



DRAWING WITH AUTOCAD 2000 - INTRODUCTION

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This is the first of a series of tutorial introductions to CAD. The overriding aim with these tutorial guides is to give you a firm conceptual understanding of CAD, the scope of techniques available for design and presentation, and strategies for approaching a variety of CAD tasks.

In this first tutorial, we will go through the process of starting up AutoCAD on the computer and then look at some of the basic drawing and editing commands. In the next tutorial, we will begin the process of drawing something very simple in order to learn how to attack a drawing task with this system. These tutorials are deliberately verbose, explaining concepts and ideas as you along. It is important, therefore, that you take the time to read and understand those explanations.

These tutorials are written for *AutoCAD 2000* running under the *Windows 95/98* operating environment. Some of the later tutorials make use of enhancement packages that are sold in conjunction with AutoCAD to address specific disciplinary CAD modelling tasks.

GETTING STARTED

It is assumed here that the reader is familiar with the general operation of Windows 95/98. If not, then there is a separate handout that explains the Windows operating environment, how to manipulate files and windows, how to execute application programs and how to use the *Help* facility in Windows.

With the release of AutoCAD 2000, *AutoDesk* has altered its marketing strategy by packaging its base product with various enhancement packages related to specific disciplines (as well as selling the base product separately). Within our Faculty, we use two of those packaged products: *Architectural Desktop* (for building design) and *Land Development Desktop* (for landscape design applications). Other packages include *Mechanical Desktop* (for product design) and *Civil Desktop* (for advanced land engineering).

*Architectural Desktop 2.0*

This tutorial can be run on any of those desktop packages, but it concentrates on the general characteristics of AutoCAD 2000. Later tutorials deal with the specific tools available in those other packages. All the screen shots included in this tutorial came from the Architectural Desktop package.

To run AutoCAD 2000, simply select *Start > CAD Applications > Architectural Desktop 2.0* from the Windows Start Menu. In the *Startup* dialogue box, pass your mouse pointer over the four options available: open an existing drawing; start a new drawing with minimal preset values; start a new drawing based on a template file which effectively pre-selects a large number of options including title blocks, text styles and things like that; or you can use a wizard that walks you through the process of selecting drawing units and paper size. For this tutorial, start a new drawing from scratch using metric units.

LAYOUT OF THE AUTOCAD SCREEN

This installation of AutoCAD is almost exactly as it is shipped from the distributor: there are virtually no local adjustments. The layout of the screen is depicted in Figure 1. The key components of the screen are:

