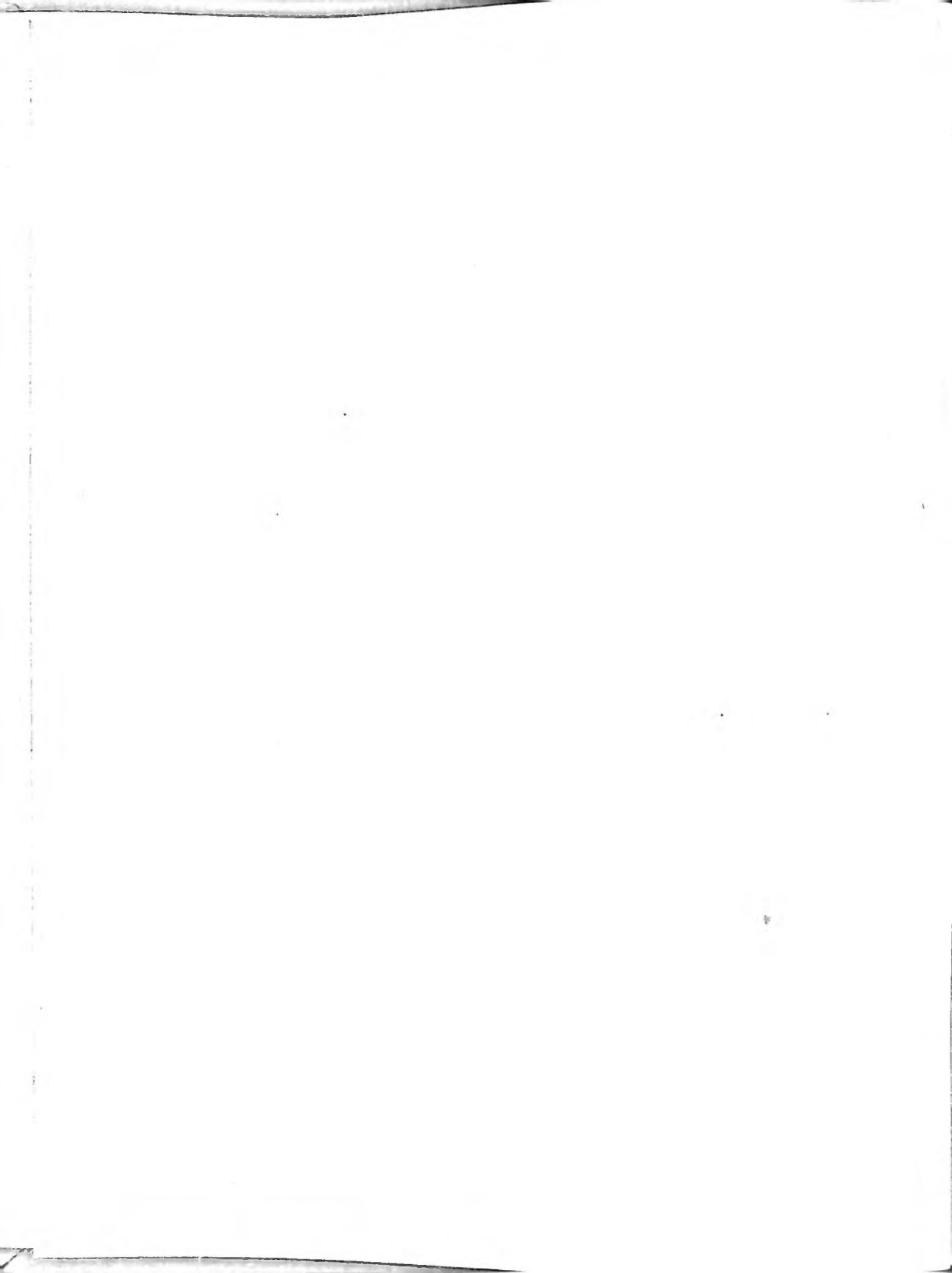


AUTOCADTM USER REFERENCE



AUTODESK, INC.



The AutoCAD™ Drafting Package

User Reference

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Preface

The AutoCAD™ drafting package is a Computer Aided Drafting application for your microcomputer. CAD applications are tremendously powerful tools. The speed and ease with which a drawing can be prepared and modified using a computer offers a phenomenal timesaving advantage over "hand" preparation. AutoCAD brings this sophisticated technology, previously available only on large and costly systems, to the microcomputer user.

There is virtually no limit to the kinds of line drawings you can prepare using AutoCAD. If it can be created by hand, it can be generated by computer. Here are a few possibilities:

- o Architectural drawings of all kinds
- o Interior design and facility planning
- o Work-flow charts and organizational diagrams
- o Graphs of any sort
- o Drawings for electronic, chemical, civil, and mechanical engineering applications
- o Plots and other representations of mathematic and scientific functions
- o Line drawings for the fine arts

Version 2.1 is the latest enhanced edition of AutoCAD. It is available for a variety of microcomputers based on the 8086 family of microprocessors, and runs under the PC-DOS and MS-DOS operating systems, Version 2.0 or later.

Whether you are a professional or a home computer user, you can easily learn to use AutoCAD for your drafting. No technical computer knowledge is required to use AutoCAD effectively; practice and a thorough understanding of its features are the keys to proficiency.

This guide presents the full set of features offered by AutoCAD. However, some features described herein are provided only with the optional Advanced Drafting Extensions packages (ADE-1, ADE-2, and ADE-3). Features included in one of these packages are noted as such in the text or marked with a "+1", "+2", or "+3", respectively. The higher ADE packages require the lower ones to be present, so a feature marked with "+1" is also present in ADE-2 and ADE-3 versions of AutoCAD.

Some features differ depending on the characteristics of the computer system or display device being used. Consult the accompanying AutoCAD Installation Guide for your computer for exceptions, additions, and other machine-dependent information.

We endeavor to keep this manual as up-to-date as possible. However, we suggest that you list the file README.DOC on your release disk for any last-minute additions or corrections.

Another publication provided with AutoCAD is the AutoCAD Standard Menu User Guide. It describes use of AutoCAD's comprehensive command menu.

The last manual is the AutoLISP Programmer Reference. It describes AutoCAD's programming language, and extension of the "variables and expressions" feature of the ADE-3 package.

Thus, the complete documentation set for AutoCAD consists of the following:

- o AutoCAD User Guide (this manual)
- o AutoCAD Installation Guide
- o AutoCAD Standard Menu User Guide
- o AutoLISP Programmer Reference
- o The README.DOC file on the AutoCAD release disk

In addition, each of the optional symbol libraries available for use with AutoCAD comes with a user guide for that library.

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Chapter 1

INTRODUCTION TO AutoCAD

The AutoCAD™ drafting package is a powerful drawing aid. It follows your instructions to produce the exact drawing you want quickly. It offers features that let you correct drawing errors easily and even make large revisions without redoing an entire drawing. It produces clean, precise final drawings. AutoCAD works for you. It doesn't put anything into your drawing "on its own". A completed AutoCAD drawing looks virtually identical to the same drawing carefully prepared by hand. ("Virtually", because AutoCAD, when used with the proper equipment, can greatly improve accuracy.) Your drawing is configured exactly as you specify, with every element appearing just where you want it.

Sample drawings have been included in this chapter and on the chapter dividers to demonstrate AutoCAD's wide range of uses. All illustrations in this manual were created using AutoCAD.

1.1 Overview

The following is a brief overview of AutoCAD usage. Complete operating instructions begin in Chapter 2.

A *graphics monitor* is used to display your drawing. You immediately see the effect of every change you make on this monitor.

AutoCAD provides a set of *entities* for use in constructing your drawing. An entity is a drawing element such as a line, circle, text string, etc. You enter *commands* to tell AutoCAD which entity to draw. Commands can be typed on the keyboard, selected from a screen menu, or entered with a push of a button from a menu on a digitizing tablet or from a multi-button pointing device. Then, responding to prompts on the display screen, you supply certain parameters for the chosen entity. These parameters always include the point in the drawing where you want the entity to appear; sometimes a size or rotation angle is also required. After you supply this information, the entity is drawn and appears on the graphics monitor. You can then enter a new command to draw another entity or to perform another AutoCAD function.

Other AutoCAD functions let you modify the drawing in a variety of ways. Entities can be erased, moved, or copied to form repeated patterns. You can change the view of the drawing displayed on the graphics monitor or display information about the drawing. AutoCAD also provides drawing aids that help you position entities accurately. When you want a paper copy of your drawing, you can *plot* it on a pen plotter or printer plotter. A simple command format allows you to accomplish all these functions easily. In some cases all you have to do is enter a command, and the function is performed immediately. The most that's required is that you follow the command with some minimal, easily entered specifications. AutoCAD prompts you, indicating the type of information needed.

The Advanced Drafting Extensions packages (ADE-1, ADE-2, and ADE-3) are available as AutoCAD options. These provide additional commands and capabilities, brief descriptions of which are included at the end of this chapter. This manual includes complete details of all three optional packages.

The remainder of Chapter 1 introduces important concepts and terminology that you must understand in order to make effective use of AutoCAD. Chapter 2 describes general AutoCAD operating procedures and the various techniques for entering commands and data; Section 2.6 provides a summary of all AutoCAD commands. Detailed command descriptions begin in Chapter 3.

1.2 Equipment Requirements

In addition to a basic computer system, (including processor, keyboard, text display screen, and disk drives), AutoCAD requires a graphics monitor capable of reasonably high resolution. A plotter (or a printer with graphics capability) can be connected to the system to produce a "hard copy" of a drawing.

Graphic Input (Pointing) Devices

We suggest you add a *pointing device* such as a mouse, digitizing tablet, or TouchPen™. Any one of these devices provides the means for instant command and point entry. Keyboard entry is relatively easy, but pointing at the screen and pushing a button is even easier. In addition to locating points and entering commands, you can use a digitizing tablet to "copy" existing drawings. Descriptions of the types of devices available are given below; additional information can be found in Chapter 2 and in Chapter 12. Not all pointing devices are available on all systems. Supported configurations for your computer and instructions for installing AutoCAD on it can be found in the accompanying AutoCAD Installation Guide / User Guide Supplement.

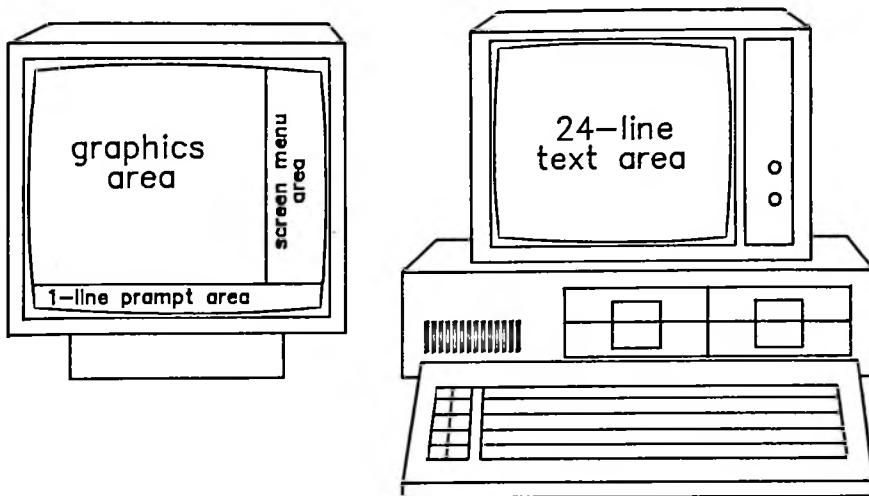
Mouse As you move a *mouse* around the tabletop, crosshairs track its movement on the screen. To select the point or menu item at which the crosshairs are positioned, push the button on the mouse. If the mouse has multiple buttons, you can also specify commands directly using the extra buttons (see Section 2.7).

Tablet Point and menu item selection using a digitizing tablet are similar to the mouse operation described above. However, the tablet's stylus is moved around only on the tablet's surface. The tablet offers two additional capabilities beyond those of the other pointing devices; you can align the tablet with the coordinate system of an existing paper drawing so that you can use AutoCAD to produce an exact copy of it, and you can set aside up to four areas of the tablet for use with *tablet menus* (see Chapter 12).

TouchPen™ With a TouchPen, you can point directly at an area of the graphics monitor's screen to enter points or to select commands from a *screen menu*. Crosshairs appear on the screen and follow the pen until you select a point or menu item by pressing the button on the pen.

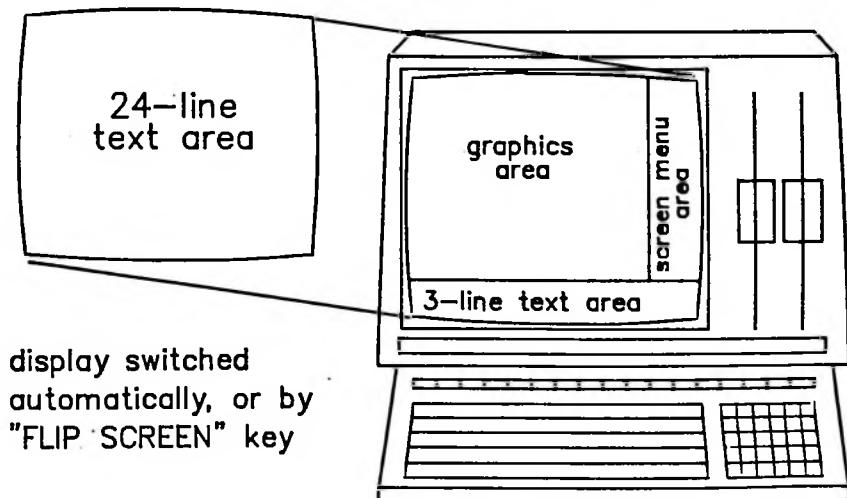
Display Monitor

On some computers, AutoCAD uses two display monitors; one for command prompts and text output, and the other for graphics. On these systems, the graphics monitor can also display a screen menu along its right edge and a one-line prompt area across the bottom, as in the following illustration. See Section 1.4.1 for a discussion of the screen menu.



TYPICAL DUAL-SCREEN CONFIGURATION

On other systems, a single monitor is used for both graphics and text purposes. Here, three lines at the bottom of the screen are reserved for command entry and prompts, and the right edge can contain a screen menu. When AutoCAD is run on such a single-monitor system, it remembers a full 24 lines (or more) of text, just like the regular text display. If information has scrolled off the three-line display, you can use a function key to "flip" to the text display and review it. AutoCAD automatically switches to the text display when it outputs a large amount of information. It automatically returns to the graphic display when it draws anything. A single-screen system is illustrated below.



TYPICAL SINGLE-SCREEN CONFIGURATION

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For users with the ADE-1 package, the graphics display can include a mode/coordinate status line, showing the current state of various AutoCAD mode switches and the location of the screen crosshairs (see Chapter 8). For some display devices, users with ADE-2 can also choose whether or not to display the screen menu and the text prompt area on the monitor (see Appendix D).

1.3 Concepts and Terminology

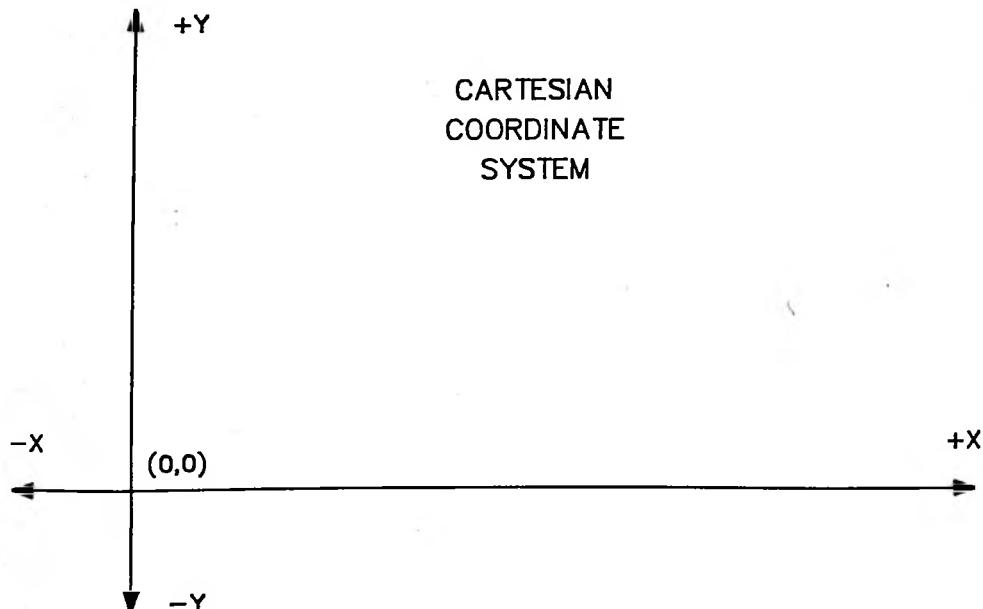
This section presents some special terms and concepts you'll encounter in this manual and while working with AutoCAD. These items are best understood by working with the program. Read this discussion now, then refer back to it if questions arise while using the program.

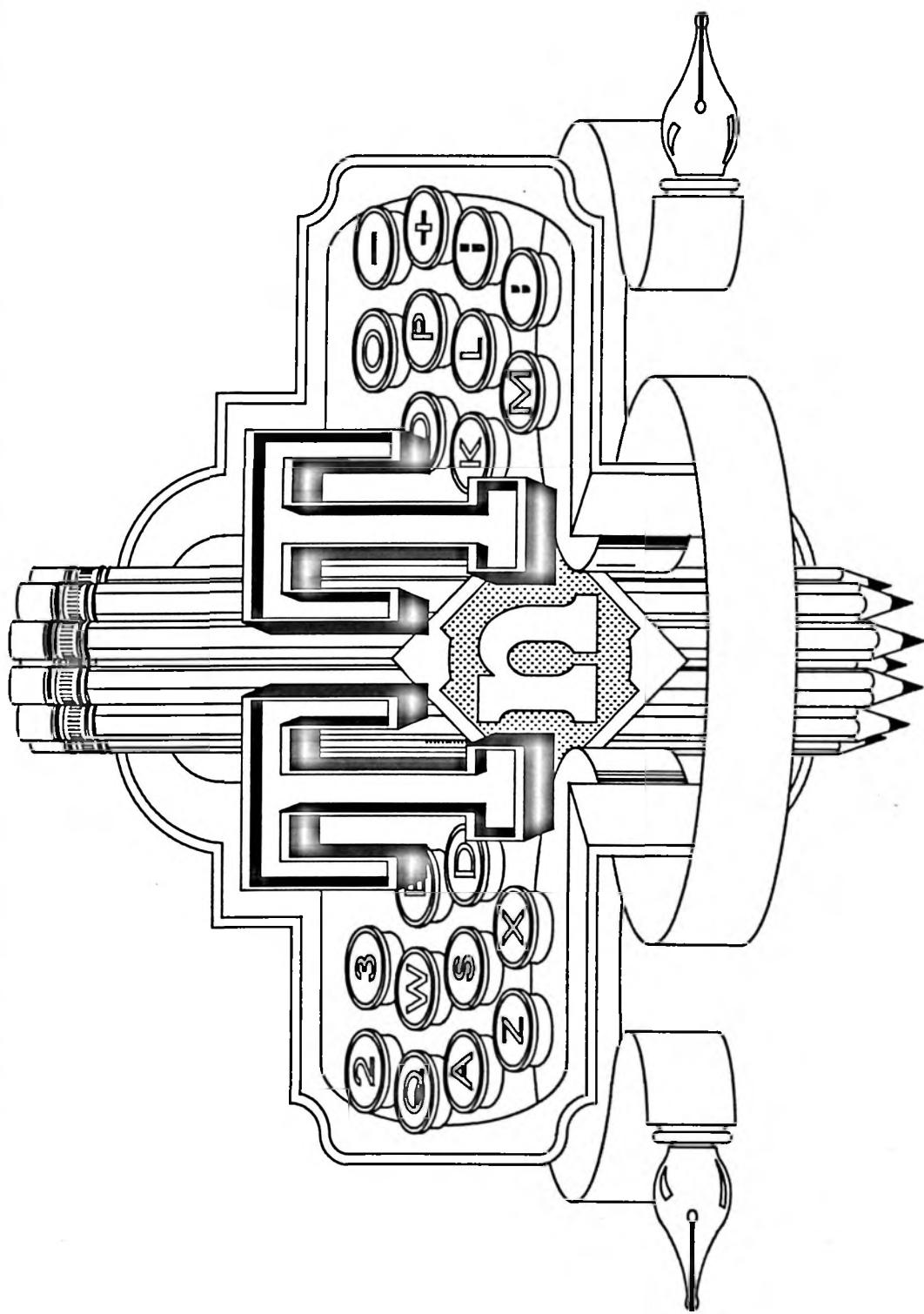
AutoCAD Drawing

An AutoCAD drawing is a file of information that describes a graphic image. It can be any size you desire, specified by any unit of measurement you want, and it corresponds exactly to a drawing prepared on paper. That is, *entities* in the drawing (elements such as lines, circles, text, etc.) are positioned in the drawing file exactly where they would be on paper.

Coordinates

A Cartesian coordinate system is used for locating points in the drawing; to position entities, for instance. An *X* coordinate specifies horizontal location and a *Y* coordinate specifies vertical location. Thus any point on the drawing can be indicated by an *X* and *Y* coordinate pair of the form (x,y) . The $(0,0)$ point is normally at the lower-left corner of the drawing. The figure below shows a Cartesian coordinate system.





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Drawing Units

As noted, entities in the drawing are positioned on coordinate points. For example, you draw a line by specifying the coordinates of its two endpoints. The distance between two points is measured in *units*. Thus, a line drawn between the points (1,1) and (1,2) is one unit in length.

A unit can correspond to whatever form of measurement your drawing requires. It can be inches, feet, centimeters, angstroms, whatever. When the drawing is plotted you can specify a scale factor to plot each unit the exact size you want.

Display

The term *display* is used in two ways. In this manual, *display* usually refers to the portion of the drawing that is currently being displayed. Occasionally, it is used to indicate the graphics monitor screen upon which your drawing is shown.

Zooming and Panning

The display can be *zoomed* in or out to magnify or shrink the visible image of the drawing. When the display is zoomed out, you can see a large portion of the drawing; zooming in can "blow up" a small portion of the drawing and display more of its details. You can zoom in to draw intricate parts of your drawing with exacting detail and then "back off" to look at the finished drawing. AutoCAD's "zoom ratio" is about ten trillion to one, more than adequate for most applications.

The graphics monitor is used as a *window* through which you can look at all or part of your drawing. Keep in mind that coordinates refer to fixed locations in the drawing, not to the physical location on the display screen. This means that the absolute size of a unit always remains constant, (e.g., the points (1,1) and (1,2) are always one unit apart), but the apparent distance between points on the screen varies with different zoom levels. When the drawing is zoomed out, the distance between coordinates appears to be small. (The line drawn between (1,1) and (1,2) may be only a quarter inch long measured on the screen.) Alternatively, when the drawing is zoomed in, the distance between coordinates appears larger. (The one-unit line may appear to be several inches long.) In both cases, the absolute distance between the coordinates is constant. Only the screen display changes.

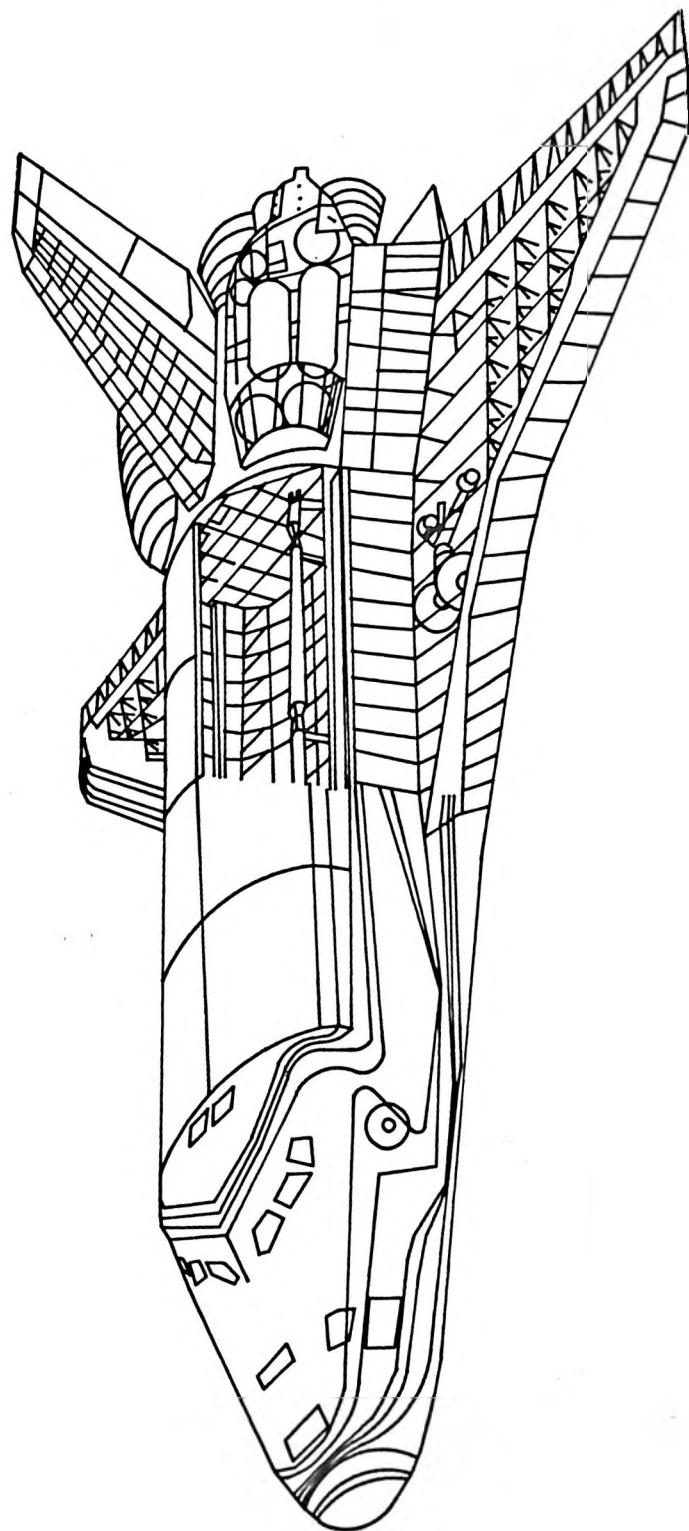
Similarly, you can *pan* across the drawing in any direction. Panning allows you to view a different portion of the drawing without changing its magnification.

Drawing Limits and Extents

AutoCAD assumes that you are drawing in a rectangular area. The *drawing limits* are the borders of this rectangle, in drawing coordinates. You can select whatever limits make sense for your drawing. For example, if you are drawing a printed circuit board that is 8 inches high by 10 inches wide, you can choose to consider a drawing unit to be one inch and the lower left corner of the board to have coordinate (0,0). You might then set your drawing limits to:

Lower left corner: (0,0)
Upper right corner: (10,8)

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If your drawing grows beyond your original plans or you find that the drawing limits are too restrictive, you can easily change the drawing limits. You can also turn AutoCAD's limits-checking off entirely.

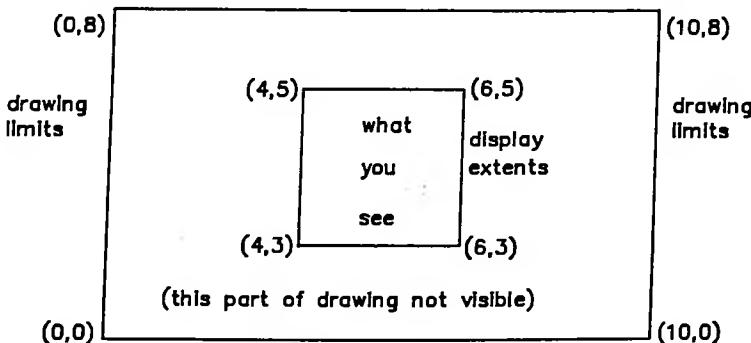
Another property of a drawing is how much of the area defined by the drawing limits currently contains information. Imagine a rectangle surrounding all the objects in your drawing; the smallest such rectangle defines the current *drawing extents*.

Display Extents

As described previously, panning permits the display window to be moved around the drawing, and zooming permits it to show large or tiny portions of the drawing by varying a magnification factor. AutoCAD keeps track of the current screen location by maintaining another set of borders, called the *display extents*. These are the borders of the *current display*, in drawing coordinates. For example, to show a magnified view of the center of an 8 inch by 10 inch printed circuit board, the display extents might be:

Lower left corner: (4,3)
Upper right corner: (6,5)

This is shown pictorially below.



Zooming and panning change the values of the display extents. When this occurs, the drawing is regenerated to show only the portion of the drawing that falls within the new display extents.

Resolution

Physical resolution refers to the amount of detail that can be represented. This is determined by the equipment you use. For digitizing tablets and plotters, resolution is usually specified as "dots per inch". Digitizer resolution determines the accuracy with which you can indicate closely spaced points. Plotter resolution determines the smoothness and size accuracy of your plotted drawing. The resolution of your display device is specified as "dots X by dots Y". Higher resolution means a smoother looking display.

The physical resolution of a device only affects the work done on that device, not AutoCAD's internal resolution. For example, by zooming in to a small part of a drawing, a point can be

specified far more accurately than the display screen would normally allow. The accuracy of the final plotted output is limited only by the resolution of the plotter, not the resolution of the display screen.

You can *snap* (lock) coordinates you enter to the nearest point on an optionally visible grid. The spacing of the grid points is called the *snap resolution*. It is completely independent of the resolution of input or output devices. In printed circuit drafting, for example, it is common to make all points align on 0.1 inch centers. In that case, you can set AutoCAD to a snap resolution of 0.1. Snap resolution can be set to any "fineness" your drawing requires. It can be changed to any value at any time or turned off entirely for "free style" drawings.

1.4 AutoCAD Features

This section introduces the variety of features that AutoCAD provides. Please read this information carefully. Drawing speed increases with your knowledge of the program.

1.4.1 Program Operation

Main Menu

AutoCAD operates on two levels to reduce both the work required to generate a drawing and the time needed to learn the system. At the outer level, AutoCAD provides a menu-driven interface (the *Main Menu*) that allows you to initiate various tasks, such as creating new drawings, modifying stored drawings, and producing plots. The main menu is the first thing displayed on the screen when you begin executing AutoCAD and it is also the means by which you terminate an AutoCAD session. It provides access to various parts of AutoCAD, such as the interactive Drawing Editor and the plotter interface. This menu is distinct from the menus available from within the Drawing Editor, discussed below.

Interactive Drawing Editor

The *Drawing Editor* is to your drawing what a text editor is to a document. When you create a new drawing or edit an existing one, AutoCAD automatically loads the Drawing Editor. The Drawing Editor displays your drawing and provides commands to create, modify, view and plot drawings. When you have finished working with a given drawing, you can save or discard any changes you have made before returning to the Main Menu.

Database Storage

All information about your drawing, the size and position of every element, the size of the drawing itself, its display characteristics (e.g., whether you're zoomed in or out), etc. is automatically updated with each command. This information is stored in your drawing file when you exit AutoCAD.

Point and Command Entry

You can specify points in the drawing in a variety of ways. From the keyboard, you can designate points by typing in absolute coordinates or coordinates relative to the last point specified; or by keyboard pointing, which utilizes cursor control keys to move crosshairs around on the graphics monitor so you can visually position to the desired point. As

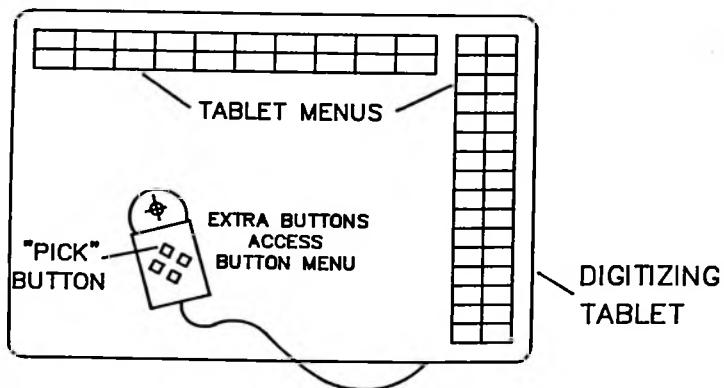
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described in Section 1.2, you can also use a graphic input device to designate points. Designated points can be locked (*snapped*) to a user-defined grid to ensure accuracy. If you have the ADE-2 package, you can also lock onto various features of existing entities, such as the midpoint of a line, or the center point of a circle.

You can also enter commands in any of several ways. You can type a command in directly or select a command from any of the menus described below. You can construct your own custom menus; see Section 2.7 and Appendix B.

- | | |
|-------------|--|
| Screen Menu | A menu can be displayed on the graphics monitor while the Drawing Editor is active. This menu allows command entry by simply pointing to the command on the display screen with a pointing device or with the keyboard. |
| Tablet Menu | Up to four menus of AutoCAD commands can be placed on the digitizing tablet, permitting a command to be entered by simply pointing to it with the stylus and pushing a button. |
| Button Menu | If your tablet stylus or mouse has multiple buttons, you can use the extra buttons to enter often-used commands. (See Section 2.7 of this manual and the AutoCAD Installation Guide / User Guide Supplement for your computer.) You can also use an auxiliary function box; such a device has buttons for command selection, but it cannot be used as a pointing device. |

The figure below illustrates tablet and button menus.

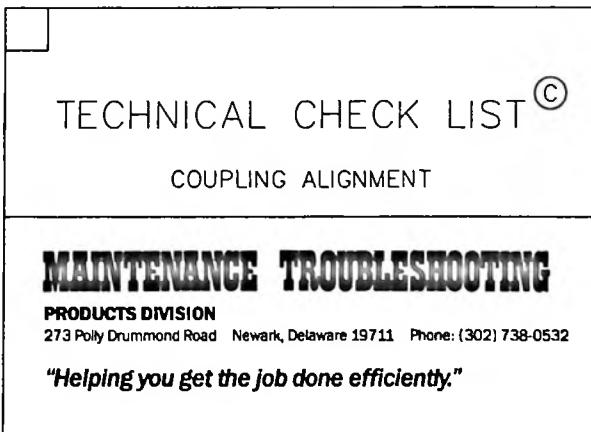
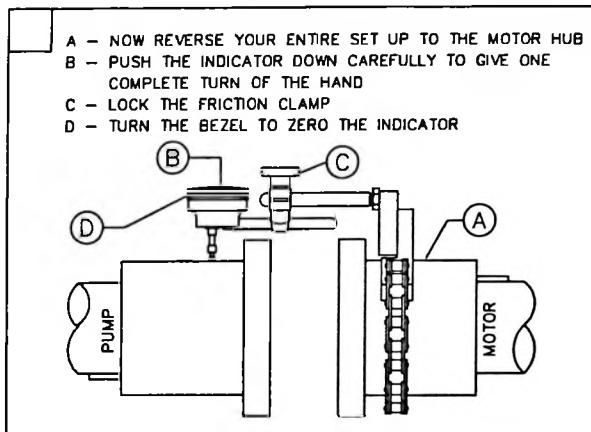
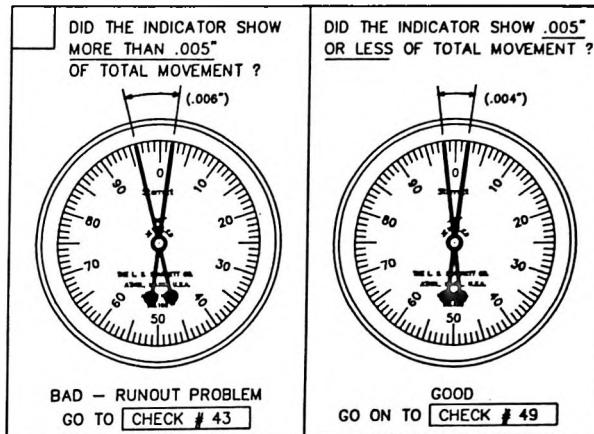


Plotting

You can plot a hardcopy of the drawing at any stage in its development. "Check-plots" can be generated while the drawing is in progress to check for positioning and dimensioning errors that might not be immediately apparent on screen. When the drawing is complete, the final plot is done to produce the finished drawing. Plot output may be sent to a pen plotter or a printer with graphics capability.

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1.4.2 Objects Within a Drawing

Objects within a drawing may be simple or complex. AutoCAD provides the ability to build a complex object from simple ones and then manipulate it as a unit. You can construct rectangular or circular *arrays* (patterns) of objects, and you can even *insert* entire drawings into the one you're currently working on. In addition, you can assign different portions of a drawing to different *layers*.

Entities

Entities are predefined elements you can put into a drawing with a single command. AutoCAD offers the following entity types:

Lines	Traces	Points	Circles	Attributes (+2)
Arcs	Text	Solids	Shapes	Polylines (+3)
Blocks				

Lines, arcs, and circles can be drawn with various dot-dash linetypes. You can draw text items in a variety of fonts, with any size, and oriented at any angle. In addition, you can create text *styles* to apply mirroring, obliquing, or a horizontal expansion or compression factor to the text characters.

Traces are solid lines of any width you specify. Shapes are small objects you can define outside AutoCAD and insert into the drawing at a specified point, with a specified scale.

Blocks are compound objects formed from groups of other objects. Attributes (+2) can attach variable text information to each instance of a Block. Polyline (+3) are connected line and arc segments, with optional dot-dash linetypes, width, and taper.

Drawing Insertion

This powerful feature lets you treat an existing AutoCAD drawing (stored on disk) as a Block, and insert it into the drawing that you are currently creating or modifying. Thus you can interactively construct a drawing *part*, store it in a regular AutoCAD drawing file, and then easily insert as many copies of it as you like in subsequent drawings. Using this mechanism, you can construct a custom library of symbols and components used often in your work.

Parts you create for insertion can contain any number and type of entities. Once inserted, a part is treated as one entity and can be moved or erased as a single unit. (See Chapter 9.)

Layers, Colors, and Linetypes

You can assign various portions of your drawing to different *layers*. You can define as many layers as you like. The layering concept is similar to the transparent overlays used in many drafting applications. Layering allows you to view and plot related aspects of a drawing separately or in any combination. For instance, a drawing file can contain the floor plan for a house on one layer, electrical wiring on a second layer, and plumbing on a third layer. You can plot (or display) the floor plan and wiring together and then re-plot the floor plan with the plumbing included as well.

A *color* and a *linetype* are associated with each drawing layer. The color is a number from 1 to 255 that selects the actual color in which items are drawn on the graphics monitor. A

linetype is a specific sequence of alternating line segments and spaces. Using these properties, you can draw attention to important details in your drawing, highlight recent changes, or visually indicate the relationships among entities. In printed circuit drawings, for example, you can assign a different color to the traces on each side or layer of the board. Similarly, in drawings of mechanical parts or architectural site plans, you can assign a special linetype to center lines or property boundaries.

Standard names have been assigned to the first few color numbers, but the actual colors displayed depend on the display device you use. (Consult your AutoCAD Installation Guide / User Guide Supplement.) For monochrome devices, the color number has no effect. Note, however, that use of color numbers makes sense even if your graphics monitor is monochrome; when sending output to a multi-pen plotter, you can assign each color number to a different pen.

Prototype Drawings

There are a number of sizes, limits, and modes that apply to a drawing and are saved with it in the drawing file. When you begin a new drawing, the initial values for these numbers can either be AutoCAD's defaults or they can come from a *prototype drawing*. A standard prototype drawing is supplied with AutoCAD. You can create any number of additional prototype drawings (one for each type of drawing your work requires, perhaps), in which you select the defaults you prefer. A prototype drawing can even contain a pre-drawn border or title box, pre-initialized layer names, a collection of custom text styles, or any other convention you generally use in a particular type of drawing. Then, when starting a new drawing, you can tell AutoCAD which prototype drawing to use for the initial drawing environment.

You can adjust the defaults set by a particular prototype drawing by editing that drawing. Simply use the normal AutoCAD commands to establish the desired limits, modes, etc., and save the updated version of the prototype drawing.

1.4.3 Auxiliary Features

Help Display

A *Help* display is available to remind you of command names and the options available for entering points and other data. Help on the format of specific commands is also available.

File Directory Access

You can list disk directories and delete, rename, or copy files without exiting AutoCAD.

Drawing Interchange Capability

You can save an AutoCAD drawing in the form of an ASCII text file that can be easily processed by user-written programs or transferred to a different computer. These *drawing interchange (DXF) files* can also be created by user-written programs for AutoCAD to turn back into drawing files. Translations between AutoCAD and other CAD systems' database formats, and special-purpose analysis and modification of AutoCAD drawings can be accomplished by means of this mechanism.

DOS 2.0 Pathname Support

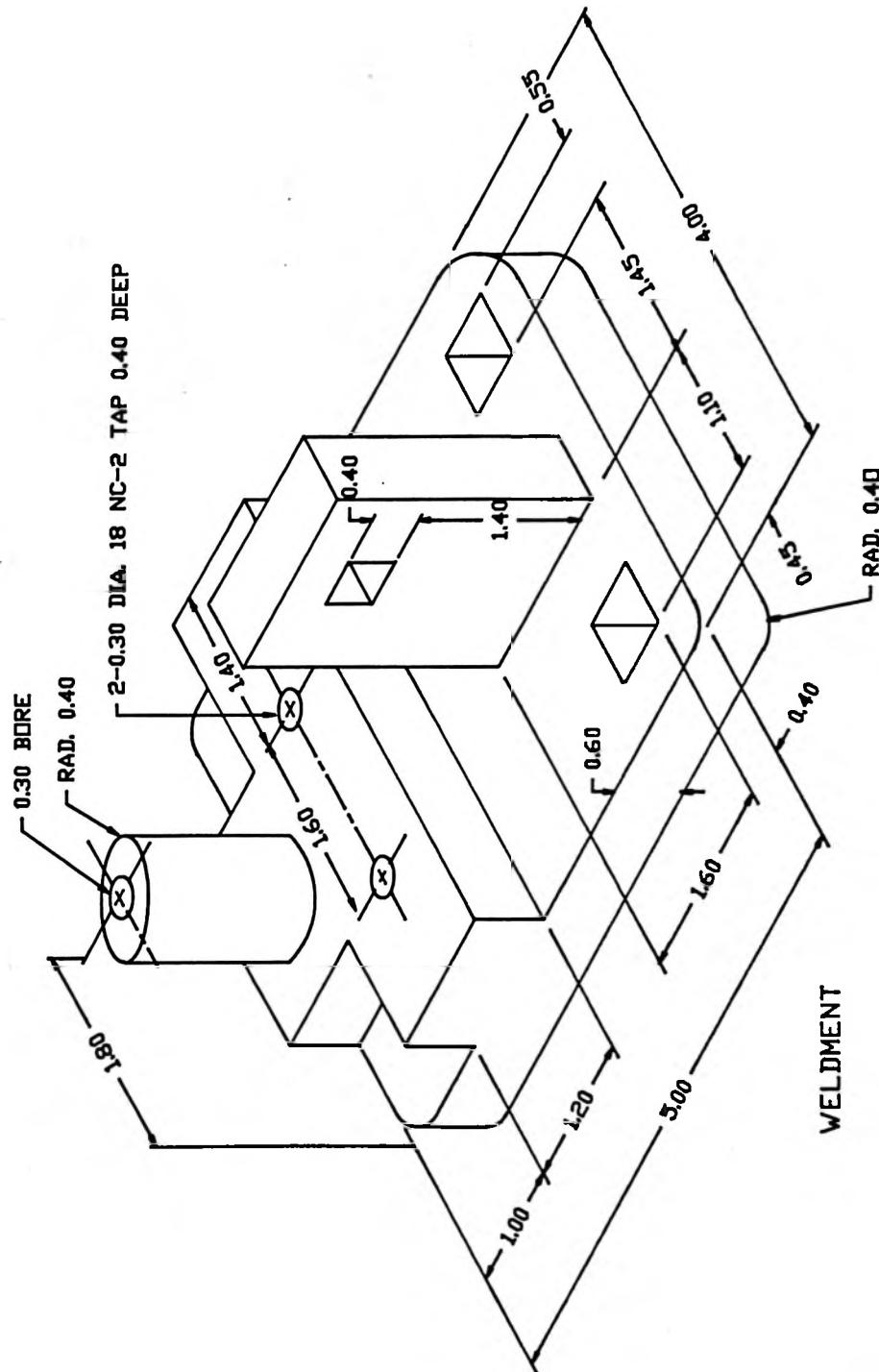
AutoCAD makes full use of the tree-structured directories provided by the MS-DOS or PC-DOS operating system, version 2.0 or higher. You can maintain several directories of drawings using just one copy of the AutoCAD program and its auxiliary files. If you prefer, you can ignore the tree-structured directory entirely, and keep the AutoCAD program files and all your drawings in the current directory on the logged disk drive.

1.4.4 Advanced Drafting Extensions

Three packages of advanced features, ADE-1, ADE-2, and ADE-3, are available as options. They are described below. The ADE-2 package requires ADE-1, and ADE-3 requires both of the others. Elsewhere in this manual, features of these packages are noted as such or marked with a "+1", "+2", or "+3", respectively.

ADE-1

- o Semi-automatic *dimensioning* is provided for such applications as mechanical engineering and architectural drafting. You can use it to easily add dimension lines and arrows for linear dimensions, angular dimensions, and circle/arc diameters and radii. You can even have AutoCAD measure the dimension and supply the proper text. The arrows and text are automatically positioned to fit.
- o Smooth arcs, or *fillets*, can be drawn to connect two lines. The lines are extended or trimmed, as necessary, so that they end precisely on the arc. Similarly, you can direct AutoCAD to *chamfer* the intersection of two lines, trimming the lines by specified amounts and connecting them with a short line.
- o Lines, traces, and arcs can be split or broken into two pieces, or one end can be cut off. A portion of a circle can be deleted, creating an arc.
- o You can display *axes*, or ruler lines, on the graphics monitor with any desired spacing of the axis ticks.
- o You can specify the format of coordinates, distances, and angles displayed by AutoCAD or entered from the keyboard. Two forms of "feet and inches" specifications are included in the choices. If you select one of these, a drawing unit is set equivalent to one inch. You can specify angles in terms of decimal degrees, grads, or radians, or in degrees, minutes, and seconds.
- o An object can be *cross-hatched*, or filled with a pattern. The pattern can be chosen from a library supplied with the AutoCAD software, read from a user-supplied disk file, or defined "on the fly". You can scale and rotate the pattern as you like.
- o A freehand *sketch* facility is provided. This feature requires a pointing device; it allows you to draw a series of short connected lines, with a specified resolution, quickly and easily. This capability is very useful for tracing maps, drawing complex curves, signatures, and many other applications. You can edit the constructed lines before storing them in the drawing database.
- o A *continuous status line* is provided to display the condition (on or off) of various modes and the location of the screen crosshairs. You can disable the status line if you wish.

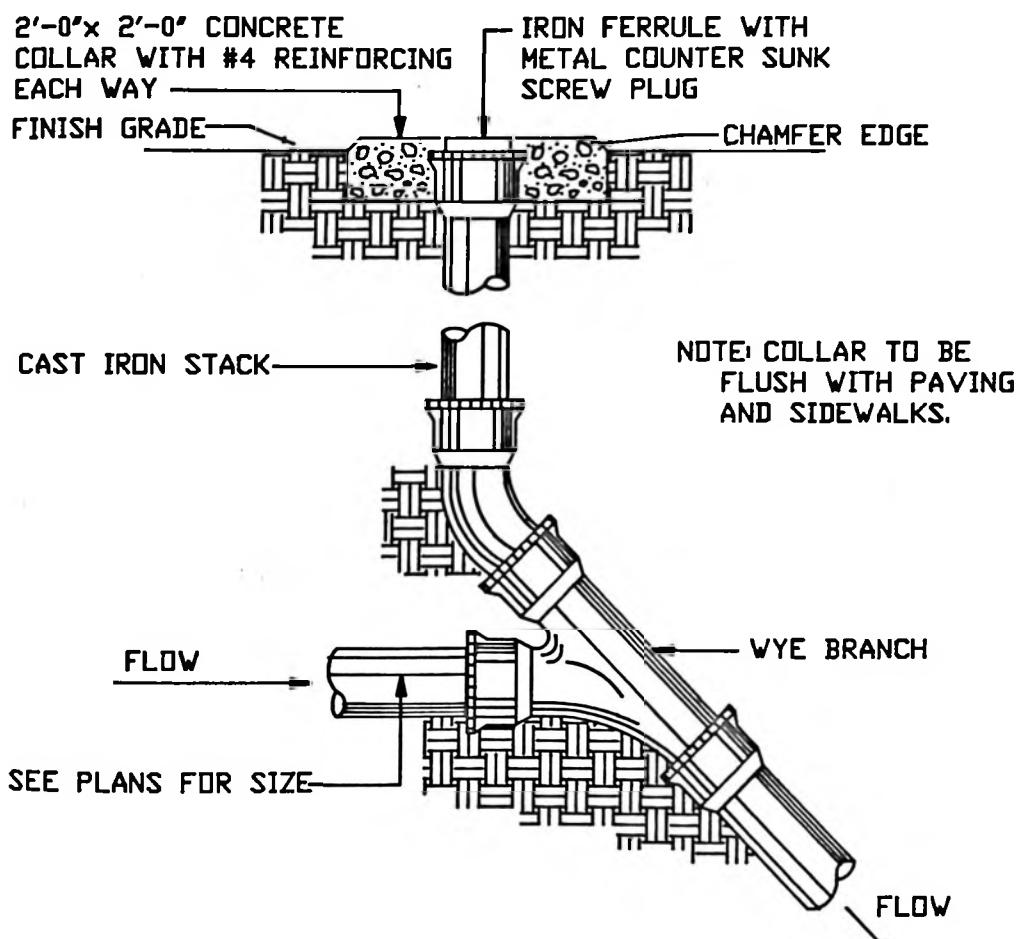


ADE-2

- o *Object (geometric) snap* is provided, allowing you to snap to reference points (endpoints, midpoints, etc.) of existing objects in the drawing as well as to grid points.
- o You can vary the aspect ratios and rotation of the snap grid, visible grid, and axis ruler lines.
- o Isometric grid and snap capabilities are provided.
- o Dynamic specification or *dragging* of certain classes of objects is available, allowing you to visually vary their locations and sizes on the graphics monitor before completing the entry.
- o The ability to define and retrieve *named views* is provided. This feature allows you to associate a name with a particular display of the drawing and to return to that view quickly and easily.
- o Existing objects can be *mirrored* about a user-specified horizontal or vertical line. The original objects can be retained or deleted.
- o A *slide* capability is provided, allowing you to save the current display from the monitor. Subsequently, you can rapidly recall the display for demonstration or other purposes, much as you would use a photographic slide. The ability to view slides is a standard AutoCAD feature, but the ability to create them is part of the ADE-2 package.
- o The ability to define and extract *Attributes* is provided. Attributes are text information that you can associate with each occurrence of an object in a drawing. Attributes have *tags* and *values*; the tag is the same for all occurrences of a given Attribute, but the value may vary from one occurrence to another. You can *extract* Attribute information from the drawing for processing by other programs or for transfer to a database. Many of the calculations that you can do with a drawing interchange (DXF) file can be accomplished more easily by using the Attribute extract feature.
- o When configuring AutoCAD, you can enable or disable the screen menu and text prompt areas. This feature allows you to control the portion of the screen available for graphics. (Not available for all display devices.)

ADE-3

- o 3D Level 1TM. You can create *three-dimensional visualizations* by "extruding" the objects in a two-dimensional drawing and then viewing the entire drawing from any direction in three-space. You can optionally request the suppression of "hidden" lines.
- o *Polylines* composed of connected line and arc segments can be drawn and edited. Polyline can have dot-dash linetypes and can be wide or tapered, and can form closed polygons. The area and perimeter of such polygons can be retrieved easily.
- o You can perform *curve fitting* on existing Polyline.
- o You can fillet or chamfer an entire Polyline.
- o You can define integer, real, point, and string *variables*, and can use them in expressions whenever AutoCAD requests input from you.



DROP CLEANOUT DETAIL

SCALE: NONE

- o Specified drawing layers may be *frozen*, to save time by eliminating them from subsequent regenerations of the drawing.
- o Objects can be highlighted during object selection (on most display devices).
- o You can add new commands to AutoCAD. These *external commands* operate by executing user-supplied programs (or system utility programs) from within AutoCAD.

1.4.5 Symbol Libraries

Optional libraries of commonly used symbols are available for many drafting applications. Using AutoCAD's drawing insertion mechanism, you can include these symbols in your drawings at any position, scale, and orientation.

Chapter 2

GETTING STARTED

This chapter presents the basic information you must learn in order to create, view, modify, and plot drawings using AutoCAD.

2.1 Notational Conventions

AutoCAD has been designed to operate on a wide variety of computer systems. When describing many of its features, we would like to state which key to press on the keyboard; however, this is difficult due to the lack of standard key names.

For instance, many computers have a CTRL key, but on others this key is marked ALT. In this manual, we have chosen the CTRL notation, so that "CTRL X" means "hold down the CTRL key and press the X key". Similarly, we will often refer to the RETURN key. On your keyboard, though, this key may be marked ENTER, SEND, NEXT, NEW LINE, etc.

Other keys used by AutoCAD are different on nearly every keyboard, so we will refer to them by the functions they perform, such as FAST CURSOR or FLIP SCREEN. Consult your AutoCAD Installation Guide / User Guide Supplement to see which keys on your computer have been assigned to these functions and for other machine-dependent information.

In the examples of command dialogues presented here, user input is underlined. When input is *Italicized*, you should supply input of the indicated type. Command names and parameters are shown in upper case, but you may usually enter them in any combination of upper and lower case.

It is important to distinguish the digit "0" from the capital letter "O". As you can see, they appear distinctly different in this text. The digit "1" and the small letter "l", however, look almost identical, so we will use a capital "L" wherever there might be confusion.

The space bar and RETURN key are used to terminate most command and data fields entered from the keyboard and usually may be used interchangeably. One exception to this occurs when entering text strings; since spaces may be included in the text, RETURN must be used to terminate the string. Some user-defined commands (ADE-3 feature) may also require termination by means of the RETURN key.

Default values are provided for many command parameters. They are displayed within corner brackets, "<>", in the command prompts and you can select the default by giving a null response (simply press the space bar or RETURN).

2.2 Loading AutoCAD

Before you use AutoCAD for the first time, make a working copy of the AutoCAD release disks and store the master disks in a safe place. The discussion below assumes that you have done so and that you have followed the software configuration instructions in your AutoCAD Installation Guide / User Guide Supplement.

The AutoCAD release disks contain numerous files. Some comprise the program itself, while others contain drivers for various graphic input/output devices. Still others are library files containing menus, text fonts, pattern definitions, and the like.

The files needed to load AutoCAD are listed below.

