is greatly enhanced by making quick reviews of the material you've already covered.

You can use the AutoCAD 2002 Assistant to become cofortable withthe AutoCAD software and speed up your learning process by:

- Working through each chapter in the order they are presented.
- Working through an entire chapter each time you sit at a workstation to use AutoCAD.
- Reviewing the material covered in the last chapter BEFORE you move on to the next chapter.

You'll find reviewing the information from your last learning session is fast and easy when you utilize the right hand page of the Assistant. To review what you learned (or didn't learn) in your last session simply:

- Flip through the right hand pages of the last chapter you covered. By looking at the figures and hand notes you will build your confidence regarding what you remember. You will also have a quick reference for those things you don't remember.
- When you see a topic or function you don't remember well, look up the figure number on the left hand page for a more detailed review.

Once you've worked through each chapter of Assistant, you can use AutoCAD to complete exercises from the Fundamentals of Graphic Communications text.

Using Assistant

The simple format of the *Assistant* will guide you through the initial process of completing the practice sessions. Several drawing files are provided with the *Assistant* to aid your assimilation of the information presented.

The drawing files provided with the *Assistant* begin with "DE" and end with the extension of ".dwg." These files should be copied into the main AutoCAD folder on your computer (see Appendix A).

These drawing files are "read-only." Depending on the method used when you initially open the files, you may receive a message from AutoCAD that informs you of the "read-only" status (see Fig. B). Always respond to this prompt by simply selecting *Yes*.

When closing the example drawing files, AutoCAD will ask you if you want to save your changes (see Fig. C). Simply respond *No* each time AutoCAD prompts you to save one of the *Assistant* drawing files.

See Appendix A for instructions on downloading the *Assistant* drawing files if these files have not been previously downloaded for you.

Opening and Closing Drawing Files

When you start AutoCAD, you may encounter two AutoCAD windows called: AutoCAD Today and Active Assistant. Whether these windows appear depends upon the AutoCAD system settings on your computer.

AutoCAD Today

When you begin AutoCAD (normally accomplished by double-clicking the AutoCAD icon (Fig. D) on your Windows Desktop) you will likely be presented with the AutoCAD Today window (Fig. E). AutoCAD Today is a feature that was added in AutoCAD 2000i to enhance Internet access. Once you have worked through Sections 1 to 5 of the *Assistant*, AutoCAD Today is well worth exploring. Until you have progressed through these first four sections of the *Assistant* we recommend that you simply close the AutoCAD Today window and use the standard *Open* function (Fig. F and Fig. G) to access the *Assistant* practice files.

Active Assistant

You will also likely be presented with AutoCAD's Active Assistant window upon first beginning AutoCAD (Fig. H). The Active Assistant was also added with AutoCAD 2000i as a dynamic help function providing command-related information. It is initially set to activate each time you select a command. You may find this somewhat annoying when trying to follow the practice sessions of the *Assistant*.

To limit the display of the Active Assistant:

- Place the mouse pointer inside the Active Assistant window.
- Right-click the mouse, and select *Settings* from the shortcut menu (see Fig. I).
- Select *On Demand* in the *Active Assistant Settings* window, and then select the *OK* button (see Fig. J).

  This action suppresses the automatic activation of the Active Assistant so it will not be seen each time you select an AutoCAD command.

Figure B

Always respond "yes" when asked to open Assistant drawings as "read only"!

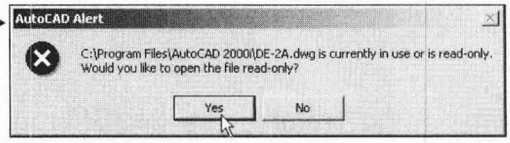
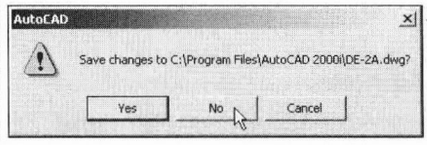


Figure C



Always respond "no" when asked to save changes to Assistant drawings.