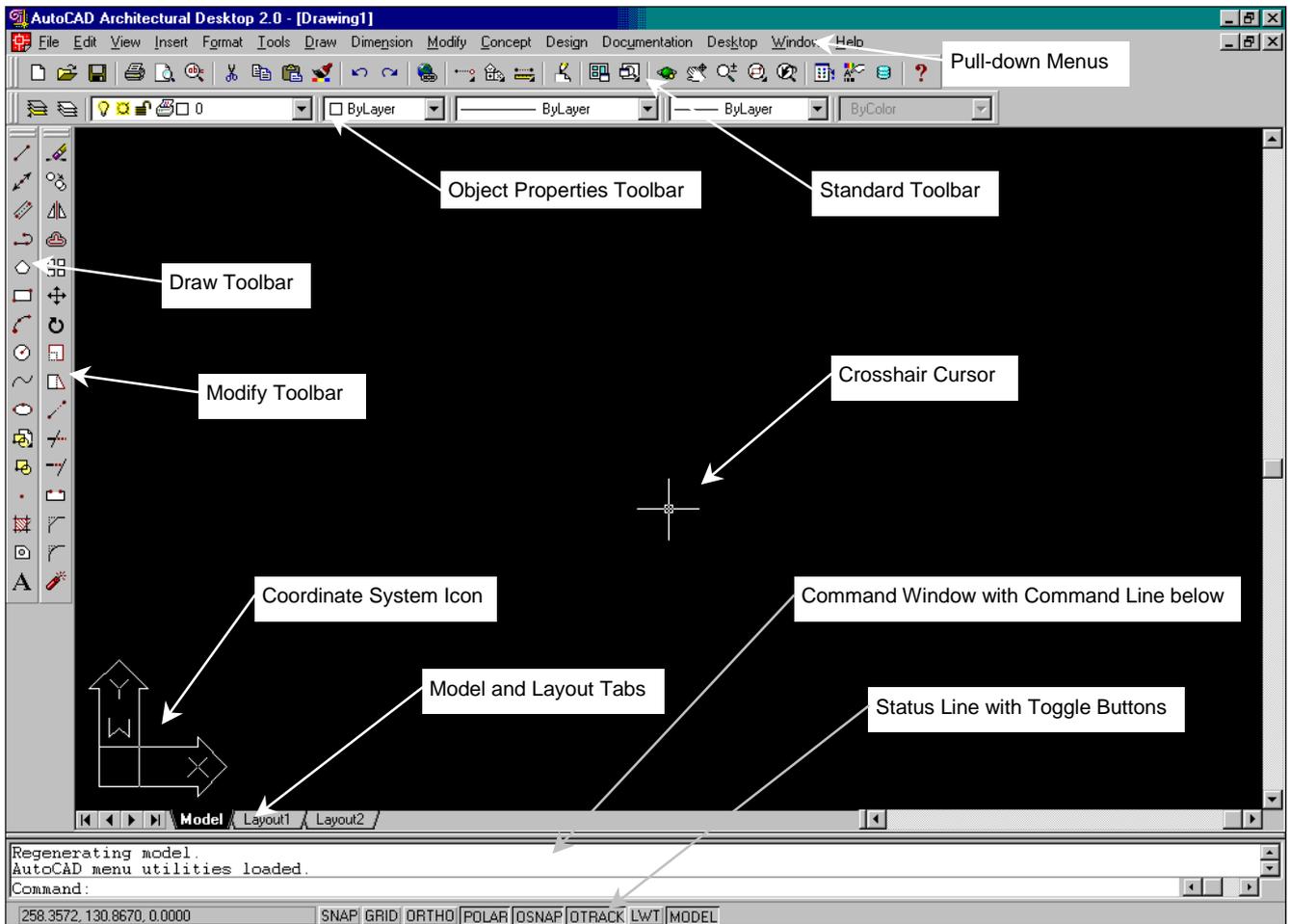


Figure 1 Layout of the AutoCAD Application Window

- **Drawing Window:** covering most of the screen and contains the *drawing cursor* that is used for pointing, and moves as the mouse is moved about. The *axis icon* in the lower left corner shows the X and Y axis directions.
- **Pull-Down Menus:** across the top of the screen operate in the same way as all *Windows* menus. Note that the *Concept*, *Design* and *Documentation* menus are part of *Architectural Desktop* and should be ignored for the purpose of this tutorial.
- **Docked Toolbars:** in the initial setup, these are arranged across the top of the screen below the pull-down menus and down the left side of the drawing area. These consist of sets of button tools, each performing some function (to be explained later). As you pass the mouse pointer over a button it becomes highlighted, and if you pause for a moment, a *tool tip* appears to identify the button.
- **Floating Toolbars:** as well as docking a toolbar along one of the edges of the drawing area, you can have any toolbar "floating" anywhere on the screen. In the initial setup, there are no floating toolbars on the screen.

If you position the mouse pointer at the top or bottom of any of the docked toolbars, hold down the mouse button and drag it onto the drawing area, then that toolbar will switch to a floating one.

- **Command Window:** immediately below the drawing area is a command input area with the standard AutoCAD command prompt (“Command:”) on the left. If you type anything on the keyboard, then it will appear on that line.
- **Status Line:** across the very bottom of the screen. This displays status information (such as the current coordinate location of the cursor in the drawing area). The labelled boxes at about the middle of the status line are “toggle settings” that can be turned on or off by clicking with the mouse ... we will come to these in due course.