ACAD.EXE	- The main executable program
ACAD.HLP	- Help text file
ACAD.HDX	- Help index file
ACAD.OVL	- Drawing Editor and overlays
ACAD1.OVL	- ADE-1 overlays (if applicable)
ACAD2.OVL	- ADE-2 overlays (if applicable)
ACAD3.OVL	- ADE-3 overlay (if applicable)
ACADL.OVL	- More ADE-3 overlays (if applicable)
ACAD.PGP	- ADE-3 external command list (if applicable)

The following files are not supplied on the AutoCAD release disks, but are created during the software configuration process. These are also required in order to execute AutoCAD.

ACAD.CFG	- Configuration data file
ACADD.S.OVL	- Video display driver overlay
ACADDG.OVL	- Digitizer (tablet/mouse) driver overlay
ACADPL.OVL	- Plotter driver overlay
ACADPP.OVL	- Printer plotter driver overlay

You can take advantage of the PATH capability provided by the MS-DOS or PC-DOS operating system by placing all of these files in a directory listed on the PATH command. Other directory conventions can also be used; see Appendix B for details.

AutoCAD can be loaded fairly easily, but the method depends on the type of disk storage available on your computer.

Hard Disk Systems

If you have copied the AutoCAD release disks onto a hard disk, the process of loading and using AutoCAD is very easy. Simply "boot-up" your computer system by following the directions supplied with it, and log onto the disk partition or directory into which you have copied the AutoCAD disks. (If you have copied AutoCAD into a directory listed on the DOS PATH command, you can execute AutoCAD while logged into any directory you like.) Respond to the system command prompt by typing "ACAD" and then press RETURN.

C>ACAD

After a few seconds, AutoCAD's Main Menu appears.

Floppy Disk Systems

Loading AutoCAD from floppy disks is a bit more complicated because the AutoCAD program is very large and may not fit on one disk. Your computer system must have at least two floppy disk drives in order for AutoCAD to be run from floppy disks.

The ACAD.EXE execution file is needed only to begin execution, so it may be supplied on a disk by itself. The overlay files must all be present throughout execution of AutoCAD, and are supplied together on one disk.

To load AutoCAD from floppy disk, follow the steps outlined below. (This procedure assumes that AutoCAD has been configured for your equipment; if this has not yet been done, follow the instructions in the "Software Installation" chapter of your AutoCAD Installation Guide / User Guide Supplement instead.)

1. "Boot-up" your computer system by following the directions supplied with it.
2. Place the working copy of your AutoCAD "EXE" disk in Drive A.
3. Place a properly formatted disk for storing your drawing files in Drive B.
4. Respond to the system command prompt by typing "ACAD", then press the RETURN key.

A>ACAD

5. If the "EXE" and "OVL" files were supplied on separate disks, the following prompt is displayed:

Can't find overlay file ACAD.OVL.
Enter new drive letter or '.' to quit:

If this message appears, remove the "EXE" disk, and place the "OVL" disk in Drive A. Respond to the prompt by typing "A:" followed by RETURN.

6. After a few seconds, AutoCAD's Main Menu appears.

WARNING

You should never remove the "OVL" disk from its drive while AutoCAD is executing. Also, never remove the drawings disk from its drive while a drawing on that disk is being edited.

2.3 Library Files

AutoCAD uses various library files for such purposes as menus, text fonts, and pattern definitions. The names of menu, text font, and shape files are stored in a drawing; AutoCAD must locate and read these library files each time you edit that drawing.

If you don't put a drive or directory prefix on the file name, AutoCAD searches for library files first in the current directory on the logged disk drive. If the file is not found there, AutoCAD then searches the directory in which the ACAD.OVL file was found. You can instruct AutoCAD to search one more directory in between these two; see Appendix B for details.

If, after searching all these directories, AutoCAD is still unable to find a needed library file, it will ask you for another file name to use. You can respond with a simple file name, in which case the search outlined above will be repeated, or you can specify a fully qualified file name, with a drive letter and/or directory prefix. In the latter case, only the specified directory will be searched.

2.4 The Main Menu

After the program is loaded, AutoCAD's Main Menu appears on the text display screen. The menu provides access to various parts of AutoCAD, such as the interactive Drawing Editor with which drawings are created. The menu display looks like this:

Main Menu

0. Exit AutoCAD
1. Begin a NEW drawing
2. Edit an EXISTING drawing
3. Plot a drawing
4. Printer plot a drawing

5. Configure AutoCAD
6. File utilities
7. Compile shape/font description file
8. Convert old drawing file

Enter Selection:

A menu selection is made by simply typing in the number for the desired task and pressing RETURN or the space bar. This results in a short dialogue prompting you for additional information. If you make a mistake during the dialogue, press CTRL C to return to the "Enter Selection:" prompt.

2.4.1 Task 0 - Exit AutoCAD

When you select Main Menu Task 0, AutoCAD terminates and returns control to the operating system. After this entry the operating system prompt reappears.

2.4.2 Task 1 - Begin a New Drawing

To start a new drawing, select Main Menu Task 1. AutoCAD then asks for the name of the drawing to be created.

Enter selection: 1

Enter NAME of drawing:

The name you enter becomes the file name used to store the drawing on disk. All drawings are given the file type ".DWG", and this is automatically appended to the name you enter (don't type the .DWG). The drawing name may be from 1 to 8 characters long and may consist of letters, digits, and the special characters "\$" (dollar), "-" (hyphen), and "_" (underscore). Some examples of valid drawing names are:

OFFICE
803
PCB_1207
\$-ZONK-\$

The following are *improper* names for drawings.

FURNITURE	<i>(name too long)</i>
100%	<i>(invalid character)</i>
ANY.OLD	<i>(do not specify file type)</i>

You can preface the name with a drive letter or a directory path specification, as in the following examples.

B:OFFICE
\PLANS\BRIDGE
C:/PARTS/ELECT/CAP

The drawing will be saved on the disk or directory indicated by the prefix; if you don't specify a prefix, the drawing will be saved in the current directory on the logged disk.

When you begin a new drawing, AutoCAD sets up the initial environment for it based on a *prototype drawing*. You can select a default prototype drawing when you configure AutoCAD; this procedure is described in Appendix D. If you haven't selected a default prototype drawing, the standard prototype drawing ACAD.DWG will be used. The environment it establishes is outlined in Appendix A.

The prototype drawing that AutoCAD actually uses when creating your drawing depends on the method with which you respond to the "Enter NAME of drawing:" prompt. Three methods are possible:

1. Drawing-name
2. Drawing-name=Prototype-name
3. Drawing-name=

The first method is the simplest; if you enter just the name of the new drawing, AutoCAD uses the configured default prototype. You can specify a different prototype by means of the second method, where the drawing name and prototype name are separated by an equals sign. The last method, wherein no prototype name follows the equals sign, tells AutoCAD not to use any prototype drawing, but rather to set all environment parameters to default values.

The following is an example of method 2.

Enter NAME of drawing: WIDGET=ANSI

This would create a new drawing named "WIDGET", with its initial environment established from the prototype drawing named "ANSI". This has exactly the same effect as if you used DOS to copy ANSI.DWG to WIDGET.DWG, and then used Task 2 to edit WIDGET. The "ANSI" prototype drawing might contain pre-named layers, a standard title box, etc. An example of method 3 would be:

Enter NAME of drawing: LIGHTING=

Here, drawing "LIGHTING" would be created without a prototype drawing.

Once the new drawing and prototype drawing have been named, AutoCAD is ready to load its Drawing Editor. First, however, it checks to see if a drawing with the same name as the new drawing already exists in the output directory. If so, the following message is displayed:

** Warning! A drawing with this name already exists.
Do you want to replace it with the new drawing? <N>

If you respond "N" (the default), the old drawing is left untouched and AutoCAD's Main Menu is displayed again. If you respond "Y", the Drawing Editor is loaded, allowing you to create the new drawing; later, when you exit from the Drawing Editor, the new drawing replaces the old one on the disk.

2.4.3 Task 2 - Edit an Existing Drawing

To make changes and additions to an existing drawing, or to just display a drawing on the monitor, select Main Menu Task 2. A drawing name is requested, just as described above for creating a new drawing. For example:

Enter selection: 2
Enter NAME of drawing: B:PC

If no drawing is found with the specified name, AutoCAD displays the message:

** No drawing with this name is on file.
Press RETURN to continue.

When you press the RETURN key, AutoCAD's Main Menu is displayed again. If you misspelled the drawing name the first time, select Task 2 again and retype the drawing name. If you've forgotten the drawing name, you can get a list of existing drawings by means of Main Menu Task 6.

In one AutoCAD session, you may create, edit, and plot many drawings or process the same drawing repeatedly. To make repeated editing easy, AutoCAD remembers the most recent drawing name you supplied, and considers that the "default drawing". You can select it when the drawing name prompt appears simply by pressing RETURN.

Enter NAME of drawing (default B:PC): (*new name or RETURN*)

As a convenience, you can specify the default drawing name on the command line when you first load AutoCAD, as in:

A>ACAD B:PC

Ordinarily, the initial display of the drawing is the same as the display when you last saved the drawing. However, if your copy of AutoCAD includes the ADE-2 package and the drawing contains named views (described in Chapter 6), you can specify one of those views to be the initial display. Simply enter the view name following the drawing name, separating the two with a comma. For example:

Enter NAME of drawing: OFFICE.RECEPTION

If the view name you specify is not found in the drawing, the message:

View name *xx* not found.

appears (where *xx* is the name you specified) and the drawing is displayed as you last saved it. In any case, all mode switches (grid on/off, etc.) are restored to the values in effect when you last saved the drawing.

2.4.4 Task 3 - Plot a Drawing

Main Menu Task 3 is used to produce a "hard copy" plot of a drawing on a pen plotter. When AutoCAD prompts:

Enter NAME of drawing:

respond with the name of an existing drawing. AutoCAD then enters its plot routine, which is described in detail in Chapter 13 of this manual.

NOTE: You can also invoke the plot routine while you are editing a drawing, by means of the Drawing Editor's PLOT command.

2.4.5 Task 4 - Printer Plot a Drawing

Main Menu Task 4 is used to produce a "hard copy" plot on a printer plotter. When AutoCAD prompts:

Enter NAME of drawing:

respond with the name of an existing drawing. AutoCAD then enters its printer plot routine, which is described in detail in Chapter 13 of this manual.

NOTE: You can also invoke the printer plot routine while you are editing a drawing, by means of the Drawing Editor's PRPLOT command.

2.4.6 Task 5 - Configure AutoCAD

Before AutoCAD can be used, it must be properly installed on your computer system. The "Configure AutoCAD" function, Task 5, is used to select the drivers for your graphics equipment and to set various AutoCAD defaults to suit your needs. This function is invoked automatically when you first install AutoCAD on your computer; you may use it occasionally thereafter to change defaults, etc. See Appendix D.

Some cabling or other hardware changes may be necessary to connect the graphics equipment to your computer; see your AutoCAD Installation Guide / User Guide Supplement for instructions.

2.4.7 Task 6 - File Utilities

Main Menu Task 6 passes control to AutoCAD's disk file utility submenu, from which you can list the contents of a disk, delete selected files, change the name of a file, or copy a file. This facility is described under the FILES command in Section 3.7.

2.4.8 Task 7 - Compile Shape/Font File

Main Menu Task 7 converts Shape descriptions into a form usable by AutoCAD's Drawing Editor. This task is needed only when a Shape or Font file is created or modified; it is fully described in Appendix B.

2.4.9 Task 8 - Convert Old Drawing File

Because Version 2.0 of AutoCAD included major changes to the internal organization of drawing files, a facility has been provided for easy upgrade of drawings created with versions older than 2.0. If you edit or plot an old-format drawing (using Menu Tasks 2, 3, or 4), AutoCAD automatically converts the drawing to the new format. This conversion process is also necessary for drawing files that are to be inserted in other drawings; however, conversion is not automatically performed during drawing insertion. Task 8 is provided to convert these drawings to the new format explicitly.

When you select Task 8, AutoCAD prompts with:

Enter NAME of drawing:

Reply with the name of the drawing file to be converted. Wild-card characters "?" and "*" are permitted; for example, a response of "B:/*" converts all drawing files in the logged directory on drive B.

NOTE: When you convert a single file, the old-format file is retained, with its file type changed from ".DWG" to ".OLD". However, if you convert several drawing files at a time by using wild-card characters in the name you enter, the old files are not retained. New-format drawing files cannot be used with an older version of AutoCAD.

For further information on upgrading to Version 2.1 from a version older than 2.0, refer to Appendix E. If you are upgrading from Version 2.0 to Version 2.1, no drawing conversion is required.

2.5 Drawing Editor Usage

When you enter the Drawing Editor by selecting Main Menu Task 1 or 2, AutoCAD clears the screen and then displays your drawing as you last left it. (If this is a new drawing, the screen remains blank.) On the right-hand edge of the screen there is a menu of commands, and at the bottom is the prompt:

Command:

This signifies that AutoCAD is in Command mode and is ready to accept a command. You can now use the AutoCAD commands to create, view, modify and make plots of drawings. Choose the operation you want to perform and use the keyboard or menu to enter the appropriate command. Entering a command puts the program into Data Entry mode. In this mode you are prompted to specify coordinates (points) and supply other data needed to complete the operation. After the required information is provided, the function is performed. The display changes accordingly and the program returns to Command mode.

As described in Chapter 1, some AutoCAD installations use a single monitor for both text and graphics, while others use separate monitors. In a single-screen system, a small area below the graphics image is provided for Drawing Editor prompts and command/data entry. While the Drawing Editor is active, a "toggle" control key permits you to flip between graphics mode

and text mode; we'll call this the "FLIP SCREEN" key. Consult your AutoCAD Installation Guide / User Guide Supplement to see which key on your computer has been assigned the FLIP SCREEN function.

2.6 AutoCAD Command Summary

The following is a list of all valid AutoCAD commands, presented in alphabetical order. Use this list for reference. Detailed descriptions of the commands can be found in the noted sections of this manual.

APERTURE +2	Controls the size of the target box for object snap (Section 8.6).
ARC	Draws arcs of any size (Chapter 4).
AREA	Finds a polygon's area and perimeter (Section 5.3).
ARRAY	Makes multiple copies of selected objects in a rectangular or circular pattern (Section 5.2).
ATTDEF +2	Creates an Attribute Definition entity for textual information to be associated with a Block Definition (Chapter 11).
ATTDISP +2	Controls the visibility of Attribute entities on a global basis (Chapter 11).
ATTEDIT +2	Permits editing of Attributes (Chapter 11).
ATTEXT +2	Extracts Attribute data from a drawing (Chapter 11).
AXIS +1	Displays a "ruler line" on the graphics monitor (Section 8.3).
BASE	Specifies origin for subsequent insertion into another drawing (Section 9.4).
BLIPMODE	Controls display of marker blips for point selection (Chapter 6).
BLOCK	Forms a complex object from a portion of the current drawing (Chapter 9).
BREAK +1	Erases part of an object, or splits it into two objects (Section 5.2).
CHAMFER +1	Creates a chamfer at the intersection of two lines (Section 5.2).
CHANGE	Alters properties of selected objects (Section 5.2).
CIRCLE	Draws circles of any size (Chapter 4).
COPY	Draws a copy of selected objects (Section 5.2).
DBLIST	Lists database information for every entity in the drawing (Section 5.3).
DELAY	Delays execution of the next command for a specified time (Command scripts, Section 10.3).
DIM +1	Adds dimension notations to a drawing (Section 10.1).
DIST	Finds the distance between two points (Section 5.3).

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DRAGMODE +2	Allows control of the dynamic specification ("dragging") feature for the CIRCLE, INSERT, and SHAPE commands (Chapter 6).
DXBIN +3	Inserts specially-coded binary files into a drawing. Special-purpose command for programs such as CAD/camera (Appendix C).
DXFIN	Loads a drawing interchange file (Appendix C).
DXFOUT	Writes a drawing interchange file (Appendix C).
ELEV +3	Sets elevation and extrusion thickness for subsequently-drawn entities. Used in 3D visualizations (Chapter 14).
END	Exits the Drawing Editor after saving the updated drawing (Section 3.2).
ENDREP	Used with REPEAT command to define a repeat group (Section 5.2).
ENDSV	Exits the Drawing Editor after saving the updated drawing and the current display (Section 3.2).
ERASE	Erases entities from the drawing (Section 5.2).
FILES	Performs disk file utility tasks (Section 3.7).
FILL	Controls whether Solids, Traces, and wide Polylines are automatically filled on the screen and the plot output (Chapter 6).
FILLET +1	Constructs a smooth arc of specified radius between two lines (Section 5.2).
GRAPHSCR	Flips to the graphics display on single-screen systems. Used in command scripts and menus (Section 10.3).
GRID	Displays a grid of dots, at desired spacing, on the screen (Section 8.2).
HATCH +1	Performs cross-hatching and pattern-filling (Section 10.2).
HELP	Displays a list of valid commands and data entry options or obtains help for a specific command (Section 3.1).
HIDE +3	Regenerates a 3D visualization with "hidden" lines removed (Chapter 14).
ID	Displays the coordinates of a specified point (Section 5.3).
INSERT	Inserts a copy of a previously drawn part (object) into the current drawing (Section 9.3).
ISOPLANE +2	Selects the plane of an isometric grid to be the "current" plane for orthogonal drawing (Section 8.1).
LAYER	Creates named drawing layers and assigns color and linetype properties to those layers (Chapter 7).
LIMITS	Changes the drawing boundaries and controls checking of those boundaries (Chapter 6).
LINE	Draws straight lines of any length (Chapter 4).

LINETYPE	Defines Linetypes (sequences of alternating line segments and spaces) and loads them from libraries (Chapter 7 and Appendix B).
LIST	Lists database information for selected objects (Section 5.3).
LOAD	Loads a file of user-defined Shapes to be used with the SHAPE command (Section 4.10).
LTSCALE	Specifies a scaling factor to be applied to all Linetypes within the drawing (Chapter 7).
MENU	Loads a file of Drawing Editor commands into the menu areas (screen, tablet, and button) (Section 2.7).
MIRROR +2	Reflects designated entities about a user-specified axis (Section 5.2).
MOVE	Moves designated entities to another location (Section 5.2).
MSLIDE +2	Makes a slide file from the current display (Section 10.4).
OOPS	Restores erased entities (Section 5.2).
ORTHO	Constrains LINE drawing so that only lines aligned with the current grid can be entered (Section 8.4).
OSNAP +2	Enables points to be precisely located on reference points of existing objects (Chapter 8).
PAN	Moves the display window (Chapter 6).
PEDIT +3	Permits editing of polylines (Section 5.2).
PLINE +3	Draws connected line and arc segments, with optional width and taper (Chapter 4).
PLOT	Plots a drawing on a pen plotter (Chapter 13).
POINT	Draws single points (Chapter 4).
PRPLOT	Plots a drawing on a printer plotter (Chapter 13).
PURGE	Removes unused Blocks, Text Styles, Layers, or Linetypes from the drawing (Chapter 3).
QTEXT	Enables Text entities to be identified without drawing the text detail (Chapter 6).
QUIT	Exits the Drawing Editor and returns to AutoCAD's Main Menu, discarding any changes to the drawing (Section 3.2).
REDRAW	Refreshes or cleans up the display (Section 6.4).
REGEN	Regenerates the entire drawing (Section 6.5).
REGENAUTO	Allows control of automatic drawing regeneration performed by other commands (Chapter 6).

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RENAME	Changes the names associated with Text Styles, Named Views, Layers, Linetypes, and Blocks (Chapter 3).
REPEAT	Used with ENDREP to draw rectangular patterns of one or more entities (Section 5.2).
RESUME	Resumes an interrupted command script (Section 10.3).
RSCRIPT	Restarts a command script from the beginning (Section 10.3).
SAVE	Updates the current drawing file without exiting the Drawing Editor (Chapter 3).
SCRIPT	Executes a command script (Section 10.3).
SHAPE	Draws pre-defined shapes (Section 4.10).
SHELL +3	Allows access to other programs while running AutoCAD (Chapter 3).
SKETCH +1	Permits free-hand sketching (Section 12.5).
SNAP	Specifies a "round-off" interval for digitizer point entry so entities can be placed at precise locations easily (Section 8.1).
SOLID	Draws filled-in polygons (Chapter 4).
STATUS	Displays statistics about the current drawing (Chapter 8).
STYLE	Creates named text styles, with user-selected combinations of font, mirroring, obliquing, and horizontal scaling (Chapter 4).
TABLET	Aligns the digitizing tablet with coordinates of a paper drawing to accurately copy it with AutoCAD (Section 12.4).
TEXT	Draws text characters of any size, with selected styles (Chapter 4).
TEXTSCR	Flips to the text display on single-screen systems. Used in command scripts and menus (Section 10.3).
TRACE	Draws solid lines of specified width (Chapter 4).
UNITS +1	Selects coordinate and angle display formats and precision (Section 3.6).
VIEW +2	Saves the current graphics display as a Named View, or restores a saved view to the display (Chapter 6).
VPOINT +3	Selects the viewpoint for a 3D visualization (Chapter 14).
VSLIDE	Displays a previously-created slide file (Section 10.4).
WBLOCK	Writes selected entities to a disk file (Chapter 9).
ZOOM	Enlarges or reduces the display of the drawing (Chapter 6).

2.7 Command Entry

You can enter a command in any of the following ways:

2.7.1 From the Screen Menu

The screen menu appears on the right side of the graphics monitor while the Drawing Editor is active. The menu may contain so many items that displaying all of them at once could be impossible. Therefore, the menu is normally split into "submenus" designed so that selection of one item may cause a menu of options for that item to appear on the screen.

A pointing device makes entry of a command from the screen menu very easy. Just move the pointer until the screen crosshairs reach the right edge of the screen, and then move up or down until the desired menu item is lighted. Then press the pointer's "pick" button.

If you don't have a pointing device, you can access the screen menu via the keyboard. Pressing the MENU CURSOR key causes a menu item to light up. Use the UP CURSOR and DOWN CURSOR keys to move to different items. When you press space or RETURN, the lighted menu item is selected. You can escape without selecting anything by pressing the ABORT CURSOR key.

2.7.2 From a Tablet Menu

A tablet menu is a portion of the digitizing tablet allocated for command entry. Usually, a printed form is attached over the menu area to assist you in locating menu items. To select a command, position the tablet stylus over the desired item on the tablet menu and press the "pick" button. For further details, see Section 12.3.

2.7.3 From a Button Menu

If you have a tablet stylus or mouse with multiple buttons, you can enter often-needed commands from the extra buttons (the ones not used for point selection). Consult your AutoCAD Installation Guide / User Guide Supplement to find out which buttons on your pointing device are available for selection of button menu items. Also see the AutoCAD Standard Menu User Guide.

2.7.4 From the Keyboard

To enter a command from the keyboard, simply type in the command name, followed by the space bar or RETURN.

2.7.5 Repeated Commands

Regardless of the method used to enter the last command, you can press the space bar or RETURN at the next "Command:" prompt to repeat that command. Some commands have a shorter data entry form when repeated in this manner; the descriptions of individual commands explain any "repeated command" assumptions.

2.8 Data Entry

After a command is entered, you're usually required to supply additional information that the program needs to perform the function. In the case of Entity Draw commands, for instance, you must indicate the point in the drawing where the entity should appear. Some entities also require a numeric value that specifies height or width.

AutoCAD prompts you for the information it needs. The various prompts, the information they require, and the methods of responding to them are given on the following pages. (This section describes the data types commonly required by many of the AutoCAD commands. The specific command descriptions beginning in Chapter 3 include any unique requirements.)

If the data you enter does not match the type of data required by the command, the message:

Invalid

appears. You will be re-prompted for the item or returned to the "Command:" prompt to start over.

NOTE: There is often more than one way to respond to a prompt. There are, for example, a number of ways to specify a point. All the choices are described here. The individual command sections of this manual show which prompts to expect, but they don't always repeat the various response methods. You may therefore want to occasionally refer to this discussion while you're learning AutoCAD. We encourage you to experiment with the data entry methods described here. As mentioned earlier, your speed increases with practice and an understanding of the program's features.

2.8.1 Coordinates

Prompt: Point

When AutoCAD prompts you with the message "Point:" it wants the coordinates of a point in the drawing. After the point is specified, a small marker is drawn at that location for your reference. Such markers, or *blips*, disappear when you next regenerate or redraw the display (see Chapter 6). You can also instruct AutoCAD not to draw blips at all; see the BLIPMODE command in Chapter 6.

If limits checking is enabled, all points you enter are checked to see if they lie outside the drawing limits. If a point is outside the drawing limits, the message:

**** Outside limits**

is displayed and the point rejected. The limits check (and the drawing boundaries themselves) are controlled by the LIMITS command (Chapter 3).

Points are the most common type of data to be entered, and there are several different ways you can specify them.

2.8.1.1 Absolute Coordinates

You can specify a point by typing the actual X and Y values on the keyboard, separated by a comma. For example: "3.5,7.225" specifies the point with an X coordinate of 3.5 and a Y coordinate of 7.225.

2.8.1.2 Relative and Polar Coordinates

You can specify a point as a distance from the last coordinates specified, by typing an "@" prior to the X and Y values. For example, if the last point specified was (10,6) and you enter:

@2.5,-1.3

the result is to specify the point (12.5,4.7).

You can also use *polar coordinates* to specify a point as a distance and angle from the previous point. The format is "@distance<angle". For instance:

@4.625<30.5

specifies the point that is 4.625 units from the last specified point, at an angle of 30.5 degrees. For further information on how angles are defined, see Section 2.8.3.

2.8.1.3 Last Coordinates

A special case of relative coordinate specification is the character "@" by itself, which causes the coordinates of the last entered point to be used. It is exactly like the relative specification "@0,0".

2.8.1.4 Pointing

To specify a point using a pointing device, move the pointer until the crosshairs on the screen are at the point you wish to specify, then press the pointer's "pick" button. The coordinates of the point are entered as if you typed them on the keyboard.

2.8.1.5 Keyboard Pointing - Cursor Control Keys

The display crosshairs can also be moved using the keyboard. On systems with no pointing device, this is the only way to select a point on the screen directly. The keyboard can also be used to get an absolutely steady and precise position after moving to an approximate location with another pointing device.

First press the SCREEN CURSOR key to tell AutoCAD that you will be pointing from the keyboard. Crosshairs appear in the last place you selected, or at the lower left corner of the screen.

To move the crosshairs, use the UP CURSOR, DOWN CURSOR, LEFT CURSOR, and RIGHT CURSOR keys (usually the four arrow keys to the right of the alphabetic keyboard). To make the crosshairs move faster, press the FAST CURSOR key once or twice; to slow them down again, use the SLOW CURSOR key. When you have moved the crosshairs to the desired location, select the point by pressing the space bar or RETURN. If you want to stop without selecting a point, use the ABORT CURSOR key.

If a pointing device is present, turning on the keyboard pointer makes the crosshairs start at the location the entry device was pointing to. This allows you to use the keyboard as a "precision" pointing device, to home in on one exact dot after the crosshairs have been moved to the general location with another pointing device.

2.8.2 Numeric Values

Many prompts require entry of a number. Examples of such prompts are as follows:

Prompt:	Height	Columns
	Width	Rows
	Radius	Column Distance
	Value	Row Distance

When entering values for these prompts from the keyboard, you may use the following characters:

+ - 0 1 2 3 4 5 6 7 8 9 E .

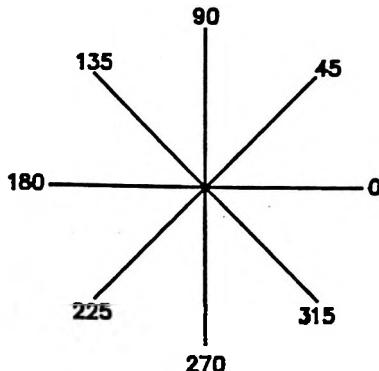
Examples are +35.7, -10, 7.2E+6 (which is "scientific notation form" for the value 7200000). For "columns" and "rows", the number must be an integer, however.

Whenever AutoCAD expects a distance, you may point to a location as explained previously. In some cases, the prompt "Second point:" appears; when you have pointed to a second location, AutoCAD uses the distance between the points as the input. In most cases, though, AutoCAD measures from some obvious base point to the point you gave. For instance, after you have given the center of a circle, AutoCAD asks for the radius. If you respond with a point, you surely want the radius to be the distance from the center to that point, placing that point on the circumference. When you specify a distance in this manner, AutoCAD adds a rubber-band line to the screen crosshairs, with one end anchored at the base point, so you can always tell where the base point is.

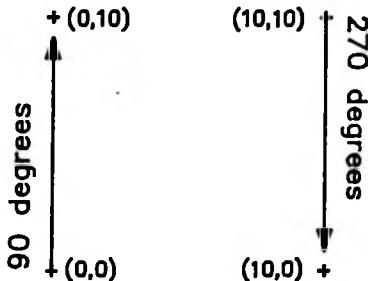
2.8.3 Angles

Prompt: Angle

Angles in AutoCAD are normally specified in decimal degrees. If the ADE-1 package is present, you have the option of using grads or radians, or degrees, minutes, and seconds (see Section 3.6). When the angle is used to specify an orientation (bearing) the following standard applies: angles increase in a *countrerclockwise* direction, and zero degrees is directly to the *right* of the starting point.



You can enter an angle numerically, from the keyboard. Follow the entry with space or RETURN. Alternatively, you can "show" AutoCAD the angle by pointing, indicating the start point and the end point of a line at the desired orientation. Note that the order in which these points are entered does matter. Pointing to (0,0) then (0,10) specifies 90 degrees (straight up), while (0,10) then (0,0) specifies 270 degrees (straight down).



In some cases, the location of the start point is pretty obvious; in such cases, if you designate a point in response to the "Angle" prompt, AutoCAD assumes that you are just specifying the end point. AutoCAD adds a rubber-band line to the screen crosshairs, with one end anchored to the base point to help you visualize the angle.

2.8.4 Displacements

Prompt: Displacement

When AutoCAD asks for a displacement it needs an X,Y vector. You can specify a displacement vector with the pointing device by entering two points to indicate "from and to".

To specify a displacement using the keyboard, enter the X and Y displacement values as if they were absolute X,Y coordinates and press RETURN. For example, "2,3" indicates a displacement of 2 units horizontally and 3 units vertically. If a prompt asking for a second point appears, press RETURN to ignore it.

2.8.5 Modifiers

Prompt: On/Off (etc.)

Some commands allow you to enter a modifier in place of a numeric value, invoking an alternate form of the command, as in:

Command: GRID On/Off/Value(X): ON

The modifier "ON" turns the grid on with the previous spacing. To enter a modifier, type it and press space or RETURN. Or select it from a screen, tablet, or button menu, just like a command.

2.8.6 File Names

Several commands ask for a file name. Enter the file name without a file type ("xxx") field; AutoCAD automatically adds the appropriate file type. You can include a drive or directory prefix, as in:

```
B:MY-FILE      or  
\JAN-85\ORG-CHRT   or  
/JAN-85/ORG-CHRT
```

NOTE: For most commands that request a file name, AutoCAD reads the required information from the file at the time of the command. The file itself is not needed for subsequent editing sessions. However, for the MENU, STYLE, and LOAD commands, only the file name is saved with the drawing; AutoCAD must locate and read the file each time you edit the drawing.

A fully-qualified file name (with drive/directory prefix) directs AutoCAD to a particular directory; no further searching is attempted. If a simple (unqualified) file name is given, AutoCAD will search for the file in the current directory, and, if that fails, in the directory in which the file ACAD.OVL was found.

2.8.7 Special Input Formats (+1)

If the ADE-I package is present, you can instruct AutoCAD to interpret the drawing units as inches and to allow input of coordinates, distances, and displacements in terms of feet and inches. Entry of angles in terms of grads or radians, or in degrees, minutes, and seconds can be accomplished as well. Use the UNITS command (Section 3.6) to select these options.

2.8.8 Variables and Arithmetic Expressions (+3)

If the optional ADE-3 package is present, you can define integer, real, point, and string *variables*, assign values to them as you please, and use those values whenever AutoCAD requests data from you. You can also perform arithmetic operations on the variables and use arithmetic expressions in response to prompts from AutoCAD. These advanced features are especially useful in custom menus and are described in Section 10.5.

2.9 Command/Data Error Correction

You can correct errors made while typing in command names or required data in three ways:

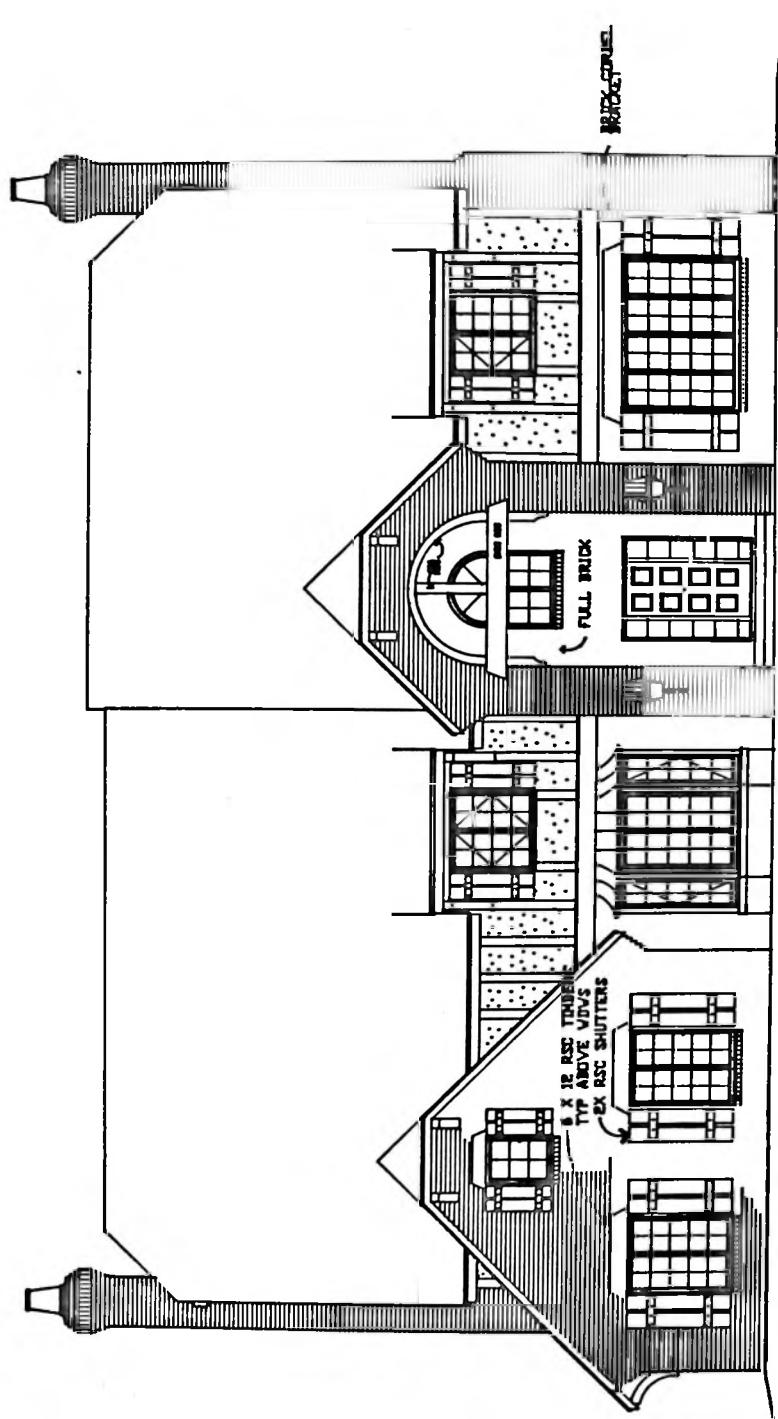
- | | |
|-----------|---|
| Backspace | Deletes one character at a time. (CTRL H has the same effect.) |
| CTRL X | Deletes all characters on the line. |
| CTRL C | Cancels the current command entry and restores the "Command:" prompt to the screen. You can cancel a command entirely at any time; that is, either while typing in the command name or during data entry. |

The keys for these functions may be different on your keyboard; see your AutoCAD Installation Guide / User Guide Supplement.

2.10 Echo to Printer

If you wish, you can instruct AutoCAD to echo all prompts, status listings, and keyboard input to your printer. The CTRL Q key is provided for this purpose. CTRL Q acts as an on/off toggle key; if printer echo is off, pressing CTRL Q once will turn echoing on. Pressing CTRL Q while printer echo is on will turn echoing off.

NOTE: AutoCAD's printer echo feature is completely separate from any printer echo feature offered by your computer's operating system. In particular, if the operating system's printer echo is enabled when you begin running AutoCAD, CTRL Q will not turn the operating system's printer echo off, but rather turn AutoCAD's printer echo on *as well*, resulting in double echoing.



Chapter 3

UTILITY COMMANDS

Detailed descriptions of AutoCAD's commands begin in this chapter. In general, we avoid describing the uses and effects of the commands in three-dimensional visualization mode; complete information on 3D mode (the ADE-3 package's "3D Level 1") is postponed until Chapter 14 to avoid unnecessary complexity in the manual.

This chapter fully describes a few commands that control basic AutoCAD functions or provide essential services.

3.1 HELP Command - User Assistance

You can use the **HELP** (or "?") command to obtain a list of AutoCAD commands in case you've simply forgotten a command name. The list also includes the various point entry formats. If you need a quick refresher course on the format or options available for a specific command, this is also available. When an invalid command is entered, AutoCAD prints a message to remind you of the availability of the Help facility.

The **HELP** command format is:

Command: **HELP** (or ?)
Command name (RETURN for list):

If you just want the list of AutoCAD commands on the general help screen, press space or RETURN in response to the "Command name" prompt. For detailed information about a specific command, reply with the desired command name. For instance, to obtain help on the **ZOOM** command, you should enter:

Command: **HELP**
Command name (RETURN for list): **ZOOM**

If there is more help information than will fit on the screen at once, AutoCAD will prompt:

Press RETURN for further help.

after each screen-full of help text. When you press RETURN, the help display will continue. If you would rather abort the help display at this point, enter **CTRL C**.

The help text is stored in the disk file **ACAD.HLP**, which is supplied with the AutoCAD software. Using a text editor, you can revise this file to include any additional information you wish displayed by the **HELP** command. If you want to do this, see Appendix B for further information.

3.2 Drawing Editor Exit

Whether you're creating a new drawing or editing a previously stored drawing, you are using AutoCAD's Drawing Editor. When you've finished editing the drawing, you must exit from the Drawing Editor and return to the Main Menu; this ensures that your changes are saved properly on disk. Three commands, with different functions, are available to accomplish this.

3.2.1 END Command

Command: END

The END command returns to the Main Menu and updates the drawing file. The old copy of the drawing will have its type changed from ".DWG" to ".BAK" (any previous ".BAK" file will be deleted). The updated drawing is now the ".DWG" file, so you can further edit it with AutoCAD if desired.

3.2.2 QUIT Command

Command: QUIT

The QUIT command returns to the Main Menu, but does not update the drawing. If you were just browsing, or have made some horrible error and wish to discard all the changes made in your editing session, you may enter the QUIT command. Because an inadvertent QUIT command could cause the loss of a lengthy editing session, AutoCAD asks you:

Really want to discard all changes to drawing?

You must answer this "YES" or "Y" to actually discard the session. After a QUIT command has been used, the Main Menu is displayed; the ".DWG" file is left unchanged. Any previously existing ".BAK" file is preserved unchanged also.

3.2.3 ENDSV Command

The ENDSV command exits the Drawing Editor and saves your drawing file, just as the END command does. In addition, the ENDSV command saves another file, called a vector file. This file is maintained by AutoCAD while you are editing your drawing; it contains information for just the portion of your drawing that is visible on the screen.

Normally, the vector file is deleted when you exit the Drawing Editor, but you can use the ENDSV command to save it. When you begin editing an existing drawing, AutoCAD checks to see if that drawing's vector file exists; if so, AutoCAD uses it to quickly reproduce the last display from the previous session.

The format for the ENDSV command is simply:

Command: ENDSV

Special notes:

1. If, during this editing session, you deleted or modified any existing entities in the drawing, the vector file cannot be saved. In such a case, ENDSV prints the following message:

Vector file not reusable -- not saved.

All other tasks are performed and the Drawing Editor is exited normally, as if you had entered the END command. If you re-edit your drawing without deleting or modifying any entities, you can then save the vector file.

2. The vector file has the same name as the drawing file, but a type of ".SRF".
3. As with ".BAK" backup files, vector files can take up a great deal of room on your drawings disk. If your disk space is scarce or the "initial view" speed improvement is not that important to you, simply use the END command. You can delete the vector files whenever you like, with no ill consequences.

3.3 SAVE Command - Updating Without Exit

As described above, you can use the END and ENDSV commands to save your new or modified drawing. Both commands exit the Drawing Editor, returning to the Main Menu. However, it is often desirable to save your changes periodically without exiting the Drawing Editor, thereby protecting your work from possible power failures, editing errors, and other disasters. You can do this with the SAVE command:

Command: **SAVE** File name: (*name*)

SAVE writes the current state of your drawing to disk, just like the END command, but remains in the Drawing Editor for further editing. The default output file name is *current-drawing.DWG*, but you can specify a different file if you like. Do not include a file type, however; file type ".DWG" is assumed.

If you use the default file name (the current drawing name), the ".BAK" backup file is cycled each time you use the SAVE command, and once more when you finally END. Thus, if you END after using SAVE at least once, or if you use SAVE more than once, the original copy of your drawing (as it was when you first began the editing session) will not be in the ".BAK" file.

If you specify a different file name and a drawing with that name already exists, AutoCAD prints the following message:

A drawing with this name already exists.
Do you want to replace it? <N>

Enter "Y" to overwrite the information in the drawing file; enter "N" or a null response to cancel the SAVE command, leaving the file unchanged.

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3.4 STATUS Command

There are many defaults, modes, and extents used by AutoCAD. The STATUS command reports the current values of many of them, as in:

Command: STATUS

40 entities in SAMPLE

Limits are	X: 0.0000	Y: 11.0000	(Off)
	Y: 0.0000	8.5000	
Drawing uses	X: 3.1250	12.5000	** Over
	Y: 1.3333	7.4500	
Display shows	X: 2.0000	7.3333	
	Y: 4.0000	8.0000	
Insertion base is.	X: 5.0000	Y: 6.0000	Z: 0.0000
Snap resolution is	X: 0.2500	Y: 0.2500	
Grid spacing is	X: 0.5000	Y: 0.5000	

Current layer: MY-LAYER-4

Color: 7 (white) Linetype: CONTINUOUS

Current elevation: 0.0000 thickness: 0.0000

Axis off Fill on Grid on Ortho off Qtext off Snap off Tablet off

Object snap modes: Endpoint, Midpoint, Tangent

Free RAM: 13283 bytes Free disk: 67584 bytes

I/O page space: 124K bytes

Some of these parameters (such as the drawing limits and display extents) have already been described. Others relate to AutoCAD features that have not yet been mentioned; it is instructive, however, to note the content and organization of the status display at this point. If your copy of AutoCAD includes the ADE-1 package, all coordinates and distances are displayed in the format specified by the most recent UNITS command (see Section 3.6).

The notation "(Off)" on the "Limits are" line means that limits checking is currently turned off. The notation "** Over" on the "Drawing uses" line means that the drawing extends outside the drawing limits. Items relating to ADE-1 or ADE-2 features (e.g., "Object snap modes") are listed only if the appropriate package is present in your copy of AutoCAD.

The "free RAM" and "I/O page space" reported relate to the amount of computer memory left for AutoCAD to use as temporary storage; they can usually be ignored. The "free disk" amount refers to space left on the drive that contains your drawing file. AutoCAD terminates execution of the Drawing Editor if it runs out of disk space, after first saving the work you've done so far.

3.5 LIMITS Command

Using the LIMITS command, you can designate the drawing limits (boundaries) for the current drawing and control the checking of those limits. The drawing limits perform three functions in AutoCAD:

1. When the limits check is on, they specify the range of coordinates that you can enter without receiving an "Outside limits" error.
2. They govern the portion of the drawing covered by the visible grid (GRID command, Chapter 8).
3. They are one of the factors that determine how much of the drawing is displayed by the ZOOM All command (Chapter 6).

Other than possibly changing the area of the screen covered by the grid, the LIMITS command does not affect the current display on the screen. The command format is:

Command: **LIMITS**
On/Off/Lower left corner <current value>:

You can respond to the prompt with any of the following:

- | | |
|---------|--|
| On | This turns the limits check on, retaining the current values of the limits themselves. While it is on, attempts to enter points outside the drawing limits are rejected (although an entity, such as a circle, may start within the limits, but extend outside them). The limits check is simply an aid to help you avoid drawing "off the paper". |
| Off | This turns the limits check off, but remembers the values of the limits for the next time the check is turned on. |
| a point | This specifies a new value for the lower left drawing limit, and results in a prompt for a new upper right limit: |

Upper right corner <current value>:

- | | |
|--------|--|
| RETURN | This retains the current value of the lower left drawing limit, and prompts for a new upper right limit as shown above. A RETURN in response to that prompt retains the current upper right limit. |
|--------|--|

The report generated by the STATUS command indicates whether the limits check is currently on or off.

When you create a new drawing, its initial drawing limits and the on/off state of the limits check are governed by the prototype drawing. At any time during the editing process, you can use the LIMITS command to change the drawing limits or turn the limits check on or off. For example, if the old limits were 0-10 in both X and Y dimensions, and you wished to change them to 4-7 in X and 0-14 in Y, you would use the following command:

Command: **LIMITS**
On/Off/Lower left corner <0.0000,0.0000>: 4.0
Upper right corner <10.0000,10.0000>: 7.14

Note that the lower left corner need not be (0,0); you can change it to any desired point, including negative coordinates.

3.6 UNITS Command - Format Control (+1)

In this user guide, all examples of coordinates, distances, and angles use ordinary decimal notation. However, in some disciplines, other forms of notation are preferred. The precision of displayed values may also be subject to personal preference. In order to accommodate these varied requirements, AutoCAD's ADE-1 package provides the "units" capability.

3.6.1 Coordinate Format Selection

When you begin a new drawing, its default display format and precision are governed by the prototype drawing. You can use the UNITS command to select the display format and precision you prefer, and you can change these selections as often as you wish while drawing. The UNITS command is invoked as follows:

Command: UNITS

The following menu is displayed:

System of units:

1. Scientific
2. Decimal
3. Engineering
4. Architectural

Enter choice, 1 to 4 <default>:

Choose the coordinate/distance format you prefer. The default shown is the format currently in effect. You may retain this format by simply pressing RETURN. To illustrate the various output formats, we will show how a distance of 15.5 drawing units would be displayed in each format:

Scientific:	1.55E+01
Decimal:	15.50
Engineering:	1'-3.50"
Architectural:	1'-3 1/2"

Note that the Engineering and Architectural formats produce "feet and inches" displays; these formats assume that each drawing unit represents one inch.

Once you have selected the format, AutoCAD asks for the precision. The prompt depends on which display format you have selected. For formats 1, 2, or 3, the prompt is:

Number of digits to right of decimal point, 1 to 8 <default>:

and you should enter the desired number or press RETURN to use the default. For format 4 (architectural), the prompt is:

Denominator of smallest fraction to display
(1, 2, 4, 8, 16, 32, or 64) <default>:

and you should enter one of the indicated values or press RETURN to use the default.

3.6.2 Angle Format Selection

After you have selected the format and precision for coordinates and distances, the UNITS command proceeds to angles, and presents the following menu:

System of angle measurement:

1. Decimal degrees
2. Degrees/minutes/seconds
3. Grads
4. Radians

Enter choice, 1 to 4 <default>:

Choose the angle measurement format you prefer. The default shown is the format currently in effect. You may retain this format by simply pressing RETURN.

Since most computer text displays do not include special symbols for the various angle measures, AutoCAD uses the following conventions: Decimal degrees are displayed as unadorned decimal numbers, grads are suffixed with a lowercase "g", and radians are suffixed with a lowercase "r". The degrees/minutes/seconds display is a compromise of the form:

123d45'56.7"

where:

d	= degrees
'	= minutes
"	= seconds

For example, an angle of 42.5 degrees would be displayed in the various formats as:

Decimal degrees	42.5
Degrees/minutes/seconds	42d30'0.00"
Grads	47.2222g
Radians	0.7418r

Whichever angle format you specify, AutoCAD next permits you to select the precision with which angles should be displayed. The prompt is:

Number of fractional places for display of angle (0 to 8) <default>:

You can select from 0 to 8 decimal places. If you have specified degrees, minutes, and seconds format for angles, the display appears as follows:

Decimal places	Display	Example
0	Degrees only	159d
1-2	Degrees & minutes	159d10'
3-4	Degrees, minutes, & seconds	159d10'12"
5-8	Fractional seconds; 1-4 decimal places	159d10'12.326"

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NOTE: The number of decimal places displayed may be constrained by the field width available for AutoCAD to display in.

3.6.3 Feet and Inches Input

When AutoCAD prompts you for a distance, displacement, spacing, or coordinates, you can always reply with a normal decimal number or scientific notation. If a "feet and inches" display format is in effect, you may also use that form of input. However, "feet and inches" input format differs slightly from the output format because it cannot contain a blank.

If feet are specified, they must be followed by an apostrophe; inches, if any, must follow with no intervening space. If display format 4 (architectural) is in effect, fractional inches may be specified. They are separated from the whole inches by any printable character except a digit, double-quote, or slash (we suggest a hyphen). The numerator and denominator are separated by a slash, and the denominator must be a power of 2 (up to 1024). The inches may be followed by a double-quote.

The following are legal inputs for the various display formats:

Scientific	Decimal	Engineering	Architectural
1.2E+02	1.2E+02	1.2E+02	1.2E+02
120.0	120.0	120.0	120.0
		10'	10'
		10'0"	10'0"
35.5	35.5	35.5	35.5
		2'11.5"	2'11.5"
			2'11-1/2"
5.0	5.0	5.0	5.0
		0'5"	0'5"
		5"	5"

Note that "feet and inches" input is never valid when the display format is Scientific or Decimal, and that fractional inches may only be input when the display format is Architectural.

3.6.4 Angle Input

When angles are entered from the keyboard, AutoCAD assumes that they are specified in the current units; they should not be suffixed with "g" or "r" if you have selected the grads or radians format. If you have selected the "degrees, minutes, and seconds" angle display format, AutoCAD will accept angles entered from the keyboard in the following format:

DdM'S"

where D, and M are integers, and S may include a decimal point. If S is omitted, M may include a decimal point. The letter d may actually be any character other than a digit, apostrophe, double-quote, or period.

3.7 FILES Command - Directory Access

It is sometimes useful to list a disk directory without exiting AutoCAD, or even while still using the Drawing Editor. The FILES command allows you to list, delete, rename, and copy files from within the Drawing Editor. (Note that Main Menu Task 6 allows you to perform the same functions.)

Command: **FILES**

The same file utility is used by the Drawing Editor and by the Main Menu. Regardless of the way you invoke it, AutoCAD's file utility displays the following menu:

File Utility Menu

0. Exit File Utility Menu
1. List Drawing files
2. List user specified files
3. Delete files
4. Rename files
5. Copy file

Enter selection:

3.7.1 Listing Drawing File Names

Selection 1 from the File Utility Menu searches for and lists the names of AutoCAD drawing (*.DWG) files. AutoCAD needs to know which disk drive to search, so it prompts:

Enter prefix:

You can reply with a drive letter or directory name, or press RETURN to specify the current directory of the logged drive.

If the disk contains many drawing files, the entire list may not fit on the screen at once. If this situation occurs, the list is presented one screen at a time; AutoCAD pauses after each screen and displays:

-- More --

To continue the list, press the RETURN key. When the list is complete, AutoCAD displays the total number of files, and repeats the "Press RETURN" message, as in:

74 files
Press RETURN to continue

When you press RETURN, the File Utility Menu is displayed again.

3.7.2 Listing Other File Names

Using File Utility Menu Selection 2, you can direct AutoCAD to search for a file whose name you supply explicitly. When you choose this menu item, AutoCAD prompts:

Enter file search specification:

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Enter a file name, with wild-card characters "?" and "*" if desired. A matches any character in that position, whereas "*" matches all characters up to a period or to the end of the file name. If you wish to scan a disk drive other than the default, include the drive specifier in your response, as in "B:WIDGET.BAK".

For instance, to list all the AutoCAD menu files in the "WORK" directory on disk drive B, enter:

B:/WORK/*.MNU

You can use this technique to list the names of files used or created by AutoCAD for various purposes. The type of file is indicated by the three-letter "type" portion of the file name, following a period. For instance, for a file named "HOUSE-14.DWG", "DWG" is the file type.

The file types used by AutoCAD are listed below.

BAK	- Drawing file backup
DWG	- Drawing file
DXB	- Binary drawing interchange file
DXF	- Drawing interchange file
DXX	- Attribute extract file (DXF format)
LIN	- Linetype library file
MNU	- Menu file
PAT	- Hatch pattern library file
SCR	- Command script file
SHP	- Shape/font definition source file
SHX	- Shape/font definition compiled file
SLD	- Slide file
TXT	- Attribute extract or template file (CDF/SDF format)
\$RF	- Vector file

To list all the files in the current directory of the default disk drive, enter a file search specification of:

.

3.7.3 Deleting Files

File Utility Menu Selection 3 allows you to delete specified files. This can be very handy. If you are running out of disk space, for example, you can make some room by deleting unneeded files (perhaps some ".BAK" backup files). The prompt is:

Enter file deletion specification:

As with selection 2, you should supply an explicit file name, optionally containing the "?" and "*" wild-card characters. If no wild-cards are used (the name is unambiguous and refers to a single file), the specified file is simply deleted. However, if wild-card characters are included, AutoCAD prompts you with each matching file name, and asks whether you want to delete it. For example, if you specified "B:?X.*", the following dialogue might ensue:

Delete B:EX.BAK? <N> Y
Delete B:EX.DWG? <N> RETURN
Delete B:2X.DWG? <N> Y

A "Y" response deletes the file, whereas "N" retains it on the disk. As indicated, retention of the file is the default.

WARNING: Deleting a file permanently erases it from the disk. Be sure that you no longer need the file or have a copy of it on another disk.

3.7.4 Renaming Files

Selection 4 on the File Utility Menu permits you to change the name of an existing file. A sample dialogue is shown below.

Enter current filename: B:WIDGET.DWG
Enter new filename: B:THINGO.DWG

These responses cause AutoCAD to change the name of file WIDGET.DWG (on drive B) to THINGO.DWG.

You can even use this technique to move a file from one directory to another, simply by renaming it. For example:

Enter current filename: B:/DIR1/FILE1
Enter new filename: B:/DIR2/FILE1

would move the FILE1 from the DIR1 directory on drive B to the DIR2 directory. This technique *cannot* be used to move a file from one disk drive to another.

3.7.5 Copying Files

Using selection 5 from the File Utility Menu, you can make a copy of an existing file. A sample dialogue is shown below.

Source filename: ABC.DWG
Destination filename: ABC.SAV

Here, a copy of file ABC.DWG is written to ABC.SAV. You can copy a file from one disk drive to another, as in:

Source filename: ABC.DWG
Destination filename: B:ABC.DWG

This would copy file ABC.DWG from the current drive to drive B.