Figure D

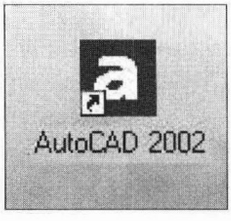


Figure E

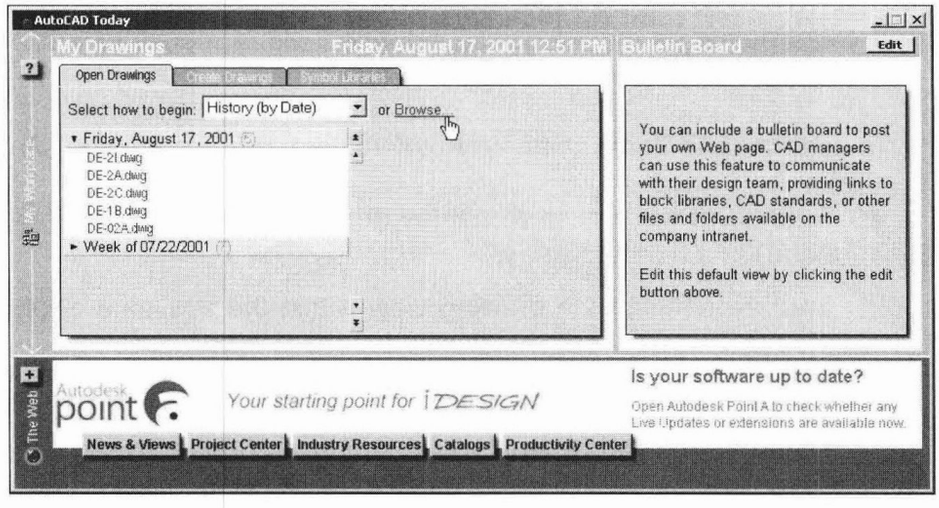


Figure F

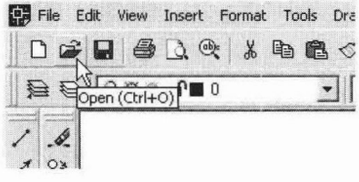
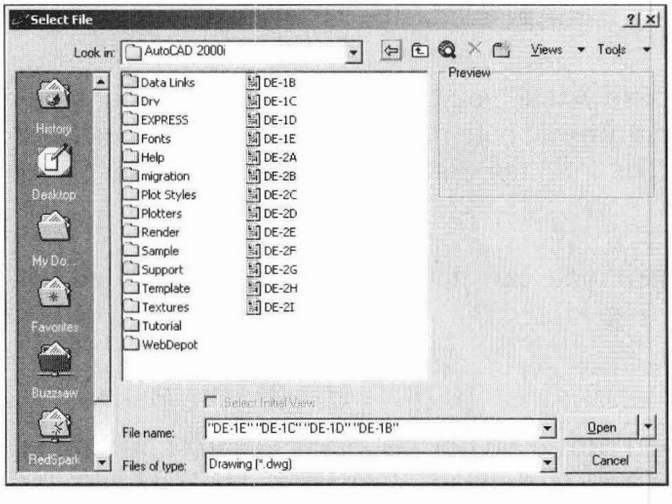


Figure G



Assistant drawing files must be downloaded from the web. See Appendix A.

Figure H

Point inside window, right-click

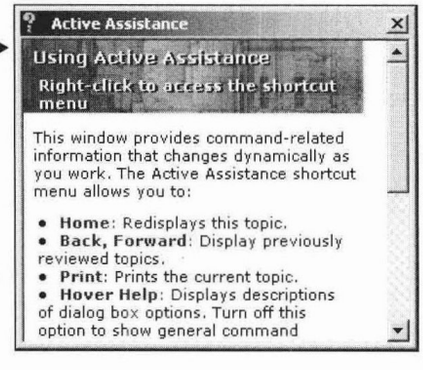
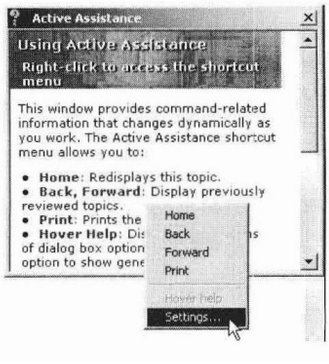


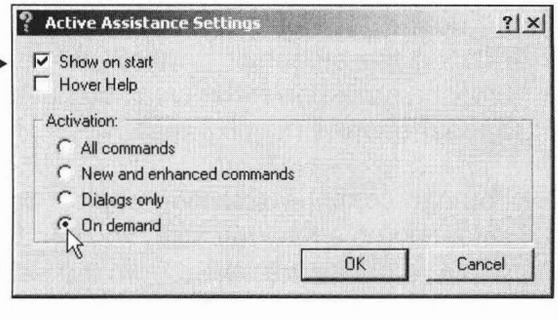
Figure I



Select "Settings."