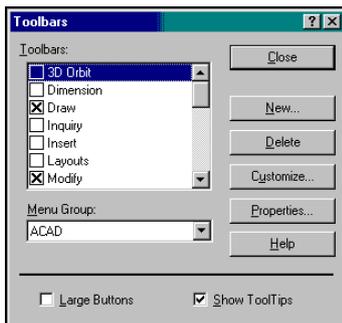
Exploring the AutoCAD Window Layout

Before proceeding, experiment with these various parts of the screen (*if you inadvertently execute a command, it doesn't matter because you can cancel it by pressing the <Esc> key on the top left of the keyboard!!*). Move the drawing cursor around the drawing area and note how the coordinate position is updated on the status line. Explore the pull-down menus by clicking them with the mouse. Try typing something at the Command prompt to see what happens, and try toggling one of the settings on the status line.

Moving and Re-Sizing Toolbars

Next, try re-positioning the toolbars using each of the techniques as follows:

- if you move the mouse pointer to the top of any docked or floating toolbar and then hold down the left mouse button, you can drag it to any position on the screen;
- as soon as you drag a toolbar to the edge of the screen, it changes shape and, if dropped, will attach itself as a docked toolbar to that edge;
- if you drop it anywhere away from the edge, then it becomes a floating toolbar;
- if you position the mouse pointer on the edge of any floating toolbar (so that it changes to double-headed arrow), then you can hold down the left mouse button and “drag” the toolbar to a different shape;
- to bring up other toolbars that are not already on the display, pull down the View menu and select Toolbars ... you will get the *dialogue box* illustrated on the left ... click the check box next to the Toolbar that you would like to open. The buttons in this dialogue also allow you to customise the toolbars, but you are left to experiment with that yourself.
- if you position the mouse pointer over any toolbar button and pause for a second or two, then a pop-up box identifies that button!



Toolbars dialogue box

Before proceeding, it may be helpful to put everything back roughly where it appears in Figure 1!!

THE AUTOCAD COMMAND SYSTEM

The first thing to understand about AutoCAD is how the command system operates.

Issuing AutoCAD commands ...

Everything in AutoCAD is achieved by issuing a *command*. Generally, commands are entered by clicking a *toolbar button* or selecting the command from a *pull-down menu*. In rare cases, a command has to be typed in at the *Command* prompt (where there is no equivalent toolbar or menu option). In some cases, a particular command can be entered in any one of those three ways depending on your personal preference. To complicate the matter further, some commands can be typed in with an abbreviated *alias*, which is often the quickest way to get the command started.

AutoCAD Prompts ...

Once a command has been issued, AutoCAD displays a prompt (or instruction) on the command line indicating your options for proceeding with that command.

You should get into the habit of watching those prompts very carefully so that you know where you are at any time.

The quickest way to learn is to have a go, and try to make sense of it yourself!!

To see how the command process operates, watch the command line as you click on any toolbar button. You will see the corresponding command printed at the command prompt and then, as AutoCAD responds to the command, it will display some kind of instruction or prompt (on the same line) and then wait for you to respond. Remember to press the <Esc> key if you wish to cancel the command (unless you want to carry it through to see what happens).

A note about interpreting AutoCAD's command prompts ...

The Command Line Prompt always follows the same "pattern". AutoCAD first tells you what it expects you to do ("enter a point", "type in a value", etc). If there are other alternative actions you can take as part of that command sequence, then those are listed next within square brackets, each separated by a slash character. You can select any of those alternative options by typing the capitalised letters only of the option. At the end of the prompt line, if appropriate, AutoCAD displays the default answer to the prompt within angle brackets. If you simply press the *Enter* key, AutoCAD will use that default value.