3.8 SHELL Command - Access to Operating System (+3)

The ADE-3 package provides a SHELL command that lets you execute utility programs while remaining in AutoCAD's Drawing Editor. The command format is simply:

Command: SHELL

After a few seconds, the following prompt appears:

DOS command:

You can reply with any command that would be a valid response to the operating system's command prompt. Typical commands include:

```
DIR A:  
TYPE .MY.SCR  
EDLIN MY.SCR  
COPY *.* B:
```

When the utility program is finished, AutoCAD's "Command:" prompt reappears.

If you give a null response (RETURN) to the "DOS command:" prompt, a prompt of the form "drive>>" (a normal DOS prompt with an extra ">" appended) appears. You can now enter multiple DOS commands, just as you would at the normal DOS prompt. To return to AutoCAD, type "EXIT" -- this exits the operating system's command handler.

AutoCAD normally uses all the available user memory in your computer, and must relinquish some of it for the SHELL command and the program you want to run. If there is not enough free memory for the SHELL command, the message:

Insufficient memory for SHELL command.

is displayed, and the normal AutoCAD "Command:" prompt reappears. If the memory is sufficient to run the SHELL command, but not for the program you wish to run, a similar message is displayed.

AutoCAD's SHELL command is implemented by means of the "External commands" feature of the ADE-3 package, described in Appendix B. If the need arises, you can adjust the amount of memory the SHELL command requires (its "memory reserve"). Appendix B describes how to do this.

IMPORTANT NOTES:

1. Since AutoCAD is still running, numerous files are open, and disk check programs such as CHKDSK may incorrectly report these open files as errors or lost blocks. Do not attempt to correct these apparent errors while AutoCAD is running! For instance, do not use the /F option on CHKDSK.
2. When you use the SHELL command, AutoCAD switches to text mode and (for some display devices) saves the image of your drawing in a graphics memory area. If the program you run by means of the SHELL command writes in that graphics memory area, AutoCAD may restore the drawing incorrectly when your program has completed. Frequently, a REDRAW operation will correct the image.

3.9 MENU Command

AutoCAD's menu facility makes it possible for people to use AutoCAD with almost no training, especially if there is one person in the company who becomes an expert on the commands. This person can construct menus (and a Help file) that are tailored to the company's work; other people can use the menus to do routine work, picking up the rest of the commands as they need them.

AutoCAD is distributed with a standard menu file named ACAD.MNU, the contents of which are described in Appendix A and in the accompanying AutoCAD Standard Menu User Guide. You can modify this file or create custom menus for your application as described in Appendix B.

When you create a new drawing, the menu file referenced by the prototype drawing is automatically loaded. The MENU command, described below, permits you to load a new menu while editing. AutoCAD stores the name of the last menu you used in each drawing file; when you re-edit a drawing, the last menu file used for that drawing is automatically loaded.

AutoCAD actually supports a total of seven menus: the screen menu, the pointer device button menu, an auxiliary "function box" menu, and up to four digitizer tablet menus. A menu file may contain sections for each of these device menus.

The MENU command directs AutoCAD to load a new menu from a disk file. The command format is:

Command: **MENU** File name: *(file name)*

The file name should not include a file type; ".MNU" is assumed.

The appropriate sections of the file are loaded into the various device menu areas. In the case of the screen menu, items from the file appear in the menu area of the screen. If you have configured tablet menus, the newly loaded tablet menus are available for use from their associated tablet areas. If your pointing device has multiple buttons, the buttons not used for point selection can be used to select items directly from the newly loaded button menu.

It is sometimes useful to disable the menu feature altogether (for instance, to blank the screen menu). You can do this by using the MENU command and simply responding to the "File name:" prompt with a RETURN.

For information on construction of a menu file, see Appendix B.

3.10 Managing Named Objects

Several types of objects associated with a drawing are referred to by name. These objects are:

- o Blocks
- o Layers
- o Linetypes
- o Text styles
- o Named views

Usually, you supply the object's name when you create it, as in the "LAYER New" or "VIEW Save" commands, although there are a few instances where AutoCAD automatically creates standard objects with default names. (Named views and the VIEW command are features of the ADE-2 package.)

As your drawings become more complex, you may wish to change the names of some of these objects in order to keep them meaningful and appropriate, to make them easier to enter from the keyboard, or to avoid conflicts with names in INSERTed drawings. The ability to change object names is provided with the RENAME command.

Similarly, during the process of creating a drawing with AutoCAD, you may create Blocks, layers, linetypes, or Text styles that you never actually use, or that become superfluous when all entities referring to them have been deleted. Retaining these objects makes your drawing file larger than it needs to be, so a PURGE command is provided to allow you to delete them from a drawing.

The RENAME and PURGE commands are described in this section.

3.10.1 RENAME Command

You can change the name of a Block, layer, linetype, Text style, or named view by means of the RENAME command. The format is:

Command: RENAME
 Block/LAyer/LType/Style/View: (select one)
 Old (object) name: (name)
 New (object) name: (name)

("View" only appears in the option list if your copy of AutoCAD includes the ADE-2 package.) Select the type of object to be renamed by responding with one of the choices. "LAyer" and "LType" may be abbreviated "LA" and "LT", while the other options may be abbreviated to one character each. Once you have selected the type of object whose name is to be changed, RENAME prompts you for the old and new names. Object names may be up to 31 characters long, and may contain letters, digits, and the special characters "\$" (dollar), "-" (hyphen), and "_" (underscore). Letters are converted to upper case.

Special Notes

1. Some standard objects created by AutoCAD cannot be renamed. These include layer "0" and the "CONTINUOUS" linetype. The "STANDARD" text style, however, may be renamed if you wish.

2. Although Shape entities (Chapter 4) are referred to by name, a Shape's name resides in its shape definition file and cannot be changed with the RENAME command.

3.10.2 PURGE Command

When you begin editing an existing drawing (Main Menu Task 2), the Drawing Editor keeps track of the drawing's named objects and notes whether or not each of them is referenced by other drawing components as they are being loaded. Immediately after entering the Drawing Editor, you can use the PURGE command to selectively delete any unused named objects. The format is:

Command: **PURGE**
Purge unused Blocks/LAyers/LTypes/SShapes/STyles/All: *(select one)*

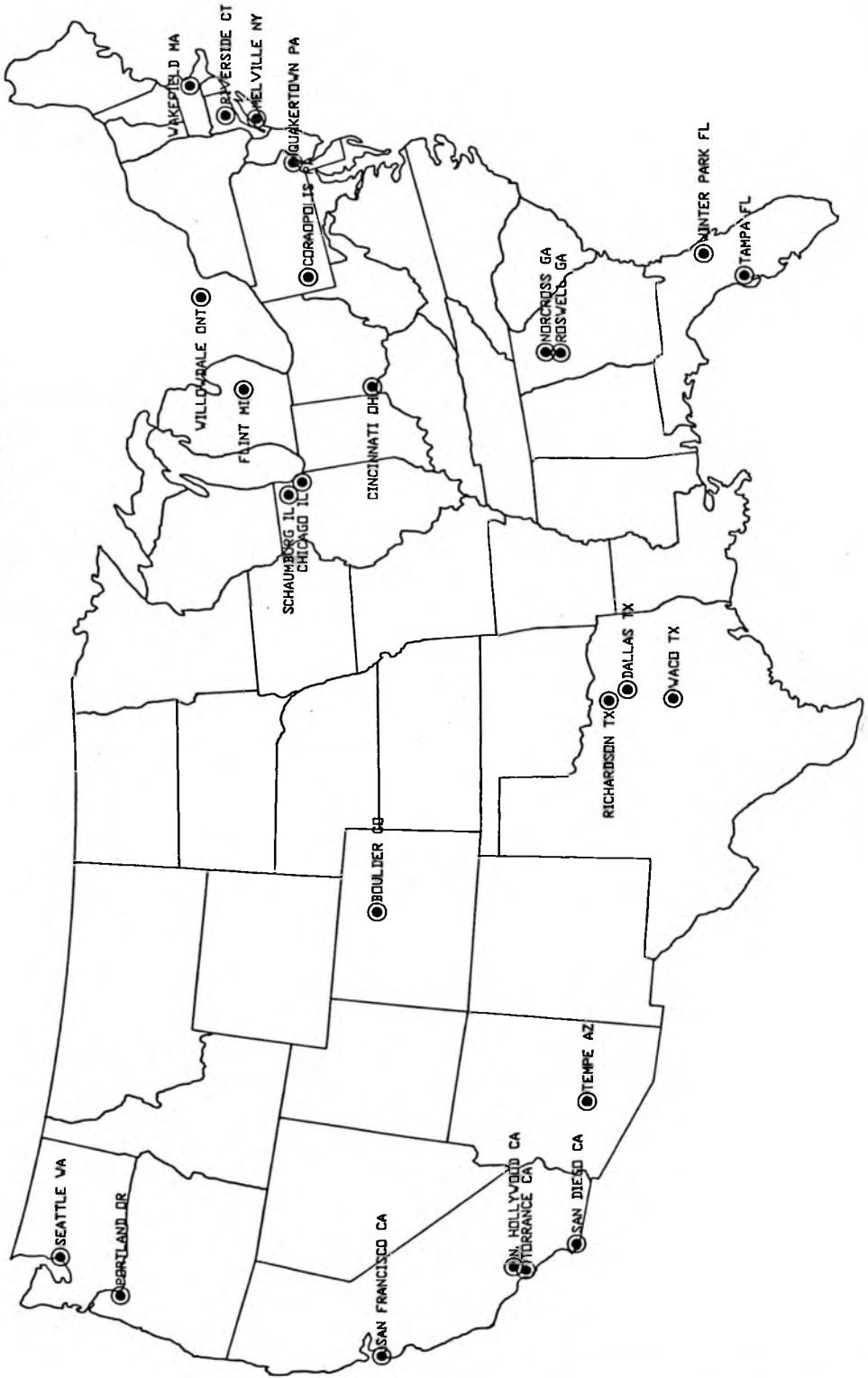
You can select one type of object to be purged, or reply "All" to purge all named object types. Each option can be abbreviated to just the letters that are capitalized in the prompt.

AutoCAD prompts you with the name of each unused object of the specified type, and asks whether that object should be purged.

Special Notes

1. The "SShape" option is provided to allow you to purge references to Shape files that have been specified via the LOAD command (Chapter 4) but which are no longer needed.
2. The PURGE command only works if it is the first command issued after you enter the Drawing Editor.
3. Some standard objects created by AutoCAD cannot be purged, even if they are currently unused. These include layer "0", the "CONTINUOUS" linetype, and the "STANDARD" text style.
4. Named views (ADE-2 feature) are never "referenced" by any other component in the drawing, so PURGE is not provided for them. However, the VIEW command (Chapter 6) provides a Delete option.

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Chapter 4

ENTITY DRAW COMMANDS

This chapter introduces AutoCAD's drawing entities and the commands used to enter them. These commands all require that you specify a point in the drawing where the entity should appear, using any of the Point Entry methods described in Section 2.8.

Two entity types, Blocks and Attributes, are not included in this chapter. Blocks are named collections of entities (even entire other drawings), which can be manipulated as a whole and inserted wherever you like; they are discussed in Chapter 9. Attributes (+2) contain textual information associated with Blocks; Chapter 11 describes Attributes in detail. All other commands that add objects to a drawing (dimensions, for instance) form the objects from the entities described in this chapter.

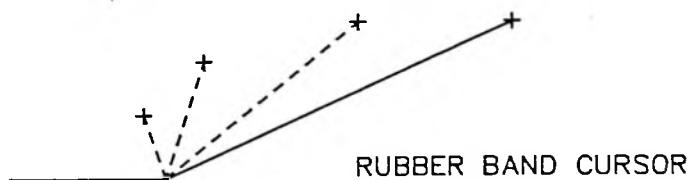
4.1 LINE Command

The most fundamental drawing entity is the Line. To draw one, enter the LINE command. You are then asked to identify both endpoints of the line. For example:

Command: **LINE** From point: **1,1**
To point: **5,2**
To point: **(space or RETURN)**



As you point to each "to" point, a "rubber band" cursor is displayed in addition to the normal crosshairs. This helps you see where the resulting line will go. A rubber band cursor is shown below; the dashed lines represent previous cursor positions.



RUBBER BAND CURSOR

Many times, you may wish to enter a series of connected lines. You could specify the start of one line at the same point as the end of the previous line, but this would be inefficient. To save time, the LINE command remains active and asks for a new "To point" after each point you specify. When you are finished entering a connected series of lines, enter a space or RETURN to terminate the LINE command. Note that you must use one of these replies to end the LINE command; any invalid responses to the "To point:" prompt just cause that prompt to be reissued.

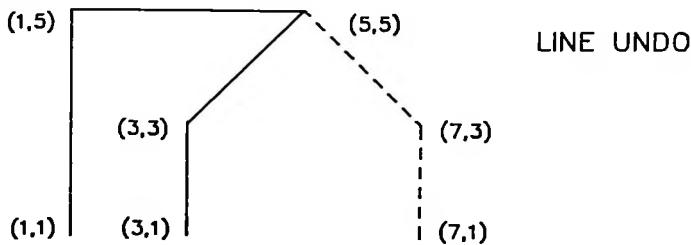
There is no difference between a set of lines entered with a single LINE command and lines drawn using multiple LINE commands; each line is a separate entity. You can use the ADE-3 package's PLINE command, described later in this chapter, to draw a sequence of lines and arcs to be treated as one entity called a Polyline.

4.1.1 Line Undo

When drawing a sequence of lines, you may sometimes want to erase the most recent segment and continue from the end of the previous segment. You can do this without exiting the LINE command by responding to the "To point:" prompt with a "U". Multiple U's backtrack through the line segments, erasing the most recent one each time. For example, the sequence:

```
Command: LINE From point: 1,1
To point: 1,5
To point: 5,5
To point: 7,3
To point: 7,1
To point: U
To point: U
To point: 3,3
To point: 3,1
To point: (space or RETURN)
```

results in the figure shown below. The dashed lines are the segments that were undone; they are included in the illustration for clarity, but are simply erased in actual use.



4.1.2 Closing Polygons

If the sequence of lines you are drawing forms a closed polygon, an additional convenience is provided. AutoCAD draws the final line segment automatically if you respond to the "To point:" prompt with a "C" (for "close"). This ensures that the first and last segments meet precisely.

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For example, you can use either of the following command sequences to draw a diamond on the display:

Command: LINE

From point: 6.5

To point: 5.4

To point: 4.5

To point: 5.6

To point: 6.5

To point: RETURN

Command: LINE

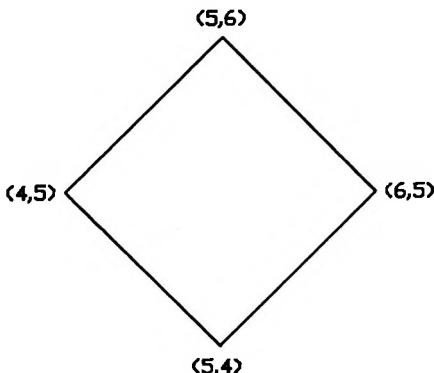
From point: 6.5

To point: 5.4

To point: 4.5

To point: 5.6

To point: C



4.1.3 Line/Arc Continuation

One additional capability is provided for the LINE command. If you respond to the "From point:" prompt with a space or RETURN, the start of the line is set to the end of the most recently drawn Line or Arc. This provides an easy means of resuming a LINE command that was interrupted for some reason, and it simplifies the construction of tangentially connected lines and arcs.

Command: LINE

From point: (just press space or RETURN)

The subsequent dialogue depends on whether a Line or Arc was more recently drawn. If a Line is more recent, the starting point of the new line has now been set, and the "To point:" prompts appear as usual.

If an Arc was more recently drawn, its end defines the starting point and the direction of the new line. Only the length of the new line needs to be specified, so AutoCAD prompts with:

Length of line: (value)

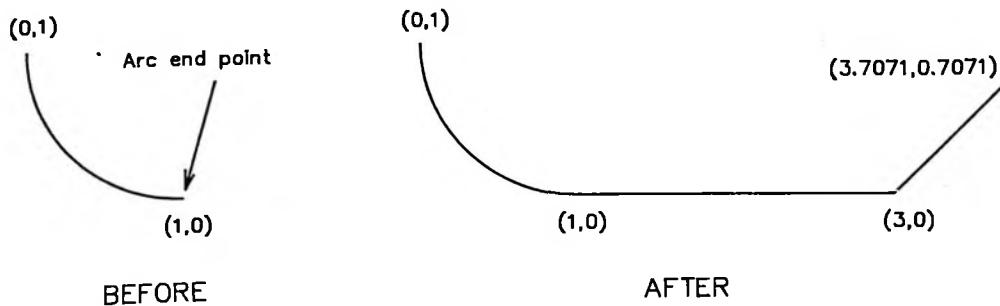
and then continues with the normal "To point:" prompts.

AutoCAD -- (4) ENTITY DRAW COMMANDS

For example, let us assume that an Arc was more recently drawn (using the ARC command, described later in this chapter). The sample dialogue:

Command: LINE
From point: RETURN (*to attach to end of arc*)
Length of line: 2
To point: @1<45
To point: RETURN (*to end LINE input*)

would result in the following:



4.2 POINT Command

To place a point in the drawing, enter the command POINT. You are then asked to identify the location of the point. As an example:

Command: POINT Point: 5,6

places a point in the drawing at location (5,6). Points can act as "nodes" for object snap purposes. (Object snap is an ADE-2 feature described in Section 8.6.)

Points plot as "dots" as wide as the pen tip you use on the plotter.

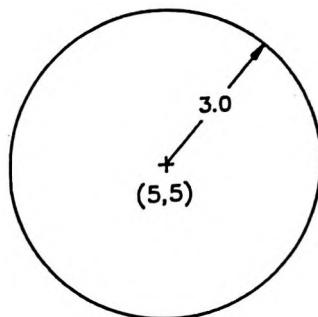
4.3 CIRCLE Command

You can draw a circle in one of four ways using the CIRCLE command.

4.3.1 Center and Radius

You can enter the circle's center point and radius. This is the default method, as indicated by the prompts.

Command: CIRCLE 3P/2P/<Center point>: 5.5
Diameter/<Radius>: 3



If you wish, you can specify the radius simply by designating a point on the circle's circumference.

4.3.2 Center and Diameter

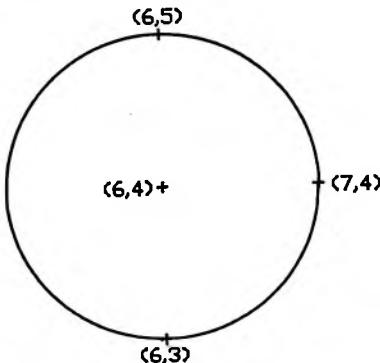
If you'd rather specify the diameter than the radius, simply enter the "D" modifier in response to the "Diameter/<Radius>:" prompt. AutoCAD then asks for the diameter. For instance, the following command sequence draws the same circle as in the above example.

Command: CIRCLE 3P/2P/<Center point>: 5.5
Diameter/<Radius>: D
Diameter: 6

4.3.3 Three-point Circles

You can also draw a circle by entering three points to be on its circumference. Respond "3P" to the "3P/2P/<Center point>:" prompt to tell AutoCAD that you wish to do this. For example:

Command: CIRCLE 3P/2P/<Center point>: 3P
First point: 6.5
Second point: 7.4
Third point: 6.3



4.3.4 Two-point Circles

If you respond to the "3P/2P/<Center point>:" prompt with "2P", AutoCAD lets you specify the circle by means of two endpoints of the circle's diameter. For instance, the following sequence draws the same circle as in the above example.

Command: CIRCLE 3P/2P/<Center point>: 2P

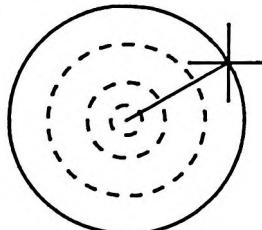
First point: 6,5

Second point on diameter: 6,3

4.3.5 Dynamic Circle Specification (+2)

With the exception of the "center and diameter" form of the CIRCLE command, the ADE-2 package allows you to "drag" the circle to the desired size visually. To do this, type "DRAG" when prompted for the last parameter of the command. Then move the pointing device, and the circle will adjust accordingly on the screen. When you are satisfied with the circle's appearance, press the pointer's "pick" button. The "DRAG" request can be supplied by a menu item. Dragging occurs only if DRAGMODE (Chapter 6) is ON. An example of "center and radius" circle dragging is shown below; the dashed circles illustrate the change in size of the circle as the crosshairs are moved outwards.

CIRCLE
DRAGGING



4.4 ARC Command

Arcs are partial circles, and are drawn using the ARC command. Eight different methods of specifying an arc are provided, allowing for personal preferences and for the varying circumstances under which arcs are needed. You may specify:

1. Three points on arc
2. Start point, center, end point
3. Start point, center, included angle
4. Start point, center, length of chord
5. Start point, end point, radius
6. Start point, end point, included angle
7. Start point, end point, starting direction
8. Continuation of previous line or arc

In this list, "center" refers to the center point of the circle of which the arc is a part.

The default method is "three points on arc," which is similar to the "3P" method of specifying a circle. The other arc specification methods are invoked by typing a letter, followed by space or RETURN, to indicate which information you want to enter next. The ARC command option letters have the following meanings:

A	-	included Angle
C	-	Center
D	-	starting Direction
E	-	End point
L	-	Length of chord
R	-	Radius

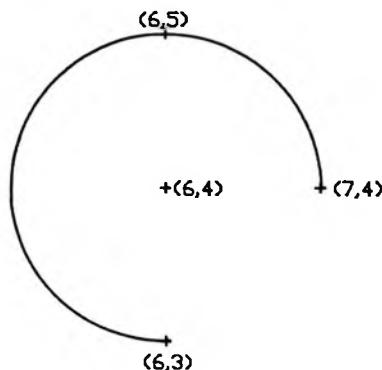
Prompts indicate which options are available at each step. Each method is discussed separately below. If you have the ADE-2 package, you can enter "DRAG" when prompted for the last parameter of each method to dynamically specify that parameter on the screen.

4.4.1 Three-point Arcs

This is the default method of arc specification, and is similar to the "3P" method of specifying a circle. The first and third points are the arc endpoints. For example:

Command: **ARC** Center/<Start point>: 7.4
 Center/End/<Second point>: 6.5
 End point: 6.3

draws the arc shown on the right.



A three-point arc may be specified from either direction. The last point is the end point used to attach a subsequent continuation line or arc.

4.4.2 Start, Center, End

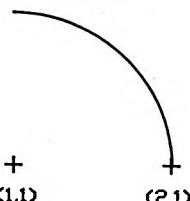
This specifies an arc drawn *counterclockwise* from the start to the end, with a specified center point.

Command: ARC Center/<Start point> 2.1
 Center/End/<Second point>: C
 Center: 1.1
 Angle/Length of chord/<End point>: 1.2.3

+ (1,2.3)

+
(1.1)

(2,1)



As illustrated in this example, the end point you specify is used only to determine the angle at which the arc ends; the arc does not necessarily pass through this point. The arc's radius is determined by the start point and center point.

It is sometimes convenient to give the center point first. For instance, you can give the center, radius, start angle, and end angle by using relative coordinates as in:

Command: ARC Center/<Start point> C
 Center: 1.1
 Start point: @1<0
 Angle/Length of chord/<End point>: @1<90

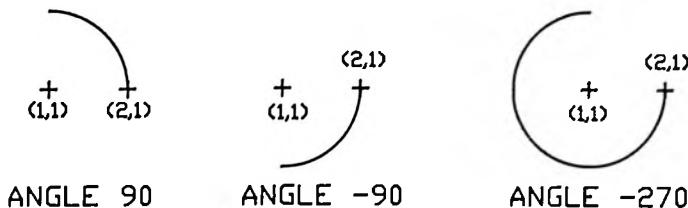
This draws the same arc as in the previous example. Note that a relative coordinate for the end point is based on the center, even though the start point was entered later. This is what is usually needed.

4.4.3 Start, Center, Included Angle

This draws an arc with the specified center and start point, spanning the indicated angle. Ordinarily, the arc is drawn *counterclockwise* from the start point. However, if the specified angle is negative, the arc is drawn clockwise. For example:

Command: ARC Center/<Start point> 2.1
 Center/End/<Second point>: C
 Center: 1.1
 Angle/Length of chord/<End point>: A
 Included angle: 90 or -90 or -270

The three different angle specifications result in the three arcs shown below.

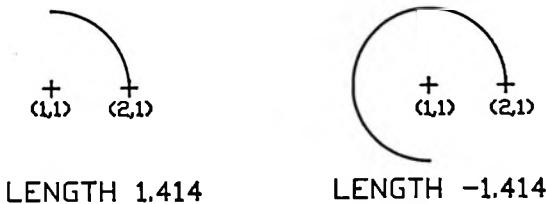


4.4.4 Start, Center, Length of Chord

A chord is a straight line connecting an arc's start point and end point. For this type of specification, the chord length is used to compute the ending angle. The same start, center, and chord length could apply to four different arcs; AutoCAD resolves this by always drawing this type of arc *counterclockwise* from the start point. By default, it draws the minor (less than 180 degree) arc, but a negative value for the chord length causes the major arc to be drawn. For example:

```
Command: ARC Center/<Start point> 2.1
Center/End/<Second point>: C
Center: 1.1
Angle/Length of chord/<End point>: L
Length of chord: 1.414 or -1.414
```

The two different chord length specifications result in the arcs shown below.

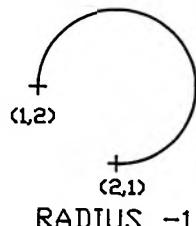
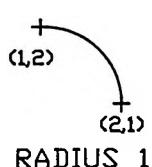


4.4.5 Start, End, Radius

Since the same values for these three variables could apply equally to four different arcs, AutoCAD always draws this type of arc *counterclockwise* from the start point and normally draws the minor arc. A negative value for the radius instructs AutoCAD to draw the major arc instead. For instance:

```
Command: ARC Center/<Start point> 2.1
Center/End/<Second point>: E
End point: 1.2
Angle/Direction/Radius/<Center point>: R
Radius: 1 or -1
```

The results of the two radius specifications are shown below.

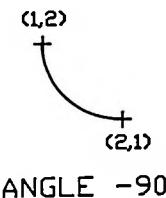
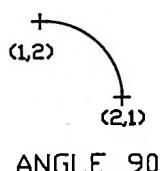


4.4.6 Start, End, Included Angle

This type of arc is normally drawn *counterclockwise* from the start point to the end point. If you specify a negative value for the included angle, however, the arc is drawn clockwise. For example:

```
Command: ARC Center/<Start point> 2.1
Center/End/<Second point>: E
End point: 1.2
Angle/Direction/Radius/<Center point>: A
Included angle: 90 or -90
```

The results for the two angle specifications are shown below.

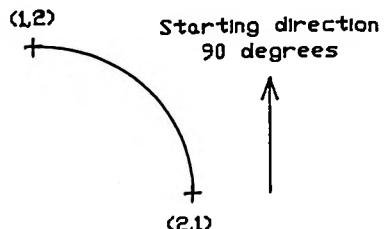


4.4.7 Start, End, Starting Direction

This method begins the arc in a specified direction. You can use it to draw an arc tangent to another entity. It will create any arc, major or minor, clockwise or counterclockwise. For instance, the sequence:

```
Command: ARC Center/<Start point> 2.1
Center/End/<Second point>: E
End point: 1.2
Angle/Direction/Radius/<Center point>: D
Direction from start point: 90
```

draws the arc shown at the right.



The direction can be specified by pointing to a single point. AutoCAD determines its direction from the starting point.

4.4.8 Line/Arc Continuation

This is actually a special case of the "start point, end point, starting direction" method discussed above. If you respond to the first prompt with a space or RETURN, the arc's start point and direction are taken from the end point and ending direction of the last arc or line drawn.

Command: ARC Center/<Start point> RETURN
 End point: 0.1

This command is especially useful after a line is drawn, to draw an arc tangent to that line.

4.5 TRACE Command

Often, lines must be solid with a specified width. In AutoCAD, such lines are called *traces*. Traces are entered just like lines, except that you use the command TRACE and, before entering the points, you are asked how wide the trace is. In repeat mode, all connected traces have the same width as that assigned to the first trace. You may enter two points when asked for the width of the trace to "show" AutoCAD the trace width.

Command: TRACE Width: .3
 From point: 1.1
 To point: 4.1
 To point: 4.4

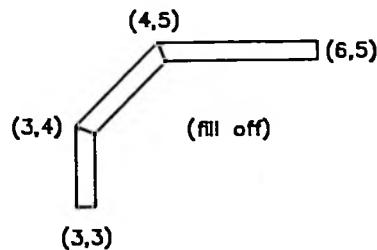
The end points specified are on the center line of the trace. The TRACE command automatically calculates the correct ending angle for connection to the next segment of the trace. For this reason, the trace is not drawn until either the next segment point is specified, or you press RETURN to stop trace entry. The starting and ending trace segments have a 90 degree end angle.

Traces are solid-filled unless Fill mode is off (see Section 6.6). If Fill is off, only the trace outline is drawn. When the final drawing is ready, you can turn Fill on for the slower but more appropriate solid fill.

For example, the command sequence:

Command: TRACE Width: .25
 From point: 6.5
 To point: 4.5
 To point: 3.4
 To point: 3.3
 To point: RETURN

generates the Trace shown at the right.



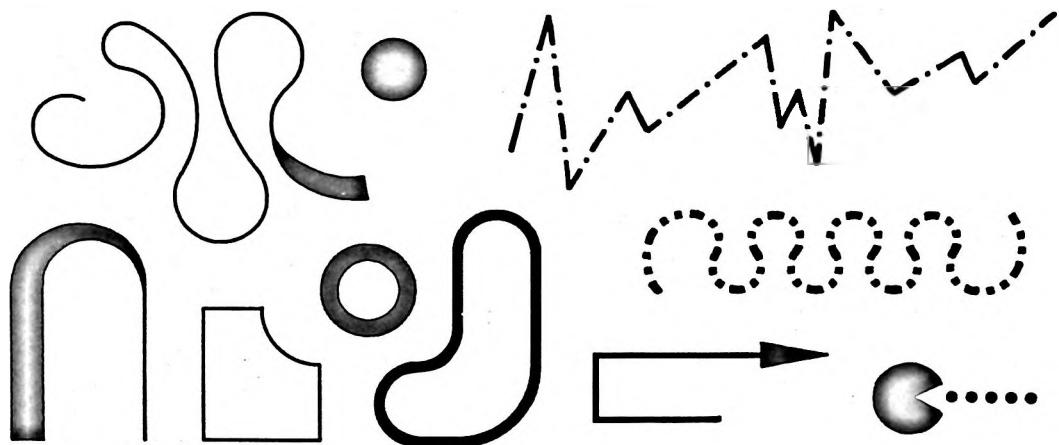
If you have the optional ADE-3 package, you may wish to use Polylines rather than Traces. Polylines offer far more flexibility than Traces and are described in the next section.

4.6 PLINE Command - Polylines (+3)

The ADE-3 package provides an entity called a *Polyline*. A Polyline is a connected sequence of line and arc segments, and is treated by AutoCAD as a single entity. Polylines have the following additional properties:

- o They can be drawn with dot/dash linetypes.
- o They can be wide (like Traces) or tapered.
- o A wide Polyline can be used to form a filled circle or a "doughnut".
- o The sequence of lines and arcs can form a closed polygon.
- o Polylines can be edited to delete vertices, or to join several Lines, Arcs, and Polylines into one Polyline.
- o Fillets and chamfers can be added where desired.
- o Curve fitting can be performed on a Polyline.
- o The area and perimeter of a Polyline can be retrieved.

The figure below illustrates several Polylines.



This section describes the **PLINE** command, with which you can draw Polylines. Editing of Polylines is described in Section 5.2.

To draw a Polyline, enter the **PLINE** command.

Command: **PLINE**
From point:

When you respond with the starting point of the Polyline, the current line-width is displayed:

Current line-width is *nnn*

This width will be used for all segments of the Polyline until you select a different width. You may now begin entering points and other specifications.

4.6.1 Straight-Line Segments

Initially, the PLINE command expects you to be entering straight line segments, and issues the following prompt:

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:

As the prompt indicates, several options are available to you. The default response is displayed within "<>" corner brackets, and indicates what will happen if you simply enter a point. If you enter a point in response to this prompt, AutoCAD interprets it as the endpoint of a line segment, and draws a straight line from the previous point to the point you have specified, just as in the LINE command. It then prompts again for another line segment.

The other responses to the PLINE prompt are options that modify the action of the command. To select an option, enter just the capitalized letters indicated in the prompt. For instance, enter "W" to select the "Width" option. Each option is described below.

Arc	This switches the PLINE command to arc mode, and results in a different prompt, to be described shortly.
Close	This option causes AutoCAD to draw a line from the current position back to the starting point of the Polyline, thereby creating a closed polygon. The PLINE command then terminates. The width of the closing line will be whatever line-width was in effect when you selected the "Close" option. Use of the "Close" option is different from drawing an explicit line back to the starting point; the effects on editing, curve fitting, and beveling of wide lines are different. The "Close" option is almost always preferable.
Length	This option allows you to draw a line segment at the same angle as the previous segment, specifying just the length of the new segment. If the previous segment was an arc, this will produce a line segment tangent to that arc.
Undo	This option removes the most recent line or arc segment added to the current Polyline. You can "Undo" any number of times, until there is just one point left in the Polyline. (If you exit the PLINE command while it contains just one point, no Polyline will be created.) The next prompt and the assumed direction for subsequent arc segments will be based on the last remaining segment after the deletion(s). That is, if the last remaining segment is an arc, the PLINE command will switch to arc mode (described below).
Width	This option allows you to specify the width of the following Polyline segment. Zero width produces a simple line that appears with the minimum displayable width regardless of the current display magnification. Widths greater than zero produce wide lines similar to Traces, solid-filled if Fill mode is on (see Section 6.6). AutoCAD prompts for both the starting width and the ending width of the segment, allowing you to taper the line, as in:

Starting width <0.0000>: 0.2
Ending width <0.2000>: 0.4

When you select a starting width, that becomes the default for the ending width. The ending width, in turn, is used as a uniform width for all subsequent segments until you change the width again. The starting and ending points of wide line segments are at the center of the line, as shown below.



Halfwidth This option allows you to specify the width from the center of a wide Polyline segment to one of its edges, or half the total width. The prompts are:

Starting half-width <current>:
Ending half-width <current>:

You may find the "Halfwidth" option most useful when "showing" AutoCAD the width by pointing, since one end of the rubber-band cursor is anchored to the center point of the end of the wide line segment. All the comments for "Width", above, also apply to "Halfwidth".

4.6.2 Arc Segments

When you respond to the PLINE command's line-drawing prompt with the "Arc" option, AutoCAD switches to arc mode for the current Polyline, and prompts with:

Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>:

If you respond with a point, it is interpreted as the endpoint of the arc. The Halfwidth, Undo, and Width options are the same as described above for straight line segments. You can set the width of an arc segment to any value from zero up to the diameter of the arc.

The arc starts at the previous point and is, by default, tangent to the previous segment of the Polyline. If this is the first segment of the Polyline, the direction will be that of the most recent Line, Arc, or Polyline segment drawn. This is not often appropriate; try one of the other options, described below, when an arc is to be the first segment of a Polyline.

Angle This option allows you to specify the included angle for the arc (that is, the angle the arc is to span). AutoCAD prompts with:

Included angle:

By default, the arc is drawn *counterclockwise*. If you want a clockwise arc, use a negative value for the included angle. The next prompt asks for further specifications:

Center/Radius/<Endpoint>:

CEnter Note that two letters are required to distinguish the "CEnter" option from the "CLose" option.

Ordinarily, AutoCAD draws the arc segment so that it is tangent to the previous Polyline segment, and thus calculates the center point automatically. The "CEnter" option allows you to override this action by specifying an explicit center point for the arc. AutoCAD prompts with:

Center point:

After you enter the center point, AutoCAD requests additional information by prompting:

Angle/Length/<End point>:

Here, "Angle" refers to the arc's included angle, and "Length" refers to the length of its chord.

CLOSE

This is similar to the "Close" option for the PLINE command's straight line mode, but causes the Polyline to be closed with an arc segment rather than with a straight line. Note that two letters are required to distinguish "CLOSE" from "CEnter".

Direction

Ordinarily, AutoCAD draws the arc segment so that it is tangent to the previous Polyline segment, and thus sets the starting direction for the arc equal to the ending direction of the previous segment. The "Direction" option allows you to override this action by specifying an explicit starting direction for the arc. AutoCAD prompts with:

Direction from starting point:

You can "show" AutoCAD the desired direction by specifying a single point; AutoCAD will interpret this as a direction from the starting point. The next prompt is:

End point:

Line

This option switches the PLINE command back to straight line mode and results in the appropriate prompt.

Radius

This option allows you to specify the radius of the arc. AutoCAD prompts with:

Radius:

The next prompt is:

Angle/Length/<End point>:

The "Angle" option lets you enter the included angle for the arc, and "Length" lets you specify the length of the arc's chord.

Second pt

This option allows you to specify the second point of a three-point arc. The prompts are:

Second point:

End point:

As you can see, you can specify Polyline arc segments by any of the means allowed in the ARC command described earlier in this chapter, although there are some differences in the order of specifications and in the defaults. (The start point, for example, is implicit.) There is one additional method of specifying an arc segment for a Polyline: radius, included angle, and direction of chord. The default chord direction is the ending direction of the previous segment of the Polyline; this is the only case in which a non-tangent arc is the default.

Suppose we are drawing a line at an angle of 10 degrees and want it to bulge out with a 225-degree arc before proceeding with another line segment in the original direction (like a capital Omega on a slant). The "radius, included angle, direction of chord" method is the way to do this, as shown below.

Command: PLINE

From point: 1,1

Current line-width 0.0000

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: @1<10



That draws the line segment we want. Now, we'll switch to arc mode to draw the arc segment.

Arc/Close/Halfwidth/Line/Undo/Width/<Endpoint of line>: A

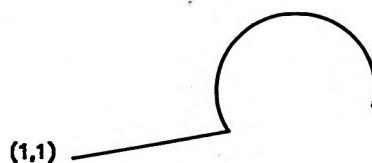
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/<Endpoint of arc>: R

Radius: 0.5

Angle/<End point>: A

Included Angle: -225 (clockwise)

Direction of chord <10>: RETURN (use line's direction)

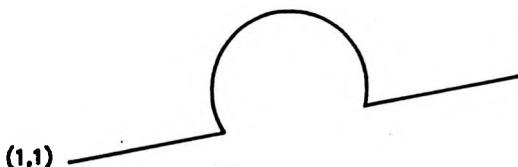


We now return to straight line mode and draw the final segment.

Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/<Endpoint of arc>: L

Arc/Close/Halfwidth/Line/Undo/Width/<Endpoint of line>: @1<10

Arc/Close/Halfwidth/Line/Undo/Width/<Endpoint of line>: RETURN



4.6.3 Dynamic Specification

For each of the arc specification methods, you can enter "DRAG" before specifying the last parameter. AutoCAD will then "drag" the arc around on the screen so that you can see clearly where the arc will go.

4.6.4 Doughnuts and Filled Circles

Using the PLINE command, you can construct solid-filled circles and "doughnuts" in just a few simple steps. Although PLINE does not permit you to create a 360-degree arc, you *can* draw a 180-degree arc segment and then close the Polyline with a matching arc segment. If the line-width is equal to the arc's diameter, a filled circle will be drawn (subject to Fill mode, described in Section 6.6). If you use a narrower line-width, a "doughnut" will be drawn. Remember that the arc's radius is measured from its center line. The widening occurs on both sides of the center line, so the resulting filled circle will have a radius equal to twice the wide arc's radius.

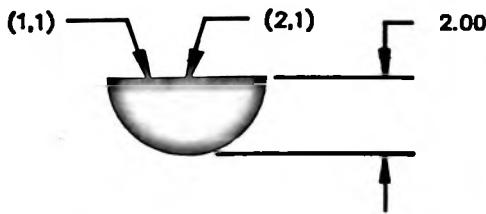
For example, suppose we want to draw a filled circle with a radius of 2 drawing units. We could use the following command sequence:

```
Command: PLINE
From point: 1,1
Current line-width 0.0000
Arc/Close/Halfwidth/Line/Undo/Width/<Endpoint of line>: W
Starting width <0.0000>: 2
Ending width <2.0000>: RETURN
```

This sets the width equal to our intended circle's radius. Now let's draw a semi-circular arc.

```
Arc/Close/Halfwidth/Line/Undo/Width/<Endpoint of line>: A
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: CE
Center point: 2,1
Angle/Length/<End point>: A
Included Angle: 180
```

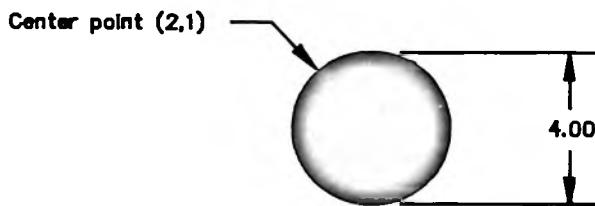
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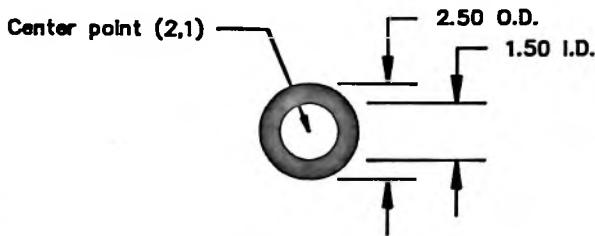
Note that the start point of the arc, (1,1) in this example, is halfway between the arc's center point and its outer edge.

The last step is to close the Polyline with another arc segment.

Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: CL



Now suppose we wanted an open "hole" in the center of the circle. To draw such a "ring" or "doughnut", we could use the same command sequence, except that we'd set a different line-width. For instance, if we changed the width from 2 units to 0.5 units, the above command sequence would produce the following figure:



4.7 SOLID Command

The SOLID command allows you to draw solid filled regions by entering them as quadrilateral or triangular sections.

First enter the two endpoints of a starting edge. Then enter either two points of the next edge, or one point and the RETURN key to specify a triangular section. You can continue to enter edges to make up the solid. When done, reply to the "Third point:" prompt by pressing RETURN.

Command: **SOLID**

First point: **4.8**

Second point: **7.8**

Third point: **4.7**

Fourth point: **7.7**

Third point: **5.6**

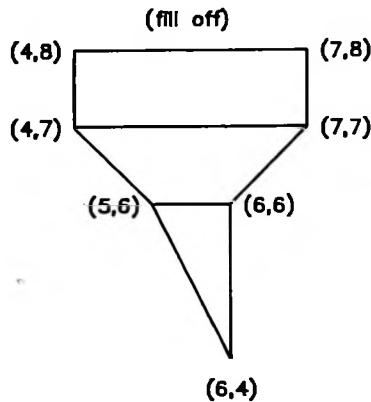
Fourth point: **6.6**

Third point: **6.4**

Fourth point: **RETURN** (triangular section)

Third point: **RETURN** (end of the solid)

The above points draw the solid shown below.



Solids are solid-filled unless Fill mode (Section 6.6) is off. When Fill is off, only the outlines are drawn, permitting faster operation while constructing the drawing. When the drawing is complete, you can turn Fill on and perform a REGEN operation to fill all Solids, Traces, and wide Polylines.

4.8 TEXT Command

You can add text to a drawing by means of the TEXT command. Text entities can be drawn with a variety of character patterns, or *fonts*, and can be stretched, compressed, obliqued, or mirrored by applying a *style* to the font. Each text string can be rotated and justified to fit your requirements. Text can be of any size.

The TEXT command interaction begins:

Command: TEXT

Starting point (or ACRS):

Respond with one of the indicated options, the actions of which are summarized in the following table:

Starting point	Left justifies the text baseline at the designated point
A (Aligned)	Prompts for two endpoints of the baseline, and adjusts the text to fit between them
C (Centered)	Asks for a point, and centers the text baseline at that point
R (Right)	Prompts for a point, and right justifies the text baseline at that point
S (Style)	Asks for a new Text style, then returns to the "Starting point:" prompt

These options are described in detail below.

4.8.1 Left Justified Text

In AutoCAD, text is ordinarily left justified at the starting point you specify. That is, the left end of the text baseline is placed at the starting point. Before the text can be drawn, however, AutoCAD must determine the desired text height, the rotation angle for the baseline, and the text string itself. It prompts for this information as follows:

Height <default>:

Rotation angle <default>:

Text:

The text height specifies how far above the baseline the capital letters extend, in drawing units. Lower case letters have "descenders", which go below the baseline, and some special characters may extend above the height or below the baseline. You can "show" AutoCAD the height by designating a point; the height will be the distance between this and the starting point. If you give a null response, the default height will be used. (The default is the height used for the previous text item with the same style.)

The rotation angle specifies the orientation of the text baseline, with respect to the starting point. The last angle specified will be used if you give a null response. You can "show" AutoCAD the angle by entering a point; the text baseline will run from the starting point to this point. If you designate a point to the left of the starting point, the text is drawn upside down.

Respond to the "Text:" prompt with the text string itself. Spaces are permitted; enter the desired characters, and then press the RETURN key.

The following example illustrates the complete prompt sequence for normal (left justified) text:

Command: TEXT Starting point (or ACRS): 2.1
Height <0.20>: .25
Rotation angle <0>: 0
Text: Master Bedroom

4.8.2 TEXT C - Centered Text

To center the text baseline at a specified point, respond to the "Starting point:" prompt with "C" followed by space or RETURN. AutoCAD then prompts:

Center point:

Enter a point. The remainder of the command is as above.

4.8.3 TEXT R - Right Justified Text

To right justify the text baseline at a specified point, respond to the "Starting point:" prompt with "R" followed by space or RETURN. When the prompt:

End point:

appears, enter a point. The remainder of the command is the same as for left-justified text.

4.8.4 TEXT A - Aligned Text

You can specify both the text height and orientation by defining two endpoints of the baseline. To do this, respond to the "Starting point:" prompt with "A" (for "aligned"). AutoCAD asks for the two endpoints and computes a text height such that the text just fits between those two points. Text entered in this fashion is considered "left justified" for the purposes of repetition (see below), although it entirely fills the width you specify.

Command: TEXT Starting point (or ACRS): A
First text line point: 1.1
Second text line point: 10.2
Text: Mona Lisa

4.8.5 TEXT Command Repetition

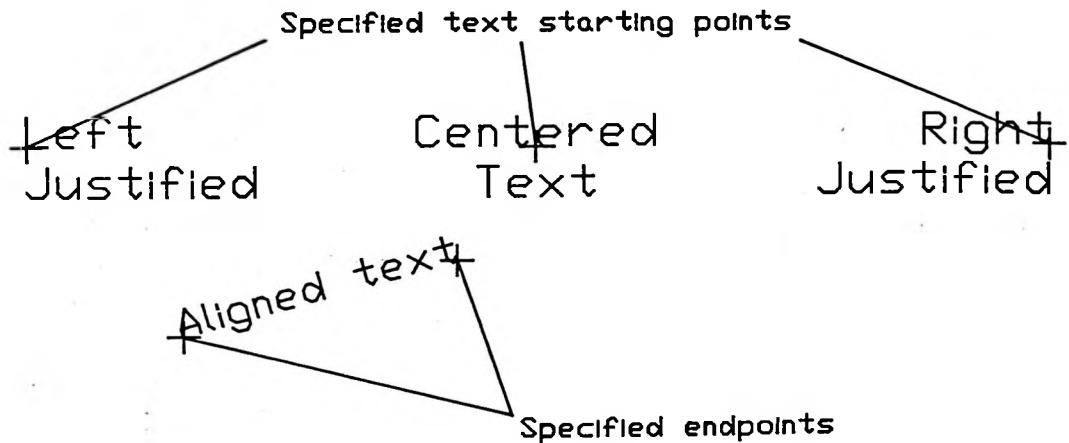
If you repeat a TEXT command (by pressing space or RETURN when prompted for the next command), AutoCAD assumes that another line of text is to be placed below the previous line, at the same angle, with the same height, and justified in the same manner (left, right, or center). For example:

Command: TEXT Starting point (or ACRS): C
 Center point: 3.2.8
 Height <0.20>: .25
 Rotation angle <0>: 0
 Text: Dining

Command: (space or RETURN to cause command repeat)
 Text: Room

In this example, the second line ("Room") will appear below the first ("Dining"). Since the first line was centered, the second line will also be centered. The vertical distance between lines is specified in the text font definition (discussed in the next section and in Appendix B).

The illustration below shows each type of text alignment.



4.8.6 TEXT S - Selecting a Text Style

The *text style* determines the appearance of the text characters. AutoCAD's Text styles are discussed in detail later in this chapter. Text is normally generated using the same style as the previous text (initially, the standard style). If you want to change styles for the current and succeeding text, respond "S" to the "Starting point (or ACRS):" prompt. AutoCAD then asks:

Style name <default>:

You can reply with the name of an existing Text style, as created using the STYLE command (described later in this chapter) or give a null response to retain the default style. AutoCAD then repeats the "Starting point (or ACRS):" prompt.

4.8.7 Control Codes and Special Characters

It is often desirable to overscore or underscore a piece of text, or to include a special character in it. You can specify these operations by including control information in the text string, using the double-percent character pair "%%" to introduce each control sequence. The following control sequences have been defined:

- | | |
|-------|--|
| %%o | - Toggle overscore mode on/off |
| %%u | - Toggle underscore mode on/off |
| %%d | - Draw "degrees" symbol |
| %%p | - Draw "plus/minus" tolerance symbol |
| %%c | - Draw "circle diameter" dimensioning symbol |
| %%% | - Force a single percent sign |
| %%nnn | - Draw special character number "nnn" |

For example, the text string:

%%uUnderlined %%oand%%u Overscored%%o

would be drawn as:

Underscored Overscored

Note that overscore and underscore modes can be in effect at the same time. Both modes are turned off automatically at the end of the text string.

AutoCAD stores text characters internally using a standard code called ASCII (American Standard Code for Information Interchange) with numeric values from 1 to 126; Appendix B includes a table of this character set. Some useful symbols may not be included on your keyboard, however, and some have no standard codes. You can add nonstandard symbols to AutoCAD's character set starting with character number 127 (see Appendix B).

You can include these nonstandard symbols (and any standard symbols not included on your keyboard) in text strings by using the "%%nnn" control sequence. Here, "nnn" represents a decimal number up to three digits long -- we recommend that you always use exactly three digits to avoid confusion. For instance, "%%065" draws character number 65, the letter "A".

One useful symbol not in the standard character set is the "degrees" symbol. Each of the character fonts supplied with AutoCAD has had this symbol added as character number 127; you can include it in text strings using the control sequence "%%127", or more simply, "%%d". Thus, the text string "98.6%%dF" is drawn as:

98.6°F

A single percent sign in text is handled as a normal character. The triple-percent "%%%>" control sequence is provided for those odd occasions when a control sequence must follow a percent sign in the actual text, or when two percent signs are needed in the text. For instance, if you need a degrees symbol following a percent sign, use "%%%%>%d". Here, the first three percents force one percent sign to be drawn, while the fourth and fifth percents introduce the "degrees" symbol. Similarly, the string "%%%%%" produces text with two percent signs. If a text string contains an unknown "%%%" control sequence, AutoCAD simply ignores the "%%%" and the character following it.

4.9 Text Styles and Fonts

A text *font* defines the pattern used to draw text characters. Text entities can be drawn using any number of character fonts. Several such fonts are supplied with the AutoCAD software; samples of each are shown below.

TXT:	Standard font	V	A
	ABC123\$&?	E	B
SIMPLEX:	Smoother font	R	C
	ABC123\$&?	T	1
COMPLEX:	Multi-stroke	I	2
	ABC123\$&?	C	3
ITALIC:	<i>Italicized</i>	A	\$
	ABC123\$&?	L	&
			?

You can construct additional fonts, as described in Appendix B.

It is often desirable to stretch or compress the characters, apply a slant to them, or draw them backwards or upside-down. This could be done using a specially-designed font for each such combination, but AutoCAD provides an easier method called *text styles*.

A Text style is composed of the following pieces of information:

- o A style name of your choice (up to 31 characters)
- o The name of an associated font file
- o A fixed text height (or zero)
- o A width (expansion/compression) factor

- o An obliquing (slant) angle
- o A "draw backwards" indicator
- o A "draw upside-down" indicator

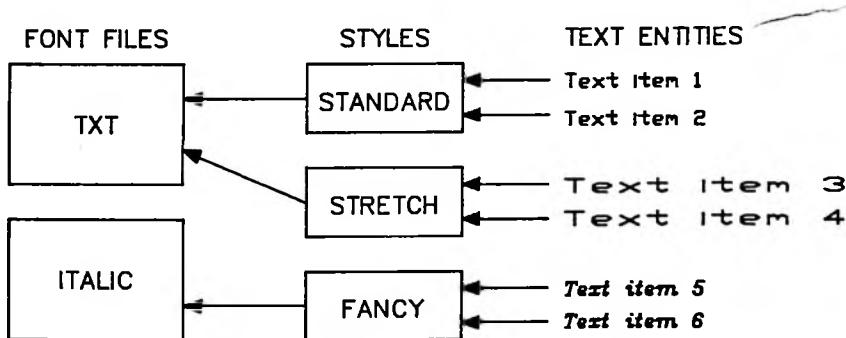
Text styles are created using the **STYLE** command, to be described shortly. When you draw a Text item (using the **TEXT** command), you can specify which style to use for that text. Essentially, the style is a quick, easy way to tell AutoCAD how to draw your text, without answering half a dozen prompts for each **TEXT** command.

When you begin a new drawing, a standard Text style is automatically created. It is used for all Text items until you create another style and request its use (via the "S" option on the **TEXT** command). The standard Text style has the following default properties (although these can be changed by the prototype drawing):

Style name:	STANDARD
Font file:	TXT
Height:	0 (not fixed)
Width factor:	1
Obliquing angle:	0
Backwards:	No
Upside-down:	No

You can change the properties of the standard Text style, just as for any other Text style, via the **STYLE** command. You can even change the style name, using the **RENAME** command described in Chapter 3. However, you cannot **PURGE** this Text style.

The following diagram illustrates the relationship between font files, Text styles, and Text entities.



As the arrows indicate, the Text items refer to the Styles, which in turn refer to the font files to determine how to form the characters. Several styles may use the same font file. Here, styles STANDARD and STRETCH both use font file TXT, but the STRETCH style has a width factor greater than 1 applied to it.

The style is used as a template from which your Text item inherits its height, width factor, obliquing angle, and backwards/upside-down properties. If these portions of a style definition are changed, nothing happens to existing Text items that were created using that style. However, if a style's font file is changed, all text using that style is regenerated using the new font. This happens immediately unless REGENAUTO (Chapter 6) is off.

The "fixed height" style parameter deserves special mention. AutoCAD normally prompts for the height each time the TEXT command is used. If you reply with space or RETURN, it defaults to the height used for the last text item. In some applications, however, text of a particular style is always the same height, so the "Height:" prompt would be superfluous. For such applications, you can simply specify the fixed height as part of the Text style; then the TEXT command does not prompt for the height when this style is used. If, on the other hand, you want to be able to specify the height of each Text item individually, set the fixed height in the Text style to zero.