Figure J

Select "On demand," then "OK".



- Then you can close the Active Assistant window by selecting the close button in the upper right corner of the Active Assistant window.

Viewing Drawing Files

Standard Microsoft Windows Pan and Zoom viewing features are available in AutoCAD. The easiest method of panning and zooming within the drawing display is determined by the type of mouse you are using.

Using a Mouse with a Mouse Wheel

If you are using a mouse device equipped with a mouse wheel, you can use the wheel for easily accessing the Pan and Zoom display functions.

To use *Pan* and *Zoom* when an AutoCAD drawing is open you can:

- Hold down the mouse wheel (by depressing it), and then move the pan cursor (the small white hand) around the display.

  You are panning the drawing view within the available display area. When you release the mouse wheel the *Pan* functions stops.

- Zoom in and out within the display area by rolling the mouse wheel forward and backward.

  Rolling the mouse wheel forward zooms in on the drawing and rolling the mouse wheel backward zooms out. Point the "crosshairs" at the location you want to zoom in to or out from.

Note: The appropriate mouse driver must be installed on your computer for the mouse wheel to operate as described here.

Using a Mouse with no Wheel

If you are using a mouse that does not have a mouse wheel, you can use the Pan and Zoom features by using AutoCAD's *Pan Realtime* and *Zoom Realtime* commands. These commands are easily accessed by selecting their respective buttons from the Standard Toolbar at the top of the AutoCAD window (Fig. K and Fig. L).

Once *Pan Realtime* or *Zoom Realtime* is selected, you can right-click the mouse and switch between them by selecting the appropriate action (Fig. M).

Selecting Objects

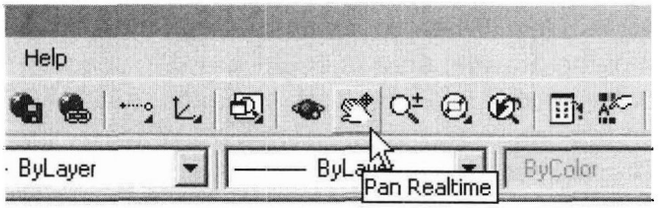
When selecting objects on an AutoCAD drawing you can select objects one at a time or by making a multiple selection. AutoCAD has several multiple selection options. The easiest option to use for multiple selections is a *Crossing* window.

Single object selection

Placing the crosshair pickbox on an object and left-clicking the mouse selects the object. The crosshair pickbox is the small box at the center of the crosshairs. When an object is selected it will appear dashed (highlighted).

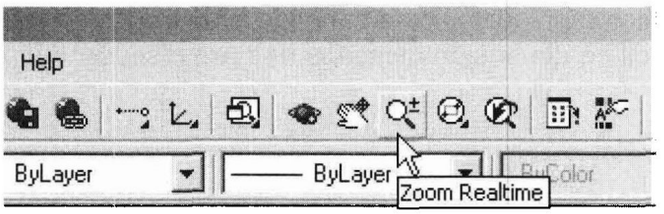
You can continue selecting objects one at a time and then select a command to edit them. If you select an object by mistake, you can hold down the *Shift* key on the keyboard and select the object again removes the object from the selection set.

Figure K



◀ If your mouse has no "wheel", use Pan Realtime button for panning around the display area.