To develop your understanding of this process further, choose any of the drawing tools, click the button while watching the command prompt, and then pick points in the drawing area in response to the prompts. *You should be able to draw things with very little further explanation!*

WORKING WITH THESE TUTORIALS

The left margin in this tutorial will be used to add commentary to the text, identify the key topics being discussed in particular paragraphs (to enable scan reading and facilitate finding things later) and to include small illustrations where appropriate.

The purpose of this first tutorial is to enable you to become familiar with AutoCAD so that you can begin your first drawing. Please note a couple of pieces of advice regarding the use of these tutorials:

- *when following the instructions in this tutorial, be certain to observe the prompts on the command line ... this will help you develop a better understanding of the logic that underlies this system;*
- *these tutorials deliberately inter-mix explanations, instructions and comments so that you don't slip into a pattern of simply following instructions like a robot - it is important that you patiently read everything as you work through the material, endeavouring to understand each step of the process - don't treat the tutorials simply as "recipes for drawing"!*

The following table shows the conventions that are used in these notes to aid you in following the steps necessary to use AutoCAD. **It is important that you understand these so that you know how to interact with AutoCAD.**



Line Tool in the Draw Toolbar



An example of a Tool Button that has a Flyout Toolbar associated with it ...

[Draw.Line]	When I want you to select a tool from a toolbar, I will enclose its name in square brackets. The Toolbar name is given first, followed by the Tool name (separated by a full-stop). Where it seems appropriate, a graphic in the left margin will be used to help identify the Tool button. Note: The process of finding the correct Tool button is complicated by the fact that some <i>Tools</i> (particularly on the Standard Toolbar) are grouped into sets of related <i>Tools</i> known as <i>Flyouts</i> and are only visible if the <i>Flyout</i> is displayed. An arrow in the lower-right corner of the button (as in the example on the left) identifies a Tool that has a <i>Flyout</i> attached to it. Display the <i>Flyout</i> by holding the left mouse button down while pointing to the Tool button.
<u>F</u> ile > <u>N</u> ew	This form is used to identify things that are to be picked from the <i>pull-down menus</i> . The ">" symbol means "and then pick", so the example could be read "pick the <i>F</i> ile (pull-

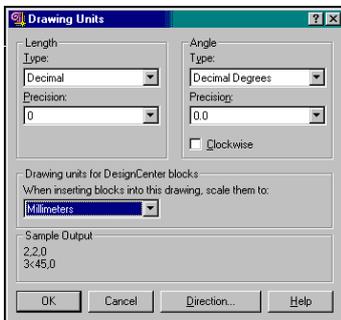
	down menu) and then pick the <u>New</u> option". <i>Note that the underlined characters show the keyboard short-cut for that menu selection, in this case <Alt-F-N> (hold down the Alt key while pressing F and then N).</i>
circle ↵	Bolded type is used to indicate entries that are to be <i>typed on the keyboard in response to an AutoCAD prompt</i> . These may be one character or several characters. The symbol "↵" represents either the <Enter> key or the <Space Bar>, since both are considered equivalent in AutoCAD and perform exactly the same function (that is, <i>they indicate the completion of a user action</i>).
<F2>	Bolded items in angle brackets represent <i>keyboard keys</i> that are to be pressed once in order to invoke a particular function in AutoCAD - in this example you would press the keyboard key labelled "F2".

AutoCAD allows us to model design objects in their real-world size in XYZ coordinate space. This is referred to as **Model Space**. We can also work in one or more layout spaces where we arrange scaled views of the design object on a model of a sheet of fixed size paper. This is often referred to as **Paper Space**. In both cases, we are working with scaled views on the screen, in one case of the design object itself, in the other, of the paper-based drawing we eventually plan to plot.

PREPARING TO DRAW WITH AUTOCAD

When working with CAD we always operate with the real-world sizes of the objects we are designing. In Australian building practice, we use millimetres as our basic unit of measure to describe architectural dimensions. That means that we typically deal with large whole numbers with no decimal places. For this tutorial, we will define a drawing size to suit a "model world" which is 20,000 x 15,000 millimetres, being large enough to model a fairly modest-size building. The choice of model world size is entirely arbitrary: its purpose, as you will see, is merely to define an area within which to work.