The obliquing angle is an offset from 90 degrees. A positive offset, such as 15 degrees, results in characters that "lean" to the right, while a negative offset produces a slant to the left. Samples using the SIMPLEX font are shown below.

0° obliquing (normal)
15° (forward) obliquing /
-20° (backward) obliquing \

4.9.1 STYLE Command

The **STYLE** command is used to create and modify Text style definitions. The format is:

```
Command: STYLE Text style name (or ?): (name)
Font file <default>: (file name)
Height <default>: (value)
Width factor <default>: (scale factor)
Obliquing angle <default>: (value)
Backwards? <Y/N> (Y or N)
Upside-down? <Y/N> (Y or N)
```

Responding "?" to the first prompt causes AutoCAD to list the currently-defined Text styles. To create a new style or modify an existing one, respond with the desired style name. The ensuing prompts all have defaults displayed within corner brackets; pressing space or RETURN in response causes the default to be used. For existing styles, the defaults are the current values associated with that style; for a new style, the defaults are the same as those described above for the standard Text style.

Style names may be up to 31 characters long, and may contain letters, numbers, and the special characters "\$ - _". The name is converted to upper case before use.

When replying to the "Font file:" prompt, do not include a file type; ".SHX" is assumed. The specified font file is read, and its character definitions are loaded automatically (unless the file is already in use by another Text style). It's okay for several styles to use the same font file.

Changing Existing Style Parameters

With the exception of the associated font file, you can change the parameters of an existing style without affecting existing Text entities; the changes are applied only to new Text entities created using that style.

If you select a new font file for an existing Text style, all Text items with that style will be redrawn using the new font when the drawing is next regenerated. If REGENAUTO (Chapter 6) is ON, such a change in font causes an automatic regen.

You can use the **RENAME** command (Chapter 3) to change the name of an existing Text style. If any existing Text items use the old style name, they are given the new name automatically.

4.9.2 Notes on the VERTICAL Text Font

A special font named **VERTICAL** is supplied with AutoCAD. This font is a simple one based on the standard **TXT** font, but oriented vertically with the characters centered below one another. AutoCAD actually treats this font as though its letters were lying on their sides. Text drawn with this font must be entered with a rotation angle of 270 degrees in order for the characters to be properly oriented. Thus, when using a pointing device to designate the rotation angle, simply point straight down from the text starting point.

You can use **TEXT** command repetition (Section 4.8) with this font; each successive text string is drawn to the right of the preceding one. However, care is needed when attempting to use width factors, obliquing angles, or backwards/upside-down modes; they affect this font differently because of its rotation.

4.10 Shapes

Shapes are special entities that you can define using lines, arcs, and circles. To draw a Shape, you first LOAD a disk file containing the *Shape definition*; you can then use the SHAPE command to put Shapes from this file into your drawing. You can specify the scale and rotation to be used for each Shape as you add it to your drawing. This section of the manual briefly describes Shapes and explains how to use the LOAD and SHAPE commands to enter a Shape into your drawing.

Although the *use* of Shapes is simple, their definition is not. For instructions on defining Shapes, see Appendix B.

NOTE: Many users can skip this section of the manual. *Blocks* (Chapter 9) provide the primary means of defining and using a library of drawing parts. They are more versatile in every way, and far easier to learn and apply, than Shape definition. Shapes, however, are more efficient for AutoCAD to store and draw; a variation of them is used to define text fonts. User-defined Shapes may help in cases where a simple part must be inserted a very large number of times and where speed is of the essence.

Shapes are defined by special files with a file type of ".SHX" (see Appendix B). Shape definitions must be LOADED before they can be used.

4.10.1 LOAD Command

Before you can use a set of Shapes in a drawing, you must LOAD the Shapes into the drawing. This is accomplished by the command:

Command: LOAD

Name of shape file to load (or ?): (Shape file name)

where "*Shape file name*" is the name of the Shape definition file. Do not include a file type; ".SHX" is assumed. You must explicitly load the Shape file the first time you need it; for subsequent editing sessions, AutoCAD automatically loads the file. Note that the .SHX file must be available each time you edit the drawing.

If you respond to the LOAD command's prompt with "?", AutoCAD will display a list of the currently-loaded Shape files.

4.10.2 SHAPE Command

After the Shape definition files have been loaded, you can place a Shape in the drawing with the command:

Command: SHAPE Shape name (or ?): (Shape name)

Starting point: (Shape origin)

Height <1.0>: (number or point)

Rotation angle <0.0>: (number or point)

The Shape name is specified in the file LOADED into memory. AutoCAD searches the loaded Shape files for a matching name, and displays the message "****Invalid****" if none is found.

The name of the last Shape you drew in this editing session is provided as the default Shape name; to draw the same Shape again, give a null response to the first prompt.

The values specified for height and angle are used to control the way the Shape is drawn. Their exact effect varies for each Shape. We suggest that you maintain documentation on the use of all Shapes in your library. You can enter the height and angle for a Shape either as numbers or as two points that "show" AutoCAD the height (as the distance between the points) or the angle (as the direction from the first point to the second). A default height of 1.0 is provided, and the default rotation angle is determined by the current Snap grid rotation, if any. If Ortho mode is on, the angle is forced to be orthogonal. To select the defaults, give a null response.

Dynamic Shape Specification (+2)

The ADE-2 package offers a special feature for Shape insertion. If you respond to the "Starting point", "Height", or "Rotation angle" prompts by entering "DRAG", and if DRAGMODE (Chapter 6) is ON, you can move the Shape into position, scale it, or rotate it, all using your pointing device. At each step, interim versions of the Shape are displayed, so that you can position it in your drawing visually.

We suggest that you avoid dragging very complex Shapes, since drawing, erasing, and redrawing such Shapes as you move them around can take a long time.

Obtaining a List of Shapes

If you forget the name used to refer to a Shape, you can obtain a list of the currently loaded (available) Shapes by issuing the SHAPE command, and responding to the "Name:" prompt with a "?". For example:

Command: SHAPE Shape name (or ?): ?

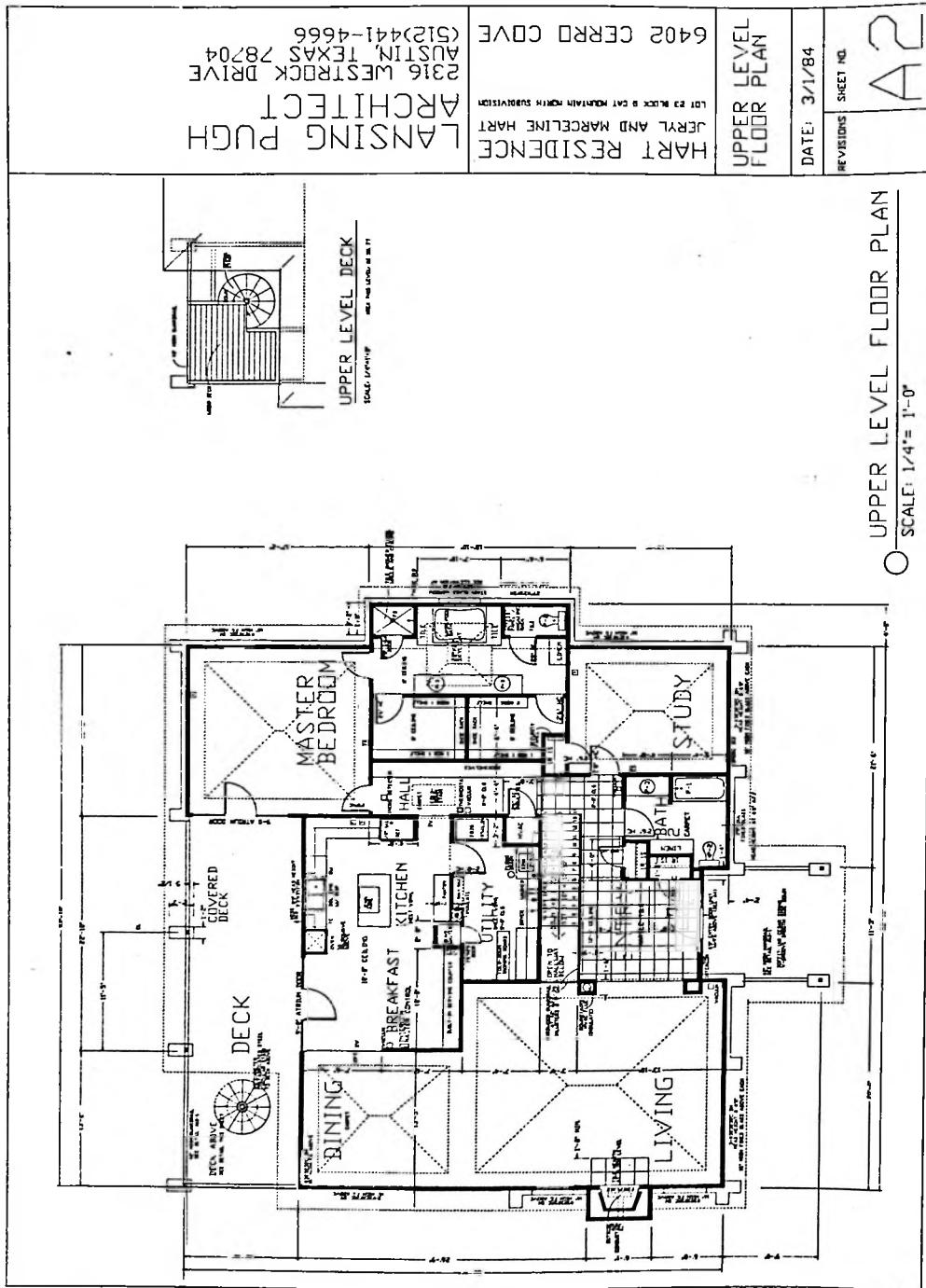
Available Shapes:

File: PC

FEEDTHRU	DIP8
DIP14	DIP16
DIP24	DIP40
DIP64	
...	

Here, seven Shapes are listed. All were loaded from file PC.SHX.

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Chapter 5

EDIT AND INQUIRY COMMANDS

5.1 Entity Selection

Most of the commands described in this chapter ask you to select one or more objects for processing. This collection of objects is called the *selection-set*. You can add objects to, or remove objects from, the selection-set interactively; if you have the optional ADE-3 package, AutoCAD will highlight the selected objects on the screen to assist you. (The method of highlighting depends on the display hardware, and highlighting may not be implemented on some displays; see your AutoCAD Installation Guide / User Guide Supplement for details.)

When AutoCAD needs a selection-set, it prompts with:

Select objects or Window or Last:

Respond with one of the following:

a point This is known as "object pointing". It immediately scans the drawing, searching for an entity that crosses the designated point. That entity is then selected.

It is best not to specify a point at the intersection of two or more entities, since it is unpredictable which of the objects will be selected. To select a Solid, Trace, or wide Polyline, point to one of its edges, not to its solid-filled region.

M (Multiple) As noted above, normal object pointing scans the drawing immediately each time you designate a point. In a complex drawing with many entities, a noticeable delay may accompany each such scan. If you wish, you can cause AutoCAD to scan the drawing just once for a group of points; to do this, enter the "M" option. AutoCAD then repeats the "Select/Remove objects. . ." prompt. Designate as many points as you like, and then press RETURN to begin the scan.

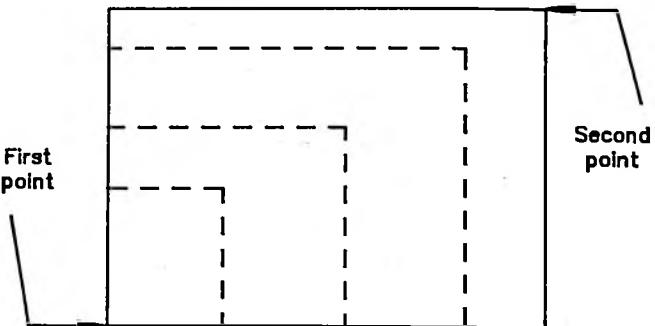
W (Window) This allows you to designate all the objects contained in a rectangular area, or "window". AutoCAD prompts you for two corner points describing the rectangle:

First point:

Second point:

Enter two points. AutoCAD displays a "box" cursor rather than the normal crosshairs so that you may see the window more clearly when pointing to the second corner. A box cursor is shown below. The dashed lines indicate the change in size of the box as you move the cursor.





Only the objects currently visible on the screen are considered for selection. If only part of an object is visible, it is selected if every part of it that's visible is within the window. AutoCAD highlights all the selected objects.

- L (Last)** This is an easy way to select the most recently drawn object that is currently visible on the screen. Only one object is designated, no matter how many times the "L" option is used when constructing a particular selection-set.
- U (Undo)** If you've added something to the selection-set inadvertently, use this option to remove it. You can step back through the selection-set; each "U" removes the group of entities most recently added to the selection-set.
- R (Remove)** The object selection process begins in "add" mode. That is, any newly designated objects are added to the selection-set. To switch to "remove" mode, use the "R" option. The "Select objects . . ." prompt changes to "Remove objects . . ." for "remove" mode.
- A (Add)** If you have switched to "remove" mode to remove objects from the selection-set, you can use the "A" option to return to "add" mode. As noted above, the object selection process begins in "add" mode.
- null reply** After each of the above options has been processed, the "Select/remove objects . . ." prompt reappears, and you can further manipulate the contents of the selection-set. To indicate that you are satisfied with the selection-set as it stands, enter a space or RETURN in response to the "Select/remove objects . . ." prompt.
- CTRL C** This aborts the selection process, discards the selection-set, and returns all highlighted objects to normal.

Whenever you add items to the selection-set, AutoCAD displays a message of the form:

n1 selected, n2 found (n3 duplicate)

to tell you how many items have been added. The last portion of the message is present only if some of the new items were already in the selection-set.

Likewise, when you remove items from the selection-set, the following message is displayed:

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n1 selected, n2 found, n3 removed.

A few examples will make the object selection process clearer. First, suppose you want to edit or inquire about the object you most recently drew. Simply enter:

Command: *(an edit/inquiry command)*

Select objects or Window or Last: L 1 found.

Select objects or Window or Last: RETURN *(to end selection process)*

... processing continues ...

That's all there is to it. Similarly, if you want to process one specific object, you can point to it as follows:

Command: *(an edit/inquiry command)*

Select objects or Window or Last: *(point to object)* 1 selected, 1 found.

Select objects or Window or Last: RETURN *(to end selection process)*

... processing continues ...

To process all the objects in a window, enter:

Command: *(an edit/inquiry command)*

Select objects or Window or Last: W

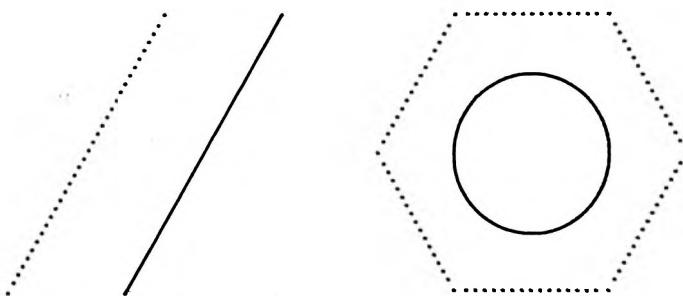
First point: *(point)*

Second point: *(point)* nnn found.

Select objects or Window or Last: RETURN *(to end selection process)*

... processing continues ...

Now let's look at a more complex example. In the figure below, the seven dotted lines are to be processed.



The six lines forming a hexagon can be designated by means of a window, but the window would include the circle, and we don't want to process that. We can do this editing easily, with only one call to the editing command itself, in the following manner:

Command: *(an edit/inquiry command)*

Select objects or Window or Last: *(designate line, point "A")* 1 selected, 1 found.

Select objects or Window or Last: W

First point: *(point "B")*

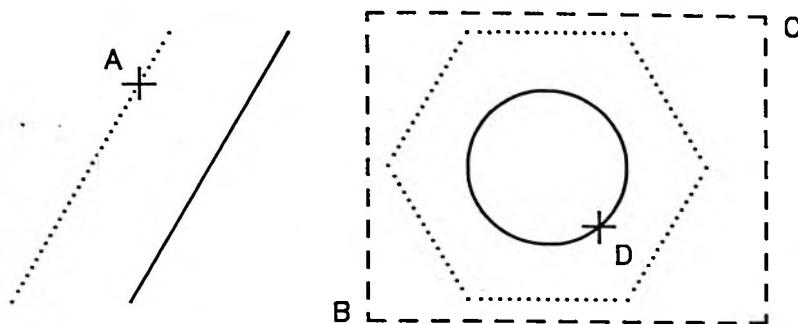
Second point: *(point "C")* 7 found.

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At this point, our selection-set contains 8 objects (everything except the one solid line). The circle is included in the selection-set because it was inside the window defined by points "A" and "B". Since we don't want to process the circle, we'll remove it from the selection-set by switching to "remove" mode and pointing to the circle.

Select objects or Window or Last: **R** (*to switch to "remove" mode*)

Remove objects or Window or Last: **(point "D")** 1 selected, 1 found, 1 removed.



Now the selection-set contains all the objects we want to process, and nothing else. We can now end the selection process and let the editing or inquiry command continue.

Remove objects or Window or Last: **RETURN** (*to end selection process*)
... processing continues ...

Of course, you could construct the same selection-set by pointing to each of the seven desired lines individually, or by windowing all the objects and then removing just the two unwanted ones. You have complete freedom to use whatever selection technique seems most appropriate for the situation at hand.

5.2 Edit Commands

Use these commands to modify your drawings.

5.2.1 ERASE Command

The ERASE command lets you specify entities that you want permanently removed from the drawing. The command format is:

Command: ERASE
Select objects or Window or Last: *(desired objects)*

A handy feature is the "ERASE Last" variation of the ERASE command. You can use this to erase the most recently drawn entity. You can use repeated "ERASE Last" commands to "step back" through the drawing, erasing the most recently drawn entity each time. For example, if you have just drawn a line followed by two circles, the command sequence:

Command: ERASE
Select objects or Window or Last: L
Select objects or Window or Last: RETURN *(end of selection)*
Command: RETURN *(to cause command repetition)*
Select objects or Window or Last: L
Select objects or Window or Last: RETURN *(end of selection)*

erases the two circles.

5.2.2 OOPS Command

The OOPS command restores entities that have been inadvertently ERASEd. Whenever the ERASE command is used, a list of entities erased is saved. The command:

Command: OOPS

restores all entities erased by the last ERASE command. Once another ERASE is done, the list of entities erased by the previous ERASE command is discarded, so OOPS cannot be used to restore them.

5.2.3 MOVE Command

The MOVE command lets you move one or more entities from their present location on a drawing to a new location. After you define the selection-set of objects to be moved, AutoCAD asks you to enter a displacement vector to indicate how far the objects are to be moved, and in what direction. You can do this by designating two points, giving the move-from point and then the move-to point, or by simply entering an x,y distance in response to the first prompt and RETURN in response to the second prompt. The directed distance you indicate is applied to all selected entities. Note that you are supplying a relative displacement; if you use the two-point method, the first point does not have to be on one of the selected objects, although that may help you visualize the displacement. The command format is:

Command: **MOVE**

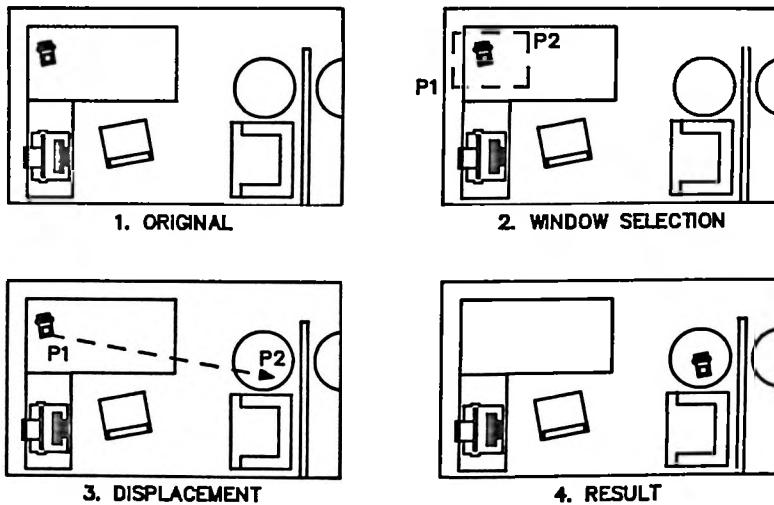
Select objects or Window or Last: *(show what to move)*

Base point or displacement: *(1st point or x,y distance)*

Second point of displacement: *(2nd point or just RETURN)*

If you have the ADE-2 package, you can enter "DRAG" when prompted for the second point of the displacement. This allows you to drag the selected objects into position visually.

The following sequence of figures shows the use of the MOVE command.



5.2.4 COPY Command

To copy existing objects, enter the COPY command. COPY is similar to the MOVE command described above, but it places copies at the specified displacement, leaving the originals intact.

Command: COPY

Select objects or Window or Last: (show what to copy)

Base point or displacement: (1st point or x,y distance)

Second point of displacement: (2nd point or just RETURN)

As for the MOVE command, you can use the ADE-2 package's "DRAG" feature to specify the second point of the displacement.

5.2.5 MIRROR Command (+2)

The ADE-2 package's MIRROR command lets you make mirror images of existing objects in your drawing, either deleting or retaining the original objects. The format is:

Command: MIRROR

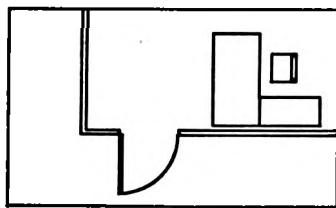
Select objects or Window or Last: (items to be mirrored)

First point of mirror line: (point)

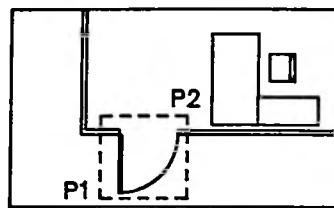
Second point: (point)

Delete old objects? <N> (Y or N)

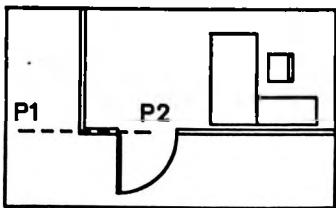
The mirror line you designate is the axis about which the selected objects are mirrored; it must be either horizontal or vertical. To assist you in specifying the mirror line, AutoCAD turns Ortho mode (Chapter 8) on temporarily if it is not already on. The following figures illustrate use of the MIRROR command. We have elected to delete the old objects.



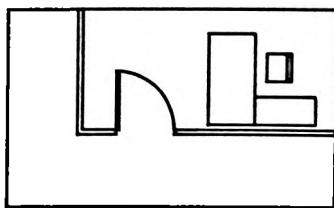
1. ORIGINAL



2. WINDOW SELECTION



3. MIRROR LINE SPECIFICATION



4. RESULT

5.2.6 CHANGE Command

The CHANGE command allows you to modify the properties of existing entities. First you must indicate which objects you want to change.

Command: CHANGE

Select objects or Window or Last: (desired objects)

When you have chosen the objects to be changed, the following prompt appears:

Change point (or Layer or Elevation):

You can reply with a point, or with an "L" to change the layer of the objects you have selected. The "Elevation" option is displayed only if the ADE-3 package is present, and relates to 3D visualizations. This topic is covered in Chapter 14. The other options are explained below.

5.2.6.1 Changing Layers

You can change the layer upon which the selected entities reside. To do this, respond to the "Change point . ." prompt with an "L" followed by space or RETURN. AutoCAD then prompts for the new layer, as in:

Command: CHANGE

Select objects or Window or Last: (desired objects)

Change point (or Layer or Elevation): L

New layer: (layer name)

Each selected object is moved to the new layer, acquiring the color and linetype associated with that layer. This form of the CHANGE command works for all entity types.

5.2.6.2 Changing Entity Properties

The properties of existing Lines, Circles, Text, Attribute Definitions, and inserted Blocks can be changed by responding to the "Change point . ." prompt with a point. AutoCAD considers this point the *change point*, and uses it to change some property of the entity you have chosen. The operation performed depends on the type of entity selected, as outlined below. (In the following descriptions, "change point" is abbreviated "CP".)

Line

The endpoint closer to the CP is changed to the designated CP. If Ortho mode (Section 8.4) is on, only orthogonal lines will result from a CHANGE.

Circle

The radius is changed so that the circumference of the circle passes through the specified CP. (The distance from the circle's center to the CP becomes its new radius.)

Text

The CP becomes the new text location. (If you press RETURN when prompted for the CP, the text is not moved.) AutoCAD then prompts you for a new text style, height, rotation angle, and text string. You can specify the height and angle by entering a point relative to the CP, or relative to the original location if no CP was specified. Note that to

retain the old text string, you must press RETURN when the "New text:" prompt appears.

- | | |
|----------------------|---|
| Attribute Definition | You can change the text properties of an Attribute Definition just as you can change a Text entity. In addition, the Attribute Definition's tag, prompt string, and default value can be changed. The Attribute Definition must not yet be part of a Block, however. (Attributes are an ADE-2 feature.) |
| Block | The CP becomes the Block's new origin. (If you press RETURN when prompted for the CP, the Block is not moved.) AutoCAD then asks for a new rotation angle. You can enter the angle by designating a point relative to the CP, or relative to the Block's origin if a CP was not specified. |

If you have the ADE-2 package, you can enter "DRAG" when prompted for the change point. This allows you to drag the selected object into position on the screen.

5.2.6.3 Changing Multiple Entities

If the selection-set supplied to the CHANGE command contains multiple Line entities, the change point is applied to all those lines. Thus, given three lines that do not meet precisely, you can make them meet by means of the CHANGE command. Simply select all three lines, and designate the change point as shown in the following figure.



If the selection-set contains multiple objects other than Lines, the change point is ignored for those objects. Instead, AutoCAD prompts for a new insertion point (for a Block), a new radius (for a Circle), etc. If you have the ADE-3 package, the object currently being changed is highlighted. If the ADE-2 package is present, you can enter "DRAG" when prompted for the original "Change point . ." to drag every object as you change it, or you can enter "DRAG" when prompted for an individual object's new location.

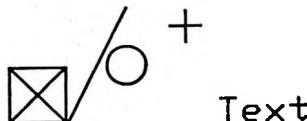
5.2.6.4 Examples

Examples of several CHANGE command sequences follow. The initial drawing consists of a line, a circle, a text string, and a Block composed of an "X" within a square. The text string has center alignment; the Block's insertion point is at its center, and it has an initial rotation of 0 degrees. The drawing is shown before and after each change. The command sequence is shown below each pair of figures, and the change point is indicated by a small cross.

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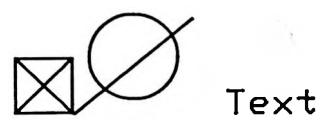
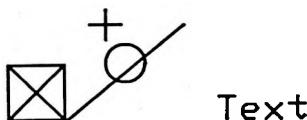
Initial drawing



Command: **CHANGE**

Select objects or Window or Last: *(point to line)*

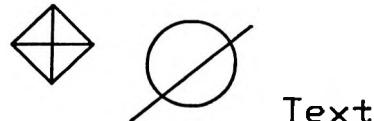
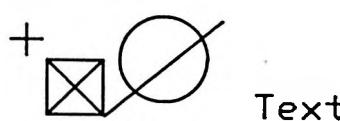
Change point (or Layer or Elevation): *(select indicated point)*



Command: **RETURN** *(to repeat the CHANGE command)*

Select objects or Window or Last: *(point to circle)*

Change point (or Layer or Elevation): *(select indicated point)*

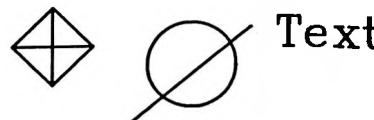
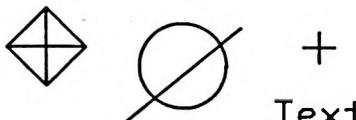


Command: **RETURN** *(to repeat the CHANGE command)*

Select objects or Window or Last: *(point to Block)*

Change point (or Layer or Elevation): *(select indicated point)*

New rotation angle <0>: 45



Command: **RETURN** *(to repeat the CHANGE command)*

Select objects or Window or Last: *(point to text)*

Change point (or Layer or Elevation): *(select indicated point)*

Text style: STANDARD.

New style or RETURN for no change: FANCY

New height <.15>: 2

New rotation angle <0>: RETURN

New text <Text>: RETURN

5.2.7 BREAK Command - Partial Erase (+1)

The ADE-1 package's BREAK command erases part of a Line, Trace, Circle, or Arc, or splits the object into two objects of the same type. BREAK does not work on Polylines; to edit Polylines, see the PEDIT command later in this chapter.

To break an object, enter the BREAK command. Then select the object to be broken, and point to the two ends of the desired break.

Command: BREAK

Select object: (specify object to be broken)

Enter first point: (Point to one end of deletion)

Enter second point: (Point to other end of deletion)

Any of the standard object selection methods (pointing, windowing, or Last) may be used. Pointing, however, results in a slightly different prompt sequence. If you point to the object, AutoCAD responds with:

Enter second point or F:

In this case, the "Enter first point:" prompt is skipped; AutoCAD assumes that the point used to select the object is also the point where you want the break to begin. If this is not actually the case, respond "F" to let AutoCAD know that it should ask for both the first and second end points of the deletion.

The second point need not be anywhere near the object; AutoCAD finds the nearest point on the object. If you want to cut off one end of a line, trace, or arc, the second point can be somewhere beyond the end to be cut off. If you simply want to split the object into two objects, without deleting anything, enter the same point for both the first and second points of the break. You can do this easily, by entering "@" (last coordinate) when AutoCAD asks for the second point.

The exact effect of the BREAK command when the two break points are not equal depends on the type of object being broken.

Line If both specified points are within the ends of the line, the line is split into two lines. If either point is at one end of the line, or if the second point is beyond one end of the line, that end is cut off.

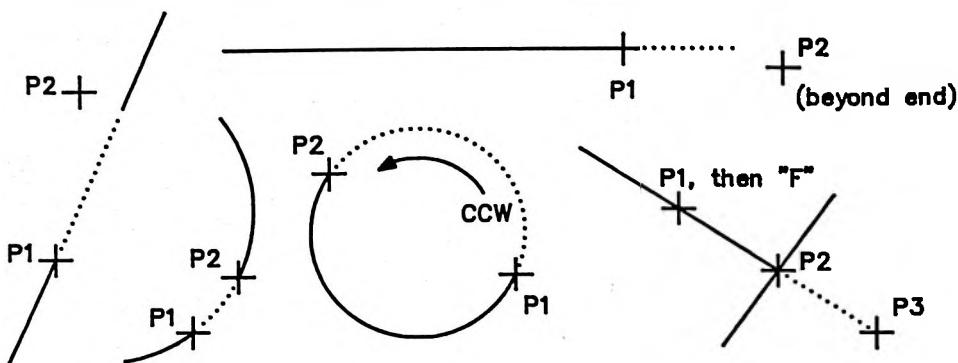
Trace A trace is broken the same way as a line. The broken ends are cut square.

Circle A circle is changed to an arc by removing a piece going *counterclockwise* from the first to the second point.

Arc As with a line, an arc is divided in two if both specified points are within the angle of the arc. The arc is shortened if the second point is off the end.

NOTE: If the second point is too far off the end, it may be nearer the other end, and the wrong part of the arc could be deleted.

Examples of breaking entities follow. The dotted lines indicate which portion of each figure is erased.



5.2.8 FILLET Command (+1)

The ADE-1 package's FILLET command connects two lines by means of a smoothly fitted arc of a specified radius. It adjusts the lengths of the lines so that they end exactly on the arc.

Command: **FILLET**

Polyline/Radius/<Select two lines>: (Point to two lines)

By default, FILLET expects you to select two lines or two straight segments of the same Polyline. Alternatively, you can enter one of the other options, to be described shortly. (The "Polyline" option is displayed only if the ADE-3 package is present.) After pointing to the two lines, *do not* press RETURN. The FILLET command knows that you will select only two points. You can select the lines with a Window specification, but this is risky; if there are more than two lines in the window, two lines are selected arbitrarily (usually the last two lines drawn).

If the two lines you select are actually segments of a Polyline, they must be adjacent or separated by one other segment. If they are separated by one segment, the FILLET operation will delete that segment and replace it with a fillet arc.

The radius to be used for fillets is remembered as part of the drawing file. The initial fillet radius for a new drawing is governed by the prototype drawing. To change the fillet radius, reply "R" to the prompt.

Command: **FILLET**

Polyline/Radius/<Select two lines>: R

Enter fillet radius: (value)

You can enter the radius numerically, or you can "show" AutoCAD the desired radius by designating two points. AutoCAD will set the radius equal to the distance between those points.

FILLET works by extending the two selected lines, if necessary, until they intersect. The lines are then trimmed and the fillet arc created. (A fillet radius of zero can be used to adjust two lines so that they end precisely at the same point.) If both lines are on the same drawing layer, the fillet is placed on that layer. If the lines are on different layers, the fillet is

placed on the current layer. If no intersection point is found within the drawing limits (and the limits check is ON), the command is rejected.

Given a fillet radius of 0.5, the effect of a FILLET command is shown below.

Command: FILLET
 Polyline/Radius/<Select two lines>: (Point to the two lines)



5.2.8.1 Filleting an Entire Polyline (+3)

If the ADE-3 package is present, you can use the FILLET command's "Polyline" option to apply fillets to an entire Polyline, or to remove fillets from a Polyline. The command sequence to do this is:

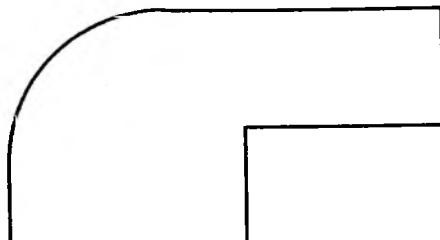
Command: FILLET
 Polyline/Radius/<Select two lines>: P
 Select polyline: (Select a single Polyline)

If you have set a nonzero fillet radius, AutoCAD will insert fillet arcs at each vertex where two line segments meet. If two line segments are separated by one arc segment, and the two line segments do not diverge as they approach the arc segment, the arc segment is removed and replaced by a fillet arc.

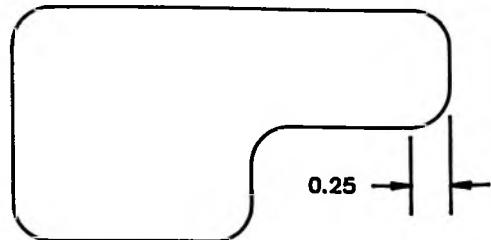
If the fillet radius is zero, the result is similar, except that no fillet arcs are inserted. If two line segments are separated by one arc segment (possibly from a previous FILLET command), that arc is removed and the lines are extended until they intersect. (However, if the lines diverge as they approach the arc segment, nothing is changed.)

For example, given the Polyline shown at the left, the following command sequence will produce the filleted Polyline shown at the right.

Command: FILLET
 Polyline/Radius/<Select two lines>: R
 Enter fillet radius: 0.25
 Command: RETURN *(to cause command repetition)*
 Polyline/Radius/<Select two lines>: P
 Select polyline: (do so)
 6 lines were filleted



BEFORE



AFTER

Parallel line segments, segments that are too short to be filleted, those that intersect outside the drawing limits (when the limits check is on), and those that diverge as they approach a separating arc segment cannot be filleted. Upon termination, the FILLET command reports the count of each such condition (if any were encountered). For example:

2 were parallel 8 were out of limits 3 were too short 2 were divergent

5.2.9 CHAMFER Command (+1)

The ADE-1 package's CHAMFER command trims two intersecting lines a specified distance from the intersection and connects the trimmed ends with a new line segment. Its operation is similar to that of the FILLET command.

Command: CHAMFER

Polyline/Distances/<Select first line>: (Point to one line)

Select second line: (Point to intersecting line)

By default, CHAMFER expects you to select two lines or two straight segments of the same Polyline. Alternatively, you can enter one of the other options, to be described shortly. (The "Polyline" option is displayed only if the ADE-3 package is present.) After pointing to the two lines, *do not* press RETURN. The CHAMFER command knows that you will select only two points. You can select the lines with a Window specification, but this is risky; if there are more than two lines in the window, two lines are selected arbitrarily (usually the last two lines drawn).

If the two lines you select are actually segments of a Polyline, they must be adjacent or separated by one arc segment. If they are separated by an arc segment, the CHAMFER operation will delete the arc and replace it with a chamfer line.

The distance that lines are to be trimmed from their intersection point when chamfering is remembered as part of the drawing file. The initial chamfer distance for a new drawing is determined by the prototype drawing. To change the chamfer distance, reply "D" to the prompt.

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Command: **CHAMFER**

Polyline/Distances/<Select first line>: **D**

Enter first chamfer distance <default>: **(value)**

Enter second chamfer distance <default>: **(value)**

You can enter each distance numerically, or you can "show" AutoCAD the desired distance by designating two points. AutoCAD will measure the distance between those points and use that value. The default for the first distance is the most recent chamfer distance specified. The default for the second distance is whatever you chose for the first distance, so symmetrical chamfers are the norm.

CHAMFER works by extending the two selected lines, if necessary, until they intersect. It then trims the first line by the first distance, trims the second line by the second distance, and connects the trimmed ends with a straight line. If the first and second distances are both zero, the two lines are extended, but no chamfer line is drawn; you can use this feature to adjust two lines so that they end precisely at the same point.

If both lines are on the same drawing layer, the chamfer is placed on that layer. If the lines are on different layers, the chamfer is placed on the current layer. If no intersection point is found within the drawing limits (and the limits check is ON), the command is rejected.

The following command sequence sets the first and second chamfer distances to 0.5 and 0.25, respectively, and then chamfers two lines. The effect of these CHAMFER commands is shown in the figure below.

Command: **CHAMFER**

Polyline/Distance/<Select first line>: **D**

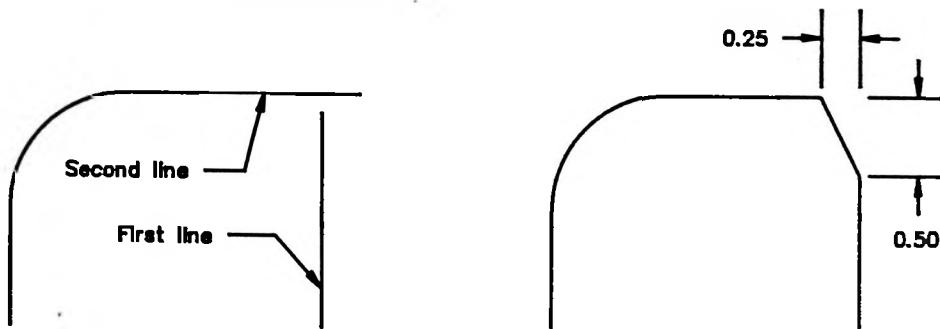
Enter first chamfer distance <0.0>: **0.5**

Enter second chamfer distance <0.5>: **0.25**

Command: **RETURN** (*to cause command repetition*)

Polyline/Distance/<Select first line>: **(Point to first line)**

Select second line: **(Point to second line)**



BEFORE

AFTER

5.2.9.1 Chamfering an Entire Polyline (+3)

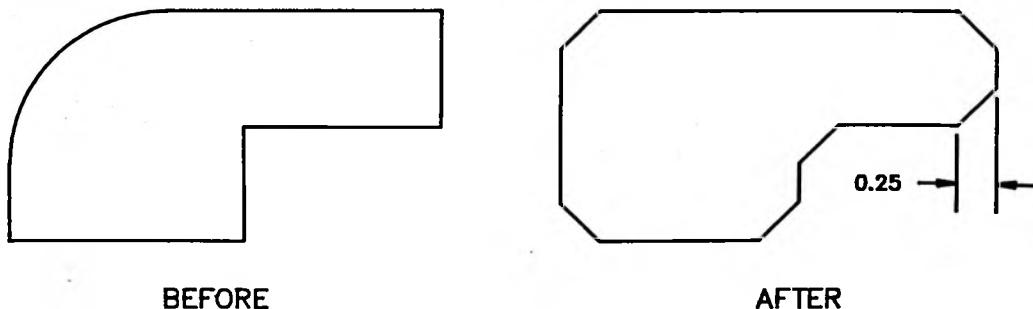
If the ADE-3 package is present, you can use the CHAMFER command's "Polyline" option to apply chamfers to an entire Polyline. The command sequence to do this is:

Command: CHAMFER
 Polyline/Distance/<Select first line>: P
 Select polyline: (Select a single Polyline)

The CHAMFER command follows the Polyline from the beginning, treating each straight segment, in turn, as the "first line". The "second line" is the following straight segment, which must be adjacent to the current segment or be separated from it by one other segment. If the Polyline is "closed", its first segment will be treated as the "second line" when CHAMFER reaches the last segment. Therefore, you may find the "CHAMFER Polyline" option appropriate only when the chamfer is to be symmetric (that is, when the first and second chamfer distances are equal).

For example, given the Polyline shown at the left, the following command sequence will produce the chamfered Polyline shown at the right.

Command: CHAMFER
 Polyline/Distance/<Select first line>: D
 Enter first chamfer distance <0.0>: 0.5
 Enter second chamfer distance <0.5>: RETURN (*to set second = first*)
 Command: RETURN (*to cause command repetition*)
 Polyline/Distance/<Select first line>: P
 Select polyline: (do so)
 6 lines were chamfered



As with the FILLET command, a count is printed of any conditions that prevent AutoCAD from chamfering the entire Polyline.

Chamfers added to a Polyline become new segments of that Polyline. Although you can un-fillet a Polyline by filleting with a zero radius, zero chamfer distances will not remove chamfers from a Polyline; AutoCAD has no way to distinguish a chamfer from any other straight segment of the Polyline.

5.2.10 ARRAY Command

The ARRAY command allows you to make multiple copies of selected objects in a rectangular or circular pattern. Each resulting object can be manipulated independently. The command format is:

Command: ARRAY

Select objects or Window or Last: (Show what to copy)

Rectangular or circular array (R/C):

The operation of the ARRAY command differs depending on which type of array (rectangular or circular) you choose to create. Therefore, each case is discussed separately.

5.2.10.1 Rectangular Arrays

If a rectangular array is selected, AutoCAD asks for the number of (horizontal) rows and (vertical) columns to be constructed. The default for each of these is 1. A rectangular array is constructed by replicating a "cornerstone" element (the objects you select) the appropriate number of times. Therefore, an array with a column and row count of 1 is meaningless and is rejected.

If the number of rows and columns you have specified would result in a very large number of replications, the ARRAY command asks whether that's really what you want:

This command will repeat the selected items *nnn* times.

Do you really want to do this?

Only a reply beginning with "Y" will allow the command to continue.

The next prompt is:

Unit cell or distance between rows:

At this point, you may enter a number to indicate the distance between adjacent rows in the array; AutoCAD then asks for the distance between columns. (Negative numbers indicate rows to be added downward and columns added to the left.) Alternatively, you may respond to the "Unit cell . . ." prompt by designating two opposite corners of a rectangle to "show" AutoCAD the row and column spacing in one operation.

Once the row and column spacing have been specified, the ARRAY command commences construction of the array. You can terminate this process before completion by entering CTRL C.

For example, suppose you have drawn a circle and a line as follows:

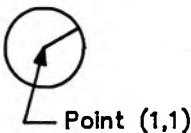
Command: CIRCLE Center point: 1,1

Radius (or D): .25

Command: LINE From point: 1,1

To point: @25<30

To point: RETURN



and you want to make an array 5 columns wide and 2 rows high. You can use the following command sequence:

Command: **ARRAY**

Select objects or Window or Last: *(window the two objects)*

Rectangular or circular array (R/C): **R**

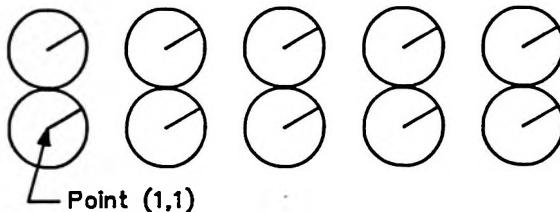
Number of rows: **2**

Number of columns: **5**

Unit cell or distance between rows: **.5**

Distance between columns: **.75**

The resulting figure is shown below.



5.2.10.2 Circular Arrays

If you reply to the array type prompt with "C" to create a circular array, the next prompt is:

Center point of array:

Respond with the point around which the selected objects are to be rotated to form the array.
Next the prompt:

Angle between items (+=CCW, -=CW):

appears. You should reply with the desired angular spacing between items, in degrees. If the angle is positive, successive items are inserted in the counterclockwise direction; if negative clockwise. Angles of zero degrees and those greater than or equal to 360 degrees are rejected.

The next prompt is:

Number of items or -(degrees to fill):

If you enter a positive number, this is taken to be the number of elements in the array (counting the original "cornerstone" element). A negative response is used to indicate the

number of degrees the array is to occupy. For convenience, a zero reply to this prompt defaults to a full circle, and is equivalent to a reply of "-360".

In order to construct the circular array, AutoCAD determines the distance from the center point to a reference point on each entity selected. The points used depend on the type of entity:

Point	- insertion point
Circle, Arc	- center point
Block, Shape	- insertion base point
Text	- starting point
Line, Trace	- one end point

(The measurement point for Lines and Traces is not predictable.) The replicated entity is placed at the measured distance from the array's center, after rotating the designated angle between items.

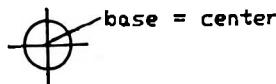
Note that a different reference point is used for each object being replicated. Thus, the objects may shift with respect to each other in each replication. To avoid this, place the desired objects in a Block (see Chapter 9) and replicate that.

The last prompt is:

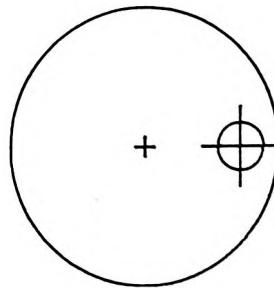
Rotate objects as they are copied? <N>

A "Y" response causes the replicated objects to be rotated in accordance with the rotation of the array itself. This rotation is illustrated in the following example:

Suppose we have defined a Block that looks like:



and that we've inserted one of them in the drawing:



Then we enter the command sequence:

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Command: ARRAY

Select objects or Window or Last: (point to the Block)

Rectangular or circular array (R/C): C

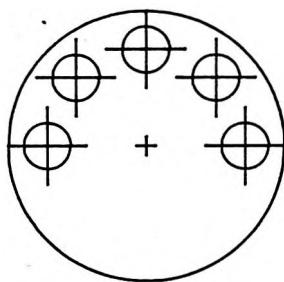
Center point of array: (select center of large circle)

Angle between items (+=CCW, -=CW): 45

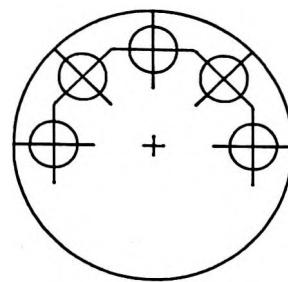
Number of items or -(degrees to fill): 5

Rotate objects as they are copied? <N>

The result is shown below, for both possible replies to the "Rotate objects . ." question. Note that the "number of items" is 5, counting the original cornerstone element.



BLOCK NOT ROTATED



BLOCK ROTATED
AROUND ITS BASE

5.2.11 REPEAT and ENDREP Commands

The REPEAT and ENDREP commands provide an alternate method of creating rectangular patterns. These commands create REPEAT and ENDREP entities surrounding the objects in the pattern; thus it is sometimes difficult to manipulate individual repeated objects at a later date. The ARRAY command, described in the previous section, is usually preferable for creating rectangular arrays.

To begin a Repeat group, enter:

Command: REPEAT

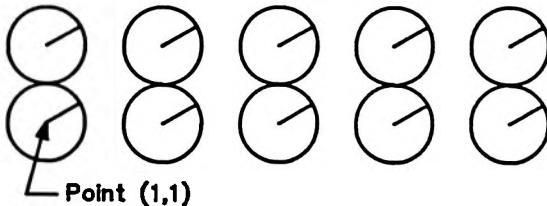
Next use other AutoCAD commands to draw the basic pattern (one occurrence of the pattern). You may use as many commands as desired. For instance, to produce the same figure as in the rectangular ARRAY example, you could enter:

Command: CIRCLE 3P/2P/<Center point>: 1,1
Diameter/<Radius>: .25

Command: LINE From point: 1,1
To point: @.25<30
To point: RETURN

When you have drawn the basic pattern, enter the command ENDREP. AutoCAD asks for the number of columns and rows and for the spacing between them; then it draws the repeated items.

Command: ENDREP
Columns: 5 Rows: 2
Column distance: 0.75
Row distance: 0.5



You can enter the column and row distances by "showing" AutoCAD two opposite corners of a rectangle defining one cell of the pattern. You can use REPEAT and ENDREP within other REPEAT/ENDREP groups without limit.

5.2.12 PEDIT Command - Polyline Editing (+3)

The ADE-3 package's PEDIT command allows you to edit Polylines in numerous ways. Using PEDIT, you can:

- o change the entire Polyline to have a new uniform width.
- o change the width and taper of individual segments of a Polyline.
- o close an open Polyline, or open a closed one.
- o remove all kinks and curves between two vertices.
- o break a Polyline into two Polylines.
- o join any number of contiguous Lines, Arcs, and Polylines into a single Polyline.
- o move selected vertices of a Polyline, or add new vertices.
- o fit a curve to all vertices in the Polyline, with optional specification of the tangent at each vertex.

To begin editing a Polyline, enter the PEDIT command.

Command: **PEDIT**

Select polyline:

Select a single entity by any of the object selection methods. AutoCAD checks to see whether this entity is a Polyline. If the selected entity is a Line or an Arc, AutoCAD prompts:

Entity selected is not a polyline.
Do you want it to turn into one?

If you respond "Y", the object will be converted into a single-segment Polyline, which you can then edit. You can use this operation when starting to join Lines and Arcs into a Polyline.

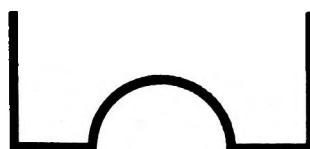
Once the Polyline has been selected, AutoCAD prompts with a list of PEDIT options.

Close/Join/Width/Edit vertex/Fit curve/Uncurve/eXit <X>:

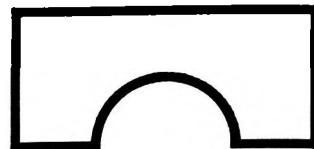
("Close" will be replaced by "Open" if the Polyline is currently closed.) To select an option, enter just the capitalized letter for that option shown in the prompt; for instance, "E" for "Edit vertex". The individual options are described below.

C (Close)

This option creates the closing segment of the Polyline, connecting the last segment with the first as shown in the following figure.



Before CLOSE



After CLOSE

If you have drawn a segment back to the starting point explicitly, closing the Polyline will have no visible effect.

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O (Open) This option removes the closing segment of the Polyline. If you have drawn a segment back to the starting point explicitly, opening the Polyline will have no visible effect.

J (Join) This option will find Lines, Arcs, and other Polylines that meet this Polyline at either end, and will add them to the Polyline. You can use this option only if the Polyline is open (see above). When you select the "Join" option, AutoCAD prompts:

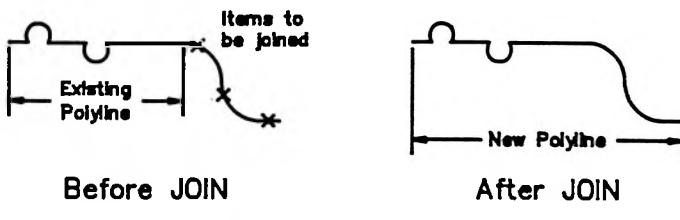
Select objects or Window or Last:

and allows you to select the candidate objects for joining to the Polyline. The Polyline itself may be included in the selection-set without any ill effect.

Once you have chosen the candidate objects, AutoCAD searches them to find any Line, Arc, or Polyline that shares an endpoint with the current Polyline, and merges that object into the Polyline. It then repeats the search using the new endpoints of the Polyline, until the search fails (either because all the candidate objects have been joined or because some do not meet the Polyline).

To be joined, an object must have a virtually exact match with one of the Polyline's endpoints; AutoCAD does not use guesswork to extend objects that are "pretty close" to the Polyline. If a line crosses the end of a Polyline in a "T", it will not be joined. If two lines meet a Polyline in a "Y", one of them will be selected arbitrarily and joined.

An example is shown below.



If it is unclear which entities have been joined to the Polyline, you can highlight them by means of the LIST command, described later in this chapter. When you point to the Polyline, all its segments will be highlighted; you can then enter CTRL C to avoid an actual listing.

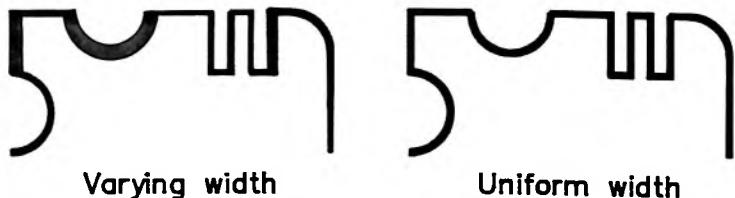
W (Width) This option lets you specify a new uniform width for the entire Polyline. AutoCAD prompts:

Enter new width for all segments:

You can enter the width from the keyboard, or you can "show" AutoCAD the width by designating two points. As usual, AutoCAD

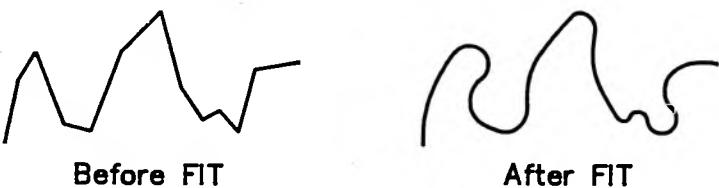
will compute the distance between those points and use that as the width. Once you have specified the width, the Polyline is redrawn accordingly.

As shown in the following figure, you can use the "Width" option to remove width variations from a Polyline.



E (Edit vertex) This option allows you to select one vertex of the Polyline and perform various editing tasks on that vertex and the segments that follow it. The details of Polyline vertex editing are covered in the next section.

F (Fit curve) This option computes a smooth curve fitting all the vertices of the Polyline, using any tangent directions that you have specified. The curve consists of a pair of arcs joining each pair of vertices. AutoCAD inserts extra vertices in the Polyline to accomplish this.



If the resulting curve does not appear as you expected, use the "Edit vertex" option to assign tangent directions or add more vertices (to better define the curve), and try again.

U (Uncurve) This option removes any extra vertices inserted by the "Fit curve" operation described above, and straightens all segments of the Polyline. Any tangent information you have assigned to the Polyline's vertices is retained for use in subsequent "Fit curve" requests.

X (eXit) This option exits the PEDIT command and returns to AutoCAD's "Command:" prompt. This option is the default response for the main PEDIT prompt.