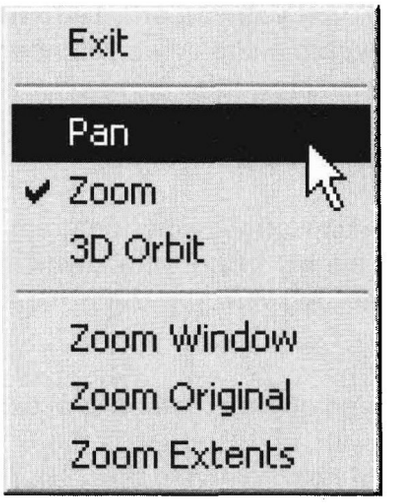
Figure L



◀ If your mouse has no "wheel", also use Zoom Realtime for zooming. Remember: after you select Zoom Realtime, place the mouse cursor in the drawing area, left-click, hold down, and move mouse cursor up and down to zoom in and out.

Figure M

Right-click shortcut menu displayed when using Pan Realtime or Zoom Realtime. Just right-click mouse when either command is active. ▶



Multiple object selection

Multiple objects can be selected by using a *Crossing* window. When you want to select more than one object, position the crosshairs on the right side of the drawing area in empty drawing space—not on an object—and left-click. When you left-click the mouse in empty drawing space, AutoCAD automatically activates one of two window selection features—a window or crossing window. By pulling the crosshairs to the left you are using the *Crossing* window feature. Any objects within, or touched by, the *Crossing* window will be selected (Fig. N).

Object appearance - Grips

Once you have selected objects using either method, the objects appear highlighted on your screen as a visual reference that they have been selected. You will also see small boxes at key points on each (see Fig. O). These boxes are called grips.

Grips combine with several of the most common editing commands to allow you to edit objects more quickly. Grips are most useful when editing dimension locations and are discussed in the dimensioning chapter of the *Assistant*.

Getting Back and Getting Help

Undo and Redo

Whenever you make mistakes in a command sequence in AutoCAD, you can undo your last command by selecting the *Undo* button (see Fig. P). You can undo back through the history of the commands you have issued all the way back to when you opened the drawing.

Be Aware! You can use the *Undo* command to undo any commands you have issued as far back as you like. But **you can only redo the last step** by using the *Redo* command button (see Fig. Q). The *Redo* command only works once to redo the last command undone! **And,** *Redo* will only work if used immediately after using *Undo*!

Getting On-line Help

As in any Microsoft Windows program, AutoCAD *Help* can be accessed at any time by pressing the *F1* key on the top row of your keyboard. *F1* activates the AutoCAD *Help* window (Fig. R). You can search AutoCAD *Help* by using the *Index* option as you normally do in any Microsoft Windows program.

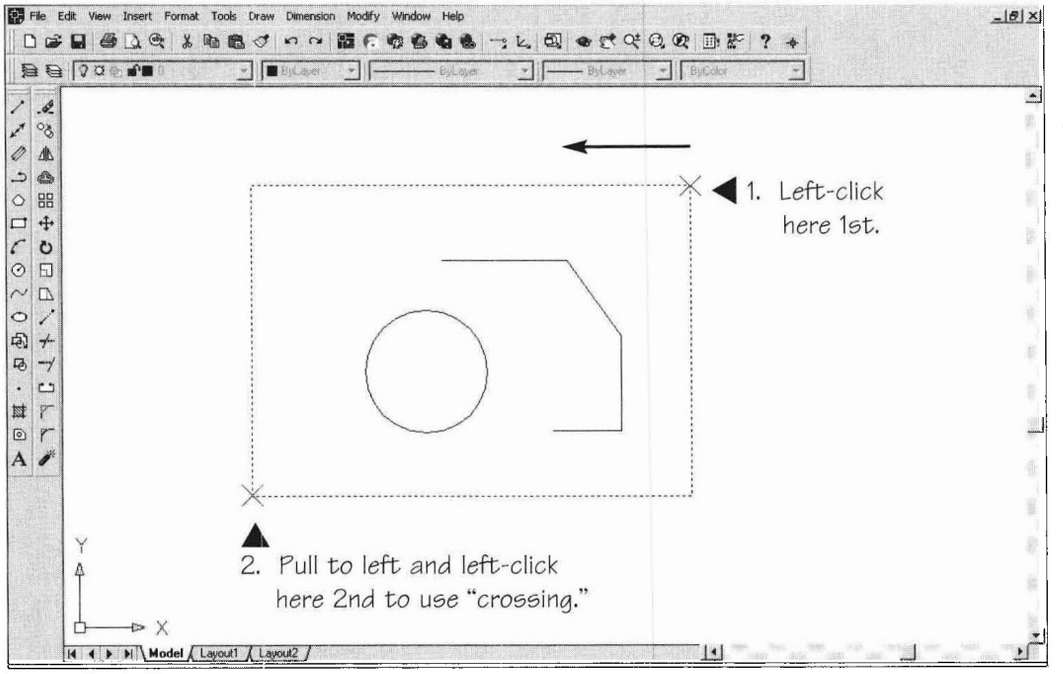
An easier and quicker method to access the appropriate *Help* screen is to select the command you want help with, and then press the *F1* key. Using this sequence, the *Help* window is opened and displays the help screen for the command that is active. This saves time since you do not need to search through long lists of commands within the *Help* window index.