To set up these parameters, proceed as described below. Note that the steps that you should take are given on the left (following the notations given in the previous section), with brief explanations where appropriate on the right.



Format > U <u>Units</u>	calls up the <i>Units Control</i> dialogue box (illustrated) - select <i>Deci</i> mal units with 0 precision and <i>Deci</i> mal Degrees with 0.0 precision and then click <i>OK</i> .
Format > D <u>rawing</u> L <u>imits</u>	executes the <i>Limits</i> command - this is the command that defines the drawing size, in this case, in model space - notice that AutoCAD prompts you, on the <i>Command</i> line, to specify the lower-left corner of drawing limits.
↵	to accept the suggested value of (0,0) as the coordinates of the lower-left corner of the drawing area - <i>note that AutoCAD always displays suggested values in angle brackets</i> .
20000,15000 ↵	to specify the upper-right corner coordinates - this represents the largest drawing area that we expect to need for this drawing - we can always change it later if necessary.

This paragraph highlights AutoCAD's flexibility in working with different drawing units.

Having set the drawing limits, we now need to instruct AutoCAD to display the whole of that area on the screen. At the present moment, it is only displaying a small portion of the drawing area (about 420 x 297 units out of 20000 x 15000!) because that is the default metric drawing size when AutoCAD is shipped. You ought to check that by moving the mouse pointer around the drawing area, noting the coordinate position displayed in the status area. To display the whole sheet, proceed as follows:



Zoom All Tool button

[Standard. Zoom All]	issue the command <i>Zoom</i> with the option <i>All</i> - check out the Command Window to see the way AutoCAD prompts you as you enter this command and then enters the response by itself!! <i>Button Tools can be programmed to execute several commands, including all the necessary responses in order to perform quite complex operations.</i>
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DRAWING WITH AUTOCAD

All drawing operations performed with AutoCAD are initiated by issuing commands through the Toolbars. The first aim of this tutorial is to make you familiar with the process of drawing by introducing the basic drawing and editing commands.

Creating New Graphics Entities

Most of the commands to create new graphic entities are in the *Draw Toolbar*. Similarly, the commands to manipulate and change those graphic entities are in the *Modify Toolbar*.

Remember that all these commands can be executed by typing the command at the Command: prompt, however, we will generally use the mouse to choose the command that we wish to use from the Toolbar or the pull-down menus.

Helpful Hint: *One last little reminder that you will find most useful! Very often with AutoCAD, you will begin to execute a command and then decide that you need to cancel that operation. To cancel any command and get back to the "Command:" prompt, simply press the <Esc> key on the top-left of the keyboard.*

The following sections describe generally how to use each of the principal drawing commands. You should work through each of these, trying to understand the process more than the specific detail of each command.



The Line Tool

[Draw.Line]	issue the <i>Line</i> command using the <i>Draw Toolbar</i> .
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Note the prompt that appears on the command line. Now if you click any point on the drawing, then the beginning of the line is fixed at that point ... after that, as you move the mouse around the line stretches with the crosshair cursor ... this is referred to as *rubber-banding*. Construct a few lines anywhere you wish, then press <Enter> to terminate the command.

Rays & Construction Lines



Construction Line tool

AutoCAD has two other types of line that are used when setting up drawings: *Construction Lines* start at a notional "midpoint" and extend infinitely in both directions (available on the Draw Toolbar); while *Rays* have a fixed start position and extend infinitely in only one direction (this command is available through the pull-down menu ... *Draw > Ray*). Both these are designed to assist drawing construction, and would not normally be plotted as part of the finished drawing (though they can be!). Experiment with both these types of lines. *We will be able to make use of these when we get on to a serious drawing task in a later tutorial!*



Circle Tool ...