After each PEDIT option (other than "eXit") is complete, the main PEDIT prompt is repeated so you can perform several editing operations with one invocation of the PEDIT command.

5.2.12.1 Vertex Editing

When you select the PEDIT command's "Edit vertex" option, AutoCAD marks the first vertex of the Polyline by drawing an "X" on the screen. If you have specified a tangent direction for this vertex, an arrow will also be drawn in that direction. AutoCAD then issues a new prompt with vertex editing sub-options.

Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit <N>:

You can abbreviate these sub-options to the capitalized letters shown in the prompt. When the selected operation is complete, the above prompt is repeated to permit further vertex editing. Each of the sub-options is described below.

N (Next) and
P (Previous)

These sub-options move the "X" marker to the next or previous vertex. They will not wrap around from the end to the start of the Polyline, even if the Polyline is closed. The default in the vertex editing prompt is whichever of these sub-options was chosen last. Therefore, you can move to a distant vertex by selecting "Next" or "Previous" once. Then simply press RETURN to step through to the desired vertex.

B (Break)

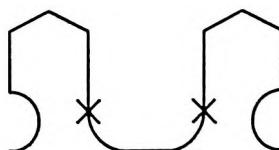
This sub-option remembers the location of the vertex that the "X" is on, and prompts:

Next/Previous/Go/eXit <N>:

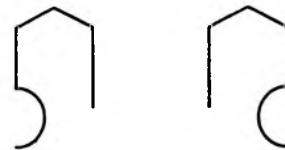
You can now move the "X" to any other vertex (or leave it where it is) and enter "Go". This will split the Polyline into two pieces at the specified vertex or vertices. Any segments and vertices between the two vertices you specified will be deleted. If either of the specified vertices is at an end of the Polyline, one truncated Polyline will result. It is not possible to delete all the vertices by selecting the two ends.

If you change your mind, enter "X" (for "eXit"). AutoCAD will cancel the break operation and return to the "Edit vertex" sub-options prompt.

For example, the following figure shows a Polyline being broken between the two marked vertices.



Before BREAK



After BREAK

NOTE: When you break a "closed" Polyline, it becomes "open" and the closing segment is removed automatically.

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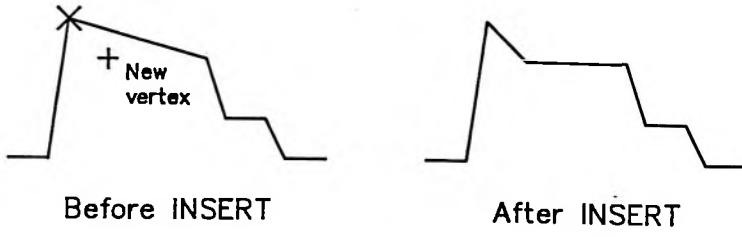
I (Insert)

Using the "Insert" sub-option, you can add a new vertex to the Polyline. AutoCAD prompts:

Enter location of new vertex:

The new vertex is added to the Polyline *after* the vertex that is currently marked with an "X".

The figure below shows a vertex being added to a Polyline. The leftmost end of the Polyline is its start point.

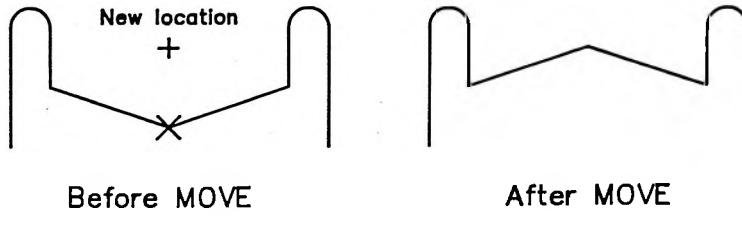


M (Move)

This sub-option allows you to move the currently-marked vertex to another location. AutoCAD prompts:

Enter new location:

The vertex that is currently marked with an "X" is moved to the location you designate. The following figure illustrates this.



R (Regen)

This sub-option regenerates the Polyline. It is used in connection with the "Width" sub-option, described below.

S (Straighten)

This sub-option saves the location of the vertex that the "X" is on, and prompts:

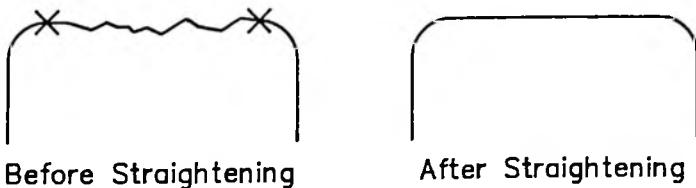
Next/Previous/Go/eXit <N>:

You can now move the "X" to any other vertex (or leave it where it is) and enter "Go". Any segments and vertices between the two vertices you specified will be deleted, and a single straight line segment will replace them. If you specify only one vertex (by entering "Go" without moving the "X" marker), the segment following that vertex will be made straight (if it is an arc).

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If you change your mind, enter "X" (for "eXit"). AutoCAD will cancel the straighten operation and return to the "Edit vertex" sub-options prompt.

In the following figure, two vertices have been selected, so the segments between them are replaced with one straight segment.



If you wish to remove an arc segment that connects two straight segments of a Polyline, and then extend the straight segments until they intersect, use AutoCAD's FILLET command, described earlier in this chapter.

T (Tangent)

This vertex editing sub-option lets you attach a tangent direction to the current vertex (marked by the "X") for later use in curve fitting. AutoCAD prompts:

Direction of tangent:

You can enter the desired tangent angle from the keyboard, or you can designate one point to "show" AutoCAD the direction from the currently-marked vertex.

W (Width)

This vertex editing sub-option lets you change the starting and ending widths for the segment that immediately follows the marked vertex. (If you're unsure which direction is forward and which is backward, the "Next" and "Previous" sub-options will show you.) Note that this is very different from the "Width" option of the main PEDIT prompt, which sets a new uniform width for the entire Polyline. AutoCAD prompts:

Enter starting width <current>:

Enter ending width <start>:

The default starting width is the current starting width for the segment. The default ending width is equal to the starting width you select.

The segment is not redrawn immediately after you specify its new width. If you wish to see it, use the "Regen" sub-option.

X (eXit)

This sub-option exits from vertex editing and returns to the main PEDIT prompt.

5.3 Inquiry Commands

These commands are used to inquire into locations and relationships between entities.

5.3.1 LIST Command

The LIST command lets you examine the data stored for an entity. The command format is:

Command: LIST

Select objects or Window or Last: (objects to list)

The information that is listed depends on the type of entity, but the entity's type, its position in the drawing, and the layer upon which it was drawn are always listed. Additional information derived from the basic parameters is often displayed as well.

The report for a Line entity, for example, includes not only the coordinates of its two endpoints, but also its length, the angle from the start point to the end point, and the change in X and Y from the start point to the end point (delta X and delta Y). Circles are described by their center point and radius, but the report includes the circumference and area as well.

For Polylines, LIST displays the coordinates and the tangent direction (if any) for each vertex. If the Polyline is closed, LIST also reports its precise area and perimeter. If the Polyline is open, LIST reports its length. It also pretends that a straight line connects the start and end points, and computes the enclosed area as well. For wide Polylines, all calculations are based on the center-lines of the wide segments, ignoring their widths.

Sometimes the information listed will not fit on the screen all at once. You can use CTRL S to pause momentarily; press any key to resume. A "printer echo" toggle key, CTRL Q, is also provided to let you send the LIST output (or any non-graphics text that AutoCAD displays on the screen) to your system's printer.

If you select many entities or a long Polyline, the LIST output may be very lengthy. You can use CTRL C to abort the listing and return to the "Command:" prompt.

5.3.2 DBLIST Command

The DBLIST command lists information about every entity in the drawing, and is primarily used in training exercises and debugging. The format is:

Command: DBLIST

Since this command can take a long time when there are a large number of entities in the drawing file, you can abort the DBLIST command and return to the command prompt by entering CTRL C. You can use CTRL S to momentarily stop the listing; pressing any key causes the listing to resume. You can also use the CTRL Q "printer echo" toggle key to send the DBLIST output to your system's printer.

5.3.3 DIST Command

The DIST command measures the distance and angle between two designated points, and displays the distance in drawing units. If your copy of AutoCAD includes the ADE-1 package, the result is printed using the current display format (see the UNITS command in Section 3.6).

Command: DIST

First point: (point)

Second point: (point)

Distance = <calculated distance> Angle = <angle>

Delta X = <change in X> Delta Y = <change in Y>

As an added convenience, if you enter a single decimal number in response to the "First point:" prompt, DIST displays that number in the format selected by the latest UNITS command (ADE-1 feature, see Section 3.6).

5.3.4 ID Command

The ID (identify) command lets you specify a point on the drawing and have the position of that point displayed in drawing coordinates.

Command: ID Point: (point to be displayed)

X = <X coordinate> Y = <Y coordinate> Z = <current elevation>

Alternatively, you can specify the point with numeric coordinates; AutoCAD draws a little blip to show you where that point is on the screen, unless BLIPMODE (Chapter 6) has been turned off. The blip vanishes when the screen is next redrawn.

5.3.5 AREA Command

The AREA command allows you to specify any number of points enclosing a space on the drawing. AutoCAD then calculates the area and perimeter of the enclosed space. The command format is:

Command: AREA

First point: (point)

Next point: (point)

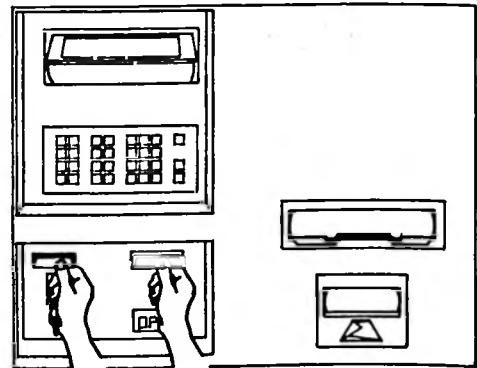
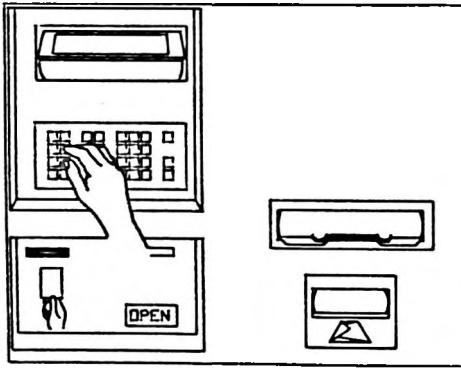
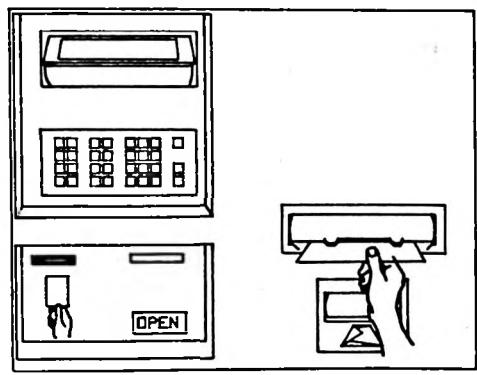
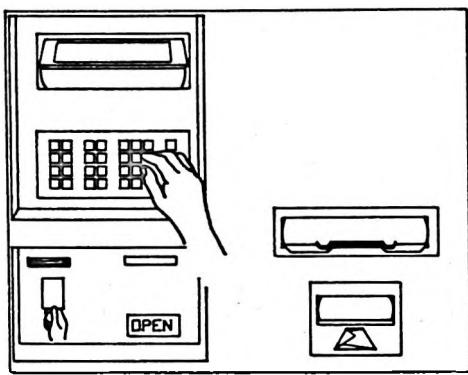
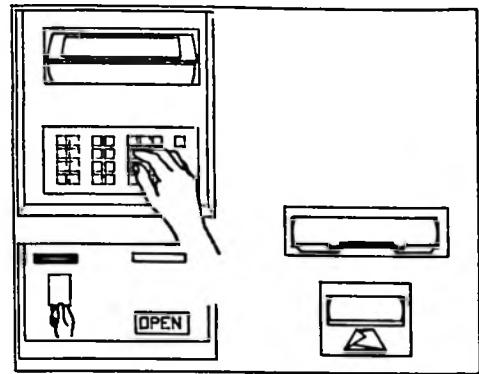
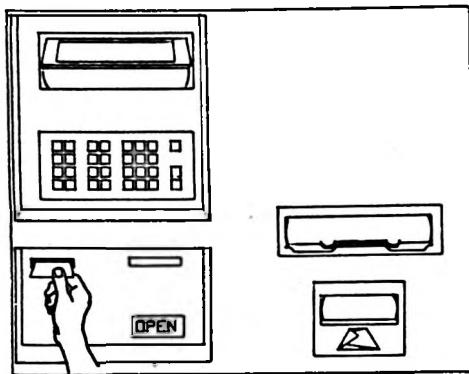
Next point: (point) and so on, finally

Next point: RETURN

Area = <calculated area>, Perimeter = <perimeter>

The enclosed space is assumed to be closed by connecting the last point specified to the first point. The polygon formed by the specified points need not be convex.

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Chapter 6

DISPLAY CONTROLS

The commands in Chapter 4 and Chapter 5 control the placement of entities within your drawing. In contrast, the commands in this chapter control how your drawing is displayed on the graphics monitor. Using these commands, you can control the position and magnification of the screen window, specify the degree to which AutoCAD elaborates time-consuming entities, and explicitly force the screen to be redrawn.

6.1 ZOOM Command

The ZOOM command acts like the zoom lens on a camera: it lets you increase or decrease the apparent size of items you are viewing, although their actual size remains constant. As you increase the apparent size of objects, you view a smaller area of the drawing in greater detail; decreasing the apparent size allows you to view a larger area.

The ZOOM command includes several options, each of which offers you a different way to designate the magnification and the portion of the drawing to be displayed. After you enter the ZOOM command, the following prompt appears:

Magnification or type (ACELPW):

You should respond by entering a magnification factor or one of the indicated ZOOM types. These options are summarized below and discussed in detail in the sections that follow.

Magnification	- A numeric zoom factor
A (All)	- Shows entire drawing (to drawing limits)
C (Center)	- Asks for center point and height
E (Extents)	- Shows entire drawing (to current extents)
L (Left corner)	- Asks for lower left corner and height
P (Previous)	- Restores previous view
W (Window)	- Asks for rectangle to be enlarged or reduced

6.1.1 ZOOM Magnification

The simplest type of ZOOM command lets you enter a display magnification factor. A magnification factor of 1 displays the entire drawing (the full view). If you enter any other number, the magnification is computed relative to the full view. For instance, if you enter a value of 2, each object appears twice as large as it does in the full view.

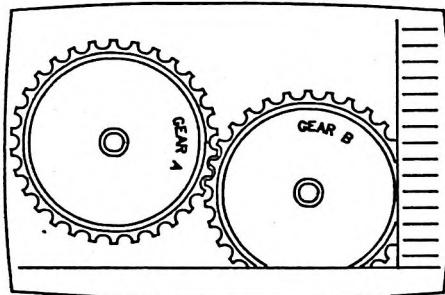
If you enter a number followed by "X", the magnification is computed relative to the current view. For example, entering "0.5X" causes each object to be displayed at 1/2 its current size on the screen. Note that only positive values may be used for magnification factors.

This ZOOM option does not change the location of the display center. Thus, when you increase the magnification, entities near the edge of the display may be forced off-screen. On the other hand, when you decrease the display magnification, areas outside the drawing limits may become visible.

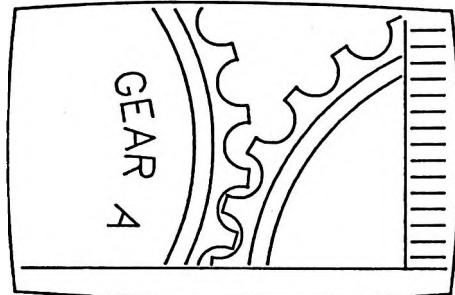
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The following example increases the display magnification by a factor of 3 relative to its current value:

Command: ZOOM Magnification or type (ACELPW): 3X



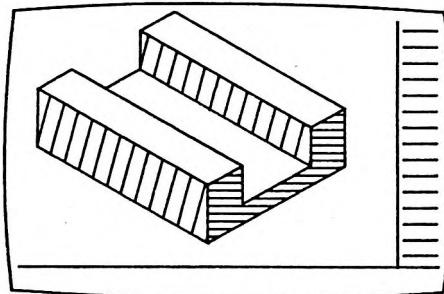
Before



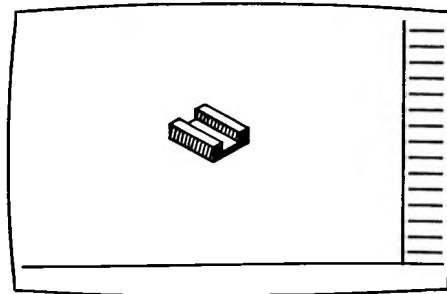
After ZOOM 3X

Similarly, the following example decreases the magnification to 1/4 that of the full view:

Command: ZOOM Magnification or type (ACELPW): .25



Full view



After ZOOM .25

6.1.2 ZOOM All

The ZOOM All command:

Command: ZOOM Magnification or type (ACELPW): **A**

changes the display so that you see all of your drawing on the screen, based on its limits or current extents (whichever are greater). If the drawing has been extended outside the drawing limits, the display also extends outside the drawing limits to show all entities in the drawing in their entirety. Occasionally, ZOOM All has to generate the drawing twice. When it does so, it displays the message:

** Second regeneration caused by change in drawing extents.

Zoom All also resets the "Drawing uses" extents that appear in the report generated by the STATUS command (described in Chapter 3). Any increase in the drawing extents is automatically reflected in the STATUS display; however, reductions in the extents are not reported by the STATUS command until you perform a ZOOM All or ZOOM Extents (see below).

6.1.3 ZOOM Extents

As noted above, the ZOOM All command displays the entire drawing surface, even if only a relatively small portion of it contains any entities. Thus, the entities may be very small when displayed. The ZOOM Extents command:

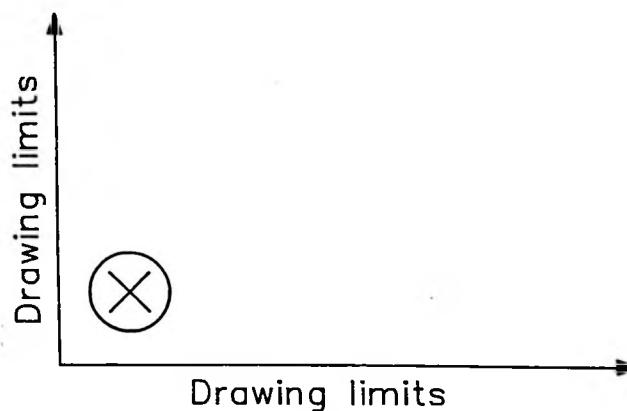
Command: ZOOM Magnification or type (ACELPW): **E**

uses only the current drawing extents, not its limits, and results in the largest possible display of all the objects in the drawing. Like ZOOM All, ZOOM Extents resets the "Drawing uses" extents for the STATUS display.

The figures on the following page illustrate the difference between ZOOM All and ZOOM Extents.

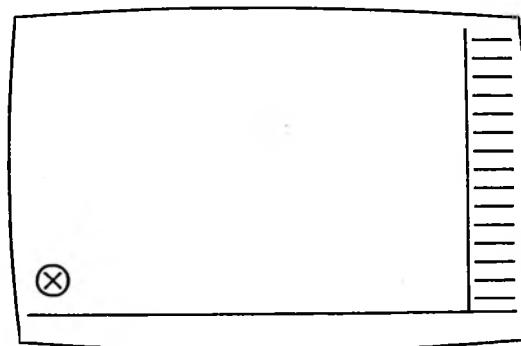
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Given the drawing shown below:

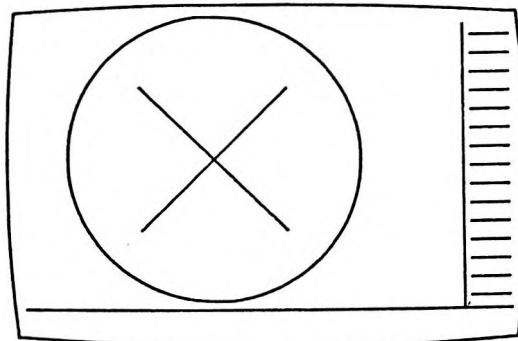


ZOOM All and ZOOM Extents would generate the following displays:

ZOOM
All



ZOOM
Extents



6.1.4 ZOOM Window

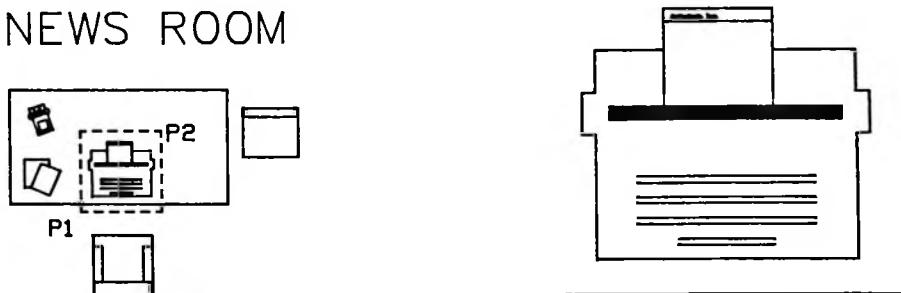
The ZOOM Window command allows you to specify the area you wish to see by entering two opposite corner points of a rectangular window. The center of the window becomes the new display center, and the area inside the window is enlarged or reduced to fill the display as completely as possible. You can enter points by coordinates or with the pointing device.

The command format is:

Command: ZOOM Magnification or type (ACELPW): W
First point: (point)
Second point: (point)

For the ZOOM Window command, a "box" cursor is displayed rather than the normal crosshairs so that you may see the window more clearly when pointing to the second corner. Use of the box cursor is demonstrated in the following example:

This display becomes this display



after issuing ZOOM W and entering the two indicated points.

6.1.5 ZOOM Center

The ZOOM Center command lets you specify your display window location by entering the desired center point of the new display. You can also optionally specify the height of the window in drawing units. The current value of the height is displayed in the prompt. If you do not specify a height value, then the magnification remains unchanged. If you specify a smaller value for the height, the magnification is increased, and if you specify a larger value, the magnification is reduced. For instance:

Command: **ZOOM** Magnification or type (ACELPW): **C**
Center point: **7,4**
Magnification or Height <5>: **2**

places the display center at (7,4) and makes the display 2 drawing units high. The example above used the absolute value 2 for the height; an "X" following the number indicates a change in magnification relative to the current value (e.g., "2X" makes the drawing twice as large).

6.1.6 ZOOM Left Corner

The ZOOM Left corner command is identical to the ZOOM Center command discussed above, except that it lets you specify the lower left corner of the display window instead of the center. For example:

Command: ZOOM Magnification or type (ACELPW): L
Lower left corner point: 2.3
Magnification or Height <5>: 2

Here again, you can specify a relative magnification by following the number with an "X".

6.1.7 ZOOM Previous

While editing or creating a drawing, you may want to zoom in to a small area, back out to view the larger area, and then zoom in to another small area. However, the ZOOM techniques described so far are not easily used to zoom back out to a prior view. You could use ZOOM All and then do repeated ZOOMs to get back to where you were, but in a detailed drawing this can be tedious.

To make this operation more convenient, AutoCAD saves the current view on a "stack" whenever it is being changed by any of the ZOOM commands described above, or by the PAN or VIEW RESTORE (ADE-2 feature) commands described in the next sections. You can return to the previous view by using the ZOOM Previous command:

Command: ZOOM Magnification or type (ACELPW): P

This "pops" the prior view off the stack and displays it. Up to three views are saved on the stack; successive ZOOM P's can restore the previous three views.

NOTE: The term "view" as used above simply means an area of the drawing defined by its display extents. If you erase some objects and issue a ZOOM P, the previous display extents are restored to the monitor, but the erased objects do *not* reappear.

6.2 PAN Command

The PAN command lets you view a different portion of the drawing, without changing the magnification. This lets you see details that were "off screen" before the PAN command. To visualize the effect of this command, imagine that you are looking at your drawing through the display window, and that you can slide the drawing left, right, up, and down without moving the window.

Of course, you must tell the PAN command what direction to move the drawing, and how far to move it. This information is called the *displacement*. You can enter a single coordinate pair, indicating the relative displacement of the drawing with respect to the screen, or you can designate two points, in which case AutoCAD computes the displacement from the first point to the second. For instance, either of the following commands would shift the displayed portion of the drawing 5 units to the left and 3 units down, as shown in the figure below.

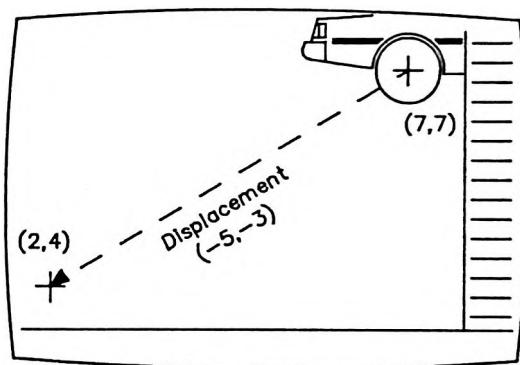
Command: PAN Displacement: -5,-3

Second point: (RETURN) (*to indicate relative displacement*)

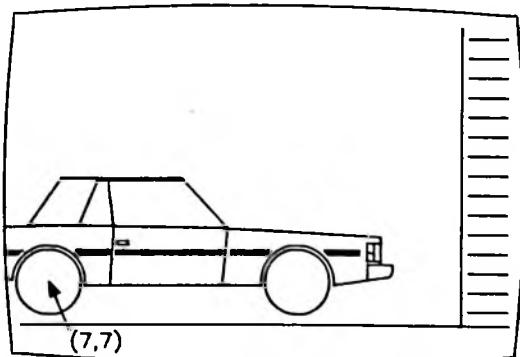
Command: PAN Displacement: 7,7

Second point: 2,4

Before
PAN



After
PAN



AutoCAD -- (6) DISPLAY CONTROLS

6.3 VIEW Command - Named Views (+2)

In many circumstances, it is necessary to switch from one portion of a large drawing to another repeatedly. You can use several ZOOM and PAN commands to do this, but the ADE-2 package's VIEW command is usually more convenient. The format is:

Command: VIEW ?/Delete/Restore/Save/Window: (*select one*)
View name: (*name*)

The valid replies to the first prompt are listed below. Each can be abbreviated to one character.

- ? This produces a list of the named views currently known for this drawing, showing their names, center points, and magnifications. If the drawing contains many named views, the entire list may not fit on the screen at once. In this case, AutoCAD will pause after each screen-full and prompt:

-- More --

To continue the listing, press RETURN.

- D (Delete) This option removes a view from the list of saved views.
- R (Restore) The view you name (if such a view exists) replaces the current display on the screen. AutoCAD remembers the center point and magnification of each named view and essentially performs a "ZOOM Center" with this information when you restore a view.
- S (Save) The current screen display is given a name you supply and is saved for later retrieval. If a view by this name already exists, the new one replaces it. Named views are remembered in the drawing file.
- W (Window) Like the "Save" option described above, this option saves a named view, but "VIEW Window" permits you to define the view by means of a window, without the need to ZOOM in on that window first. AutoCAD asks you to designate two points describing the window.

View names may be up to 31 characters long, and may contain letters, numbers, and the special characters "\$" (dollar), "-" (hyphen), and "_" (underscore). Names are converted to upper case before use.

One application of the 'VIEW command would be to "VIEW Window" each room of a floor plan and assign a name to each window (e.g., "KITCHEN", "GARAGE", "LIVING-ROOM"). From then on, whenever you want to display or edit the portion of your drawing containing the kitchen, simply enter:

Command: VIEW ?/Delete/Restore/Save/Window: R
View name: KITCHEN

When you edit an existing drawing containing named views, you can specify one of those views to be displayed when the Drawing Editor first loads the drawing. See Main Menu Task 2 in Chapter 2. You can also plot portions of a drawing by view name; see Chapter 13.

6.4 REDRAW Command

Although several commands redraw the picture on the screen automatically (for instance, when you turn a grid off or change visible layers), it is sometimes useful to explicitly request a redraw. This process "cleans up" the display by removing any marker blips and is invoked with the command:

Command: REDRAW

You can abort a redraw by pressing **CTRL C**. This can save time if you're about to issue another command that also causes a redraw.

6.5 REGEN Command

The **REGEN** command forces AutoCAD to regenerate the entire drawing and redraw the screen. This is a longer process than a simple **REDRAW**, and it is seldom necessary. The command format is:

Command: REGEN

A **REGEN** may be useful if you have changed or moved one instance of a **REPEAT** group (see Section 5.2); the other instances aren't updated until the drawing is regenerated. You can abort a regen by pressing **CTRL C**. This can save time if you're about to issue another command that causes a regen.

The **ZOOM**, **PAN**, and **VIEW RESTORE** (ADE-2 feature) commands always regenerate the drawing. Several other commands also perform regens under certain circumstances; see the **REGENAUTO** command later in this chapter.

6.6 FILL Command

On some displays and plotters, it takes a long time to fill the interiors of Traces, Solids, and wide Polylines. The **FILL** command lets you turn this interior filling on or off.

Command: FILL On/Off <current>:

You can change the setting of Fill mode whenever you like. The most recent setting is remembered in the drawing file and is the default in the above prompt. To retain the current setting, give a null response to the prompt. The initial setting of Fill mode for a new drawing is governed by the prototype drawing.

When Fill mode is **OFF**, only the outlines of Traces, Solids, and wide Polylines are displayed or plotted. Changing the Fill mode does not affect existing objects until the drawing is regenerated (see the **REGEN** command, above).

6.7 QTEXT Command

The QTEXT command is similar to the FILL command described above, but it governs the display and plotting of Text and Attribute entities by enabling or disabling "quick text" mode.

Command: QTEXT On/Off <current>:

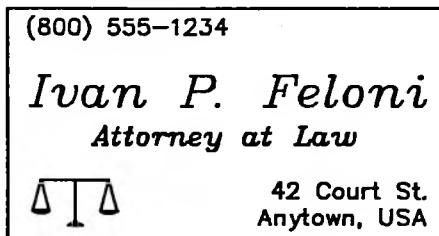
The initial on/off setting of quick text mode for a new drawing is determined by the prototype drawing. You can turn it on and off as often as you like; the most recent setting is remembered in the drawing file and is the default in the "On/Off" prompt. To retain the current setting, give a null response to the prompt.

In quick text mode, AutoCAD draws each Text and Attribute item as a simple rectangle. This mode takes effect the next time the drawing is regenerated, and affects all existing Text and Attribute items at that time. Rather than laboriously computing and drawing the strokes for each character, quick text mode uses the number of characters in the string to form a rough estimate of the text length; this estimate is then used as the length of the rectangle. The height of the rectangle reflects the height of the text.

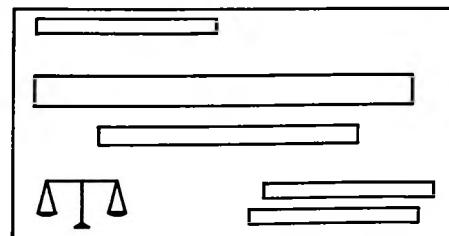
Even if QTEXT mode is on, newly-entered Text and Attribute entities are drawn completely to confirm their content and placement. When the drawing is next regenerated, however, these Text and Attribute entities will appear in the quick text format.

Specify "QTEXT ON" if your drawing contains many Text and Attribute items or if you've used a fancy font. When you are ready to plot your final output, or if you just want to see the text details again, enter "QTEXT OFF" followed by "REGEN".

The following figures demonstrate the effect of the QTEXT command:



QTEXT OFF



QTEXT ON

6.8 BLIPMODE Command

The BLIPMODE command allows you to control the generation of temporary marker blips on the screen. When Blip mode is on, a marker blip is displayed whenever you designate a point. When Blip mode is off, no such marker blips are displayed. The command format is:

Command: BLIPMODE On/Off <current>:

To erase any marker blips drawn while Blip mode is on, issue a REDRAW, REGEN, ZOOM, PAN, or any other command that regenerates or redraws the drawing.

The prototype drawing determines the initial setting of Blip mode for a new drawing. You can change the setting whenever you like; the most recent setting is remembered in the drawing file, and is indicated as the default in the "On/Off" prompt. To retain the current setting, give a null response to the prompt.

6.9 DRAGMODE Command (+2)

The ADE-2 package lets you draw certain entities (Circles, Arcs, Polylines (+3), Blocks, and Shapes) dynamically, "dragging" them into position on the screen. Also, the COPY, MOVE, and CHANGE commands can drag any existing object. To initiate dragging, you must enter "DRAG" at appropriate points in the command sequence; see the CIRCLE, ARC, PLINE, INSERT, SHAPE, MOVE, COPY, and CHANGE command descriptions for details. "DRAG" requests can also be included in menu items.

With some computer configurations, the dragging process may be time-consuming. The DRAGMODE command:

Command: DRAGMODE On/Off <current>:

lets you enable or disable dragging. When Drag mode is off, all "DRAG" requests are ignored, including those embedded in menu items. When Drag mode is on, dragging is permitted, and all "DRAG" requests are honored. When you create a new drawing, the initial setting of Drag mode is governed by the prototype drawing; you can change it as often as you like. The most recent setting is remembered in the drawing file; it is indicated as the default in the "On/Off" prompt. To retain the current setting, give a null response to the prompt.

6.10 REGENAUTO Command

Some AutoCAD commands change basic properties of many entities in your drawing, affecting not only those entities that are on the screen, but also those that are off-screen or that reside on layers that are presently off. In fact, some changes affect which entities appear on the screen. (Consider, for example, changing the font associated with a Text style from the standard "TXT" to "VERTICAL".)

When you make such changes, the command in question normally performs an automatic regen of the entire drawing to ensure that the screen display reflects the actual state of the drawing. Visual fidelity is thus maintained. In some situations, however, you may issue several commands that cause massive changes, and performing a regen after each of them is unnecessary. The REGENAUTO command:

Command: REGENAUTO On/Off <current>:

lets you control whether automatic regens are performed. Responding "Off" disables automatic regens; responding "On" enables them again. If any automatic regens are suppressed while REGENAUTO is turned off, one regen will be performed when you turn REGENAUTO back on. When you create a new drawing, the initial REGENAUTO setting is governed by the prototype drawing. You can change it whenever you like; the most recent setting is remembered in the drawing file and is the default in the "On/Off" prompt. To retain the current setting, give a null response to the prompt.

The commands affected by REGENAUTO mode, and the circumstances under which they attempt regens, are listed below.

ATTEDIT +2	If global editing of all Attributes (not just the visible ones) is selected.
BLOCK	If a Block is redefined.
INSERT	If "INSERT Block=file" is used to redefine a Block.
LAYER	If a layer's linetype or freeze/thaw state is changed.
LTSCALE	If the linetype scale is changed.
STYLE	If the font file associated with a Text style is changed.

Note that the ZOOM, PAN, VIEW RESTORE (ADE-2 feature), and REGEN commands perform explicit regens and are not affected by REGENAUTO mode.

Chapter 7

LAYERS, COLORS, AND LINETYPES

7.1 Basic Concepts

7.1.1 Layers

You can place the entities in your drawing on one or more drawing *layers*. It may be helpful to think of layers as transparent overlays, although layers are not limited to such uses. With layers, you can easily group associated components of a drawing. A layer (or a set of layers) can hold the entities related to a particular aspect of the drawing; you can then control the visibility, color, and linetype of all these entities globally.

The same drawing limits, coordinate system, and zoom factor apply to all layers in a drawing. Layers are always perfectly registered with one another; a designated point on one layer aligns precisely with the same point on every other layer.

There is no limit to the number of layers in a drawing, nor is the number of entities per layer restricted in any way. You can assign a meaningful name to each layer, and you can select any combination of layers to be displayed. You can display all layers simultaneously if you like. Layers and their properties are part of your drawing, and they are saved in the drawing database.

7.1.2 Color Numbers

Each layer in a drawing has an associated *color number*, an integer between 1 and 255. Several layers can have the same color number. To facilitate exchange of drawings between different computer systems, the first seven color numbers have been assigned standard meanings, as follows:

- 1 - Red
- 2 - Yellow
- 3 - Green
- 4 - Cyan
- 5 - Blue
- 6 - Magenta
- 7 - White

However, some display devices used with AutoCAD are not capable of displaying all these colors in this order, while other devices can display many more colors. About all we can state with assurance in this guide is that any layer that is "on" will be displayed on the monitor, and each display driver adheres as closely as possible to the above color sequence. See your AutoCAD Installation Guide / User Guide Supplement for exceptions and for information on what happens if you use color numbers between 8 and 255.

If your graphics monitor is only capable of on/off monochrome output, all color numbers will produce the same visual effect. Color numbers are useful even in this case, however, because each one can be assigned to a different pen on a multi-pen plotter.



7.1.3 Linetypes

Each drawing layer also has an associated *linetype*. Essentially, a linetype is a repeating pattern of dashes, dots, and blank ("pen up") spaces. Each linetype has a name and a definition that stipulates the particular dash-dot sequence and the relative lengths of the dashes and blank spaces. You can select from a library of standard linetypes supplied with AutoCAD (see Appendix A of this manual) or create your own linetype patterns as described in Appendix B.

Linetypes are another means of conveying visual information. If your graphics monitor provides only monochrome output, you can use various dashed and dotted linetypes instead of color numbers to differentiate one layer from another on the screen. In some drafting disciplines, conventions have been established giving specific meanings to particular dash-dot patterns.

The only drawing entities that are affected by linetypes are Lines, Arcs, Circles, and Polylines. If a line is too short to hold even one dash-dot sequence, AutoCAD draws a continuous line between the endpoints.

By default the linetype for all newly-created layers is called "CONTINUOUS", meaning, naturally, a normal solid line. You needn't concern yourself with linetypes at all if your drawing involves only lines of the continuous type.

The linetypes used internally by AutoCAD should not be confused with the dashed line capabilities provided by some plotters. Although the two types of dashed lines produce similar results, we suggest that you not use both types at the same time. See Chapter 13 for a complete discussion of plotting.

7.2 Properties of Layers

A layer has the following properties:

Layer name	This is the name you use to refer to the layer in various commands. It may be up to 31 characters long and may contain letters, digits, and the special characters "\$" (dollar), "-" (hyphen), and "_" (underscore). All layer names are converted to upper case. You can use descriptive names appropriate to your application, such as "PLUMBING" or "FRONT", or you can use shorter names like "1", "2", "3", etc.
Visibility	A layer can be either visible ("on") or invisible ("off"). Only visible layers are displayed or plotted. Invisible layers are still part of the drawing; they just aren't displayed or plotted. You can turn layers on and off at will, in any combination.
Color number	The color number defines the actual display color for a visible layer. By default, new layers are assigned color number 7. Standard meanings have been assigned to the first few color numbers.
Linetype name	This is the name of a specific dash-dot line pattern with which all Lines, Arcs, Circles, and Polylines on this layer are to be drawn. The name may be up to 31 characters long, using the same characters allowed for a layer name. Several layers can use

the same linetype. By default, new layers are given the "CONTINUOUS" linetype.

Freeze/thaw state If the ADE-3 package is present, each layer has an additional property called its *freeze/thaw state* that controls whether or not entities on that layer are considered for generation when AutoCAD regenerates the display. "Thawed" layers are generated; "frozen" layers are not. Freezing a layer is similar to turning its visibility off, but in a complex drawing freezing unneeded layers can improve AutoCAD's regeneration speed.

If the ADE-3 package is not present, layers are always considered "thawed".

7.3 The Current Layer

The Drawing Editor maintains a "current" layer. This is the layer upon which newly created entities are drawn. If you select a different layer to become current, any new entities you then draw go on that layer.

7.4 Initial Layers and Linetypes

When you begin a new drawing, a layer named "0" is created automatically by AutoCAD. By default, this layer is assigned color number 7 and the "CONTINUOUS" linetype and is set as the initial current layer. Layer "0" has some special properties regarding Blocks (described in Chapter 9) and it cannot be renamed or deleted.

The "CONTINUOUS" linetype is also defined automatically when you begin a new drawing. It cannot be renamed or deleted.

Additional layers and linetypes, different layer/color/linetype assignments, and a different initial current layer may be stipulated by the prototype drawing.

7.5 Layers and Plotting

At plot time, only layers that are "on" and "thawed" are plotted. On a pen plotter, you can assign a different pen to each color number in the drawing. This works even for single-pen plotters; you can ask AutoCAD to pause for pen changes. Also, if your plotter provides dashed lines, you can map the color number into one of these hardware line types. Note that you may get undesirable results if you use hardware line types together with AutoCAD's linetypes; we suggest that you use one or the other, but not both. See Chapter 13 for a further discussion of plotting.

7.6 Layer/Linetype Renaming and Deletion

You can change the name of a layer or linetype any time you wish, and AutoCAD provides a method for deleting unused layers and linetypes. These functions are handled by the RENAME and PURGE commands, which can operate on other named objects in your drawing, as well. They are described in Chapter 3.

As noted above, layer "0" and the "CONTINUOUS" linetype cannot be renamed or deleted.

7.7 LAYER Command

You can use the LAYER command to create new layers, select the current layer, set the color number and linetype for designated layers, turn layers on and off, and list the defined layers. These functions are selected from an option list, as shown below.

Command: **LAYER**

?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: (*select one*)

(The "Freeze" and "Thaw" options appear only if the ADE-3 package is present.) You can abbreviate your reply to two characters for "On" and "Off", and to one character for the other options. The prompts that follow depend upon the function you select; they are described individually in the sections that follow. When a function is complete, the initial LAYER command prompt reappears, allowing you to perform multiple layer-type operations before exiting the command. When you wish to exit, respond to the prompt with a null line (simply press space or RETURN). If you selected any functions that might change the display, a regen or redraw operation is performed automatically as the LAYER command exits. (Note that regens are performed automatically only if REGENAUTO is ON; see Section 6.10).

7.7.1 Layer Name Lists

Most of the LAYER command options request a list of layer names. A prompt such as:

Layer name(s) <*default*>:

requests one or more layer names. If you supply more than one name, separate them with commas. Intervening spaces are not allowed. Some functions have a default layer; a null response selects just that layer.

The functions that operate on existing layers accept "?" and "*" wild-card characters similar to those permitted in some operating system commands. A "?" matches any character in the corresponding position, while a "*" matches any and all characters from that position to the end of the name. Thus, given the following existing layers in an architectural drawing:

```
FL-1-PLUMBING
FL-1-ELECTRICAL
FL-1-WALLS
FL-2-PLUMBING
FL-2-ELECTRICAL
FL-2-WALLS
```

the layer list "FL-?-WALLS" would select the "-WALLS" layers for both floors 1 and 2, while "FL-1*" would select all layers associated with the first floor. In order to make the wild-card feature most useful, we recommend that you set up your own layer naming conventions similar to those in this example.

The descriptions of the individual LAYER command functions will specify whether or not you may use wild-cards.

7.7.2 LAYER ? - List Layer Data

The "?" function produces a list of the currently defined layers, showing their names, on/off states, color numbers, and linetypes. When the prompt:

Layer name(s) for listing <*>:

appears, supply a list of existing layer names, including wild-card characters if you wish. For example, the command sequence:

Command: LAYER
 ?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: 1
 Layer name(s) for listing <*>: BASEMENT

lists information about layer "BASEMENT". If you give a null reply to the last prompt, AutoCAD lists information about all existing layers (the default name list, a lone "", matches all names).

The layer display has the following format:

Layer name	State	Color	Linetype
0	On	7 (white)	CONTINUOUS
TITLE-BLOCKS	Off	7 (white)	CONTINUOUS
FL-1-PLUMBING	On	3 (green)	DASHED
FL-1-WALLS	Frozen	12	CONTINUOUS
FL-2-PLUMBING	On	5 (blue)	CONTINUOUS
FL-2-WALLS	On	1 (red)	DOT
BASEMENT	On	240	CONTINUOUS
DIMENSIONS	Off	7 (white)	CONTINUOUS

Current layer: 0

As indicated in this sample, if a color number has a standard meaning, its color name is listed as well.

If your drawing has many layers, the entire list may not fit on the screen at once. In this case, AutoCAD will pause after each screen-full and prompt:

-- More --

Press RETURN to continue the listing.

7.7.3 LAYER Set - Select Current Layer

As you draw new entities, they are placed on the "current" layer and displayed using the color number and linetype associated with that layer. You can select a different current layer by using the LAYER command's "Set" function. When the prompt:

New current layer <current>:

appears, respond with the name of an existing layer. Wild-card characters are not permitted in this case, since you must select one layer unambiguously. The prompt includes the name of

the present current layer; it is retained if you give a null response to the prompt. For example, if layer "FOUNDATION" is the current layer and you want to begin drawing on layer "FLOOR-3", the command sequence is:

Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: SET
New current layer <FOUNDATION>: FLOOR-3

If the layer you select to become the current layer is presently OFF, AutoCAD turns it ON automatically, using the color number and linetype previously assigned to that layer. If the layer is presently frozen, it cannot be made the current layer.

7.7.4 LAYER New - Create New Layers

If you want to create your own layers, you can do so using the LAYER command's "New" function. When the prompt:

New layer name(s):

appears, reply with a list of layer names to be created. The list may not contain wild-card characters or existing layer names. Each layer thus created is turned on and is assigned color number 7 and the "CONTINUOUS" linetype. For example, the following command sequence creates two new layers:

Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: NEW
New layer name(s): BACKGROUND.FOREGROUND

The "New" function does not change the current layer, and it has no effect on existing entities. A null response to the "New layer name(s):" prompt simply returns to the main LAYER command prompt without creating any new layers.

7.7.5 LAYER Off - Turn Layers Off

Entities associated with an "off" layer are not displayed on the graphics monitor and they are not plotted. The entities still exist in the drawing; they are just invisible. To turn selected layers off, use the LAYER command's "Off" function. Respond to the prompt:

Layer name(s) to turn Off:

with a list of the layers you wish turned off. The list should contain only existing layer names, and it may include wild-card characters. All layers with matching names are turned off. Their color numbers and linetypes are remembered, and they are restored when the layers are again turned on. For example, to turn off all layers whose names begin with the letter "X", enter:

Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: OFF
Layer name(s) to turn Off: X*

No default layer is provided for the "Off" function. It is possible to turn off the current layer, but this is rarely desirable. To do so causes no harm, but it can be confusing if you

don't realize what has happened; new entities you draw are added to the drawing but are not displayed until the layer is again turned on. If you select the current layer due to a wild-card match or other entry, AutoCAD asks:

Really want layer (name) (the CURRENT layer) off? <N>

to warn you about the situation. An "N" response (the default) leaves the current layer as it was. If you do want to turn off the current layer, simply reply "Y". Just remember to turn it back on or select a new current layer before drawing anything new.

7.7.6 LAYER On - Turn Layers On

Layers that have been turned off can be turned on again using the LAYER command's "On" function. AutoCAD prompts:

Layer name(s) to turn On:

and you can reply with a list of existing layer names. Wild-card characters are permitted. No default is provided; a null reply simply returns to the main LAYER command prompt. The following example turns on two existing layers:

Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: ON
Layer name(s) to turn On: FRONT BACK

Each designated layer is turned on (made visible) using the color number and linetype previously associated with it. If the layer is presently frozen, turning it on is not sufficient to make it display again; you must also thaw the layer.

7.7.7 LAYER Color - Set Color Number

You can change the color number associated with specific layers by using the "Color" function of the LAYER command. The prompt sequence is:

Color:
Layer name(s) for color *n* <current>:

Respond to the first prompt with a legal color number (an integer between 1 and 255). Or, if you prefer, reply with one of the standard color names (e.g., "YELLOW") rather than a number.

After you specify the color, AutoCAD asks for a list of layer names to which the color should be applied. Reply with the names of existing layers, using wild-cards if you wish. A null list means just the current layer. For instance, to assign color number 4 to all layers whose names have a "-" as the fourth character, you can use:

Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: COLOR
Color: 4
Layer name(s) for color 4 (cyan) <XYZ-1>: ??*-*

Ordinarily, the specified layers are given the color you designated and are then turned on. If you'd prefer to assign the color but turn the layers off, precede the color with a minus sign (-). For example:

Command: LAYER?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: COLORColor: -1Layer name(s) for color -1 (red) <XYZ-1>: ABC

would set layer "ABC" to color number 1 and turn that layer off. The same thing would happen if you responded to the "Color" prompt with "-RED".

7.7.8 LAYER Ltype - Set Linetype

You can change the linetype associated with specific layers by using the "Ltype" function of the LAYER command. The first prompt is:

Linetype (or ?) <CONTINUOUS>:

Respond with the name of an existing linetype; the "CONTINUOUS" linetype is used if you give a null response. If the linetype has not yet been used in the current drawing, AutoCAD automatically loads it from the ACAD.LIN library file. (Linetypes can be loaded from other files by means of the LINETYPE command, described in the next section.)

AutoCAD then asks for a list of layer names to which the linetype should be applied.

Layer name(s) for linetype xxx <current>:

Reply with the names of existing layers, using wild-cards if you wish. A null list means just the current layer.

For example, assuming that a linetype named "DASH-3" has been loaded or exists in ACAD.LIN, you can assign it to all layers having names with 3 or fewer characters as follows:

Command: LAYER?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: LTYPELinetype (or ?) <CONTINUOUS>: DASH-3Layer name(s) for linetype DASH-3 <XYZ-1>: ???

To obtain a list of the loaded linetypes, respond to the "Linetype (or ?)" prompt with a "?". The name of each linetype loaded in the current drawing is displayed, along with a brief description; this description may take the form of a diagram created with periods and underscores, as shown below.

Loaded linetypes:

Name	Description
CONTINUOUS	Solid line
DASHED	-----
CENTER	_____
DASHDOT	-. . -. . -. . -. . .

If your drawing has many linetypes, the entire list may not fit on the screen at once. In this case, AutoCAD will pause after each screen-full and prompt:

-- More --

Press RETURN to continue the listing.

7.7.9 LAYER Freeze - Freeze Layers (+3)

Using the LAYER command's "Freeze" option (a feature of the ADE-3 package), you can instruct AutoCAD to ignore the entities on specified layers when regenerating the drawing. Entities on frozen layers are not displayed or plotted, and AutoCAD spends no time calculating where they go. Therefore, you can increase AutoCAD's ZOOM, PAN, VPOINT (+3), REGEN, and entity selection performance for complex drawings by freezing the layers that are not of immediate interest.

When you select the "Freeze" option, AutoCAD asks for a list of layer names to be frozen.

Layer name(s) to Freeze:

Reply with the names of existing layers, using wild-cards if you wish. There is no default for this prompt; the current layer cannot be frozen.

For example, if you have placed all the dimensioning annotations for a complex drawing on a layer named "DIMENSIONS", but you do not need to see the dimensions while you perform some editing of details, you can freeze that layer as follows:

```
Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: FREEZE
Layer name(s) to Freeze: DIMENSIONS
```

Since freezing a layer turns it off, you may be confused about when to use "LAYER Freeze" as opposed to "LAYER Off". The only difference is a matter of efficiency. If you are frequently switching between layers, doing a bit of editing on each of them, you should use "LAYER Off" for any layers whose entities are currently obstructing your view. If, on the other hand, you are doing most of your editing on one layer (or a set of layers), and do not need to see the entities on another set of layers, you should freeze those layers. This will speed up your editing if you do many ZOOMs, PANs, or other operations that cause drawing regeneration.

7.7.10 LAYER Thaw - Thaw Layers (+3)

The "Thaw" option thaws selected layers, negating the effect of the "Freeze" option. This is a feature of the ADE-3 package. AutoCAD asks for a list of layer names to be thawed.

Layer name(s) to Thaw:

Reply with the names of existing layers, using wild-cards if you wish. There is no default for this prompt.

For example, the layer we froze in the previous example can be restored to normal by means of the following command sequence:

```
Command: LAYER
?/Set/New/On/Off/Color/Ltype/Freeze/Thaw: THAW
Layer name(s) to Thaw: DIMENSIONS
```

7.8 LINETYPE Command

7.8.1 General Notes

You can use the LINETYPE command to load linetype definitions from a library file explicitly and to create new definitions for storage in a library. Once loaded, linetypes can be assigned to layers using the "LAYER Ltype" command described in the previous section. A library of standard linetypes (file ACAD.LIN) is supplied with AutoCAD; see Appendix A for details. See Appendix B for information on creating your own linetypes.

The only drawing entities that are affected by linetypes are Lines, Arcs, Circles, and Polylines.

NOTE: The "LAYER Ltype" command automatically loads linetypes from the ACAD.LIN library file when they are needed. If you do not plan to create your own linetypes or to load linetypes from another file, you can skip to the section titled "Scanning a Linetype Library".

There are several important concepts to understand when using linetypes.

1. A linetype definition must exist in a library file before it can be loaded into your drawing.
2. A linetype must be loaded from a library file before it can be assigned to a layer. (If the linetype is in ACAD.LIN, the "LAYER Ltype" command can load it automatically.)
3. When you load a linetype, its definition is copied into your drawing; there is no need to keep the file from which it was loaded accessible thereafter.
4. If you recreate and reload a linetype using the LINETYPE command and the linetype is currently assigned to layers of the drawing, an automatic regen is performed (subject to REGENAUTO, Section 6.10) using the new definition. This is convenient when experimenting with a linetype definition, providing instant feedback of its effects on existing entities.

Using the LINETYPE command, you can explicitly load linetype definitions for use in a drawing. You can also use it to create or modify linetype definitions and to list the contents of linetype library files. The format is:

Command: LINETYPE
?/Load/Create: (*select one*)

The "Load and "?" command functions are described in the following sections; "Create" is discussed in Appendix B.

When a function is complete, it returns to the "?/Load/Create" prompt. To terminate the LINETYPE command, give a null response or press CTRL C when this prompt appears.

7.8.2 Loading Linetypes From a Library

Before loading a particular linetype, the definition for it must exist in a disk file. The file ACAD.LIN contains the library of standard linetypes supplied with AutoCAD. (If you want one of these standard linetypes, there is no need to explicitly load it; the "LAYER Ltype" command automatically loads linetypes from ACAD.LIN when they are needed.)

To explicitly load a linetype definition into your current drawing, issue the LINETYPE command and select its "Load" option. AutoCAD prompts:

Name of linetype to load:

Respond with the name of the linetype you wish to load. Linetype names may be up to 31 characters long, and may contain letters, digits, and the special characters "\$", "-", and "_". Once you have entered the name, you are asked:

File to search <default>:

Reply with the name of the disk file containing the definition of this linetype. Do not include a file type in the name; type ".LIN" is assumed. A null response causes the default file to be used. The specified linetype is loaded if it is found in the file. If the linetype is not found, AutoCAD prints an error message and cancels the load operation. If AutoCAD finds that the linetype was previously loaded, an additional prompt is given:

Linetype was loaded before. Reload it? <Y>

which should be answered with either "N", to retain the present definition, or "Y", to replace the current definition with the new one. If you answer "Y" and if the current drawing contains visible layers that already use the linetype you are reloading, the drawing is regenerated (subject to REGENAUTO, Section 6.10) when you exit from the LINETYPE command. This lets you see the immediate effects of the linetype redefinition.