Getting Started

Each of the following sections of the *Assistant* assume that you have started the AutoCAD software and that the example drawing files have been downloaded, or copied, into the main AutoCAD folder of your computer.

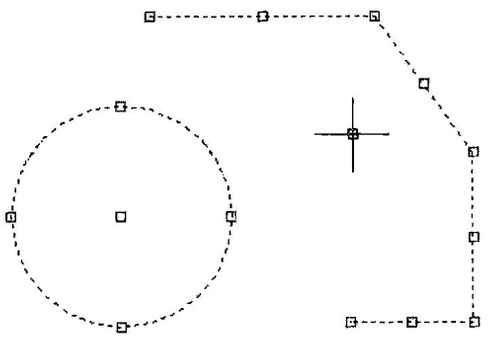
To begin using the *Assistant*, double-click on the AutoCAD icon on your Windows desktop (Fig. S) to start AutoCAD and then reference Appendix A for drawing file locations.

Figure N



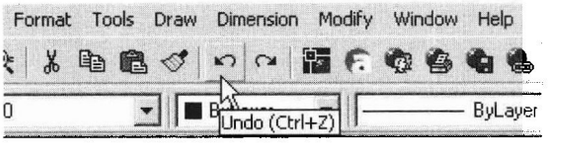
Objects selected by using a "crossing window."

Figure O



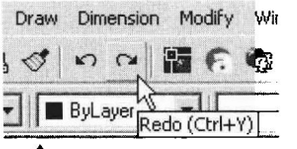
Objects highlighted to show they are selected. Little boxes are called "Grips."

Figure P



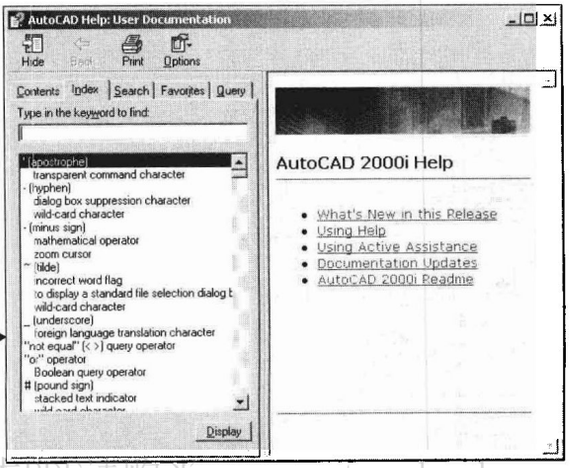
You can "Undo" back as far as you want.

Figure Q



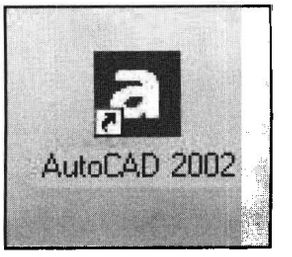
You can only "Redo" one time. So be careful when using "Undo"!