[Draw.Circle]	Issue the <i>Circle</i> command and construct a few circles experimenting with the <i>options</i> on the command line ... the <i>options</i> are separated by a slash (/) character ... you can choose an option by typing the capitalised letters at the start of each option ...
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Notice that there are several ways of constructing circles: by specifying the centre point and radius; by specifying the endpoints of a diameter; by picking

any three points on the circumference; etc. If you select the *Draw > Circle* command from the pull-down menus, then AutoCAD will automatically pre-select options before you begin the command. *Try that out to see how many different ways you can construct a circle.*



Rectangle tool



Arc tool



Ellipse tool



Regular Polygon tool

There are several other tools for drawing primitive geometric entities, including *rectangles, arcs, ellipses* and *regular polygons*. You should experiment with each one by watching the prompts and trying to discern the range of options that are available for the construction of each one. By the time you've done that, you will have a nice cluttered drawing!

Useful shortcut to remember: whenever the "Command:" prompt is on the Command Line, then you can recall the last command simply by pressing the **<Enter>** key (or the **<Space Bar>** or the **right mouse button**) ... this feature is very useful when you are doing the same operation over and over again ... try it with some of the drawing commands.

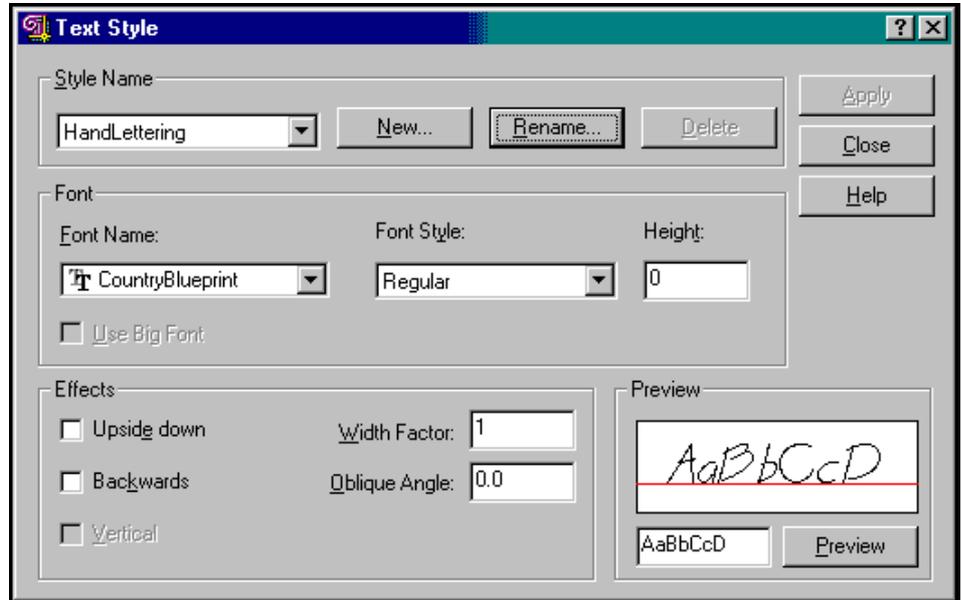
AutoCAD Text facilities ...

The last graphic primitive that I want to introduce in this tutorial is text. AutoCAD provides a very powerful set of tools to produce text on drawings. The first step always is to create a text style to be used. A style is defined by a font (which can be either a special AutoCAD font or any Windows font loaded on the computer you are using), combined with other properties like mirroring or turning the text upside down, etc. You can create quite dramatic effects with AutoCAD text.

Select **Format > Text Style** from the pull-down menus to open the dialogue shown below.

Text Style Dialogue box ...

Click the **New** button to create a new style. I called mine "HandLettering" and chose a Windows font that has a hand lettering style to it. You can try setting some of the other effects if you wish. Note that by setting the height to 0 (zero), the actual height of the text can be set when the text is created.



Once you have created a style, click **Apply** to make it the "current style" and then close the dialogue box. I want to introduce two basic commands that are used to create new text entities.