7.8.3 Scanning a Linetype Library

If you want to see what linetypes are available for loading, select the LINETYPE command's "?" option. AutoCAD will prompt with:

File to list <default>:

You can use a null reply to list the default file or supply another file name. Do not include a file type in the name; type ".LIN" is assumed. Once the file name has been specified, AutoCAD lists the linetypes defined in that file, as shown below.

Linetypes defined in file XXX.LIN:

Name	Description
DASHED	---
HIDDEN	- - - - -
CENTER	— — — — —
PHANTOM	— — — — —
DOT
DASHDOT	— • — • — • — • — •
BORDER	— — • — — — • — — • —
DIVIDE	— — • — — — • — — • —

If the list is long, it may not fit on the screen all at once. In this case, AutoCAD will pause after each screen-full and prompt:

-- More --

To continue the listing, press RETURN.

Note that the listing produced by "LINETYPE ?" is the contents of a library file; it does not necessarily reflect the linetypes currently loaded in your drawing. You can use the "LAYER Ltype ?" command sequence to display the currently loaded linetypes.

7.9 LTSCALE Command

The dash specifications in a linetype definition are in terms of drawing units. Since drawing units mean "inches" in some drawings and "kilometers" (or whatever) in others, a method is provided for adjusting the linetypes to a meaningful scale for your drawing.

A global linetype scale factor is provided for each drawing, with a default value of 1.0. You can change this scale factor with the LTSCALE command:

Command: LTSCALE

New scale factor <*default*>:

Respond to the prompt with a scale factor greater than 0. The current value is used as the default; you can retain it by pressing space or RETURN. When you change the linetype scale factor, the drawing is regenerated with the revised dash lengths, unless REGENAUTO is off (see Section 6.10).

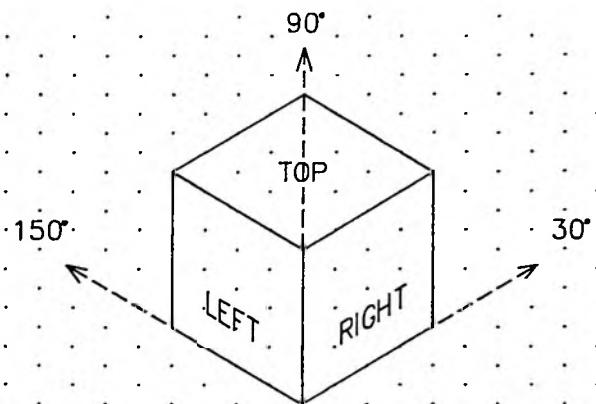
Chapter 8

DRAWING AIDS

NOTE TO ADE-2 USERS: ISOMETRIC DRAWINGS (+2)

Several sections of this chapter describe ADE-2 features designed to assist you in working with isometric drawings. A brief discussion of such drawings is provided below as a reference.

The isometric style assists you in drawing isometric views of three-dimensional objects such as the cube shown below. An isometric grid, such as the SNAP and GRID commands use, has been superimposed on the cube.



DRAWING AIDS

As indicated, the isometric Snap grid has three major axes. Assuming that rotation has not also been applied to the Snap grid, the axes are vertical, 30 degrees, and 150 degrees. Two AutoCAD features, Ortho mode and "keyboard pointing" with the cursor movement keys, can only deal with two of the three axes at a time. Therefore, AutoCAD assumes that you are drawing on one of three "isometric planes" (Left, Top, or Right), each of which has an associated pair of axes. The meaning of Ortho mode and the action of the cursor movement keys are then modified to follow the current pair of axes. The ISOPLANE (+2) command allows you to select the current isometric plane and thus the current pair of axes.

8.1 SNAP Command

Points entered by means of a pointing device can be locked into alignment with an imaginary rectangular grid by the "snap" mechanism. The *snap resolution* defines the spacing of this grid; the screen crosshairs and all input coordinates are locked ("snapped") to the nearest point on the grid if Snap mode is on. Using Snap mode, you can enter points quickly, letting AutoCAD ensure that they line up precisely. You can use the SNAP command to turn Snap mode on or off or to change your snap resolution. If your copy of AutoCAD includes the ADE-2 package, you can also invoke rotation of the grid, set differing X and Y spacing, or choose an isometric format for the snap grid. The prototype drawing governs the initial snap resolution and on/off setting for new drawings.

A change in the snap grid only affects the coordinates of points you subsequently enter. Entities already in the drawing retain their coordinates, even if they do not line up with the new snap grid.

Note that the snap grid is invisible. You can use the GRID command (described in the next section) to display a separate visible grid. You can set the spacing of the two grids to equal or related values to assist you in using the snap grid. The description of the GRID command includes details on accomplishing this. The SNAP command format is:

Command: **SNAP**
 On/Off/Value/Aspect/Rotate/Style: *(select one)*

The meaning of each option is described below. (Those marked with "+2" are ADE-2 features; they appear in the option list only if the ADE-2 package is present.) The illustrations of the various snap grids are provided for clarification; these grids would not appear on the screen unless GRID mode was on.

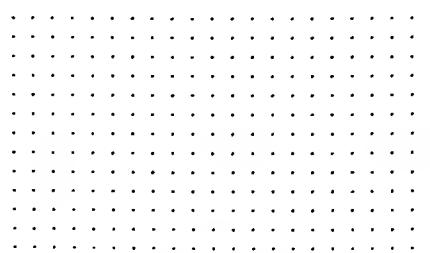
On The "On" option causes Snap mode to be activated, using the previous snap grid resolution, rotation, and style. For example:

Command: **SNAP**
 On/Off/Value/Aspect/Rotate/Style: **ON**

If the current snap resolution is 0, AutoCAD asks you to supply a new value. For instance, to change the snap resolution from 0 to 0.5, enter the following command sequence:

Command: **SNAP**
 On/Off/Value/Aspect/Rotate/Style: **ON**
 Value/Aspect: **0.5**

A standard-style snap grid is illustrated below.



NORMAL GRID

Off "SNAP Off" deactivates Snap mode, but remembers the values and modes so they can be restored if you activate it again. For example:

Command: SNAP
 On/Off/Value/Aspect/Rotate/Style: OFF

Value If you respond to the SNAP option prompt with a numeric value, Snap mode will be activated using that value as the snap resolution. For instance, to set the snap resolution to 0.5 drawing units, enter:

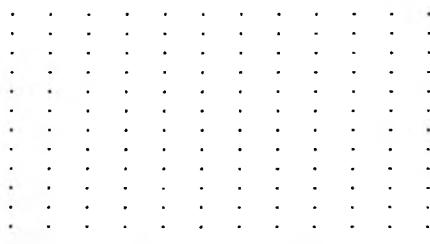
Command: SNAP
 On/Off/Value/Aspect/Rotate/Style: .5

Supplying a value of 0 is another method of turning Snap mode off. The initial snap resolution value for a new drawing is governed by the prototype drawing.

Aspect +2 You can specify differing X and Y spacings for the snap grid by responding to the option prompt with "Aspect" (or simply "A"). AutoCAD then issues prompts for the horizontal and vertical spacings, as in:

Command: SNAP
 On/Off/Value/Aspect/Rotate/Style: A
 Horizontal spacing: 1
 Vertical spacing: .5

The grid created by this command is shown below.

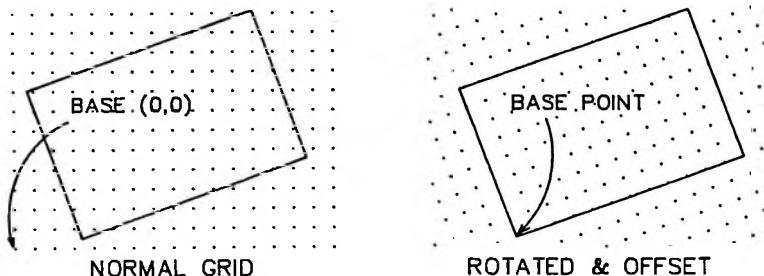


GRID WITH ASPECT OPTION

The "Aspect" option is not available if the current Snap style is "isometric". See "Style", below.

Rotate +2 Often, it is useful to rotate the snap grid with respect to the drawing and the display screen, or to set the snap grid's base point to something other than (0,0). You can accomplish these objectives using the SNAP command's "Rotate" (or "R") option. AutoCAD asks for the desired base point and rotation angle. For example, to align the snap grid with a box that's rotated 20 degrees, and place a grid point at one corner of the box, you should enter:

Command: SNAP
 On/Off/Value/Aspect/Rotate/Style: R
 Base point <0,0>: (designate corner of box)
 Rotation angle <0>: 20



The default base point is (0,0), and the default rotation angle is 0 degrees. You can specify a rotation angle between -90 and 90 degrees; a positive angle rotates the grid counter-clockwise, whereas a negative angle denotes clockwise rotation. The grid rotates around its base point.

- Style +2** The "Style" (or "S") option permits you to select the format, or style, of the Snap grid. Two choices, "standard" and "isometric", are provided. "Standard" refers to the normal rectangular type of grid (which may have differing X and Y spacing), while "isometric" refers to a grid designed for isometric drafting purposes, with grid points arranged to assist you in drawing 30, 90, 150, 210, 270, and 330-degree lines. For this type of grid, AutoCAD requests the desired vertical spacing between grid points, as shown in the following example:

Command: SNAP
 On/Off/Value/Aspect/Rotate/Style: S
 Standard/Isometric: I
 Vertical spacing: 1.5

The "isometric" Snap style is simply a drawing aid. It affects the appearance of AutoCAD's grid, axis (+1), and crosshair displays and the angles used for Ortho mode. However, it does not affect such AutoCAD features as dimensioning (+1) and distance computation.

See the note at the beginning of this chapter for an illustration of an isometric grid. The default Snap style is, of course, "standard".

A control key is provided to permit toggling Snap mode on and off, even in the middle of another command. See Section 8.8.

The screen crosshairs adapt to the Snap grid's style and rotation. That is, if the Snap grid is rotated, the screen crosshairs are rotated the same amount. If the Snap style is "isometric", the crosshairs follow the proper pair of axes.

If you have the ADE-2 package, you can also snap to reference points of objects already in the drawing. This is done using another mode called *object snap* described in Section 8.6.

Keyboard Pointing and Special Snap Modes

If you use the keyboard cursor movement keys to designate points ("keyboard pointing"), and intend to use rotated or isometric grids, read on for some special notes. Otherwise, please skip the rest of this section.

If you activate Snap mode with a rotated grid, the keyboard's cursor movement keys act as though the keys themselves were rotated the same amount as the grid. That is, if the Snap grid is rotated 30 degrees (counter-clockwise), the "cursor right" key moves to the next grid point in the 30-degree direction.

If Snap mode is activated using the "isometric" style, the cursor movement keys are interpreted differently. In this mode, AutoCAD assumes that you are drawing on one of three "isometric planes", (Left, Top, or Right), and assigns each of the four "orthogonal" directions applicable for that plane to one of the cursor keys. See the beginning of this chapter for a discussion of the isometric planes. The ISOPLANE (+2) command, discussed in Section 8.5, controls the choice of isometric planes.

8.2 GRID Command

The GRID command displays a reference grid of dots with any desired spacing. This feature allows you to have a "feel" for the sizes of drawing entities and their relationships. You can turn the grid on and off at will, and you can change the dot spacing easily. The grid is not considered part of the drawing; it is for visual reference only and is never plotted.

A grid is specified by entering the command:

Command: GRID
On/Off/Value(X)/Aspect: (select one)

The various options are described below. (Those marked with a "+2" are ADE-2 features and only appear if the ADE-2 package is present.)

On The "On" option activates the grid, using the previous dot spacing. For example:

Command: GRID
On/Off/Value(X)/Aspect: ON

When you begin a new drawing, the prototype drawing determines the initial grid spacing and on/off state.

Off If you respond to the option prompt with "Off", the reference grid will be erased from the screen. For example:

Command: GRID
On/Off/Value(X)/Aspect: OFF

Value(X) You may supply a numeric value to set the dot-to-dot grid spacing, in terms of drawing units. To set the grid spacing to 0.75 units, for instance, enter:

Command: GRID
On/Off/Value(X)/Aspect: 0.75

While not necessary, it is often useful to set the grid spacing equal to the Snap resolution or a multiple of it. If the grid value is zero (the default case), the grid spacing adjusts to the Snap spacing automatically. To specify the grid spacing as a multiple of the Snap value, put an "X" after the value. For example, if the Snap spacing is 0.1 and you want a visible grid point at every fifth snap point, use a grid value of "5X", as in:

Command: GRID
On/Off/Value(X)/Aspect: 5X

Aspect +2 Using the "Aspect" (or "A") option, you can request a visible grid with differing X and Y spacings. AutoCAD then asks for the horizontal and vertical spacings, as in:

Command: GRID
On/Off/Value(X)/Aspect: A
Horizontal spacing(X): .1
Vertical spacing(X): .25

As indicated by the "(X)" in these prompts, an "X" may follow either of these values to signify a multiple of the Snap resolution.

The "Aspect" option for the visible grid is not available if the current Snap style is "isometric".

A control key is provided to allow the visible grid to be turned on and off, even in the middle of another command. See Section 8.8.

Note that no "Rotate" or "Style" options are provided for the GRID command. The rotation, base point, and style of the visible grid always match those of the Snap grid.

If the spacing of the visible grid is set too small, you might get a display having such a dense grid that you could not see the drawing clearly. This could also occur if you later ZOOMed out to a smaller scale (shrinking the drawing). In such cases, AutoCAD displays the following message and does not draw the grid:

Grid too dense to display.

To display the grid, issue another GRID command and specify a larger spacing.

Even if the display shows space outside the drawing limits, the grid appears only within the limits. This clearly defines the drawing limits on the screen.

8.3 AXIS Command - Ruler Lines (+1)

The ADE-1 package's AXIS command instructs AutoCAD to display "ruler lines" with specified tick spacing along the edges of the graphics monitor. The command format is similar to that of the GRID command:

Command: **AXIS**

On/Off/Tick spacing(X)/Aspect:

A response of "On" or "Off" changes the status of the axis line accordingly. You can set the tick spacing in terms of drawing units simply by entering a number. Alternatively, you can enter an "X" following the number to set the tick spacing to a multiple of the Snap resolution. For instance, "10X" sets the tick spacing to ten times the snap resolution, placing an axis tick at every tenth snap point.

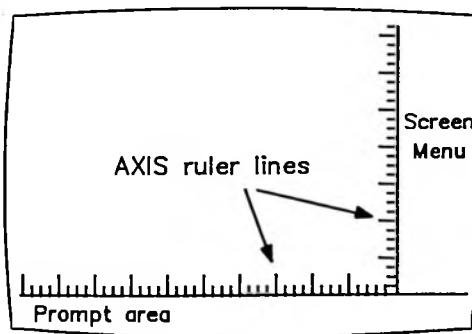
You can use the "Aspect" option to set differing horizontal and vertical spacings for the axis ticks. This option is not available when the Snap style is "isometric".

As with the GRID command, a small tick spacing (or ZOOMing out) may produce ticks so dense that you cannot discern them on the screen. If this occurs, AutoCAD displays the following message:

Axis ticks too close to display.

To display the axis, issue another AXIS command, specifying a larger tick spacing.

When a "feet and inches" display format is in effect (see the UNITS command in Chapter 3), some of the tick marks may be double size, to indicate whole inches or whole feet. The large tick marks are drawn only if the specified tick spacing is an exact fraction of an inch or a foot.



8.4 ORTHO Command

AutoCAD's Ortho mode helps you ensure that all Lines and Traces drawn using a pointing device will be orthogonal with respect to the current snap grid. Assuming for the moment that the snap grid has not been rotated, and that the Snap style is "standard", this means that all Lines and Traces will be either vertical or horizontal, but never diagonal.

Begin Ortho mode by entering the command:

Command: ORTHO On/Off: ON

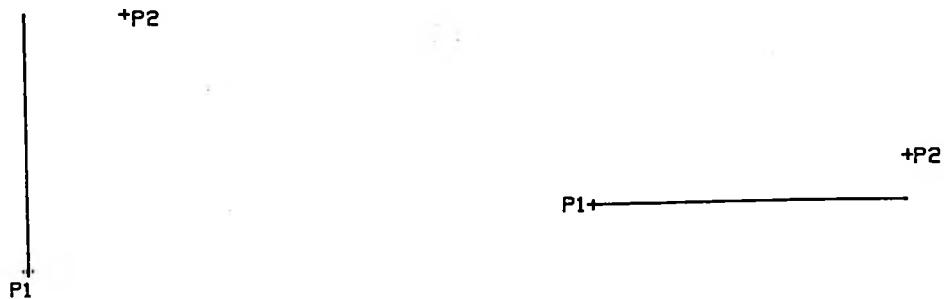
You can return to normal mode with the command:

Command: ORTHO On/Off: OFF

A control key is also provided to toggle Ortho mode on and off, even in the middle of another command. See Section 8.8.

To draw a Line or Trace in Ortho mode, enter the first point as usual. Then, as you move the pointing device toward the desired "to" point, the rubber-band line indicates where the orthogonal line will go; it is either vertical or horizontal, depending on how far the crosshairs are, vertically and horizontally, from the first point. The larger of these distances determines the orientation and length of the orthogonal line.

In the following examples, P1 is the first point specified, and P2 is the position of the crosshairs when the second point is specified.



Ortho mode also applies when you specify an angle or distance by means of two points (with a pointing device). Ortho mode does not affect explicit keyboard point entry, which by its nature is very precise.

Ortho Mode and Special Snap Modes

In the above discussion, we have assumed that the snap grid is of the standard rectangular style, and that it has not been rotated. However, as described in Section 8.1, the ADE-2 package permits rotation of the snap grid and selection of an isometric grid style. These snap grid parameters affect the behavior of Ortho mode.

If you rotate your Snap grid, Ortho mode rotates accordingly. For instance, if you specify a rotation of 45 degrees for the Snap grid (using the "SNAP Rotate" command form), Ortho mode will permit only 45, 135, 225, and 315-degree Lines and Traces.

Selection of the "isometric" Snap style changes the interpretation of Ortho mode. AutoCAD assumes that you are drawing on one of three "isometric planes", (Left, Top, or Right), and adjusts Ortho mode to force alignment with the two major axes of that plane. See the beginning of this chapter for a discussion of the isometric planes. The ISOPLANE (+2) command, discussed in Section 8.5, controls the choice of isometric planes.

8.5 ISOPLANE Command (+2)

You can set the Snap style to "isometric" by means of the "SNAP Style Isometric" command sequence described in Section 8.1. The ISOPLANE command is used to select the current isometric plane and thus the current pair of axes.

Command: **ISOPLANE**
Left/Top/Right/(Toggle):

Refer to the cube illustration at the beginning of this chapter for a better understanding of the responses to this prompt. Each may be abbreviated to one letter.

- | | |
|--------|--|
| Left | The "Left" option selects the left-hand plane, defined by the 90-degree and 150-degree axis pair. When Snap mode is on, the "up" and "down" cursor keys move along the 90-degree axis, and the "left" and "right" cursor keys move along the 150-degree axis. |
| Top | The "Top" isometric plane is the top face of the cube; it uses the 30-degree and 150-degree axis pair. When Snap mode is on, the "up" and "down" cursor keys move along the 30-degree axis, and the "left" and "right" cursor keys move along the 150-degree axis. |
| Right | Responding to the prompt with "Right" selects the right-hand plane, defined by the 90-degree and 30-degree axis pair. On this plane, the "up" and "down" cursor keys move along the 90-degree axis, and the "left" and "right" cursor keys move along the 30-degree axis. |
| RETURN | If you give a null response (just space or RETURN), ISOPLANE toggles to the next plane in a circular fashion (Left, Top, Right . . .). For example, if the current plane is "Left" and you press RETURN in response to the prompt, the current plane is changed to "Top". Similarly, if the current plane is "Right", it is changed to "Left". |

When you first select the isometric Snap style, the isometric plane is "Left". A control key is provided to switch planes even in the middle of another command. See Section 8.8.

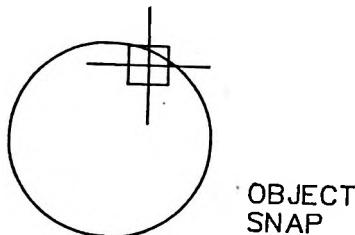
The cursor movement keys are only affected by the isometric plane when Snap mode is activated and the Snap style is isometric. If the Snap style is isometric, Ortho mode will use the appropriate axis pair even if Snap mode is off.

8.6 Object Snap (+2)

This section describes the ADE-2 package's object (geometric) snap facility. This feature lets you refer to points that are related to objects already in the drawing. This facility complements AutoCAD's basic Snap facility, which locks points to various coordinate systems. A simple use of object snap is to lock the start point of a line to the end point of a previously drawn line. Object snap allows geometric constructions much more complex than this, but the same general approach applies.

8.6.1 Basic Operation

You can specify a variety of object snap modes, as described below. If you select any combination of these modes, the object snap mechanism is activated whenever AutoCAD requests a point. A special target symbol is added to the screen crosshairs to indicate the area within which AutoCAD will search for object snap candidates. For example, the figure below shows object snap being used to select a circle.



You can adjust the size of the target by means of the APERTURE command, described later in this section.

The OSNAP command sets "running" object snap modes, which are applied to all point selections. You can override the running modes to input a single point using other object snap modes or to disable the object snap mechanism entirely for a single point.

To select a point using object snap, position the crosshairs so that the desired object falls within or crosses the target. Then press the pointer's "pick" button. AutoCAD searches the target area for an object that has a point satisfying at least one condition specified by the current object snap modes. The target may contain more than one such object, and each object may have more than one candidate snap point. If more than one potential snap point is found, the one closest to the crosshair position is chosen. This point is then identified with a "blip" on the screen and is used instead of the point at the crosshair position.

Of course, it is possible that no snap points of the specified type will be found. AutoCAD's action in such a situation depends on whether the point selection is being made using the running object snap modes or an override.

Running modes: If running object snap modes are in effect and no snap points are found, the crosshair position is used, just as though all object snap modes were off. No warning is displayed.

Override: If the running object snap modes are being overridden for this point selection, an "***Invalid***" message is displayed if no point is

found that satisfies the override conditions. Most commands abort when this occurs. The LINE command, however, ignores any invalid "To" point and prompts for another.

Regular Snap mode takes effect on points from the input device before they are considered by object snap. Ortho mode is overridden by object snap if a candidate snap point is found.

Object snap recognizes only the entities *visible* on the screen. Thus, material on turned-off layers and the "pen up" portions of dashed lines, are not seen by object snap. Object snap can snap to objects that are components of Blocks. There are restrictions on referring to certain objects that are Block members -- these restrictions are mentioned below.

8.6.2 Object Snap Modes

The following modes are available for object snap. You can abbreviate the mode names to the first three characters for the OSNAP command or for single point override specifications.

NOTE: For purposes of object snap, each segment of a Polyline (+3) is treated as a separate Line or Arc entity. For wide Polyline segments, object snap uses the center-line, ignoring the width.

Nearest	Snaps to the point on a line, arc, or circle that is closest to the position of the crosshairs, or snaps to the Point entity that is closest to the crosshairs. Arcs and circles that are part of a Block are not seen by this mode.
Endpoint	Snaps to the closest endpoint of a line or arc.
Midpoint	Snaps to the midpoint of a line or arc.
Center	Snaps to the center of an arc or circle. (Note that you must point to a visible part of the circumference to designate the arc or circle. If this mode is combined with others, you may have trouble with another object snap point being closer than the center).
Node	Snaps to a Point entity. Points may be placed overlapping joints or attachment locations in a Block definition and thus function as "snap nodes" after the Block has been INSERTed.
Quadrant	Snaps to the closest quadrant point of an arc or circle. These are the 0, 90, 180, and 270 degree points on a circle or arc (of course, only the visible quadrants of an arc may be selected). Note that the quadrant points are taken from the original definition of the circle or arc, and that if it is a member of a rotated Block the quadrant points rotate with it.
Intersection	Snaps to the intersection of two lines, a line with an arc or circle, or two circles and/or arcs. Intersections with arcs and circles that are members of Blocks are not seen by this mode, although line intersections within Blocks work normally. For this object snap mode, both objects must cross the target on the screen.
Insert	Snaps to the insertion point of a Shape, Text, or Block entity.

The modes listed above snap to individual drawing features. The next two modes snap to features with respect to the "last point" entered (i.e., the point whose value would be returned by "@"). This choice is very natural when using the following modes in line construction, for example.

Perpendicular	Snaps to the point on a line, circle, or arc that forms a normal from that object to the last point. A circle or arc used with this mode must not be part of a Block.
Tangent	Snaps to the point on a circle or arc that, when connected to the last point, forms a line tangent to that object. A circle or arc used with this mode must not be part of a Block.

You can use the object snap modes described thus far in any combination. The "Quick" mode, described below, alters the method AutoCAD uses to select candidate snap points; it can be used only in conjunction with one or more of the modes described above.

Quick	<p>As described earlier, object snap normally searches for all objects crossing the target and selects the closest potential snap point of the specified type(s). When many entities are visible on the screen, this search can result in a noticeable delay. In such situations, you can use "Quick" mode along with your other object snap modes; "Quick" object snap stops searching as soon as it finds one object with at least one point of the specified type.</p> <p>If two or more objects with candidate object snap points cross the target, "Quick" mode chooses the first one it sees. Since it is difficult to predict which object will be seen first, we recommend that you have just one such object in the target when using "Quick" object snap mode.</p> <p>If "Intersection" is one of the specified object snap modes, a full search is performed regardless of the presence of "Quick" mode.</p>
-------	---

The final mode specification is used to specify no modes in the OSNAP command or to disable object snap entirely for a single point.

None	Turns off object snap.
------	------------------------

8.6.3 OSNAP Command

Using the OSNAP command, you can select one or more object snap modes to be in effect for all subsequent point selections. These become your "running" object snap modes; they are remembered along with your drawing, are displayed by the STATUS command, and remain in effect until countermanded by another OSNAP command. When you begin a new drawing, the initial object snap modes are governed by the prototype drawing.

The OSNAP command format is:

Command: OSNAP
Object snap modes:

To disable object snap, reply with "None", "Off", or simply RETURN; all have the same effect. You can select one or more object snap modes by entering their names (three

characters are sufficient), separated by commas. For instance, to snap to the closest of the endpoint, midpoint, or quadrant, enter:

Command: OSNAP
 Object snap modes: ENDP,MIDPOI,QUAD

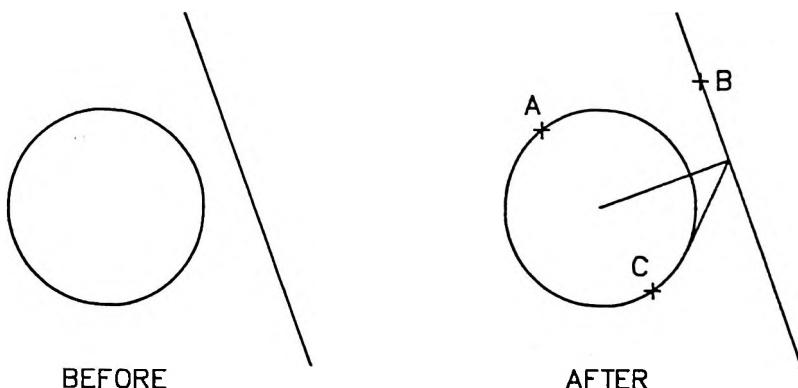
8.6.4 Single Point Override

You can invoke object snap modes for a single point by typing them on the keyboard (followed by a space or RETURN) whenever AutoCAD prompts for a point to be entered. If you want multiple modes, use commas to separate them. These specified modes completely replace the running OSNAP modes for only the next point specified, after which the object snap modes revert to those specified by the most recent OSNAP command. AutoCAD prompts with an appropriate preposition (such as "of" or "to") to confirm that the modes have been accepted. Invalid mode specifications abort point entry.

For example, you could use the following command sequence to draw a line from the center of a circle perpendicular to an existing line, followed by a line tangent to the circle.

Command: LINE
 From point: CENTER of (point on circle, "A")
 To point: PERP to (point on line, "B")
 To point: QUICK.TAN to (point on circle, "C")
 To point: (RETURN)

"Before" and "after" views of the drawing might be:



Many users will use object snap overrides only, and may never have cause to use the OSNAP command. The OSNAP command is especially handy, however, when "wiring up" drawings with many snap nodes, or when drawing structures in which many connections to intersections exist.

8.6.5 APERTURE Command

As described above, a special "target" box is added to the screen crosshairs when object snap is activated, and only the objects that cross the target box are candidates for object snap. You can adjust the size of the target box by means of the APERTURE command:

Command: **APERTURE**

Object snap target size (1-50 pixels) <default>:

Enter the number of pixels (picture elements, or dots) the target box should extend from the crosshair position. The larger the number, the bigger the target box. Values in the range 1-50 are possible. AutoCAD remembers the most recent setting of the aperture size and uses it for all subsequent editing until you change it. The initial aperture setting is governed by the prototype drawing.

If the display device has unequal horizontal and vertical resolutions (dots per inch), the target box will be rectangular, not square.

8.7 Status Line (+1)

In systems with the ADE-1 package, an area of the graphics monitor can be used to display a *status line* showing the current layer name, the status of various AutoCAD modes, and the current coordinates of the screen crosshairs. A typical status line would look like:

Layer: XYZ Ortho Snap Tablet X=1.1234 Y=2.3456

Only the first eight characters of the layer name are displayed. Mode names like "Ortho" and "Snap" appear only if the associated modes are currently on.

You can decide at configuration time whether or not to display the status line. See Appendix D for details.

If present, the coordinate display is initially "static"; that is, it is updated only when you select a point. However, you can request a "dynamic" coordinate display that is constantly updated as you move the screen crosshairs. This dynamic coordinate display has one additional feature: if a rubber-band line is currently part of the crosshairs, the display switches from X-Y coordinates to a "length and angle" display of the form:

length < angle

showing you the length and angle of the rubber-band line. The static/dynamic mode of the coordinate display and use of the "length and angle" format are governed by a control key described in the next section.

NOTE: Some graphics monitors cannot display enough text characters on one line to handle the normal status line described above. On such systems, one-letter abbreviations are used for the mode names, and the word "Layer" is also abbreviated. For example, the status line:

L: XYZ OST X=1.1234 Y=2.3456

is equivalent to the example at the beginning of this section.

8.8 Mode Toggle Control Keys

The SNAP, GRID, ORTHO, and ISOPLANE (+2) commands described earlier in this chapter, and the TABLET command (Section 12.3) can be used to change their associated modes, but there is often a need to change them during another command.

For instance, while drawing a sequence of orthogonal lines with the LINE command, you might need to draw a couple of diagonal lines and then resume Ortho mode. To do this, you could terminate the LINE command, issue the ORTHO OFF command, reinvoke LINE to draw your diagonal lines, issue ORTHO ON, and then LINE again.

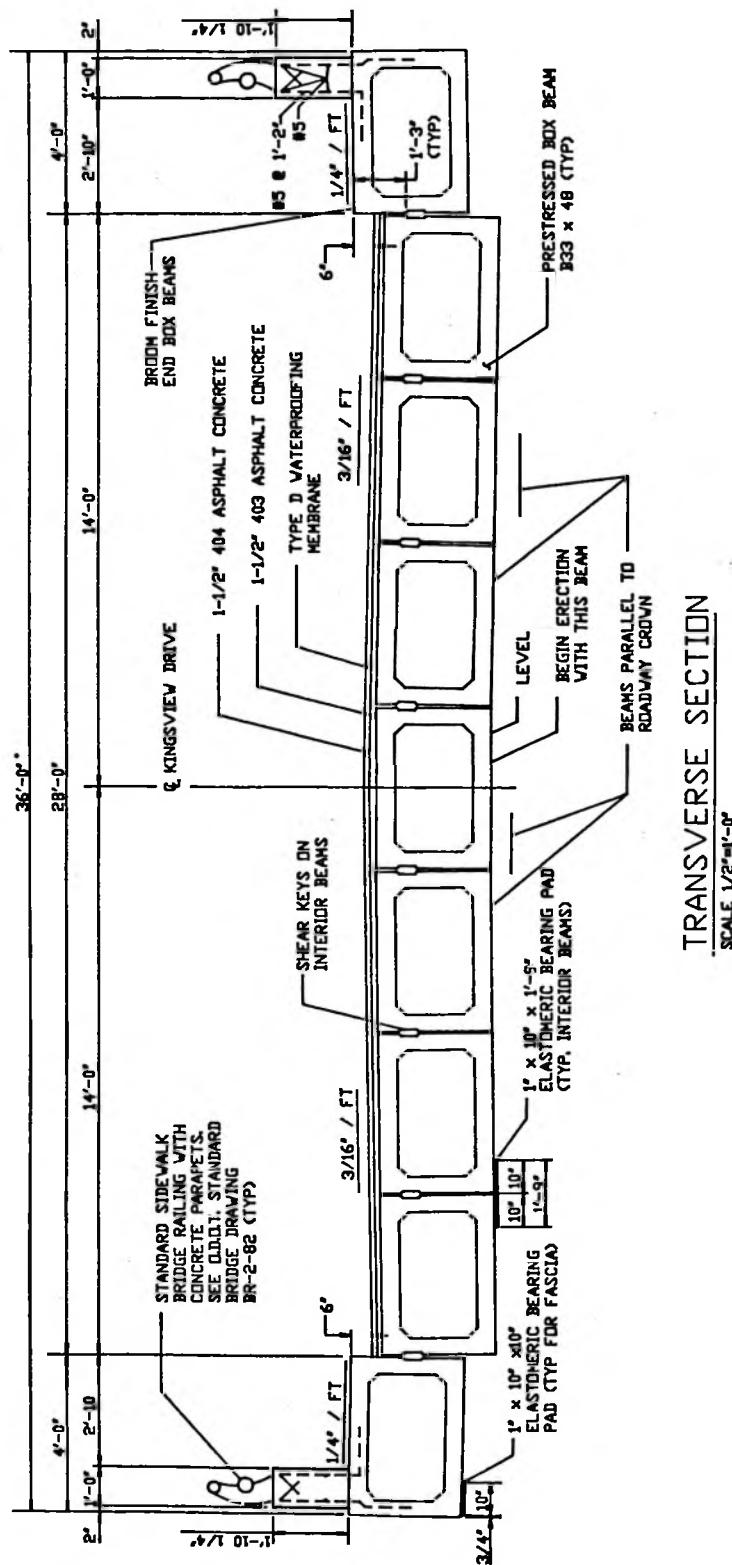
Since the above approach is rather cumbersome, control keys are provided for Snap, Grid, Ortho, Isoplane, and Tablet modes. These keys may be used at any time, even in the middle of a command. Each key "toggles" or "flips" the associated mode: if off, it is turned on; if on, it is turned off. (The control key for Isoplane mode toggles through the three possible isometric planes.) There is also a control key for the dynamic coordinate display (ADE-1 feature).

The control keys are listed below. Those marked with a "+1" are available only with the ADE-1 package; those marked with "+2" are part of the ADE-2 package.

CTRL B	Snap mode on/off (CTRL S is used by the operating system)
CTRL D +1	Selects the format of the coordinate display, in a circular fashion (dynamic length and angle, dynamic X-Y coordinate, static X-Y coordinate). The "length and angle" format is available only when a rubber-band cursor is displayed on the screen; it displays the length and angle of the rubber-band line.
CTRL E +2	Selects the next isometric plane in a circular fashion (Left, Top, Right ...)
CTRL G	Grid on/off
CTRL O	Ortho mode on/off
CTRL T	Tablet mode on/off

When any of these keys is used, a message is displayed confirming the action.

On some computers, special function keys may be assigned to these functions in addition to the standard control keys listed above. See your AutoCAD Installation Guide / User Guide Supplement for information. It is also possible to construct menu items to invoke these control functions (see Appendix B).



Chapter 9

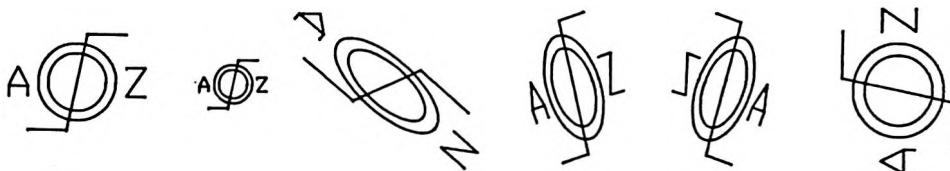
COMPLEX OBJECTS - BLOCKS

9.1 General Information

A **Block** is a set of entities grouped together into a complex object. Once so grouped, the entities are given a **Block Name**; you can use this name to insert that group of entities into your drawing wherever you like. Each instance of the Block in the drawing (a **Block Reference**) can be given different scale factors and rotation. For example, consider the following figure:



If the entities in this figure were grouped into a Block, they could then be inserted in any or all of the following ways.



A Block is treated as a single object by AutoCAD; you can MOVE, ERASE, or LIST a Block simply by pointing to any entity within it. The internal structure of a Block is irrelevant; it is considered a primitive entity just like a line and can be manipulated as such by the Edit and Inquiry commands.

You can define a Block from a set of objects in your current drawing, or from a separate drawing previously created with the Drawing Editor. Thus, you can create your own library of "parts" interactively and insert them into your drawings at any specified location. Each insertion of a Block simply uses the original definition, with a different position, scale, and rotation. This makes drawings with many uses of a single Block very compact and efficient.

Four commands are provided specifically for defining and manipulating Blocks; these commands (BLOCK, INSERT, BASE, and WBLOCK), are described in this chapter. Also, RENAME and PURGE can be used to rename and delete existing Blocks and other named objects. These commands are described in Chapter 3.

9.1.1 Blocks and Layers

A Block can be composed of entities that were drawn on several layers; the layer information is preserved in the Block. Upon insertion, each entity is drawn on its original layer, no matter what the current drawing layer happens to be. Thus, if a Block includes a circle that was drawn on layer "ABC", that circle is drawn on layer "ABC" (with that layer's color and linetype properties) when the Block is inserted, even if the current drawing layer at that time is "XYZ".

There is one exception to this rule, however. Entities within a Block that were drawn on the special layer named "0" are generated on the current layer when the Block is inserted. This allows construction of Blocks that are generated on the current layer just like lines, circles, and other built-in entities. Variable ("0") and fixed (non-"0") layers may be freely mixed within a Block.

If the layer on which a Block was inserted is frozen, that Block is not generated at all, even if portions of it reside on other (thawed) layers.

9.1.2 Nested Blocks

A Block may contain other Block References. For example, you can have a "canned" memory array that you insert on various printed circuit boards. That memory array can contain Blocks defining the various components used in the array, each with feed-through holes at appropriate places. Each feed-through hole can itself be a Block. There is no limit to the nesting complexity of Blocks, except that self-reference is not permitted.

If an inner Block includes entities on layer "0", those entities "float" up through the nested Block structure until an outer Block is found on a "fixed" (non-"0") layer; when this occurs, these layer-"0" entities are placed on that layer. If no "fixed" layer is encountered in the outer Blocks, the entities remain on layer "0". For example, consider the following:

Block A contains: Circle on layer "0"
 Block B contains: Block A on layer "0"
 Block C contains: Block B on layer "FOUNDATION"

When Block B is inserted, its circle (Block A) is drawn on the current layer. However, no matter what layer is current when Block C is inserted, its circle (Block A of Block B) is always drawn on layer "FOUNDATION".

9.2 BLOCK Command - Block Definition

The BLOCK command lets you create new Blocks "on the fly" from parts of an existing drawing. The command format is:

Command: **BLOCK** Block name (or ?): (name)

Enter the name you want the new Block to have. Block names may be up to 34 characters long, and may contain letters, digits, and the special characters "\$ - _" (dollar, hyphen, and underscore). AutoCAD converts any letters to upper case. If a Block with this name already exists, AutoCAD prompts:

Block XYZ already exists.
 Redefine it? <N>

If you reply "N" or give a null response, the BLOCK command exits without changing anything. You can respond "Y" to redefine the Block in question; the drawing will be regenerated (subject to REGENAUTO), revising any existing insertions of the Block.

The next prompt is:

Insertion base point:

Respond by designating the point to be used as the base (reference) point for subsequent insertions of this Block. A typical base point is the center of the Block or its lower left corner; this part of the Block appears at the "insertion point" when the Block is later inserted in a drawing. It is also the point about which the Block can be rotated during insertion.

After you have chosen the Block's name and its base point, the only information still needed is the set of entities to be used to form the Block. To obtain this information, AutoCAD prompts:

Select objects or Window or Last:

You can use any of the entity selection methods described in Section 5.1 to designate the components of the new Block. AutoCAD then constructs a Block Definition with the specified name, insertion base, and entities. AutoCAD provides visual confirmation of the entities you've selected by erasing them from the screen; it also deletes them from the drawing. If you don't want these entities erased, use the OOPS command (Section 5.2) to restore them to the drawing.

The following BLOCK command creates a Block named "MYPART", with an insertion base point of (2,5).

Command: BLOCK Block name (or ?): MYPART
 Insertion base point: 2.5
 Select objects or Window or Last: (do so)

The Block thus created can now be inserted by means of the INSERT command, described in Section 9.3.

NOTE: Blocks created using the BLOCK command are stored only in the drawing in which they were created, and copies of them can be inserted only in that drawing. You can use the WBLOCK command (Section 9.5) to write the Block Definition to a disk file for inclusion in other drawings.

9.2.1 BLOCK ? - Listing Defined Blocks

To list all the Block names present in a drawing, use the "BLOCK ?" command sequence.

Command: BLOCK Block name (or ?): ?

This can be handy if you've forgotten the name of a Block. If the drawing contains many Blocks, the entire list may not fit on the screen at once. In this case, AutoCAD will pause after each screen-full and prompt:

-- More --

To continue the listing, press RETURN.

9.3 INSERT Command - Block Reference

You can use the INSERT command to insert a previously defined Block into your drawing. The command interaction looks like:

Command: **INSERT** Block name (or ?): (*Block name*)
 Insertion point: (*indicate where it is to be placed*)
 X scale factor <1> / Corner / XYZ: (*number or point*)
 Y scale factor (default=X): (*number*)
 Rotation angle <0>: (*number or point*)

(The "XYZ" option in the "X scale factor . . ." prompt appears only if the ADE-3 package is present, and is associated with 3D visualizations. This topic is covered in Chapter 14.)

A copy of the specified Block is drawn with its base point at the designated insertion point. First, AutoCAD multiplies all *X* and *Y* dimensions of the Block by the *X* and *Y* scale factors you supplied. It then rotates the Block by the rotation angle, using the insertion point as the center of rotation. If you give a null response to a prompt, the default value is used; thus, you can get the Block inserted at its original scale and rotation just by pressing RETURN after all three prompts. Note that the *Y* scale equals the *X* scale by default.

The name of the last Block inserted during the current editing session is remembered and becomes the default for subsequent INSERTs.

9.3.1 Negative Scale Factors

You can specify negative values for the *X* and *Y* scale factors to insert mirror images of Blocks. For example, the object on the left was inserted with *X* and *Y* scale factors of 1, while the right-hand drawing shows the same Block, inserted with an *X* scale factor of -1.

X scale = 1
Y scale = 1

X scale = -1
Y scale = 1



9.3.2 Corner Specification of Scale

You can define the *X* and *Y* scales at the same time, using the insertion point and another point as the corners of a box. The *X* and *Y* dimensions of the box become the *X* and *Y* scale factors. If the Block Definition fits in a 1-unit by 1-unit box, the inserted object will fit in the box you have defined. (See the discussion of 1 x 1 Blocks later in this section.)

In order to use corner specification, simply enter a point in response to the "X scale factor:" prompt. You can also reply to this prompt with "C", in which case AutoCAD will prompt for the "Corner point:".

The second point should be above and to the right of the insertion point, otherwise you will get a negative scale factor and a mirror image of the original part.

NOTE: When you use the Corner method of scale specification, it is difficult to ensure that the X and Y scales come out identically. An appropriate Snap spacing can help, but this is often restrictive. If your application requires matching X and Y scales, we suggest that you specify the X and Y scales in the normal fashion, instead of using the Corner method.

9.3.3 Angle Specification via Point

If you enter a point for the rotation angle, AutoCAD measures the angle of the line from the insertion point to that point, and uses it for the rotation. If Ortho mode is on, the rotation angle is forced to be orthogonal.

9.3.4 Dynamic Insertion (+2)

The ADE-2 package offers a special feature for Block insertion. If you respond to the "Insertion point", "X scale factor", or "Rotation angle" prompts with "DRAG" and DRAGMODE (Chapter 6) is ON, you can move the part into position, scale it, or rotate it all using your pointing device. At each step, interim versions of the part are displayed, so that you can position it in your drawing visually.

Dragging of the scale factor is similar to the Corner specification method described above and the same considerations apply. We suggest that you avoid dragging very complex parts, since drawing, erasing, and redrawing such parts as you move them around can take a long time.

9.3.5 1 x 1 Blocks

A very useful convention is to draw the original part in a 1-unit by 1-unit square. Upon subsequent insertion, the X and Y scale factors become the actual dimensions in drawing units. This is particularly handy when using the "corner" method of specifying the X and Y scales.

Consider, for example, the 1 x 1 Block called "RECTANG" that is supplied with the AutoCAD software. This Block consists of a square with its lower left corner at (0,0) and its upper right corner at (1,1). Whenever you want to draw a rectangle, you can "INSERT RECTANG". The X scale factor you specify becomes the width of the rectangle, and the Y scale factor becomes the rectangle's height.

9.3.6 Example

Often, a figure is composed of a basic group of entities, duplicated to form a symmetrical object. With a little planning, you can use the BLOCK and INSERT commands to ease the construction of such figures. For example, Figure A below can be created by drawing the much simpler object shown as Figure B.

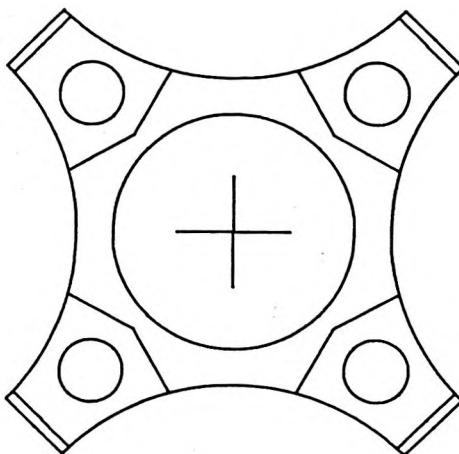


Figure A

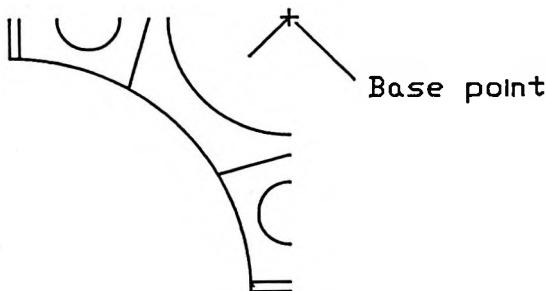


Figure B

The BLOCK command can be used to form a Block from Figure B, with the indicated base point. Then INSERT can be used to draw the Block four times, using the same insertion point for all four insertions, with rotation angles of 45, 135, 225, and 315 degrees.

Another method of creating this figure is to insert one copy of the Block and then use the ARRAY command (Section 5.2) to construct a circular pattern. The center of the pattern is the Block insertion point, and the pattern contains four items spaced 90 degrees apart from each other.

9.3.7 INSERT * - Retaining Individual Parts

No matter how complex a Block may be, it is treated as a single entity by AutoCAD; if you point to one line of it and use the MOVE command, for instance, the entire object moves, not just the line you pointed to.

If you'd prefer to simply transcribe the Block, retaining its separate entities, you can do so by preceding the Block name with an asterisk on the INSERT command. For instance:

Command: INSERT Block name (or ?): *WIDGET

Insertion point: 10,12

Scale factor <1>: 1

Rotation angle <0>: 0

inserts the entities comprising the Block "WIDGET" at the specified point, applying a scale factor of 1 and no rotation. Note that only one scale factor is requested for an "INSERT *"; it is applied to both the X and Y scales and cannot be negative.

Entities drawn on layer "0" of a Block remain on that layer when inserted with "INSERT *".

9.3.8 INSERT ? - Listing Defined Blocks

As described earlier in this chapter, the "BLOCK ?" command sequence lists the names of all defined Blocks in the current drawing. For your convenience, the "INSERT ?" command sequence does the same thing.

9.4 Entire Drawings as Blocks

As described earlier in this chapter, the INSERT command is used to draw a Block that has been previously defined. However, the INSERT command has another important function: if no Block has been defined with the specified name, INSERT searches for a drawing file having that name. If such a file is found, it is first copied into the current drawing as a Block Definition, using the file name as the Block name. Then the INSERT command continues as usual, drawing a copy of the newly defined Block.

Using this feature of the INSERT command, you can construct a parts library with a separate file for each part and easily insert those parts in as many other drawings as you like. Each part is simply a separate drawing file created with the Drawing Editor.

You can include a drive letter in the file name supplied to INSERT, as in "B:WIDGET"; the drive specification does not become part of the Block name. You can also load a drawing file and assign a different name to the Block created from it. If you want to do this, respond to the INSERT command's "Block name" prompt with:

Block name=file name

where "Block name" is the name you want given to the Block formed from file "file name". Do not specify a file type; INSERT assumes a file type of ".DWG". For instance:

Command: INSERT Block name (or ?): MUSTANG=B:EDSEL

loads the file B:EDSEL.DWG, creating a Block named MUSTANG.

When you INSERT a drawing, it is physically transcribed into the current drawing and becomes a Block Definition. This means that the part file does not have to be available when you later work on the drawing into which it was inserted. Subsequent insertions simply reference the Block Definition with different position, scale, and rotation.

Any Blocks defined in the inserted drawing become available for use in the current drawing.

Insertion of a drawing does not affect the drawing that is being inserted.

9.4.1 BASE Command

When using the BLOCK command to dynamically create a Block from objects in the current drawing, you must supply a "base insertion point" to be used by subsequent INSERT commands. Such a reference point is also necessary when inserting an entire drawing, so a means of designating this point is required.

If your current drawing is of a part that you expect to insert into other drawings, you should specify the base point for such insertions using the BASE command:

Command: BASE Base point: (point)

If you do not use the BASE command when drawing the part, the point (0,0) in the part becomes its insertion base by default.

9.4.2 Changing an Inserted Drawing

Suppose you have inserted drawing WHEEL into drawing CAR. WHEEL is now a Block in CAR, and the original drawing file WHEEL is not needed when CAR is edited. In fact, if you modify drawing WHEEL, the changes are not automatically applied to CAR.

There is, however, a way to update Block WHEEL in drawing CAR to reflect the changes you have made. You can do this by means of the

Block name=file name

form of the INSERT command. If the Block name and file name are identical, the "*file name*" may be omitted, but you must specify the "=" to redefine the Block.

After reading the new Block definition from the file, AutoCAD displays the message:

Block xxx redefined

and regenerates the drawing to update all existing occurrences of the Block. (You can suppress this regeneration by setting REGENAUTO OFF -- see Section 6.10.) The remaining prompts for the INSERT command (insertion point, etc.) are then issued. If all you want to do is change the Block Definition without creating a new insertion, simply enter CTRL C when prompted for the insertion point. For instance:

```
Command: INSERT Block name (or ?): WHEEL=
Block WHEEL redefined
(... auto regen ...)
Insertion point: CTRL C
```

loads Block WHEEL from file WHEEL.DWG and replaces the previous definition of Block WHEEL. The drawing is then automatically regenerated, applying the new definition of Block WHEEL to all existing insertions of it. The CTRL C terminates the INSERT command at this point.

9.4.3 Special Considerations

A drawing file may contain numerous "named objects", including Blocks, layers, linetypes, Text styles, and named views (+2). When you INSERT a drawing into the current drawing, the named objects in the inserted drawing (except for the named views, if any) are added to the current drawing. If the inserted drawing and the current drawing contain objects with duplicate names, the definitions in the current drawing take precedence.

For instance, suppose drawing CHILD is to be inserted in the current drawing, FAMILY, and that these drawings contain the following named objects.

	FAMILY	CHILD
Linetypes	CONTINUOUS DOT	CONTINUOUS DASH
Layers	0 1 MOTHER FATHER SON	0 1 (linetype DASH) DAUGHTER
Text styles	STANDARD FANCY (font X)	STANDARD FANCY (font Y)

The "INSERT CHILD" command begins by adding the DAUGHTER layer and DASH linetype to the FAMILY drawing. It then forms a Block named CHILD from all the entities in the CHILD drawing. Finally, the INSERT command creates one reference to the new CHILD Block, on the current layer.

Note that:

- o Layer 1 of CHILD uses the DASH linetype, but layer 1 of FAMILY does not. FAMILY's definition of layer 1 takes precedence because it is the current drawing, so entities from layer 1 of CHILD are drawn using continuous lines when inserted in this drawing.
- o The definitions of the FANCY text style are different. Again, FAMILY's definition overrides that of the inserted file, so any text items in CHILD that used the FANCY style are now drawn using font X rather than font Y. This can have a significant effect on the appearance of the text.
- o Text font files and Shape files referenced by the inserted drawing must be present at the time of the INSERT and whenever the current drawing is subsequently edited.

9.5 WBLOCK Command - Write Block to Disk

You can use the WBLOCK command to write all or part of a drawing out to a new drawing file. The command format is:

Command: WBLOCK File name: *(output file name)*
Block name:

Do not include a file type when entering the file name; a type of ".DWG" is assumed. Four different responses to the "Block name:" prompt are possible:

- Name The entities comprising the definition of the specified Block are written.
- = Shorthand for the above, when the Block and the output file have the same name.
- * The entire drawing is written. This is similar to the SAVE command (Chapter 3), except that unreferenced Block Definitions are not written. This provides a means of deleting Block Definitions once all references to them have been deleted. See the PURGE command (Chapter 3) for another method.
- Blank AutoCAD asks you to select objects and an insertion base point, as for the BLOCK command. The selected objects are written to the specified file, and they are deleted from the current drawing. You can use the OOPS command to retrieve them if you so desire.

9.6 Why Use Blocks?

Blocks provide several advantages that may not be readily apparent. The sections that follow describe these advantages.

9.6.1 Work Reduction and Organization

Using Blocks, a drawing can be built up from smaller details in a "building block" fashion. Drawing of features that appear often can be performed once, rather than repetitively.

9.6.2 Customization

Blocks can be used to construct a custom library of drawing parts suited to your particular application. By combining these Blocks with custom menus (see Appendix B), you can create a complete application environment around AutoCAD.

9.6.3 Ease of Redefinition

Drawings must often be revised when the specifications for a particular part change. This can be a laborious task if you must locate each of the affected parts and edit them. However, if you have defined the part as a Block, you can simply redefine the Block; all references to that Block are automatically updated.

9.6.4 Space Savings

Each entity you add to a drawing increases the size of the drawing file on disk. AutoCAD must remember certain information about each entity; the points, radii, scale factors, and so on, that describe its size and its location within the drawing. Consider a facilities plan with numerous desks, chairs, and other furniture. If each chair is drawn as several lines and arcs, the space required for all the chairs may be considerable.