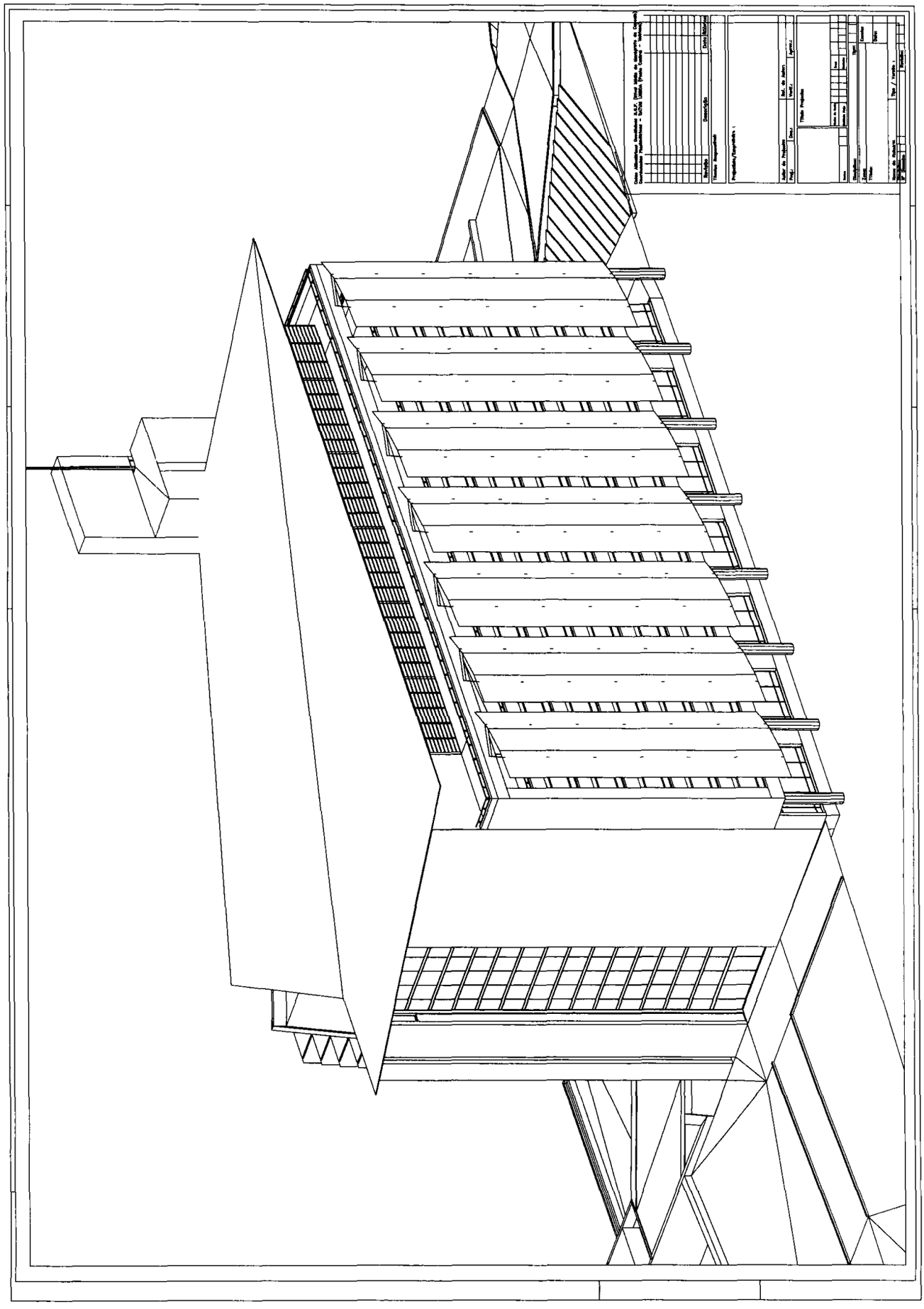
Figure R



Help! Just press "F1" key anytime!

Figure S





EXPO HEADQUARTERS MODEL.DWG, Courtesy of Autodesk, Inc.

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AutoCAD: An Engineering Graphics Communication Tool

The engineering design process generally begins with a rough idea sketch and advances toward a refined manufactured product or system. Computer-Aided-Design technology (CAD) offers several benefits to the engineering design process.