Single Line Text ...

Draw > Text > Single Line Text

this command is only available through the pull-down menus, and allows you to enter one or more single lines of text ... each line is treated as a separate entity ... notice on the *Command Line* that you are prompted to enter a start point ... pick a point somewhere in the drawing area (to

	mark the desired bottom left corner of the first character in the text string) ... you will then be asked for the text height ...
500 ↵	this specifies the text height (note that at a plot scale of 1:100, this text would be 5 mm high).
↵	by simply pressing <Enter>, you are accepting the offered value of 0 for the text angle ... AutoCAD generally offers "default" values for most requested input ... the default value is shown in angle brackets on the <i>Command Line</i> .

The Command Line prompt should now read "Enter Text:", indicating that you should enter the text that you wish to appear on your drawing. A text box will also appear at your specified starting point. You can enter as many lines of text as you wish, each terminated with <Enter>. To stop the process, simply press <Enter> twice (the second time, at the Enter Text: prompt, effectively terminates the command).

There is an important lesson in this paragraph ...

Try using the Single Line Text command again (just by pressing ↵ to recall the command), but this time, instead of typing the text height as a number, use the mouse pointer to pick two points on the screen that "shows" the text height that you want. *In general, AutoCAD will always allow you to "point" to a distance (by picking two points) just as it also lets you type in a coordinate pair (x and y values) instead of picking a point on the screen.*



Text tool on the Draw Toolbar

[Draw.Text]	this command allows you to enter several lines of text inside a dialogue box ... AutoCAD will prompt you (on the Command Line) to pick two points that form the opposite corners of a rectangle that defines the area in which the text is to be positioned ... the <i>Edit MText</i> dialogue will then appear.
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Enter a few lines of text in the dialogue box and then click *OK*. Notice that the text wraps automatically in the dialogue box to fit within the width of the rectangle that you defined. You may notice also that if there is too much text to fit in the rectangle, then AutoCAD simply allows it to spill over the lower boundary of your rectangle. Once the text is on the drawing, it is treated as a single entity by AutoCAD.

Manipulating the View in the Drawing Area

Before looking at a couple of the AutoCAD editing commands, we will take a moment to learn about some of the techniques available to control the displayed view of your drawing within the drawing area.

Introducing the Zoom Command

The first of these is the *Zoom* command. This allows you to change the scale of the view on the screen so that you can focus in on smaller details or expand out to see a larger area of the drawing. Try the following variations on the *Zoom* command:



Zoom Window tool

[Standard. Zoom Window]	click this button and then point to two locations on your drawing that mark opposite corners of a rectangle ... everything within that rectangle will then be re-drawn to fill the whole drawing area ... notice that if you hold down the left mouse button over this tool, then you get the <i>Zoom Flyout</i> ... while holding the left button down, pause over each button in the flyout and read what it does ... <i>the second-last one in the flyout is the most useful of these: it zooms out so that you can see everything in your drawing.</i>
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Real-Time Zoom Tool

[Standard. Real Time Zoom]	This tool allows you to dynamically change the size of the drawing on the screen ... click the button once and then move the mouse into the centre of the drawing area ... now, while holding down the left button, move the mouse up and down the screen and observe the effect ... press <Esc> to cancel real time zoom.
[Standard. Zoom Previous]	this tool returns you to the previous view of the drawing ... you can use it repeatedly to switch through up to 10 previous views.



Zoom Previous Tool

The Pan command ...

The second way of changing the view is to use the *Pan* command. This allows you slide your view across the drawing while keeping the display scale constant. Once understood, it is a very useful command ... try experimenting with it as follows:



Real Time Pan tool

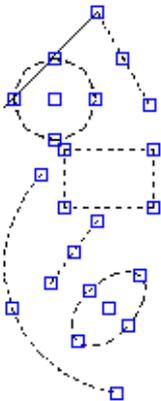
[Standard. Real Time Pan]	this tool is used to move across your drawing ... click the tool once and then move the mouse pointer into the drawing area ... again, hold down the left button and drag the drawing across the drawing area ... press <Esc> to cancel real time pan.
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Note: When using either of the *Real Time Zoom* or *Pan* tools, you can click the right mouse button to pop up a menu that allows you to switch between the tools or to exit from whichever one is active at the time.