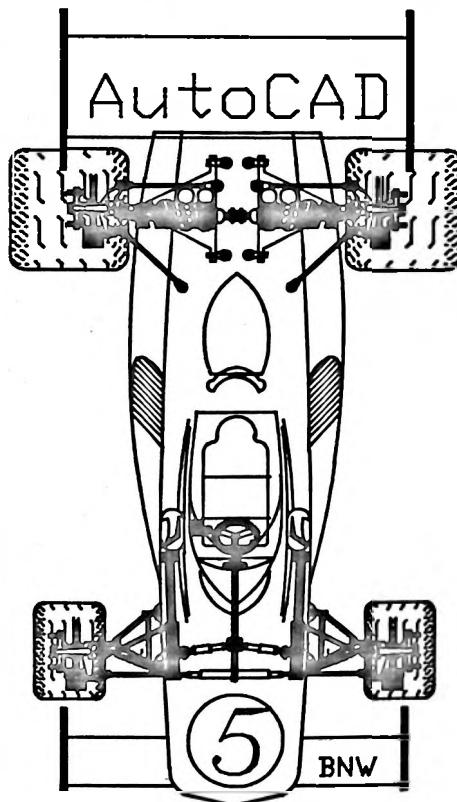
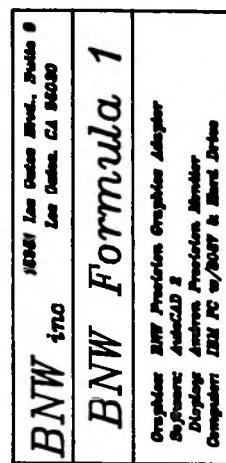
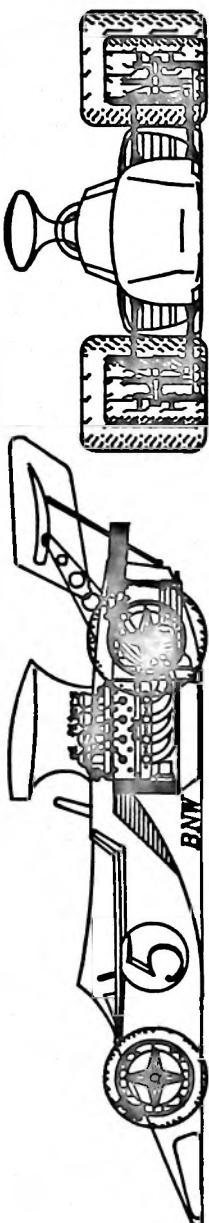
But suppose you created a CHAIR Block composed of all those lines and arcs and inserted that Block wherever a chair was needed. The Block Definition would contain all the line and arc entities, but there would be only one such definition. For each insertion of the Block, AutoCAD only needs to keep track of one entity (a Block Reference). Each chair thus takes much less disk storage. The more complex the Block Definition, the greater the space savings for each insertion.

Note that an "INSERT *blockname" inserts the Block's individual parts; no space savings will result. Also, the COPY, ARRAY, and MIRROR (+2) commands simply make *copies of the individual objects* you select, expanding the drawing file for each object just as if you had drawn it individually. While they save drawing time, these commands do not save file space unless the selected objects are Block References.

9.6.5 Attributes (+2)

If the optional ADE-2 package is present, you can attach Attributes to a Block. Attributes are textual information that can vary with each insertion of a Block and can be displayed as ordinary text or remain invisible. They can be extracted from a drawing and transferred to a database, bill of materials program, etc. Attributes are fully described in Chapter 11.

AutoCAD -- (9) COMPLEX OBJECTS - BLOCKS



Chapter 10

SPECIAL FEATURES

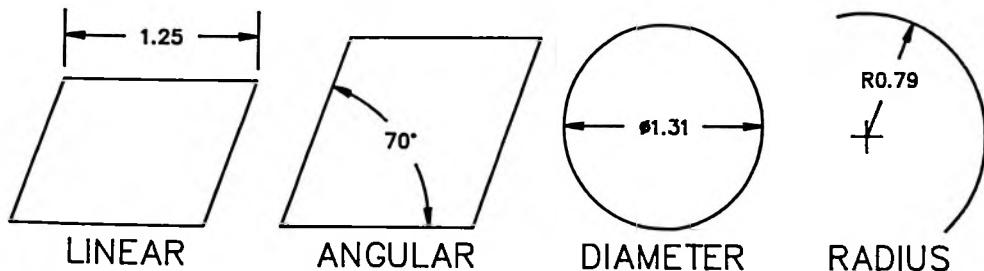
10.1 Semi-automatic Dimensioning (+1)

This section describes the ADE-1 package's dimensioning feature.

10.1.1 Introduction

In many applications, a precise drawing plotted to scale is not sufficient to convey the desired information; annotations must be added showing the lengths of objects or the distances or angles between objects. *Dimensioning* is the process of adding these annotations to a drawing. The term *dimensioning* also refers to the annotations themselves.

AutoCAD provides four basic types of dimensioning; linear, angular, diameter, and radius. A simple example of each is shown below.

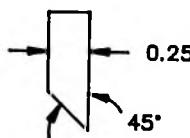


You can draw dimensions "by hand", using such commands as DIST and LIST to obtain the relevant information and then drawing the necessary lines, arcs, arrows, and text. However, AutoCAD's Dimensioning feature makes this process much simpler.

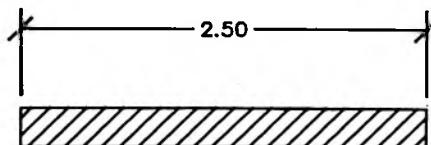
Before proceeding with the details of dimensioning with AutoCAD, we must introduce a few terms used throughout this discussion:

Dimension line

This is a line with arrows at each end, drawn at the angle at which the dimension has been measured. The dimension text is situated along this line, sometimes dividing it into two lines. Usually, the dimension line is inside the measured area, as shown in the first set of figures. Sometimes, however, it does not fit. In such cases, two short lines are drawn outside the measured area, with arrows pointing inward, as in the following figure.

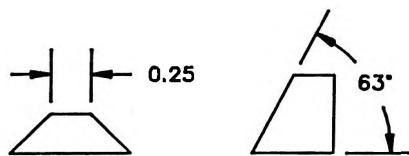


For angular dimensioning, the dimension line is actually an arc. For linear dimensioning, you can select *tick marks* to be drawn rather than arrows. Tick marks are short lines drawn through the points where the dimension line meets the extension lines, at 45 degrees with respect to the dimension line. A dimension line with tick marks is shown below.



Extension lines

If the dimension line is drawn outside the object being measured, straight *extension lines* (called "witness lines" in some texts) are drawn from the object, perpendicular to the dimension line. Extension lines are used only in linear and angular dimensioning. Because they are sometimes superfluous, a means is provided to suppress one or both of them. Dimensions with extension lines are illustrated below.



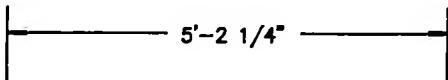
Dimension text

This is a text string that usually specifies the actual measurement. You can use the measurement computed automatically by AutoCAD, supply your own text, or suppress the text entirely. If you use the default text, you can instruct AutoCAD to append plus/minus tolerances to it automatically (see Tolerances, below).

The text is drawn using the current Text style. If you use the default text, its format is governed by the UNITS command (Section 3.6); you can select "grads", "radians", or "degrees, minutes, and seconds", format for angular dimensions, and "feet and inches"

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format for other dimension types. An example of "feet and inches" format is shown below.

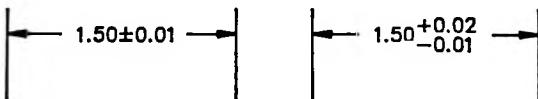


The default text may include degree, diameter, and plus/minus special symbols. AutoCAD uses the "%%" character sequences described in Section 4.8.7 to generate these symbols; you can use the same technique to include these symbols in text you supply explicitly.

NOTE: Dimension text requires a normal (horizontal) text font. If the current text style uses the VERTICAL font, you must change to a normal style before dimensioning the drawing.

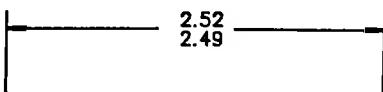
Tolerances

Dimension tolerances are plus/minus amounts that AutoCAD can append to the dimension text it generates automatically. You specify the plus and minus amounts; they may be equal or different. If the plus and minus tolerances are equal, AutoCAD draws them with a "plus/minus" symbol. If they differ, the tolerances are drawn one above the other. Examples of the two formats are shown below.



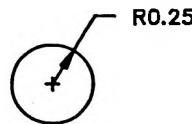
Limits

You can elect to have the tolerance values applied to the measurement. The default dimension text is then the resulting high/low limits rather than the nominal measurement with tolerances. For instance, if the nominal measurement is 2.50 units and the tolerances are +0.02 and -0.01, the limits are drawn as follows:



Leaders

For some dimensioning and other annotations, the text may not fit comfortably next to the object it describes. In such cases, it is customary to place the text nearby and draw a *leader* line from the text to the object. For instance, when diameter or radius dimensioning is desired, but the arc or circle is too small for the dimension text to fit inside, a leader can be drawn from the text to the arc or circle. An example is shown below.



Center mark/line A *center mark* is a small cross marking the center of a Circle or Arc. *Center lines* are broken lines crossing at the center and intersecting the circumference of the Circle or Arc at its quadrant points. A center mark and center lines are shown below.



CENTER MARK



CENTER LINES

Variables

The manner in which dimensions are drawn is controlled by a set of *dimensioning variables*. You can change their values to meet the requirements of a particular dimensioning situation. Some of the variables are simply "on/off" switches, whereas others hold numeric values. The dimensioning variables are discussed in detail in Section 10.1.8.

The dimension line, extension lines, arrows, leaders, and dimension text are drawn as separate entities on the current layer. Thus, after a dimension has been drawn, you can change the placement or content of the text, etc.

Dimensioning and Polylines (+3)

For the purposes of dimensioning, each segment of a Polyline is treated like a separate Line or Arc entity. In the case of wide Polylines, all measurements are based on the center line of the segment, ignoring its width.

10.1.2 DIM Command

When you want to add dimensioning to a drawing, enter the **DIM** command, as in:

Command: **DIM**
Dim:

The "Dim:" prompt indicates that you are in dimensioning command mode. In this mode, the normal set of AutoCAD commands is replaced by a special set of dimensioning commands. To return to normal command mode (as indicated by the "Command:" prompt), use the **EXIT** command or respond to the "Dim:" prompt with **CTRL C**.

The dimensioning commands can be grouped into five categories:

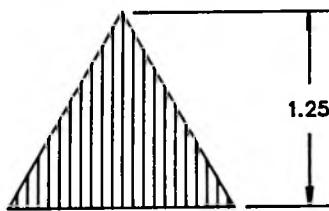
1. Linear dimensioning commands
2. Angular dimensioning commands
3. Diameter dimensioning commands
4. Radius dimensioning commands
5. Dimensioning utility commands

Each command name can be abbreviated to its first three letters. The commands in each category are listed below.

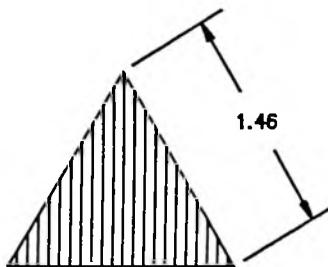
Linear Dimensioning Commands

HORIZONTAL Generates a linear dimension with a horizontal dimension line, as shown at the beginning of this section.

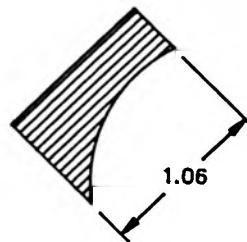
VERTICAL Generates a linear dimension with a vertical dimension line, as shown below.



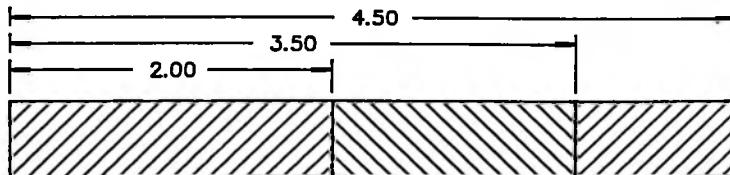
ALIGNED Generates a linear dimension with the dimension line parallel to the specified extension line origin points. This permits you to align the dimensioning notation with the object. An example is shown below.



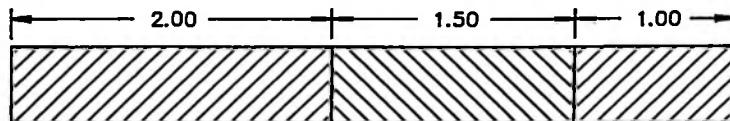
ROTATED Generates a linear dimension with the dimension line drawn at a specified angle. For example, the dimension line in the following figure is drawn at a 45 degree angle.

**BASELINE**

Continues a linear dimension from the baseline (first extension line) of the previous dimension. As shown in the figure below, the new dimension line is offset to avoid drawing on top of the previous dimension.

**CONTINUE**

Continues a linear dimension from the second extension line of the previous dimension. In effect, this breaks one long dimension into shorter segments that add up to the total measurement. An example is shown in the following figure.

**Angular Dimensioning Commands****ANGULAR**

Generates an arc to show the angle between two nonparallel lines. An example was shown at the beginning of this section.

Diameter Dimensioning Commands**DIAMETER**

Dimensions the diameter of a circle or arc. An example was shown at the beginning of this section.

Radius Dimensioning Commands

RADIUS Dimensions the radius of a circle or arc, with an optional center mark or center lines. An example with a center mark was shown at the beginning of this section.

Dimensioning Utility Commands

CENTER Draws a Circle/Arc center mark or center lines.

EXIT Returns to the normal Drawing Editor command mode.

LEADER Draws a line or sequence of lines (similar to the normal LINE command) for controlled placement of dimension text. Useful mostly for radius and diameter dimensioning.

REDRAW Redraws the entire display, erasing any marker "blips" that were present (just like the normal REDRAW command).

STATUS Displays all dimensioning variables and their current values.

UNDO Erases the annotations produced by the most recent dimensioning command.

These commands are described in detail in the following sections.

NOTE: While you are in dimensioning command mode, normal AutoCAD commands are not permitted. However, the mode toggle keys (Section 8.8) and object snap overrides (ADE-2 feature; see Section 8.6) can be used as usual. You can use **CTRL C** to cancel any dimensioning command and return to the "Dim:" prompt.

10.1.3 Linear Dimensioning

Linear dimensioning is done with the HORIZONTAL, VERTICAL, ALIGNED, and ROTATED commands. The only difference among these commands is the angle at which the dimension line is drawn. Nothing is drawn until you have answered all the dimensioning prompts.

The ROTATED command allows you to specify the dimension line angle explicitly. It prompts:

Dimension line angle <0>:

As usual, you can enter the angle in a number of ways, including designating two points. You can give a null reply to use the default angle of zero degrees.

The other linear dimensioning commands are special cases of the ROTATED command; HORIZONTAL is equivalent to ROTATED with an angle of zero (the dimension line is drawn horizontally). Similarly, VERTICAL uses a 90-degree angle to produce a vertical dimension line, and ALIGNED draws the dimension line using the angle between the extension line origins. With the exception of ROTATED, therefore, the linear dimensioning commands do not ask for an angle.

The next prompt issued by each of the linear dimensioning commands is:

First extension line origin or RETURN to select:

Valid responses are:

- | | |
|---------|--|
| a point | This specifies the start of the first extension line, subject to adjustment as specified by the DIMEXO dimensioning variable (see Section 10.1.8). |
| RETURN | This indicates that you want to select a Line, Arc, or Circle entity rather than designate a point. AutoCAD determines the appropriate extension line origins automatically, as described below. |
| Space | Same as RETURN. Allows you to select a Line, Arc, or Circle. AutoCAD then determines the extension line origins automatically, as described below. |

10.1.3.1 Manual Extension Lines

If you designate the origin point of the first extension line (by entering a point), the next prompt is:

Second extension line origin:

Designate the point at which the second extension line is to start (again, subject to the DIMEXO offset).

10.1.3.2 Automatic Extension Lines

If you are drawing a linear dimension for a Line, Arc, or Circle entity, you can instruct AutoCAD to position the extension lines automatically. To do this, respond to the "First extension line origin:" prompt by entering RETURN or a space. AutoCAD then prompts:

Select line, arc, or circle:

You must select the object by pointing to it; "Window" and "Last" selections are not permitted. The selected object must be a Line, Arc, or Circle.

If a Line or Arc is selected, its endpoints are used as the extension line origins. For a Circle, the endpoints of a diameter are used, but the orientation of the chosen diameter depends on the dimensioning command. For HORIZONTAL, VERTICAL, or ROTATED, the diameter at the specified angle is used; for ALIGNED, the point used to select the Circle defines one end of the diameter.

10.1.3.3 Dimension Line and Text

After you have specified the extension line origins (or selected an entity), AutoCAD requests:

Dimension line location:

Designate a point through which the dimension line is to pass. AutoCAD also uses this information to decide in which direction to draw the extension lines. The last prompt in the sequence is:

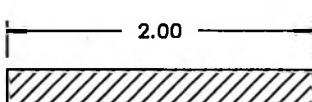
Dimension text <*measured length*>:

A null response (RETURN) causes AutoCAD to measure the dimension line and generate appropriate text. In this case, the text format is dictated by the latest setting of the UNITS command (Section 3.6), and tolerances or limits are drawn if the appropriate dimensioning variables are set (see Section 10.1.8). Alternatively, you can supply the text explicitly. You can also suppress the dimension text entirely, by entering a single blank followed by RETURN.

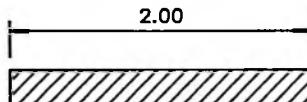
The dimension text is normally centered between the extension lines. However, if the dimension line, arrows, and text do not fit between the extension lines, they are drawn outside. The text is placed near the second extension line; thus, you can choose the text placement by judicious ordering of the extension lines. If object selection is used to produce automatic extension lines, the second extension line is the one furthest from the point used to select the object. For example, the dimensions in the figures below were drawn using object selection. The X's indicate the points used to select the objects (lines); note on which side the dimension text is drawn.



Dimension text is drawn using the current Text style, with the height specified by the variable DIMTXT (see Section 10.1.8), unless the current Text style has a fixed height, in which case that height is used. When the dimension line, arrows, and text all fit between the extension lines, the variable DIMTAD controls the text placement. If DIMTAD is "off", the text divides the dimension line; if DIMTAD is "on", the text is drawn above the dimension line. The two formats are shown below.



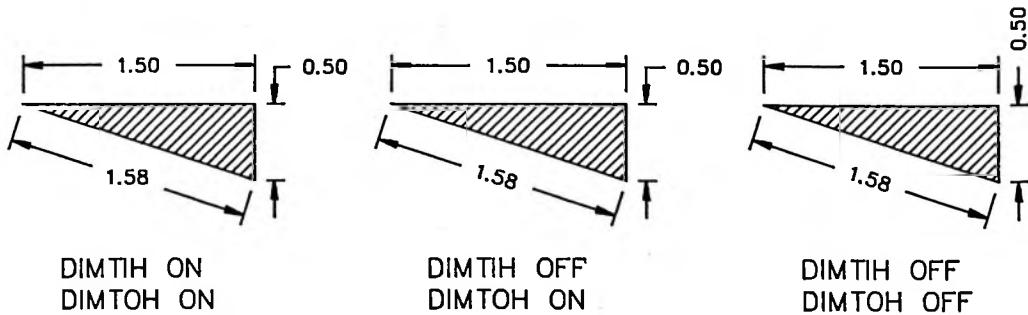
DIMTAD OFF



DIMTAD ON

The drafting standards published by the American National Standards Institute (ANSI) specify "unidirectional" text orientation as preferable, so AutoCAD's default is to draw all dimension text with a orientation of zero degrees (horizontally), even for vertical dimension lines. However, you can use dimensioning variables DIMTIH and DIMTOH to allow dimension text to be drawn at the same orientation as the dimension line. DIMTIH affects only text drawn inside the extension lines, whereas DIMTOH affects text drawn outside the extension lines. Both variables are initially "on", forcing all dimension text to be drawn horizontally. The following figures illustrate the effects of DIMTIH and DIMTOH.





10.1.3.4 Examples

The following figures illustrate the effects of HORIZONTAL, VERTICAL, ALIGNED, and ROTATED dimensioning. In all four figures, the extension line origins are designated explicitly. Points "A" and "B" are the first and second extension line origins, respectively, and point "C" is the location designated for the dimension line. For the ROTATED case illustrated below, the prompt sequence is:

Dim: ROTATED

Dimension line angle <0>: 170

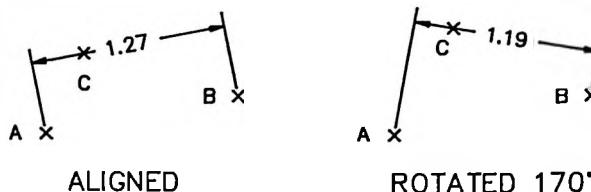
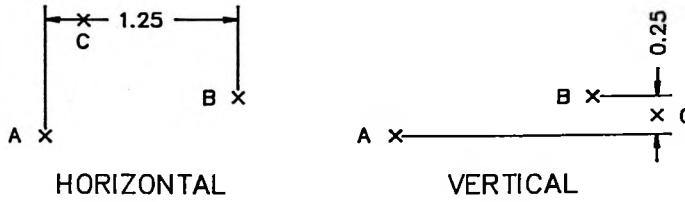
First extension line origin or RETURN to select: (point "A")

Second extension line origin: (point "B")

Dimension line location: (point "C")

Dimension text <1.19>: (RETURN)

With the exception of the "Dimension line angle" prompt, the same prompt sequence applies for the HORIZONTAL, VERTICAL, and ALIGNED cases.



The figures below illustrate use of the ALIGNED command with automatic extension lines. Entity selection is used, and AutoCAD chooses appropriate extension line origins.

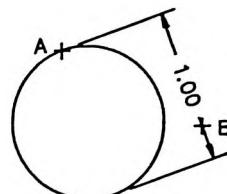
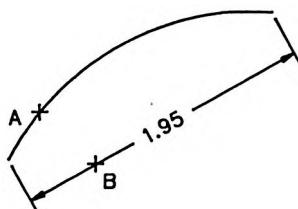
Dim: **ALIGNED**

First extension line origin or RETURN to select: **(RETURN)**

Select line, arc, or circle: **(do so, point "A")**

Dimension line location: **(point "B")**

Dimension text <measured length>: **(RETURN)**



ALIGNED, WITH AUTOMATIC EXTENSION LINES

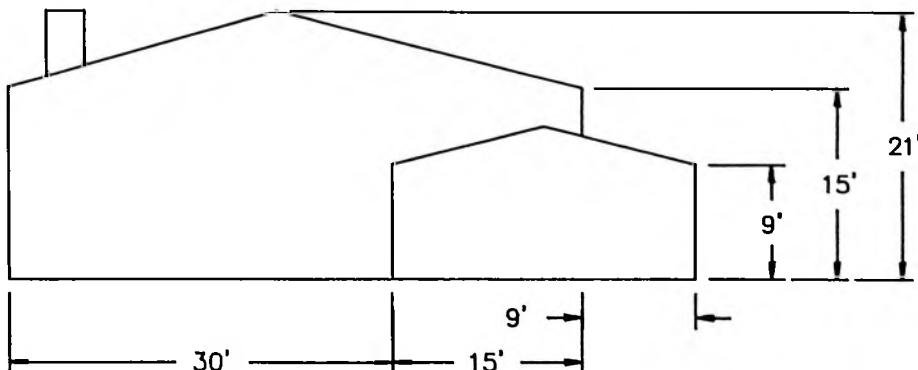
10.1.3.5 Continuing Linear Dimensions

Often, a series of related dimensions must be drawn. Sometimes several dimensions are measured from the same baseline; other times one long dimension is broken into shorter segments that add up to the total measurement. The BASELINE and CONTINUE commands are provided to simplify these operations.

Draw the first dimension using HORIZONTAL, VERTICAL, ALIGNED, or ROTATED. Then enter BASELINE or CONTINUE. AutoCAD proceeds directly to the "Second extension line origin" prompt, and then asks for the dimension text. The dimension line is placed at the same angle as the previous dimension.

If the new dimension line would overlay the previous dimension line, AutoCAD automatically offsets the new dimension line by the amount set in the DIMDLI dimensioning variable (see Section 10.1.8). The first extension line is extended accordingly.

In the following figure, the horizontal dimensions were drawn using CONTINUE, and the vertical dimensions were drawn using BASELINE.



10.1.4 Angular Dimensioning

The ANGULAR command invokes angular dimensioning. Here, the dimension line is an arc spanning the angle between two nonparallel straight lines. The lines do not have to intersect in order to do angular dimensioning on them. Nothing is drawn until you have answered all the dimensioning prompts.

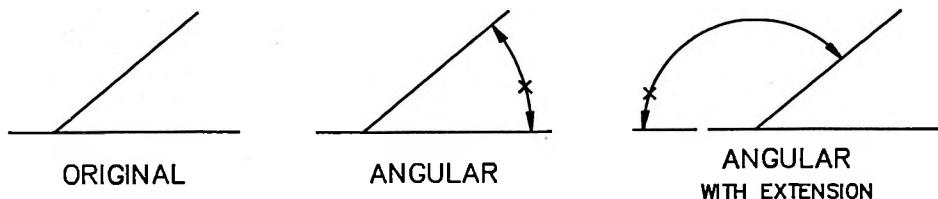
When you enter the ANGULAR command, AutoCAD requests:

Select first line:
Second line:

Point to the two lines you wish to dimension. They must not be parallel. AutoCAD then prompts:

Enter dimension line arc location:

Designate a point between the two lines. AutoCAD draws an arc passing through the point you have specified. The arc is always less than 180 degrees. Extension lines are drawn automatically if the arc does not intersect the line or lines being dimensioned. In the following figures, the X's indicate the points designated as dimension line arc locations.



Next AutoCAD asks for the dimension text:

Dimension text <measured angle>:

You can use the default text generated by AutoCAD (by pressing RETURN), enter explicit text, or suppress the text entirely (by entering a single blank followed by RETURN).

The last prompt in the sequence is:

Enter text location:

If you give a null response (space or RETURN), AutoCAD uses a default text location. It breaks the arc line and centers the text within the arc. If sufficient room is not available for the dimension lines, arrows, and text, AutoCAD displays the message:

Text does not fit. Enter new text location:

You can designate any point in the drawing for text placement. No further checking is done to see if the text will fit. Also, the dimension arc is then drawn without a break; AutoCAD assumes that you have placed the text away from the dimension arc.

Examples

In the figures below, points "A" and "B" select two lines for angular dimensioning, and point "C" designates the position of the dimension arc (and thus the portion of the figure to be dimensioned). The prompt sequence is:

Dim: **ANGULAR**

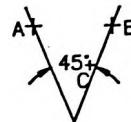
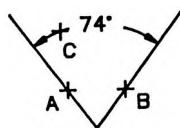
Select first line: (do so, point "A")

Second line: (select, point "B")

Enter dimension line arc location: (point "C")

Dimension text <measured angle>: (**RETURN**)

Enter text location: (RETURN to use default)



In the above examples, the dimension text fits in the default location. If the text does not fit, as in the following figure, an additional prompt appears for the text location.

Dim: **ANGULAR**

Select first line: (do so, point "A")

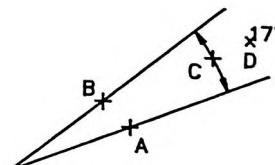
Second line: (select, point "B")

Enter dimension line arc location: (point "C")

Dimension text <measured angle>: (**RETURN**)

Enter text location: (RETURN to use default)

Text does not fit. Enter new text location: (point "D")

**10.1.5 Diameter Dimensioning**

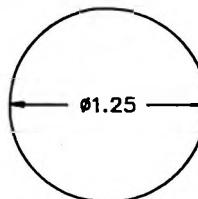
You can invoke diameter dimensioning for Arcs and Circles by means of the DIAMETER command. The first prompt is:

Select arc or circle:

Point to a Circle or Arc. The results are very similar to ALIGNED linear dimensioning, except that the dimension line runs through the center of the circle and no extension lines are drawn. The point used to select the Circle or Arc defines one end of the diameter. The next prompt is:

Dimension text <measured diameter>:

You can use the default text generated by AutoCAD (by pressing RETURN), enter explicit text, or suppress the text entirely (by entering a single blank followed by RETURN). The default text begins with a "diameter" symbol. If the diameter line, arrows, and text all fit inside the Circle or Arc, the dimension is drawn. For example:



If there is sufficient room for the diameter line and arrows, but not for the text, the text is moved outside. Also, if the variable DIMCEN is nonzero (see Section 10.1.8), AutoCAD draws a center mark or center lines. To determine the location for the text, AutoCAD prompts:

Text does not fit. Enter leader length for text:

Specify the length of the leader line to be drawn from the circle diameter point to the text. A rubber-band line is added to the screen crosshairs to help you specify the length by pointing. A null response tells AutoCAD to use the default leader length (two arrow lengths). Also, if you indicate a leader length shorter than two arrows, the default length is used instead. Note that you can specify the length of the leader but not its direction; the leader is always an extension of the diameter line, extending through the point you used to select the Circle or Arc. An example is shown in the figure on the left below.



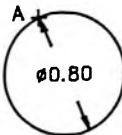
If the angle of the diameter line is greater than 15 degrees, an additional short horizontal leader (one arrow long) is drawn next to the dimension text, as illustrated in the rightmost figure above.

If the diameter of the Circle or Arc is less than four arrow lengths, the diameter line and arrows cannot be drawn inside it. In this case, no diameter line is drawn, but an arrow is added to the end of the text leader, as shown below. If variable DIMCEN is nonzero (see Section 10.1.8), AutoCAD draws a center mark or center lines.



Example

Dim: **DIAMETER**
 Select arc or circle: **(point "A")**
 Dimension text <measured diameter>: **(RETURN)**

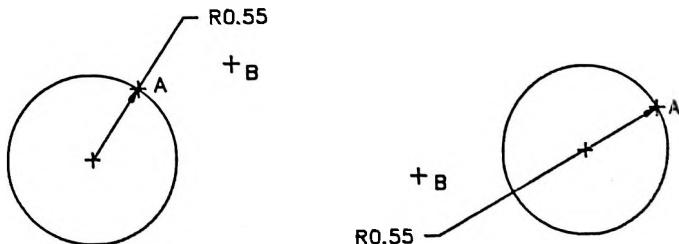
**10.1.6 Radius Dimensioning**

Radius dimensioning is performed using the RADIUS command. It is almost identical to diameter dimensioning, except that only a radius line is drawn (half of a diameter line). The radius line has only one arrow, and if the text fits inside the Circle or Arc, it is centered along the radius line. Rules similar to those for diameter dimensioning are used to determine whether the radius line and/or the text should be drawn outside the Circle or Arc. AutoCAD draws a center mark or center lines if variable DIMCEN is nonzero (see Section 10.1.8).

The default text generated by AutoCAD begins with the letter "R" to denote a radius dimension. To use the default text, give a null response (RETURN) to the "Dimension text:" prompt. You can also enter explicit text, or enter a single blank followed by RETURN to suppress the dimension text entirely.

If the dimension text does not fit inside the Circle/Arc, a leader is drawn as for the DIAMETER command. For RADIUS, however, AutoCAD allows the leader to be drawn either toward the Circle/Arc or through its center point. To draw the leader through the center point, designate its length by pointing to a location on the other side of the Circle/Arc from the point used to select the Circle/Arc. The figures below illustrate this.

Dim: **RADIUS**
 Select arc or circle: **(point "A")**
 Dimension text <measured radius>: **(RETURN)**
 Text does not fit. Enter leader length for text: **(point "B")**



10.1.7 Dimensioning Utility Commands

10.1.7.1 CENTER

The CENTER command draws a center mark or center lines for a Circle or Arc, as governed by the DIMCEN dimensioning variable (see Section 10.1.8). The prompt sequence is:

Dim: CENTER
Select arc or circle:

10.1.7.2 EXIT

The EXIT command ends dimensioning command mode and returns to normal Drawing Editor command mode. The "Dim:" prompt is replaced by the standard "Command:" prompt. (You can also return to normal Drawing Editor command mode by entering CTRL C in response to the "Dim:" command prompt.)

10.1.7.3 LEADER

Leaders must often be drawn from the dimension text to the object being dimensioned. Although the DIAMETER and RADIUS commands provide an automatic leader capability for simple cases, more complex cases occasionally arise. For instance, sometimes a leader must be routed around other objects in the drawing.

The LEADER command is provided to let you construct complex leaders. You can use it at any time; when dimensioning a diameter or radius, for example, if you see that a complex leader is necessary, suppress the DIAMETER/RADIUS text by entering a single blank followed by RETURN when prompted for the dimension text. Then issue the LEADER command:

Dim: LEADER

LEADER prompts for a "Leader start" point, followed by any number of "To" points, similar to AutoCAD's normal LINE command. You can respond "U" when prompted for a "To" point to undo the last leader segment drawn.

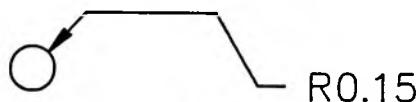
Start at the object being dimensioned; when you reach the point where you want the dimension text to be drawn, give a null response (space or RETURN) to the "To point" prompt. AutoCAD then prompts:

Dimension text <measurement>:

where the default is the measurement from the most recent dimension. Here again, you can use the default (press RETURN), supply explicit text, or suppress the text altogether (enter a space followed by RETURN).

If the length of the first leader segment is greater than 2 arrows, AutoCAD draws an arrow at the end of that segment. Otherwise, only a line is drawn.

The following figure illustrates a leader drawn by the LEADER command.



10.1.7.4 REDRAW

The REDRAW command redraws the entire display, erasing any point selection "blips" that may have been present. This dimensioning command is the same as the normal AutoCAD REDRAW command.

10.1.7.5 STATUS

In dimensioning command mode, the STATUS command lists all the dimensioning variables and their current values, as shown below. Section 10.1.8 contains detailed information on these variables.

DIMSCALE	1.0000	Overall scale factor
DIMASZ	0.1800	Arrow size
DIMCEN	0.0900	Center mark size
DIMEXO	0.0625	Extension line origin offset
DIMDLI	0.3800	Dimension line increment for continuation
DIMEXE	0.1800	Extension above dimension line
DIMTP	0.0000	Plus tolerance
DIMTM	0.0000	Minus tolerance
DIMTXT	0.1800	Text height
DIMTSZ	0.0000	Tick size
DIMTOL	Off	Generate dimension tolerances
DIMLIM	Off	Generate dimension limits
DIMTIH	On	Text inside extensions is horizontal
DIMTOH	On	Text outside extensions is horizontal
DIMSE1	Off	Suppress the first extension line
DIMSE2	Off	Suppress the second extension line
DIMTAD	Off	Place text above the dimension line

10.1.7.6 UNDO

The UNDO command erases the annotations produced by the most recent dimensioning command. If the dimension just drawn doesn't appear as you'd like it, UNDO it. Then adjust the values of dimensioning variables, if necessary, and try again.

NOTE: UNDO's memory goes back only one dimensioning command and is lost whenever you return to normal Drawing Editor command mode.

10.1.8 Dimensioning Variables

The manner in which dimensions are drawn is controlled by a set of dimensioning variables. Some of these variables are simply "on/off" switches. Their effects are listed below.

- DIMSE1 (Suppress Extension line 1) - If "on", suppress the first extension line. Default value: Off.
- DIMSE2 (Suppress Extension line 2) - If "on", suppress the second extension line. Default value: Off.
- DIMTIH (Text Inside Horizontal) - Controls the orientation of dimension text for linear dimensioning where the text fits between the extension lines. If "on", the text is always drawn horizontally. If "off", the text rotation angle equals the angle of the dimension line. Default value: On (text is drawn horizontally).
- DIMTOH (Text Outside Horizontal) - Same as DIMTIH, except that it controls text drawn outside the extension lines. Default value: On (text is drawn horizontally).
- DIMTAD (Text Above Dimension line) - For linear dimensioning where the dimension text is drawn between the extension lines and at the same angle as the dimension line, this variable controls placement of the text. If "off", the text is drawn centered along the dimension line, dividing the line in two. If DIMTAD is "on", the text is placed above the dimension line. Default value: Off.
- DIMTOL (Tolerance) - If "on", append dimension tolerances to the default text. Default value: Off.
- DIMLIM (Limits) - If "on", generate dimension limits as the default text. Default value: Off.

DIMTOL and DIMLIM can both be "off", but cannot both be "on". If you turn one of them on, the other is turned off automatically.

The following dimensioning variables govern sizes or distances. Their values may be set by entering a distance (in any format accepted by the current UNITS mode) or by designating two points.

- DIMASZ (Arrow Size) - This specifies the size of the arrows drawn at the ends of dimension lines. Multiples of the arrow size are used to determine whether dimension lines and text will fit between the extension lines. Default value: 0.18 units.
- DIMTSZ (Tick Size) - This variable specifies the size of the "ticks" drawn instead of arrows for linear dimensioning. If zero, arrows are drawn. Default value: 0 units.
- DIMTXT (Text size) - This specifies the height of the dimension text, unless the current Text style has a fixed height. Default value: 0.18 units.
- DIMCEN (Center mark size) - This variable controls the drawing of Circle/Arc center marks and center lines by the CENTER, DIAMETER, and

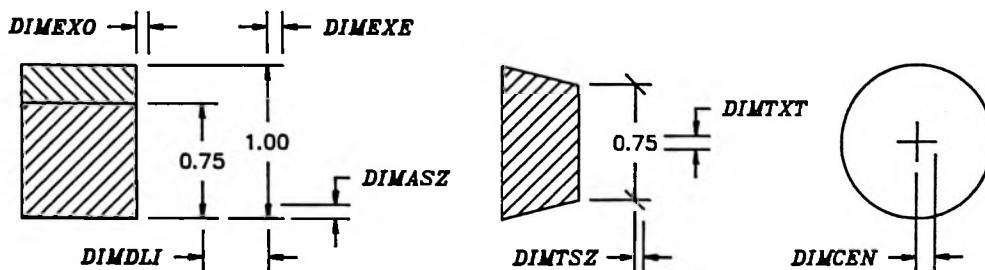
RADIUS commands. If zero, center marks/lines are not drawn. If DIMCEN is greater than zero, the value specifies the size of the center mark. If DIMCEN is negative, center lines are drawn rather than center marks; the absolute value specifies the size of the "mark" portion of the center line. (For DIAMETER, the center mark/line is drawn only if the dimension text is placed outside the Circle or Arc.) Default value: 0.09 units.

- DIMEXO** (Extension line Offset) - The extension lines are offset this amount from the origin points you specify. Thus, you can point directly at the corners of an object to be dimensioned, but the extension lines will stop just short of the object. Default value: 0.0625 units.
- DIMEXE** (Extension line Extension) - This specifies how far the extension line should extend beyond the dimension line. Default value: 0.18 units.
- DIMDLI** (Dimension Line Increment) - This variable controls the dimension line increment for continuation of linear dimensioning with the BASELINE and CONTINUE commands. Successive continuations are offset by this amount, if necessary, to avoid drawing over the previous dimension. Default value: 0.38 units.
- DIMTP** Plus tolerance. Default value: 0 units.
- DIMTM** Minus tolerance. Default value: 0 units.

The following dimensioning variable is a scale factor, and as such may be set only to a decimal number. Distance or two-point specifications are not permitted.

- DIMSCALE** This is the overall scale factor applied to all dimensioning variables that specify sizes, distances, or offsets. It is not applied to tolerances. Default value: 1.0.

The figure below illustrates the parameters governed by most of the dimensioning variables.



The most recent value of each dimensioning variable is remembered and saved along with the drawing; the defaults listed above are the initial values for a new drawing. You can obtain a list of all dimensioning variables showing their current values by means of the STATUS command, described earlier in this section.

Changing Variable Values

At almost any time while you are in dimensioning command mode, you can specify a new value for any of the dimensioning variables; to do this, simply enter the name of the variable. AutoCAD then prompts:

Current value <nnnn> New value:

You can enter a new value, or give a null response to retain the current value. The current dimensioning operation, if any, then continues. The one time you cannot enter a variable name in this manner is when AutoCAD is asking for the "Dimension text".

The dimensioning variables control the generation of subsequent dimensions; those that have already been drawn do not change when a variable is changed.

In some cases, a change in a dimension variable does not take effect until the next command. For example, once AutoCAD has requested the dimension text, a change in the arrow size or text size does not take effect until the next command. Therefore, it is best to change variable values only at the "Dim:" prompt.

10.2 Crosshatching and Pattern Filling (+1)

In many drafting applications, it is common practice to fill an area with a pattern of some sort. The pattern can help differentiate between components of a three-dimensional object, or it can signify the material composing an object. This process, called "crosshatching" or "pattern filling", can be accomplished using AutoCAD's HATCH command; it is referred to simply as "hatching" in this discussion. Hatching is a feature of the ADE-1 package.

AutoCAD is supplied with a library of standard hatch patterns in the file ACAD.PAT. You can hatch with one of these standard patterns, with a custom pattern from your own file, or with a simple pattern defined "on the fly" in the HATCH command.

Appendix A of this manual contains samples of all the standard hatch patterns supplied in ACAD.PAT. For information on defining custom hatch patterns, see Appendix B.

What are hatch patterns?

Each hatch pattern is composed of one or more *hatch lines*, at specified angles and spacing. The hatch lines may be continuous or broken into dots and dashes. The pattern is repeated or clipped, as necessary, to exactly fill the area being hatched.

Hatching generates Line entities for the chosen pattern and adds them to the drawing. Since a pattern may contain many lines, AutoCAD normally groups these lines into an internally generated Block. Thus, if you have hatched an area and decide you don't like the hatching, you can just point to any line of it with the ERASE command and all the hatching will go away. You can use "ERASE Last" when appropriate. If you wish to edit the output of the hatching process manually, or if you're debugging a pattern and want to be able to examine the individual generated lines, you can suppress this Block grouping.

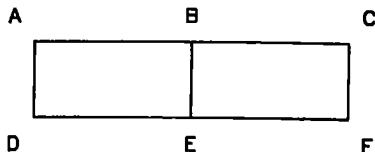
The Block generated for hatch lines does not appear in the "INSERT ?" report, and it is automatically deleted if all references to it are deleted.

NOTE: Although hatch lines may be broken into dot-dash segments, these dot-dash lines are not "linetypes" in the usual AutoCAD sense; each dot-dash segment of a hatch line is a separate entity. Hatching should always be performed on a layer having the "CONTINUOUS" linetype.

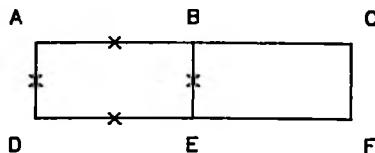
10.2.1 Defining the Boundary

Hatching works on regions of the drawing enclosed by a boundary made up of Line, Arc, Circle, Trace, and Polyline entities. When hatching an area, you must select the entities that define the boundary via the normal "Select objects" mechanism; of course, Window or Last selection may be used.

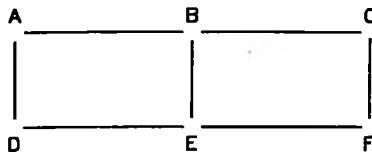
The entities forming the hatching boundary should intersect at their endpoints; overhangs can produce incorrect hatching. If the drawing requires an overhanging boundary line, this line must be drawn using two segments. Then just the appropriate segment can be included in the selection of the boundary entities, and hatching can be performed normally. For instance, suppose you want to hatch the leftmost portion (box "ABED") of the following figure.



However, if line "AC" and line "DF" run the full length of the figure, selecting them as part of the hatch boundary (as shown below) is improper and may result in incorrect hatching.



To define a proper boundary, the figure must be drawn using separate line segments for "AB", "BC", "DE", and "EF" as shown in the following exploded view.



Now the boundary entities meet properly (at their endpoints). You can select segments "AB", "BC", "ED", and "DA" individually or by means of a window, and box "ABED" will be hatched as you intended.

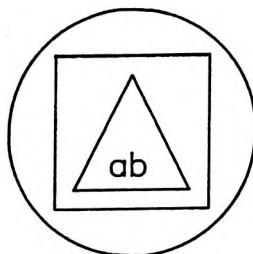
If wide Polyline segments form part of the hatch boundary, AutoCAD uses the segments' center-lines as the boundary, ignoring the widths of the segments.

Hatching of the enclosed area may be affected by the presence of other entities inside the boundary. However, hatching only knows about entities it has been told about via the object selection mechanism -- a Window selection guarantees that all internal structure is correctly seen.

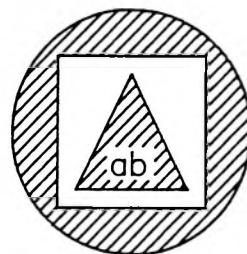
10.2.2 Hatching Styles

If the space inside the boundary is empty (or if none of the internal objects have been selected), that area is simply filled with the chosen pattern. If anything inside the boundary has been included in the entity selection, the result depends on the "style" of hatching. Three styles are available.

Let us consider a circle with a square inside it that, in turn, has a triangle inside it, as illustrated at the left below. There is also some text in the figure; the effect of Text entities on hatching is discussed later. For these examples, assume that a Window selection has been performed, so all the entities are included in the selection.



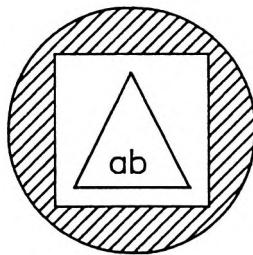
AREA TO BE HATCHED



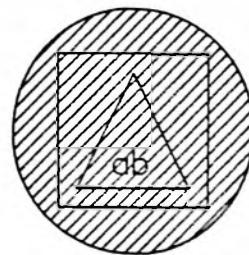
NORMAL STYLE

The **Normal** (default) style of hatching is illustrated in the figure on the right. This style hatches inward starting at the area boundary, at each end of each hatch line. If it encounters an internal intersection, it turns off hatching until another intersection is encountered. Thus, areas separated from the outer boundary by an odd number of intersections are hatched, while areas separated by an even number of intersections are not.

The second style of hatching is **Outermost**. This style also hatches inward from the area boundary, but it turns hatching off if an internal intersection is encountered and does not turn it back on. Since this process starts from both ends of each hatch line, the effect is that only the outermost level of the structure is hatched, and all internal structure is left blank. The leftmost portion of the following drawing illustrates the effects of Outermost hatching on our example figure.



OUTERMOST STYLE



IGNORE STYLE

The third style of hatching is **Ignore**. Here, all inner structure is hatched right through. In our example, each hatch line would start and end on the circle (the outer boundary), and pass right through the inscribed triangle and square as if they weren't there, as shown at the right, above. Normally the effect of Ignore style hatching can be achieved simply by not selecting the internal entities, but the style option is provided in case you wish to do a Window selection for convenience, or in case Blocks are present, preventing you from selecting the boundary without the internal structure.

Hatching Text, Attributes (+2), Shapes, Traces, and Solids

Text, Attribute (+2), Shape, Trace, and Solid entities behave specially with regard to hatching. If a hatch line would pass through such an entity, it is automatically turned off, so that these entities won't be hatched through. In fact, Text, Attribute, and Shape entities are surrounded by a magic invisible box that encloses the entity with a margin for readability, which disables all hatch lines that would otherwise pass through it. This means, for example, that you can draw a pie slice, label it with text, and then hatch it in confidence that the text will remain readable. This does not apply, of course, if Ignore style hatching is selected -- in that case the text is hatched right through.

NOTE: Although a Trace may be part of the hatch boundary, and Traces and Solids get special treatment when they are encountered inside the hatch boundary, it is not possible to draw a Trace or Solid with "Fill" mode off and then hatch that outlined object.

Composite Entities and Hatching

Hatching works correctly on inserted Blocks. The Hatch command recognizes the internal structure of a Block and hatches the figure just as though it were composed of separate entities. This is true regardless of the X scale, Y scale, and rotation with which the Block(s) were inserted. If scaling turns a Circle or Arc into an ellipse or elliptical arc, the hatch lines stop at the ellipse correctly. Note that selecting any part of a Block for hatching still selects the entire Block.

REPEATs are another matter. HATCH cannot detect that a selected entity is a member of a REPEAT, and it hatches only the first member of the group (unless the REPEAT is inside a Block). Thus, it is unwise to use REPEATs if you intend to use hatching -- use the ARRAY command instead.

10.2.3 HATCH Command

Hatching is performed using the HATCH command:

Command: **HATCH**
 Pattern (? or name/U,style) <default>:

If you want to use one of the standard patterns in the ACAD.PAT pattern library, just enter the pattern name. If the pattern you requested is not in ACAD.PAT, AutoCAD looks for it in a file whose name is the same as the pattern name, with ".PAT" appended if there is no period in the name supplied. Thus, if you answered this prompt with "PIT", and there were no such pattern in ACAD.PAT, AutoCAD would look for the pattern in a file named PIT.PAT. If you responded "B:GOOSE.OLD", AutoCAD would look for the "GOOSE.OLD" pattern in the file GOOSE.OLD on drive B.

If you respond "?" to the "Pattern" prompt, a list of all the standard patterns in ACAD.PAT is printed. This is handy when you forget the name of a pattern.

If you answer "U" to this prompt, you are allowed to define a simple pattern on the fly. AutoCAD prompts:

Angle for crosshatch lines <default>
 Spacing between lines <default>
 Double hatch area? <default>

which should be answered with the angle, the interline spacing, and a "Y" or "N" indicating whether you want a second set of lines at 90 degrees to the original lines to be drawn also. You can answer the first two prompts with numbers or by "showing" AutoCAD via two points. Each prompt uses the value from the previous HATCH command as a default; to select the default, give a null response.

Whether you use a predefined pattern or define one with the "U" option, you may specify the hatching style by appending it to the pattern name with a comma. Style codes are:

- N - Normal (same as no style specified)
- O - Fill outermost areas only
- I - Ignore internal structure

Thus, to hatch just the outermost area of a boundary with the "MUD" pattern, you would answer the "Pattern" prompt with "MUD,O". Or, to define a pattern on the fly, and use the Ignore style, enter "U,I". If you forget what the styles are, use a question mark for the style and AutoCAD will remind you (and reject the command).

AutoCAD normally groups all hatch lines generated by one HATCH command in a single Block. If you do not want this to happen, precede the reply to the "Pattern" prompt with an asterisk, "*". Thus, if you want to hatch an area with the "BRASS" pattern, and have the hatch lines entered individually, reply "*BRASS" to the prompt.

If you have selected a pattern from a file (e.g., not "U"), you will receive two additional prompts:

Scale for pattern <default>:

Angle for pattern <default>:

Each pattern is defined with an initial size, and with a rotation of zero degrees. You may expand or contract the pattern, or rotate it with respect to the axes by supplying the desired values to these prompts. Because most standard patterns have a characteristic size of 1 drawing unit, you may respond to the "Scale" prompt with the desired size of the pattern; AutoCAD will then automatically scale the pattern to that size. You may respond to the above prompts with two points to "show" AutoCAD the size or angle. The scale and angle from the previous HATCH command are provided as defaults; to use a default, give a null response to the prompt.

After you choose the pattern and style, AutoCAD prompts:

Select objects or Window or Last:

You should select, by any of the standard methods, the objects that define the boundary of the area to be hatched, and those objects internal to the boundary that you wish hatching to be aware of. Note that Window selection is particularly convenient in many hatching applications.

Once you have selected the boundary, hatching commences. Hatching may take a while, particularly when the pattern, boundary, and/or internal structure are complex; you can watch the hatch lines appear as they are generated, or take a coffee break. You can terminate the hatching process at any time with CTRL C. The pattern generated up to that point is added to the drawing. If the hatch lines were being grouped into a Block (the default case), you can use "ERASE Last" to delete them.

10.2.4 Hatch Pattern Alignment

It is often important that adjacent areas be hatched with the same pattern or with similar patterns, and that they "line up" properly. AutoCAD ensures proper alignment automatically, by generating all lines of every hatch pattern from the same reference point. This point is normally the (0,0) origin point of the drawing.

If the ADE-2 package is present, AutoCAD uses the Snap base point as the reference point for all hatching. This point is normally (0,0), but you can change it by means of the SNAP command's "Rotate" option (see Section 8.1). Thus, you can vary the alignment of your hatch patterns to better fit the areas being hatched or to meet other special requirements.

Hatching also uses the Snap rotation angle set by "SNAP Rotate" as the default rotation angle for all hatch patterns. Of course, you can override the default with any rotation angle you like during the HATCH command.

10.2.5 HATCH Command Repetition

If you repeat a HATCH command immediately (by pressing space or RETURN in response to the next "Command:" prompt), AutoCAD assumes that you want to hatch another area with the same pattern, style, scale, and angle. It skips the associated prompts and proceeds directly to the "Select objects" prompt.

10.3 Command Scripts

AutoCAD provides a *script* facility that allows commands to be read from a text file. This feature allows you to execute a predetermined sequence of commands; you can invoke these commands when you begin running AutoCAD, using a special form of the "ACAD" command, or you can run a script from the Drawing Editor by using the SCRIPT command. The script facility provides an easy way to create a continuously running display for product demonstrations and trade shows.

10.3.1 Invoking Scripts When Loading AutoCAD

To invoke a script when AutoCAD is first loaded, use the command form:

A>ACAD default-drawing script-file

The script file must be the second file named on the ACAD program call line, and it is assumed to have a file type of ".SCR". This method of script invocation reads commands from the script file starting with the Main Menu prompts, so the first line of the file should select the desired task number.

For example, suppose that every time you began creating a new drawing, you immediately turned the grid on, set the global linetype scale to 3.0, and set layer "0" as your current layer, with color "RED". All this could be accomplished using a prototype drawing, but for the purposes of this discussion, let's do it instead with the following script, stored in a file called, say, SETUP.SCR.

1	<i>(create new drawing)</i>
	<i>(blank: use default name)</i>
GRID ON	<i>(turn on grid)</i>
LTSCALE 3.0	<i>(set scale for linetypes)</i>
LAYER SET 0 COLOR RED 0	<i>(select current layer and its color)</i>
	<i>(blank to end LAYER command)</i>

Now, if you wanted to create a new drawing called B:WIDGET, you might invoke AutoCAD as follows:

A>ACAD B:WIDGET SETUP

This sets the default drawing name to B:WIDGET and begins reading commands from SETUP.SCR. This script creates a new drawing with the default name (B:WIDGET) and proceeds to issue your usual sequence of setup commands. When the end of the script file is encountered, the "Command:" prompt appears and AutoCAD awaits a command from you.

Of course, you must be very familiar with the sequence of prompts issued by AutoCAD in order to provide an appropriate sequence of responses in the script file. Note that AutoCAD's prompts (and even its command names) may change in future releases; some revision of your scripts may be necessary when you upgrade to a later version of the program. Similarly, avoid the use of abbreviations; future command additions may create ambiguities. Also note that each blank space in the script file is significant, since AutoCAD accepts either a space or a RETURN as a command or data field terminator.