As new designs are conceived, the engineer creates free-hand sketches to capture ideas and possible design solutions. Sketching takes ideas from the "virtual" realm—the short-term memory in the mind of the engineer—into a more tangible form that can be communicated and understood by others (see Fig. 1-1).

To begin the process of analyzing and refining the design, more formal graphic documents—engineering drawings—are created to communicate the specific details of the design. The sketched ideas take on greater description and precision as they are developed into more explicit engineering drawings.

The first stage of utilizing CAD technology—transferring the ideas incorporated in a free-hand sketch into a CAD drawing—is generally in the form of a "roughed-out" CAD sketch. Even rough CAD sketches give the designer extreme accuracy due to the digital environment of the CAD system. For example, AutoCAD stores information with a digital accuracy of 14 places. The data within a CAD drawing is not only accurate but also versatile since the original CAD drawings can be reworked and refined into the final design to be manufactured. The digital format also makes the information transferable to other design technologies and to the manufacturing stage.

Accessing AutoCAD Commands

Commands in AutoCAD can be accessed by several simple methods. Here, we will focus on three main methods of accessing commands: Right-click menus (also called shortcut menus); Toolbar menus; and Pull-down menus (see Fig. 1-2).

Right-click menus

Right-click (shortcut) menus—greatly enhanced in recent releases of AutoCAD—are an extremely efficient method of working with commands and command options. Right-click menus are context-sensitive, which means they provide access to options of the command in progress and to related commands. The right-click menu that appears depends upon the position of the mouse pointer at the time, if any objects are currently selected, and if a command is currently in progress.

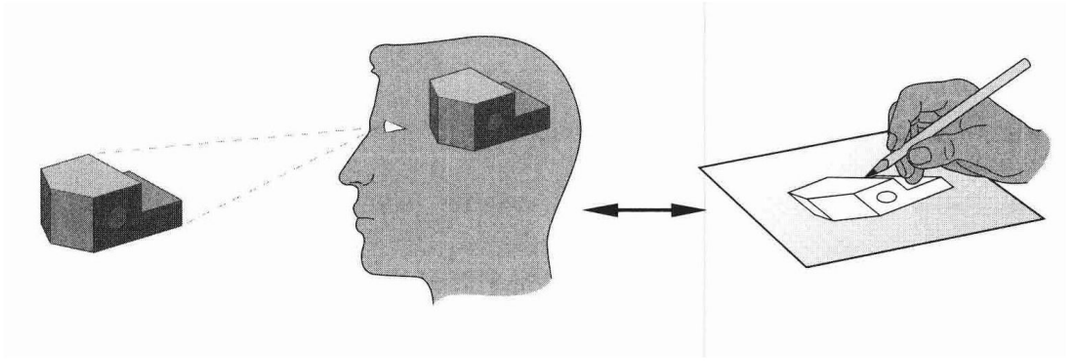
The most commonly used right-click menus are the Default menu, Command-mode menu, and the shortcut menus for Edit, Osnap, and Toolbars as you will see in the following practice steps.

Practice using Shortcut menus

You can access the Default right-click menu by right-clicking the mouse before selecting any objects or before selecting a command.