Scroll Bars ...

The third way to move around a drawing, of course, is to use the Scroll Bars on the right and bottom edge of the Drawing Window. These operate like any Windows application.

Editing with Entity Handles ...



Modifying and Changing Graphic Entities

The simplest and most intuitive way to edit entities in a drawing is to make use of *entity handles*. If you position the crosshair cursor so that the small square at the intersection of the crosshairs is over one of the entities in your drawing, and then click the left mouse button, that entity will become highlighted with small box "handles" positioned at its key points. This is illustrated in the margin for a number of different entity types. Once selected in that fashion, you can use the mouse to "pick up" one of the handles (by positioning the mouse over the handle and holding down the left mouse button), and then drag that handle to a new location in order to reshape the entity. You can also pick up a selected entity and drag it to a new location by picking up the appropriate handle. The best way to understand these techniques is to experiment with your drawing until you feel that you have begun to master the technique. You should also look up the Help facility to gain further insights into how to manipulate entity handles.

The Editing Commands ...

A more controlled approach to editing is to use one of the editing tools supported by AutoCAD: we will look at *erase*, *move* and *copy*. In each case, you need to *select* the entities upon which you want the editing command to operate. If you have already selected one or more entities (in the manner described in the previous paragraph), then those entities are assumed to be the ones upon which the command is to operate. If there are no entities "currently selected", then the command will prompt you to select some objects before it proceeds with the editing process. To understand these processes, work through the following steps making random selections of objects where required:



Erase tool



Copy tool

<Esc> <Esc>	pressing the cancel key twice will ensure that any entities that are currently selected in your drawing (with entity handles visible) will be de-selected ...
[Modify.Erase]	once this command has been issued, note that AutoCAD prompts you to on the Command Line to "Select Objects:" ... pick a few objects to delete and then press ↵ to complete the command ... the selected objects will disappear!.
[Modify.Copy]	again, when prompted, pick a few objects to copy and then press ↵ ... this time AutoCAD prompts for a "Base Point:" followed by an "End Point:" ... pick any two points on your drawing that indicate a <i>displacement vector</i> ... the selected objects will be moved over a distance and direction that matches the displacement vector defined by your two picks.

After the Copy Command has completed, try pre-selecting a couple of entities before issuing the Move command as follows:



Move tool

[Modify.Move]	this command operates exactly like the Copy command, except that the selected objects are moved rather than duplicated ... notice that there is no prompt to select the objects because they were pre-selected in this case!
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Rotate, Scale & Mirror tools.

You should now spend some time experimenting with these and the other Modify tools, including: *scale*, *mirror* and *rotate*. The other modify tools probably need further explanation, but you can try them if you wish!

Using the Right Mouse Button ...

One last comment on editing! If you select any object in your drawing and then click the *Right Mouse Button*, you will get a pop-up menu giving you several options for manipulating that object. Many of those options will be specific to type of entity that you have selected. For example, the two types of text entity will each have an option to allow you to edit the text in that entity. You are also able to bring up a properties box in which you can edit many of the specific properties of that entity. You should experiment with that technique as it will be useful in all sorts of circumstances!!

Finishing the Tutorial ...

When you have finished experimenting, simply pick *File > Exit* in order to finish this Session. Because you have worked on a drawing without storing that information on a file on the system, AutoCAD will ask if you wish to save the changes. In this case, simply click "No" (because there is no need to "save" the work that you have done in this tutorial). Normally, of course, you would want to save the changes that you have made: the *S*ave and *S*ave *A*s options in the *F*ile menu support those operations.