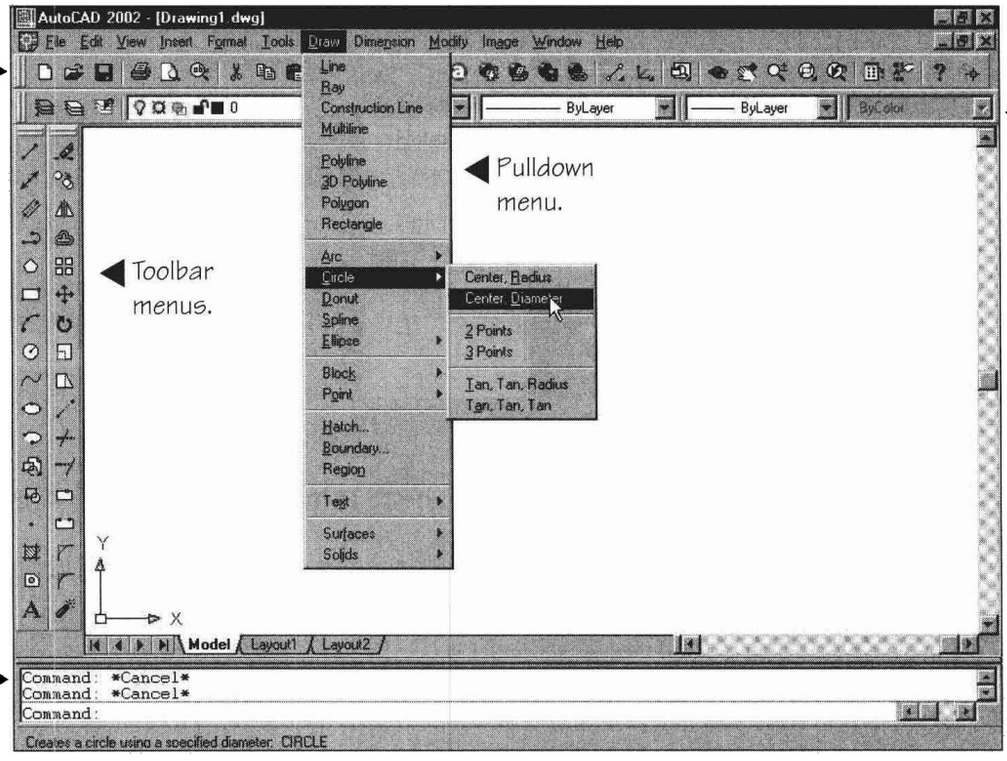
- Select the *Open* button on the Standard toolbar (see Fig. 1-2) and open the DE-1A.dwg drawing file.

Figure 1-1



◀ Sketching puts ideas into tangible form.

Figure 1-2



The "Open" button is on the Standard toolbar. ▶

◀ Pulldown menu.

◀ Toolbar menus.

◀ Object Properties toolbar.

Command Prompt. ▶

▲ Status bar

- With no objects selected and no commands active, place the mouse pointer (crosshairs) in the graphics area and right-click.

  The Default shortcut menu appears (see Fig. 1-3). The Default shortcut menu contains editing commands such as *Cut*, *Copy*, and *Paste* and display control commands such as *Pan* and *Zoom*.

- Press the *ESC* key to make this shortcut menu disappear.

  The Edit shortcut menu can be accessed after objects have been selected.

- Use the small square at the center of the crosshairs to select one or more objects (while no commands are active).

  First you can press *ESC* a few times, if necessary, to ensure that no commands are active.

- Once you have selected an object, place the mouse pointer in the graphics area and right-click.

  The Edit shortcut menu appears containing common AutoCAD editing commands such as *Erase*, *Move*, and *Copy Selection* (see Fig. 1-4). This shortcut menu may also contain editing commands specific to the selected object.

- Press the *ESC* key to make this shortcut menu disappear.

  Osnap shortcut menus are extremely helpful when selecting point locations on your sketch or drawing. Often, when AutoCAD prompts you for a center point or any other point, it is best to use the *Osnap* shortcut menu.

  An Object Snap mode such as Endpoint, Midpoint, or Center can be selected from the *Osnap* shortcut menu. To use Osnap when specifying a point, right-click the mouse while holding down the *Shift* key on the keyboard.

- Hold down the *Shift* key with your left hand.

- Right-click the mouse to produce the *Osnap* shortcut menu (see Fig. 1-5)—the mouse pointer must be in the drawing area.

  If AutoCAD were asking for a specific point location, you could select one of the many point options from this menu.

  Selecting *"Osnap Settings..."* from the *Osnap* shortcut menu will produce the *Osnap Settings* window. Here you can turn on specific *Osnap* modes so they are always active, called "Running Osnap." Running Osnaps reduce the need to repeatedly access the *Osnap* shortcut menu which saves you time while drawing and editing.

- Press the *ESC* key to make this shortcut menu disappear.

  Command-Mode shortcut menus are easily accessed by right-clicking whenever an AutoCAD command is active.

  There are various shortcut menus that can appear when you right-click while a command is in progress. The middle section of the shortcut menu that appears provides options for the currently active command (see Fig. 1-6). *Zoom* and *Pan* are also available at the bottom of Command-mode shortcut menus.