10.3.2 SCRIPT Command

The **SCRIPT** command permits you to invoke a script file while you are in the Drawing Editor.

Command: SCRIPT Script file: *(name)*

The file type ".SCR" is assumed; you should not include it in your response. The sequence of commands stored in the script file is executed, and the "Command:" prompt reappears. If a **SCRIPT** command is read from a script file, the current script file is terminated, and the file named on the command becomes the current script file.

Several additional commands are provided to make scripts more flexible. These are described below.

10.3.3 DELAY Command

A script can be used as a sort of "electronic flipchart". However, some AutoCAD operations happen rather quickly, making it difficult for people to see what's happening on the screen. For instance, if your script draws a line and then erases it, your audience might not see the action. The **DELAY** command is provided to cause a sufficient pause between such operations.

Command: DELAY Delay time in milliseconds: *(number)*

Number specifies the duration of the pause. The larger the number, the longer the delay. Due to the wide range of processing speeds for the computer systems used with AutoCAD, it is impossible for us to be precise about the delay time in this manual, but the delay is designed to be about one millisecond per increment. Thus, if you enter "DELAY 1000", the next "Command:" prompt should be delayed for about one second. As noted, the timing on your machine may be different, but it should not be off by more than a factor of two. The upper limit for the delay number is 32,767, a bit less than 33 seconds.

10.3.4 RESUME Command

Pressing **CTRL C** or the **Backspace** key on the keyboard interrupts a running script at the end of the current command and lets you issue commands using the normal methods.

If you later wish to return to the script (picking up where you interrupted it), simply enter:

Command: RESUME

Any error encountered while processing a command from a script file causes the script to be terminated. If this occurs while the Drawing Editor is active, you can use the **RESUME** command to continue the script.

10.3.5 GRAPHSCR and TEXTSCR Commands

Several AutoCAD commands (such as **HELP** and **STATUS**) automatically flip to the text display on single-screen systems. Although you can use the **FLIP SCREEN** key to manually flip back to the graphics display, there is no way to include this key in a command script, nor in a menu item. Therefore, two special commands are provided to permit scripts and menu items to flip between the text and graphics displays. These commands are:

Command: GRAPHSCR

to flip to the graphics screen, and

Command: TEXTSCR

to flip to the text screen. These commands are ignored on dual-screen systems.

10.3.6 Continuous Scripts

In some situations (trade shows, dealer showrooms, etc.) it is useful to have a script that "plays" over and over, showing various aspects of a product. This can be accomplished using scripts, but the method used depends on how the current script file was invoked.

If the current script file was named on the ACAD call line, and the last command in the script file is "END" or "QUIT Y", the script file is restarted from the beginning when the Main Menu reappears. In this case, the script file must begin with appropriate replies to the Main Menu prompts.

On the other hand, if the SCRIPT command was used to invoke the current script file, you can explicitly request the script to be rewound and restarted by including an RSCRIPT command (described below) in the script file itself.

10.3.6.1 RSCRIPT Command

You can include the RSCRIPT command in a script file to force the script to be restarted from the beginning.

Command: RSCRIPT

This command is handled by AutoCAD's Drawing Editor; it does not cause the Drawing Editor to exit. Thus, the script file being restarted should begin with Drawing Editor commands, not Main Menu responses. Also note that the RSCRIPT command is not understood by the Main Menu routine. Thus, it cannot follow an "END" or "QUIT Y" command in the script.

10.3.6.2 Examples

The following is a simple example of a continuous script, intended to be invoked by means of the ACAD program call line.

1	(new drawing)
DEMO	(drawing name)
LINE 1,1 2,1 2,2 1,2 c	(draw a box)
DELAY 500	(pause 1/2 second)
MOVE W 0,0 3,3 7,0	(move box to right 7 units)
	("7.0" was a displacement)
DELAY 1000	(pause 1 second)
QUIT Y	(cycle)

If we stored this script in a file named BOX.SCR, we could invoke it as follows:

A>ACAD_X BOX

Note that the script supplies the drawing name ("DEMO"), so the default drawing name "X" specified on the ACAD call line is ignored. The script creates drawing "demo", draws a box one unit square, and then moves the box 7 units to the right before quitting and starting over.

When constructing script files, be sure to incorporate spaces and RETURNS in the appropriate places. In this example, for instance, note the two spaces between "3,3" and "7,0" for the MOVE command. The first space ends the "3,3" data field, and the second space indicates that object selection is complete. The characters in the script file are processed just as if they had been entered from the keyboard, and every character counts.

The following is an example of a continuous script designed to be invoked with the **SCRIPT** command:

REGEN	<i>(regen the drawing)</i>
TEXT .3,2 .5 0	<i>(set to add text)</i>
Come in for a demonstration!	
DELAY 1000	<i>(pause 1 second)</i>
ERASE L	<i>(erase the text)</i>
RSCRIPT	<i>(blank to end object selection)</i> <i>(rewind and restart script)</i>

If we stored this script in a file called "SHOWOFF.SCR", we could invoke it via:

Command: SCRIPT Script file: SHOWOFF

This script regenerates the drawing and adds some text to it at a predetermined position. After a short pause, the text is erased, and the script restarts with the "regen" command.

10.4 Slide Shows

Sometimes, it is necessary to show a client various views of a complex drawing or views of several different drawings. This can be done using ordinary AutoCAD facilities (loading the various drawings and ZOOMing as necessary), but the process can be time-consuming. If you can plan your presentation in advance, AutoCAD's *slide* capability may be preferable. In fact, you can create complete "slide shows" by combining the slide feature with the script facility described earlier.

NOTE: The standard version of AutoCAD can be used to view slides, but the ADE-2 package is required in order to create new slides.

What is a slide?

In AutoCAD, a slide is a file containing a "snapshot" of the display on the graphics monitor. You can use ordinary AutoCAD commands to obtain the desired view on the monitor and then make a slide of that display. You can rapidly recall the slide for subsequent viewing.

10.4.1 MSLIDE Command - Making a Slide (+2)

The ADE-2 package's MSLIDE command is used to make a slide of the current display on the graphics monitor. First, create the display you wish to save, then enter:

Command: MSLIDE Slide file: *(name)*

A file type of ".SLD" is appended to the name you supply; do not include a file type in your response. A "redraw" operation takes place as the slide is being made, so you can watch its progress. Portions of the drawing that are off-screen or on layers that are "off" or "frozen" are not included in the slide.

10.4.2 VSLIDE Command - Viewing a Slide

To view a slide, issue the VSLIDE command:

Command: VSLIDE Slide file: *(name)*

Do not include a file type in your response; a file type of ".SLD" is assumed by AutoCAD. If the specified slide file is found, it is read, and its contents replace the display on the graphics monitor. To restore the current drawing to the monitor, issue a REDRAW command.

If you are presenting a series of slides to an audience, one additional capability may be helpful. Ordinarily, the speed with which a slide can be displayed is limited by the disk accesses necessary to read the slide file. It is possible, however, to "preload" the next slide from disk into memory while the audience is assimilating the current slide, and then quickly display the new slide from memory.

To preload a slide, place an asterisk before the file name in the VSLIDE command. The next VSLIDE command senses that a slide has been preloaded and displays it without asking for a file name. For example, consider the following script:



VSLIDE SLIDE1	<i>(begin slide show)</i>
VSLIDE *SLIDE2	<i>(preload slide 2)</i>
DELAY 2000	<i>(let folks read slide 1)</i>
VSLIDE	<i>(display slide 2)</i>
VSLIDE *SLIDE3	<i>(preload slide 3)</i>
DELAY 2000	<i>(let folks read slide 2)</i>
VSLIDE	<i>(display slide 3)</i>
DELAY 3000	<i>(let folks read slide 3)</i>
RSCRIPT	<i>(cycle)</i>

The disk access time necessary to load the next slide has been overlapped with the viewing time for the current slide. Additional delays have been used, as well.

10.4.3 Slide Notes

This slide capability is very different from other aspects of AutoCAD, and care must be taken when using it. In particular:

- o Slides cannot be edited. If normal editing commands (to draw a line, for example) are executed while a slide is being viewed, the editing does not affect the slide, but rather the current drawing (which is not visible). Therefore, we recommend that only the following commands be used while a slide is being viewed.
 - VSLIDE, to view another slide.
 - DELAY, to let the audience read the slide.
 - REDRAW, to return the current drawing to the monitor. (Note that any command that forces a redraw operation, such as GRID OFF, also returns the current drawing to the monitor.)
 - Other commands, such as MENU and SCRIPT, that affect neither the slide nor the current drawing. Commands that flip to text mode on single-screen systems, like FILES and HELP, can also be used, except that on some displays flipping to text mode and back to graphics mode forces a redraw operation.

If a slide must be revised, edit the drawing from which the slide was made, and make a new slide.

- o The layer/color relationships, zoom magnification, and other Drawing Editor conditions in effect when a slide is viewed have no effect whatsoever on the appearance of the slide. A slide is simply a "snapshot" of the screen at the time the slide was made, colors and all.
- o If you use a low-resolution graphics monitor when creating a slide file, and later upgrade to a monitor with higher resolution, you can still view the slide; AutoCAD adjusts the image accordingly. However, the slide will not take full advantage of your new monitor until you remake the slide file from the original drawing.

10.5 Variables and Expressions (+3)

If the optional ADE-3 package is present, you can define integer, real, point, and string *variables*, assign values to them as you please, and use those values whenever AutoCAD requests data from you. You can also perform arithmetic and conditional operations on the variables, and use expressions involving them when responding to AutoCAD's prompts.

These fairly advanced features are most useful when used in conjunction with custom menus (Appendix B). If you are a new AutoCAD user, skip this section for now and come back to it after you have mastered the basics.

You will find that the constructs provided in the ADE-3 package closely resemble those offered by the LISP programming language. However, the full LISP language is not provided at this time.

10.5.1 Variables

AutoCAD variables may be of four types; integer, real, point, and string. A variable's type is automatically attached to it based on the type of value assigned. Variables retain their values until reassigned or until the Drawing Editor session has ended. You can name your variables anything you wish, provided that the first character is alphabetic. The variable "pi" is preset to the value of *pi*. You can use it just like variables you define yourself.

The "setq" function is used to assign values to variables. The format is:

```
(setq variable-name value)
```

The "setq" function assigns the specified value to the variable whose name is given. It also returns the value as its function result. If you use "setq" when AutoCAD has issued a "Command:" prompt, it will set the variable and display the value assigned. Note the parentheses surrounding this expression; they are required. A few examples follow.

```
(setq k 3)
(setq x 3.875)
(setq layname "EXTERIOR-WALLS")
```

These expressions assign values to an integer, a real, and a string variable, respectively. Point variables are a bit more complicated, since they contain both *X* and *Y* components. Points are expressed as *lists* of two numbers surrounded by parentheses, as in:

```
(3.875 1.23)
```

The first item in the list is the *X* component of the point, and the second is the *Y* component. You can use another built-in function, "list", to form a list of this form.

```
(list 3.875 1.23)
```

Thus, to assign particular coordinates to a point variable, you can use one of the following expressions:

```
(setq pt (list 3.875 1.23))
(setq pt (list x 1.23))
```

The latter uses the value of variable "x" as the *X* component of the point.

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You can refer to the *X* and *Y* components of a point individually, using two more built-in functions called "car" and "cadr".

(**car pt**) (returns the *X* component of point variable "pt")
(**cadr pt**) (returns the *Y* component of point variable "pt")

Thus, if variable "pt1" is set to point (1,2) and variable "pt2" is set to point (3,4), we can set variable "pt3" to point (1,4) by using the *X* component of "pt1" and the *Y* component of "pt2", as follows:

```
(setq pt3 (list (car pt1) (cadr pt2)))
```

A "reverse" function is also provided. It reverses the order of the items in a list, and can therefore be used to swap the *X* and *Y* components of a point variable. Continuing the previous example, for instance:

```
(reverse pt3)
```

would return the point (4,1).

If you want to use the value of a variable as the response to a prompt from AutoCAD, simply enter the name of the variable, preceded by an exclamation point, "!. Suppose, for example, that you have set variable "abc" to the value 14.887024. You could then enter "!abc" any time you want to respond to a prompt with the value 14.887024. For instance:

Column distance: !abc

Similarly, if you want to begin drawing a line at point (1,4), and you have set variable "pt" equal to that point, you can enter:

Command: LINE

From point: !pt

...

NOTES: In order for expressions or variable references to be interpreted correctly, the left parenthesis, "(", or exclamation point, "!", must be the *first character* you enter in response to a prompt. Since ordinary text strings can begin with a "!" or "(", you cannot use a variable or expression to supply the text string to such commands as TEXT and ATTDEF.

10.5.2 System Variables

AutoCAD provides numerous predefined variables, many of which can be used to change various drawing modes and limits. System variables are accessed by means of the "getvar" and "setvar" functions, where:

```
(getvar "variable-name")
```

returns the value of the named system variable, and:

```
(setvar "variable-name" value)
```

assigns the specified value to the named system variable. Note that the variable name must be enclosed in double quotes. A list of AutoCAD's *system variables* follows.

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System variable	Type	Meaning
APERTURE	integer	Object snap target size.
ATTMODE	integer	Attribute display mode (0 = off, 1 = normal, 2 = on).
AUNITS	integer	Angular units mode.
AUPREC	integer	Angular units decimal places.
AXISMODE	integer	Axis on if 1, off if 0.
AXISUNIT	point	Axis spacing, X and Y.
BLIPMODE	integer	Marker blips on if 1, off if 0.
CHAMFERA	real	First chamfer distance.
CHAMFERB	real	Second chamfer distance.
DRAGMODE	integer	Dragging enabled if 1, off if 0.
ELEVATION	real	Current elevation set by ELEV command.
EXTMAX	point	Upper right "drawing uses" extents.
EXTMIN	point	Lower left "drawing uses" extents.
FILLETRAD	real	Fillet radius.
FILLMODE	integer	Fill on if 1, off if 0.
GRIDMODE	integer	Grid on if 1, off if 0.
GRIDUNIT	point	Grid spacing, X and Y.
HIGHLIGHT	integer	Object selection highlighting on if 1, off if 0.
INSBASE	point	Insertion base point (set by BASE command).
LASTANGLE	real	The end angle of the last arc entered.
LASTPOINT	point	Referenced by "@" during keyboard point entry.
LIMCHECK	integer	Limits checking on if 1, off if 0.
LIMMAX	point	Upper right drawing limits.
LIMMIN	point	Lower left drawing limits.
LTSCALE	real	Global linetype scale.
LUNITS	integer	Linear units mode (1 - 4).
LUPREC	integer	Linear units decimal places or denominator
ORTHOMODE	integer	Ortho on if 1, off if 0.
OSMODE	integer	Object snap bit-code (sum of the following): 1 = Endpoint 32 = Intersection 2 = Midpoint 64 = Insert 4 = Center 128 = Perpendicular 8 = Node 256 = Nearest 16 = Quadrant 512 = Quick
QTEXTMODE	integer	Quick text mode on if 1, off if 0.
REGENMODE	integer	REGENAUTO on if 1, off if 0.
SCREENSIZE	point	Graphics screen size in X and Y pixels.
SKETCHINC	real	Sketch record increment.
SNAPANG	real	Snap / grid rotation angle.
SNAPBASE	point	Snap origin point.
SNAPISOPAIR	integer	Current isoplane (0 = left, 1 = top, 2 = right).
SNAPMODE	integer	Snap on if 1, off if 0.
SNAPSTYL	integer	Snap style (0 = stand, 1 = isometric).
SNAPUNIT	point	Snap resolution, X and Y.
TEXTSIZE	real	Default text size.
THICKNESS	real	Current thickness set by ELEV command.
TRACEWID	real	Default trace width.
VIEWCTR	point	Center of current view.
VIEWSIZE	real	Current view height in drawing units.

For example, to turn the grid off, you could use:

```
(setvar "GRIDMODE" 0)
```

The "GRID OFF" command would have exactly the same effect, and would change the value of the GRIDMODE system variable accordingly.

10.5.3 Arithmetic Expressions

Several arithmetic, trigonometric, and geometric functions are available for use in expressions. They include:

(+ x y)	Returns the sum of <i>x</i> and <i>y</i> .
(- x y)	Returns the difference of <i>x</i> and <i>y</i> .
(* x y)	Returns the product of <i>x</i> and <i>y</i> .
(/ x y)	Returns the quotient of <i>x</i> divided by <i>y</i> .
(max x y)	Returns the maximum of <i>x</i> and <i>y</i> .
(min x y)	Returns the minimum of <i>x</i> and <i>y</i> .

Actually, each of the functions listed above can accommodate more than two arguments, and will perform the associated function for the entire set of arguments specified. For example:

```
(+ 8 3 14 300)
(- 100 20 2)
(* 3 4 5)
(/ 1000 20 2)
```

will compute the sum, difference, product, and quotient of their respective arguments, and return the values 325, 78, 60, and 25, respectively. Likewise,

```
(max 8 3 14 300)
```

will return 300, the maximum of the specified arguments.

On the other hand, the functions listed below require the number of arguments shown.

(abs x)	Returns the absolute value of <i>x</i> .
(sqrt x)	Returns the square root of <i>x</i> .
(expt x p)	Returns <i>x</i> to the <i>p</i> power.
(exp p)	Returns <i>e</i> to the <i>p</i> power.
(log x)	Returns the natural log of <i>x</i> .
(float x)	Returns the promotion of integer value <i>x</i> to real.
(fix x)	Returns the truncation of real value <i>x</i> to integer.
(sin ang)	Returns the sine of <i>ang</i> , where <i>ang</i> is an angle in radians.
(cos ang)	Returns the cosine of <i>ang</i> , where <i>ang</i> is an angle in radians.
(atan x)	Returns the arc tangent (in radians) of <i>x</i> .
(l+ x)	Returns the sum of <i>x</i> and 1. This is equivalent to (+ x 1).
(l- x)	Returns the difference of <i>x</i> and 1. This is equivalent to (- x 1).
(angle p1 p2)	Returns the angle (in radians) between points <i>p1</i> and <i>p2</i> .
(distance p1 p2)	Returns the distance between points <i>p1</i> and <i>p2</i> .
(polar p1 ang d)	Returns the point that is at distance <i>d</i> from point <i>p1</i> at bearing <i>ang</i> (in radians).

Note that none of these functions actually change the value of a variable; they simply compute things and return a result. If you want to use an expression to change the value of a variable, you must apply the result of the expression by means of the "setq" function. For example:

```
(setq x (- x 2))
```

subtracts 2 from variable "x" and stores the result back in "x".

10.5.4 String Functions

Several string functions are available for use in expressions. They are:

(itoa int)	Returns the integer value <i>int</i> converted to an ASCII string.
(atoi s)	Returns the integer conversion of string <i>s</i> .
(ascii c)	Returns the (integer) ASCII code for character <i>c</i> .
(chr int)	Returns the ASCII character represented by integer <i>int</i> .
(strcat s1 s2)	Returns the concatenation of strings <i>s1</i> and <i>s2</i> .
(strlen s)	Returns the length of string <i>s</i> .
(terpri)	Begins a new line on the display screen.

10.5.5 Conditional Expressions

You can construct *conditional expressions* that perform a given operation only if some condition is true. This is done using the "if" function. You must tell "if" what condition to test, and what operation to perform if the test is successful. You may also indicate an operation to be performed if the test fails. There are two forms of the "if" function, as shown below.

```
(if condition do-if-true)
(if condition do-if-true do-if-false)
```

"Condition", "do-if-true", and "do-if-false" can each be an expression, enclosed in parentheses. The "condition" is evaluated, and if non-nil, the "do-if-true" expression is executed. If "condition" evaluates to nil, the "do-if-false" expression is executed if it is present.

Several functions are provided for use in the "condition". These are:

(minusp num)	Returns t if <i>num</i> is negative, nil otherwise.
(zerop num)	Returns t if <i>num</i> is zero, nil otherwise.
(numberp x)	Returns t if <i>x</i> is a number (integer or real), nil otherwise.
(not a b ...)	Returns the logical "not" of <i>a, b, ...</i>
(or a b ...)	Returns the logical "or" of <i>a, b, ...</i>
(and a b ...)	Returns the logical "and" of <i>a, b, ...</i>
(= a b)	Returns t if <i>a</i> is equal to <i>b</i> , nil otherwise.
(/= a b)	Returns t if <i>a</i> is not equal to <i>b</i> , nil otherwise.
(> a b)	Returns t if <i>a</i> is greater than <i>b</i> , nil otherwise.
(>= a b)	Returns t if <i>a</i> is greater than or equal to <i>b</i> , nil otherwise.
(< a b)	Returns t if <i>a</i> is less than <i>b</i> , nil otherwise.
(<= a b)	Returns t if <i>a</i> is less than or equal to <i>b</i> , nil otherwise.
(listp a)	Returns t if <i>a</i> is a list, nil otherwise.
(null a)	Returns t if <i>a</i> is nil, returns nil otherwise.

For example, suppose you have an integer variables *i* and a string variable *s*, and that you want the string to be "Correct" if *i* is zero, and "Error" otherwise. You could accomplish this with the conditional expression:

```
(if (= i 0) (setq s "Correct") (setq s "Error"))
```

Here, "(= i 0)" is the condition to be tested; that is, "is variable *i* equal to zero?". If it is, the first "setq" will be performed; if *i* is not zero, the second "setq" will be performed.

You can base a conditional test on the value of an AutoCAD system variable, as in the following example.

```
(if (> (getvar "FILLETRAD") 0.25) (setvar "GRIDMODE" 0))
```

Here, the "getvar" function is used to obtain the value of the "FILLETRAD" (fillet radius) system variable, and that value is checked. If it is greater than 0.25, the "setvar" function is then used to set system variable "GRIDMODE" to zero (turning off the grid). In this example, no "do-if-false" expression is specified, so no action is taken if the fillet radius is less than or equal to 0.25.

10.5.6 Data Input Functions

Several functions are provided to allow you to prompt the user for input, and to use the response in an expression. These functions are most useful in custom menus. See Appendix B.

Prompting for Input

You can prompt for input using the "prompt" function, as in:

```
(prompt "string")
```

This sends "string" as an AutoCAD prompt. The string may be specified explicitly, or it may be a reference to a string variable. For example:

```
(prompt "Enter your name: ")  
and  
(setq s "Enter your name: ")  
(prompt s)
```

would have exactly the same results. Note that prompts differ from regular AutoCAD displayed output in that prompts are sent to the prompt area of dual-screen systems, as well as to the text screen. The "prompt" function always returns nil.

Obtaining a Point

To read a point input by the user, use the "getpoint" function.

```
(getpoint [prompt])
```

The "prompt" argument is optional; if present, it is displayed as a prompt. You can use this instead of a separate "prompt" function if you like. The "getpoint" function returns nil if an error is encountered. The user can employ any of the point specification methods, including special formats if enabled by the UNITS command.

Obtaining a Distance

To read a distance specified by the user, use the "getdist" function.

(getdist [base] [prompt])

The user can employ any of the distance specification methods, including special formats if enabled by the UNITS command.

If the user enters a point, AutoCAD will attach one end of a rubber-band cursor to that point and prompt for a second point. It then returns the distance between those points. If a point value is specified as the "getdist" function's optional "base" argument, that point is treated as though it was the first point the user entered, and the rubber-band cursor will be attached to that point. If the user then enters a point, AutoCAD will return the distance from the base point to the user-specified point.

If an optional "prompt" argument is specified, that string is issued as a prompt. If both a "base" and a "prompt" are desired, they can be specified in either order. The "getdist" function returns nil if an error is encountered.

Obtaining an Angle

To read an angle specified by the user, use the "getangle" function.

(getangle [base] [prompt])

This works exactly like "getdist", above, except that the angle, rather than the distance is returned if the user enters points. If an error is encountered, nil is returned. The user can employ any of the angle specification methods, including special formats if enabled by the UNITS command.

Obtaining a Real Number

To read a real number (scale factor, etc.) supplied by the user, use the "getreal" function.

(getreal [prompt])

Only decimal numbers and scientific notation are accepted by this function. Special formats enabled by the UNITS command are *not* valid responses. This function is useful for such items as scale factors. It returns nil if an error is encountered.

Obtaining an Integer

To read an integer number supplied by the user, use the "getInt" function.

(getInt [prompt])

Special input formats enabled by the UNITS command are *not* valid responses. The user must enter a simple integer. "GetInt" returns nil if an error is encountered.

Obtaining a Text String

To read a text string supplied by the user, use the "getstring" function.

```
(getstring [t] [prompt])
```

If a (non-nil) argument is supplied, the user must terminate the input string by pressing RETURN. Otherwise, the input string is terminated by either space or RETURN. If "getstring" encounters an error, it returns nil.

10.5.7 Object Snap Function

You can use the object snap mechanism within expressions by means of the "osnap" function.

```
(osnap pt "s")
```

"pt" is a point to be submitted to AutoCAD's object snap mechanism, and "s" is a string that contains the desired object snap modes. The function returns the point that is a result of applying "s" to "pt". If no appropriate point is found, nil is returned. For example:

```
(setq pt2 (osnap pt1 "mid"))
```

sets "pt2" to the midpoint of a line if the point "pt1" is within aperture range of the line. Because this is a function, it does not display an aperture box; indeed, the above example does not ask the user for a point. It just applies the "osnap" function to a point.

10.5.8 Display Control Functions

Functions are provided to allow certain display-oriented functions to be performed in an expression. These are:

- | | |
|-------------------|---|
| (redraw) | Unconditionally redraws the display. |
| (graphscr) | Flips to the graphics display on single-screen systems. |
| (textscr) | Flips to the text display on single-screen systems. |

Each of these functions is equivalent to the similarly-named AutoCAD command.

Chapter 11

ATTRIBUTES (ADE-2 FEATURE)

11.1 Introduction

This chapter describes *Attributes*, a feature of AutoCAD's optional ADE-2 package. Attributes are special AutoCAD drawing entities that contain text. You can use them to label Blocks in your drawing, then later collect them into a disk file for processing by an application such as a bill-of-materials program.

To use an Attribute, you must first create an Attribute Definition using the ATTDEF command. An Attribute Definition is an AutoCAD drawing entity that describes the characteristics of the Attribute. It is displayed on the screen as a text string, called the *Attribute tag*. After you create the Attribute Definition, you can specify it as one of the entities to be included in a Block Definition. Thereafter, each time the Block is inserted, AutoCAD prompts you for the *value* of the Attribute, which may be any text string. Thus, each occurrence of the Block can have a different value for the Attribute. The Attribute value is displayed in the position previously occupied by the Attribute tag.

You can extract Attribute information from the drawing with the Drawing Editor's ATTEXT command. This command writes the Attribute information to a disk file in a form suitable for processing by user-supplied programs or for transfer to a database package such as dBASE II. The Attribute extract feature is similar to drawing interchange files, but it is easier to use and much more powerful.

In most AutoCAD editing operations, a Block Reference together with its Attributes acts as a single entity, just as if the Attributes were Text entities included in the Block Definition. If you select a Block Reference during the ERASE command, the Block Reference and all its Attributes are erased. If you use the CHANGE command to alter the position and orientation of a Block Reference, its Attributes move and turn with it. But as we shall see later, the ATTEDIT command allows you to change Attributes independently of the Block Reference they belong to.

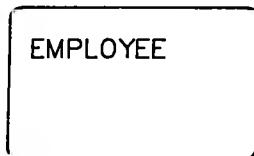
You can associate more than one Attribute with a Block, provided that each Attribute has a different tag. If a Block has more than one Attribute, AutoCAD prompts you for the value of each whenever you insert the Block. You can also, if you wish, define *constant* Attributes. A constant Attribute has the same value for every occurrence of the Block that contains it, so AutoCAD does not prompt for a value when you insert the Block.

Attributes can also be *invisible*. An invisible Attribute is not displayed or plotted; however, information about it is stored in the drawing file and written to the extract file by ATTEXT.

Example

Let's examine a simple example of Attribute usage. Suppose you're an office manager or a facility planner and want to place several desks in an office drawing. Each desk is assigned to an individual employee, and you want to draw the employee's name inside or adjacent to his or her desk. Also, when the drawing is complete, you would like to make a list of all the

desks in the office, showing their locations and occupants. To accomplish this, first draw one desk. Then use the ATTDEF command to create an Attribute Definition with a tag of "EMPLOYEE" and a prompt of "Employee name". If some of the desks in your office are currently unassigned, provide a default value of "CLERK" for this Attribute. When the ATTDEF command is complete, the word "EMPLOYEE" is drawn at the position and size you have specified inside or adjacent to the desk drawing, as shown in the following figure.



Next, use the BLOCK command to create a Block called DESK, selecting both the desk drawing and the Attribute Definition by means of a window. Both are deleted from the drawing as the Block is defined.

Later, when you insert the Block named DESK, AutoCAD prompts you as usual for an insertion point, X- and Y-scale factors, and a rotation angle. Then it displays:

Enter Attribute values

followed by the prompt:

Employee name <CLERK>:

Suppose you respond by entering RETURN to accept the default value. Then AutoCAD draws a copy of the desk, and in the place where the word "EMPLOYEE" appeared when you defined the Attribute, "CLERK" is drawn. If you want to draw Jim Smith's desk, insert the DESK Block again, but this time enter "Jim Smith" in response to the "Employee name" prompt. Your drawing now looks as follows:



Later you can invoke the ATTEXT (ATTRIBUTE EXTRACT) command to create a file whose entries contain Block names, X/Y coordinates, and Attribute values, as shown below:

DESK	110.0	250.0	CLERK
DESK	150.0	300.0	Jim Smith

The ATTEXT command allows you to include other information as well, such as the scale and rotation of the Block; it also allows you to specify a different format for the file.

Expanding on the previous example, suppose you want additional information associated with each desk. The employee's department and telephone extension could be easily added by means of Attribute Definitions similar to that for the employee's name. For facility planning and cost estimation, you might also want to know the color, manufacturer, model number, and price of each desk. Attributes could be defined to hold these pieces of information as well. You might want to make some of them invisible. Now, every time you insert a DESK, AutoCAD prompts for all these Attributes and draws the visible ones.

The drawing could now be used to produce extract files for various purposes. For office management, you might extract the locations, occupants, departments, and telephone extensions of all the desks, as in:

DESK	150.0	300.0	Jane Doe	Accounting	402
DESK	200.0	320.0	Jim Smith	Sales	511
DESK	220.0	320.0	Carol White	Sales	512

For facility planning, you might extract a different set of Attributes (such as location, color, manufacturer, model number, and price) from the same drawing, as in:

DESK	150.0	300.0	walnut	Acme Mfg.	14-1550W	179.95
DESK	200.0	320.0	walnut	Acme Mfg.	14-1550W	179.95
DESK	220.0	320.0	beige	Acme Mfg.	14-1550B	159.95

11.2 ATTDEF Command

The ATTDEF command is used to create an Attribute Definition. The Attribute Definition acts as a template for the Attribute. Besides supplying the Attribute tag, it specifies the prompt that is used to request the Attribute value when the Block is inserted, as well as a default for the Attribute value. It also describes the placement, size, and style of the text string used to display the Attribute value. The command format is:

Command: **ATTDEF**
 Attribute modes -- Invisible:N Constant:N Verify:N
 Enter (ICV) to change, RETURN when done:

Attributes have three optional modes, described below.

- | | |
|-----------|---|
| Invisible | If you select Invisible mode, the Attribute value is not displayed in the drawing when the Block is inserted. This mode is useful if you are not interested in seeing an Attribute on the screen, or if you have so many Attributes that they would clutter up your drawing or unnecessarily increase the amount of time needed to regenerate it. The ATTDISP command, described later in this chapter, allows you to override the setting of Invisible mode. |
| Constant | The Constant mode gives the Attribute a fixed value for all insertions of the Block. Unlike variable Attributes, the value of a constant Attribute cannot later be changed. |
| Verify | If an Attribute is defined with this mode on, you have a chance during the insertion process to verify that its value is correct. |

If you respond to the prompt by entering "I", "C", or "V", the corresponding mode is reversed. The "Attribute modes" line is then displayed again with an "N" changed to a "Y", and the prompt is repeated. When you decide that all the modes are as you like them, enter RETURN.

AutoCAD then prompts:

Attribute tag:

Enter the desired Attribute tag. The tag may contain any characters except blanks. Any lower-case letters in the tag are translated to upper case. The tag must not be null. It is used to identify each occurrence of an Attribute in your drawing. Once you have entered the tag, AutoCAD displays:

Attribute prompt:

Enter the prompt line that you wish to appear later when a Block containing this Attribute Definition is inserted. If you specify a null prompt (by responding with RETURN), the Attribute Tag is used as the prompt. If you have specified Constant mode for the Attribute, AutoCAD does not request a prompt, since it would be meaningless.

The next request is for:

Default Attribute value:

Enter the default Attribute value. It may be null. If you have turned on the Constant mode, AutoCAD does not ask you for a "Default Attribute value"; instead, it prompts:

Attribute value:

You should supply the value for the constant Attribute.

NOTE: If you need leading blanks in the Attribute prompt or default value, you must start the string with a single backslash character (\). If for some reason you want the first character to be a backslash, you must start the string with two backslashes.

Once you have provided the above information, ATTDEF proceeds to issue exactly the same sequence of prompts as the TEXT command, except that it does not solicit the text string itself; the Attribute tag is used in its place. You can specify any of the text alignment types (centered, right-justified, aligned). When you finish specifying the placement of the Attribute tag, it is displayed in your drawing. It is erased from the drawing when you include it in a Block Definition. Later, when you insert the Block, the Attribute value will be displayed at the same location within the Block, with the same text style and alignment.

Repeating ATTDEF Commands

You can automatically align a series of Attributes in the same way as text strings. For the second and subsequent Attributes in a series, simply enter a space or RETURN when AutoCAD prompts you for a command. Instead of requesting the location, height, and angle of the Attribute, AutoCAD automatically aligns it below the previous Attribute.

11.3 ATTDISP Command - Visibility Control

Normally, Attributes are visible unless they are defined with Invisible mode on. The ATTDISP command allows you to override the state of visibility in the Attribute Definition.

Command: ATTDISP

Normal/On/Off <current value>:

In the Normal state, Attributes are visible unless they are marked invisible. Entering "N" selects this state. Entering "On" makes all Attributes visible, and entering "Off" makes all Attributes invisible. If you change states, the drawing is regenerated (unless REGENAUTO is off).

11.4 ATTEDIT Command - Editing Attributes

ATTEDIT is a special command for editing Attributes independently of the Block Reference with which they are associated. It allows you either to edit Attributes one at a time, changing any or all of their properties, or to do a global edit on a selected set of Attributes, changing only their value strings. The global edit can be restricted to Attributes visible on the screen or it can include invisible Attributes and Attributes currently off-screen.

To invoke Attribute editing, enter the ATTEDIT command.

Command: **ATTEDIT**

Edit Attributes one by one? <Y>

Your reply controls the subsequent series of prompts, as follows:

Yes A "Y" response selects "one by one" editing of Attributes; AutoCAD prompts you individually for each one. Only the Attributes that are currently visible on the screen can be edited in this manner, but you can further restrict the set to be edited by means of the normal pointing or windowing methods, and by specification of the Block names, tags, and values of the Attributes you intend to edit. Using this mode, you can change the placement, orientation, and other properties of each Attribute, as well as the Attribute value.

No You can reply "N" to select global Attribute editing. You can use this mode to perform mass editing of all Attributes in a drawing, or you can restrict the set of Attributes to be edited (by tag, value, Block name, and on-screen visibility). Only Attribute values can be changed using global mode.

Whichever edit mode you've chosen (global or individual), you need to select the set of Attributes to be edited. You can restrict the edit to only those Attributes with specified tags or values, or to Attributes attached to Blocks with particular names. AutoCAD asks for such restrictions as follow:

Block name specification <*>:

Attribute tag specification <*>:

Attribute value specification <*>:

The response to each of these prompts may include wild-card characters ("?" and "*"). Attributes are selected for editing only if their associated Blocks match the Block name specification, their tags match the Attribute tag specification, and their values match the Attribute value specification. For each of these prompts, the default response is just an asterisk, which is a wild-card character matching any string. So if you just enter RETURN for the Block name specification, Attributes of Blocks with all names are selected. Similarly, entering RETURN for the tag and value specifications selects Attributes with all tags and values. But if, for example, you enter "EMPLOYEE" for the tag specification, then only Attributes with a tag of "EMPLOYEE" are selected for editing; if you enter "A?1*", then any Attributes whose tags have an A as their first character and 1 as their third character are selected.

It is possible for an Attribute value to be null. If you want to select only such Attributes for editing, enter "\\" for the Attribute value specification.

11.4.1 Global Editing

For global mode, AutoCAD first prompts:

Global edit of Attribute values.
Edit only Attributes visible on screen? <Y>

If you reply "N" to this prompt, AutoCAD flips to text mode (if not already there) and says:

Drawing must be regenerated afterwards.

This means that the changes you make to Attributes are not reflected immediately in the drawing. Instead, at the end of the command, the entire drawing is automatically regenerated (unless REGENAUTO is off).

The next step is to select the set of Attributes to be edited. You can restrict the edit to specific tags, values, or Blocks, as described in the previous section.

Visible Attributes Only

If you are editing only visible Attributes the next prompt is:

Select Attributes or Window or Last:

You can either point to Attributes or do a Window or Last selection to further limit the set of Attributes to edit. If you just entered RETURN for the Block name, Attribute tag, and Attribute value specifications, then all Attributes selected at this point are edited. An "X" is drawn at the starting point of all Attributes selected. AutoCAD then asks:

String to change:
New string:

Before answering the first prompt, you can examine the screen to see if the right Attributes are marked with X's. If not, you can terminate the command with CTRL C. If you like what you see, proceed with the edit, as described below.

All Attributes

If you are editing all Attributes, not just the visible Attributes, the screen is in text mode at this point (unless you have a dual-screen system), and nothing is marked with X's. The "String to change" and "New string" prompts appear, and you should proceed from there.

Respond to the prompts with a string of characters you wish to change and the desired new string. AutoCAD then examines the Attribute value of each selected Attribute, and replaces the first occurrence of the "string to change" with the "new string". No change is made if the "string to change" isn't found within the Attribute value. Note that "?" and "*" characters are interpreted literally in these strings, and not as wild-card characters.

Consider the following example:

String to change: AX
New string: MX

If one of the selected Attributes has a value of "I-AX104", it will be changed to "I-MX104".

If you enter a null value for the "string to change" (by hitting RETURN), then whatever "new string" you enter is placed at the front of each selected Attribute's value. For instance:

String to change: (RETURN)
New string: A-

would change "XYZ" to "A-XYZ", and "10" to "A-10".

11.4.2 Individual Editing

The first step in editing individual Attributes is to select the Attributes to be edited. As in global editing, you can select Attributes by limiting the tags, values, or Blocks to be considered.

The next prompt is:

Select Attributes or Window or Last:

You can either point to Attributes or do a Window or Last selection to further limit the set of Attributes to edit. If you just entered RETURN for the Block name, Attribute tag, and Attribute value specifications, then all Attributes selected at this point are edited.

Each selected Attribute is marked in turn with an "X", and you are allowed to change any of its properties. AutoCAD prompts:

Val/Pos/Hgt/Ang/Style/Lay/Nxt <N>:

The seven options listed are short for Attribute value, Position, Height, rotation Angle, text Style, Layer, and Next. If the Attribute was originally defined with Aligned text, the prompt does not include "Hgt" or "Ang". Any of the options can be selected by entering its first letter. If you choose "N" (or just enter RETURN), AutoCAD proceeds to the next selected Attribute. If you choose any of the others, AutoCAD prompts you to enter a new value. When you enter the new value, AutoCAD redraws the Attribute to reflect the change and then repeats the prompt. The "X" mark remains on the current Attribute until you use the "Nxt" (or "N") option to move to the next Attribute.

The ATTEDIT command terminates normally if you enter "N" after editing the last selected Attribute. You may also terminate the command at any time by entering CTRL C.

value

If you respond with "V" for Attribute value, AutoCAD asks:

Change or Replace? <R>

If you only want to change a few characters of the Attribute value, you may respond to the prompt with "C". AutoCAD asks for:

String to change:

New string:

Respond to the first prompt with the string of characters you want to change, and to the second with the desired replacement string. Either of these strings may be null. Note that "?" and "*" characters are interpreted literally in these strings, and not as wild-card characters.

If you answer "R" or just enter RETURN in response to the "Change or Replace?" prompt, AutoCAD prompts:

New Attribute value:

and you may type in a new Attribute value. If you simply enter RETURN at this point, the Attribute value is set to the null string.

Position

If you want to move the Attribute, respond with "P" for Position. AutoCAD prompts for a new Starting, Center, or End point depending on whether your Attribute is left-justified, centered, or right justified. If it is Aligned, AutoCAD prompts for both ends of a new text base line.

Height, Angle, Style, and Layer

Responding "H", "A", "S", or "L" to the options prompt causes AutoCAD to request new values for the Height, rotation Angle, text Style, or Layer of the Attribute.

11.5 ATTEXT Command - Attribute Extraction

The ATTEXT command lets you extract Attribute entities from your AutoCAD drawing and write them to a disk file for analysis by another program or for transfer to a database. This operation does not change the drawing in any way.

When you invoke the ATTEXT command, AutoCAD asks first for the type of output file you prefer:

Command: **ATTEXT**
CDF, SDF or DXF Attribute extract? <C>

The possible formats are:

CDF Comma Delimited Format is the default Attribute extraction format. It produces a file containing at most one record for each Block Reference in the drawing. The fields of each record are separated by a delimiter (comma by default), and character fields are enclosed in quotes (single quotes by default). Some database packages can read this format directly; in dBASE II, the operation is "APPEND FROM . . . DELIMITED". This format is also easily processed by user programs written in BASIC.

SDF This file format is the same as that produced by dBASE II's "COPY . . . SDF" operation. This is a de facto standard for input to microcomputer

database systems. At most one record is written for each Block Reference in the drawing. The fields of each record are of a fixed width; no field separators or character string delimiters are employed. Using dBASE II, this file format can be read using the "APPEND FROM... SDF" operation. This format is also easily processed by user programs written in FORTRAN.

- DXF** This is a variant of AutoCAD's Drawing Interchange File format, described in Appendix C. It contains only Block Reference, Attribute, and End of Sequence entities.

For CDF and SDF format extracts, the next prompt is:

Template file *<default>*:

This will be explained shortly, under "CDF and SDF Extract". For all extract formats, AutoCAD next prompts for the name of the output file:

Extract file name *<drawing name>*:

If you enter RETURN, the extract file will have the same name as your drawing. Otherwise, it will have the name you enter. Its file type will be ".TXT" if you have selected CDF or SDF format, or ".DXX" if you have selected DXF format. You can also specify a file name of "CON:" to send the Attribute extract directly to the text screen, or "PRN:" to send it to a printer. If you do this, make sure that your printer is connected and ready to print; attempting to print the Attribute extract when your printer is not ready may result in an error condition.

11.5.1 CDF and SDF Extract

The CDF- and SDF-format extraction processes are very similar, and are therefore described together here. Each lets you write Attribute information to a text file in a format that can be read easily by dBASE II or a similar database package, by a program of your own, or by a text editor. You can select which Attributes you want listed and what information you want included about the Blocks in which they are found. For instance, an architect can extract the dimensions, location, and orientation of each Block with a "WINDOW" Attribute for energy-use computations, etc., whereas users who wish only to count the number of "WINDOW" Attributes can extract just each occurrence of the Attribute. Thus, you can extract only the data you need; your file is not expanded unnecessarily by excess data.

The prompt sequence for CDF- and SDF-format extraction is as follows:

Command: ATTEXT

CDF, SDF or DXF Attribute extract? *<C, S, or RETURN>*

Template file *<default>*:

Extract file name *<drawing name>*: *(enter name or RETURN)*

Template File

The *template file* tells AutoCAD how to structure the extract file; it specifies which Attributes are to be extracted, what information is to be included for each Block having those Attributes, and how that information is to appear. If you have previously specified a template file, that file is the default. The template file must have the file type ".TXT", and must be located either in the current directory or on the library drive.

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You can create a template file by means of commands from dBASE II or a similar database package, or by using a text editor. Each line of the template file specifies one *field* to be written in the extract file, including the name of the field, its width in characters, and its numerical precision, if applicable. Each record of the extract file will include all the specified fields in the order given in the template file. The fields that you can specify are shown below, formatted as in a template file.

BL:LEVEL	Nwww000	(Block nesting level)
BL:NAME	Cwww000	(Block name)
BL:X	Nwwwddd	(X coordinate of Block)
BL:Y	Nwwwddd	(Y coordinate)
BL:LAYER	Cwww000	(Block insertion layer name)
BL:ORIENT	Nwwwddd	(Block rotation angle)
BL:XSCALE	Nwwwddd	(X scale factor of Block)
BL:YSCALE	Nwwwddd	(Y scale factor)
Other	Cwww000	(Attribute tag, character)
Other	Nwwwddd	(Attribute tag, numeric)

(The comment fields must not actually be present in the template file.) Each record starts with the field name. Field names may be of any length. The next nonblank character must be "C" or "N", denoting a character or numeric field. The next three digits are the field width in characters. The last three are the number of decimal places for a numeric field. The field width and decimal places correspond closely to the "w" and "d" in a FORTRAN "Fw.d" format.

The template file can include any or all of the first eight field names listed above, in whatever order you wish them to appear in each record of the extract file. In addition, the template file must include at least one Attribute tag field. The Attribute tag fields determine which Attributes, and hence which Blocks, are included in the extract file. In the extract file, each such field is filled with the corresponding Attribute value. If a Block contains some, but not all, of the specified Attributes, the values for the absent ones are filled in with blanks (if character) or zeroes (if numeric). Block References that do not contain any of the specified Attributes are excluded from the extract file. A particular field must not appear more than once in the template file.

Attribute values can be written to the extract file either as character fields or as numeric fields. The format is selected by means of the "Cwww000" or "Nwwwddd" descriptor for the appropriate Attribute tags. If numeric output is chosen for a particular Attribute tag, the corresponding Attribute value should actually contain numeric information; otherwise this field of the output record will not contain the proper information. A warning message is displayed when AutoCAD detects such an error.

An Attribute is extracted only if its tag name matches the field name specified in the template file. Lower-case letters in both names are converted to upper case before the comparison is performed.

The following is a sample template file.

BL:NAME	C008000	(Block name, 8 characters)
BL:X	N007001	(X coordinate, format nnnnn.d)
BL:Y	N007001	(Y coordinate, format nnnnn.d)
MANUFACTURER	C016000	(Manufacturer's name, 16 characters)
MODEL	C009000	(Model number, 10 characters)
PRICE	N009002	(Unit price, format nnnnnn.dd)

To generate a template file with dBASE II, first define your database. Then enter:

```
USE database
COPY TO structure-name STRUCTURE EXTENDED
USE structure-name
COPY TO structure-name SDF
```

At this point, *structure-name.TXT* has the template ready for AutoCAD. Note that the SDF form of template file is used for CDF extract as well.

To create a template file with a text editor, simply type in the field names and characteristics you want to appear in the extract file. Remember that the template file must have a file type of ".TXT".

An SDF extract using the sample template file listed above might produce an output file that looks like this (with a header line added here to show you where the fields are):

< NAME >< X >< Y >< MANUFACTURER >< MODEL >< PRICE >
DESK 120.0 49.5Acme Indust. 51-793W 379.95
CHAIR 122.0 47.0Acme Indust. 34-902A 199.95
DESK -77.2 40.0Top Drawer Inc. X-52-44 249.95

Compare this with the sample template file. Note that the order of the fields in each record corresponds to the order of the field names in the template file. Also, the width of each field is dictated by the template file entries.

If a CDF extract is performed using the same template file, the output file might appear as shown below.

```
'DESK', 120.0, 49.5,'Acme Indust.', '51-793W', 379.95
'CHAIR', 122.0, 47.0,'Acme Indust.', '34-902A', 199.95
'DESK', -77.2, 40.0,'Top Drawer, Inc.', 'X-52-44', 249.95
```

Here again, the order of the fields in each record corresponds to the order of the field names in the template file. However, the specified field widths are used only as maximum widths. Character fields are enclosed in quotes and trailing blanks are removed; a null string is denoted by two consecutive quotes (""). Numeric fields are written with one leading blank (or a minus sign if negative). The default field delimiter is the comma, and the default character string quote character is the single quote, or apostrophe. Two additional template records can be used to override these defaults:

```
C:DELIM     c        (Field delimiter)
C:QUOTE     c        (Character string delimiter)
```

The first nonblank character following the "C:DELIM" or "C:QUOTE" field name becomes the respective punctuation character. For example, if you want to surround character strings with double quotes rather than single quotes, include the following line in your template file:

```
C:QUOTE "
```

Similarly, if the template file includes the line:

```
C:DELIM /
```

AutoCAD will use slashes rather than commas to separate the fields of each record in the output file.

The field delimiter must not be set to a character that is valid for a numeric field. Similarly, the quote character must not appear in any character fields.

Nested Blocks

BL:LEVEL is the nesting level of a Block Reference. A Block inserted directly in the drawing has a nesting level of 1. A Block Reference that is part of another Block has a nesting level of 2, and so on.

For a nested Block Reference, i.e., one with BL:LEVEL greater than 1, the X and Y coordinates, scale factors, and rotation angle reflect the actual position and size of the nested Block. For example, if Block OUTER is inserted in your drawing with X and Y scale factors of 2 and a rotation angle of 30 degrees, and OUTER contains an instance of Block INNER inserted with scale factors of 1.5 and rotated 15 degrees, then the record in the extract file for this instance of INNER will have BL:XSCALE and BL:YSCALE equal to 3 and BL:ORIENT equal to 45.

In some cases of deeply nested Blocks with different X and Y scale factors and nonzero rotation angles, the inner Block References cannot be correctly represented with just two scale factors and a rotation angle. When this happens, the scale factors in the extract file record are set to zero.

Error Handling

If a field is not wide enough for the data that is to be placed in it, AutoCAD truncates the data and prints the following message:

**** Field overflow in record <{i+}record number{i-}>**

This could happen, for example, if you have a BL:NAME field with a width of 8 characters, and a Block in your drawing has a name 10 characters long.

Using the Extract File

Once the CDF or SDF extract file has been created, you can load the Attribute data into your database or process it with a program of your own. In dBASE II, use the following instructions to load an SDF extract file:

```
USE database
APPEND FROM extract-file-name SDF
```

For a CDF extract file, the dBASE II command sequence is:

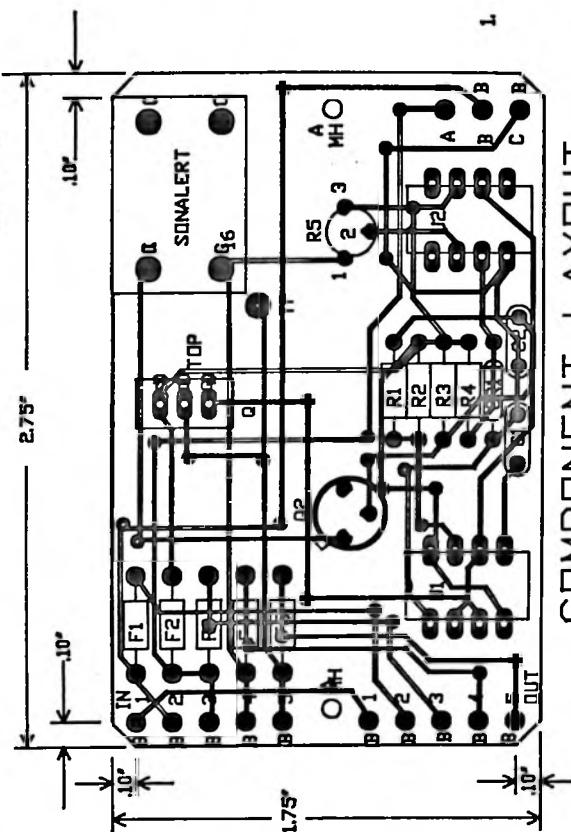
```
USE database
APPEND FROM extract-file-name DELIMITED
```

11.5.2 DXF Extract

Attribute extraction in DXF format creates a file similar to a Drawing Interchange File, the format of which is described in Appendix C. This format includes more detailed information, but is generally much more complicated and harder to interpret than the SDF format.

The prompts for DXF-format extraction are similar to those for CDF- and SDF-format extraction, except that AutoCAD does not request the name of a template file. The output file has the type ".DXX" to distinguish it from normal DXF files.

DRILL DWG, FOR 100 Hz BD,
SCALE 2 : 1



STANDARDS
(UNLESS OTHERWISE NOTED)

1. ALL UNCALLED OUT HOLES TO BE .031" DIA.
2. ALL HOLES TO BE $\pm 3\%$ TOLERANCE.
3. HOLE CALL OUT DIA. QUANTITY

	DIA.	QUANTITY
A	.090"	2
B	.070"	13
C	.054"	4
D	.035"	3

4. ALL HOLES TO BE PLATED THRU TOP AND BOTTOM SIDE OF BOARD (EXCEPT 'A' HOLES).
5. TRIM BOARD AS PER DIMENSIONS.

STANDARDS
(UNLESS OTHERWISE NOTED)

1. LAST REFERENCE DESIGNATORS USED
F5 R5 U2 Q2 C2 CRI
5 MH IN 5 MH OUT ABC MH TP
SIGNALET
2. ALL RESISTORS ARE 1/4 W, $\pm 5\%$ TOLERANCE.
3. ALL RESISTANCE IS EXPRESSED IN OHMS.
4. ALL CAPACITANCE IS EXPRESSED IN MICROFARADS.
5. REFER TO B.Q.M. FOR ALL COMPONENT VALUES.

COMPONENT LAYOUT
100 Hz BOARD
SCALE 2 : 1

Chapter 12

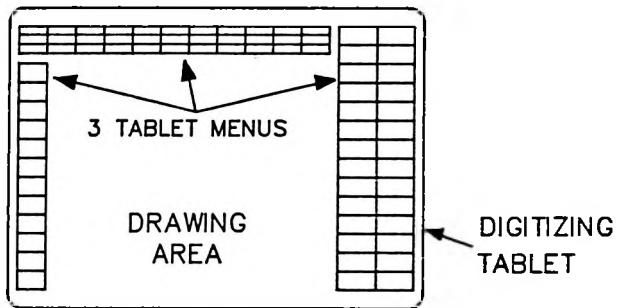
POINTING DEVICE FEATURES

This chapter describes AutoCAD features that are directly related to graphic input, or pointing, devices.

12.1 Tablet Menus

If you have a digitizing tablet, you can set aside portions of its surface as *tablet menus* for entry of commands. You can attach printed rectangular forms over these areas to assist you in locating menu items. These forms are divided into columns and rows of smaller "boxes", each assigned a number from left to right, top to bottom, starting with 1. To select a menu item, move the tablet's cursor to the appropriate box, and push the "pick" button. AutoCAD supports up to four tablet menus, each with as many boxes as you'd like, and each situated wherever you like on the tablet surface.

Although each tablet menu area must be rectangular, you can construct an L- or U-shaped menu by placing two or more menu areas adjacent to one another, as in the following figure:

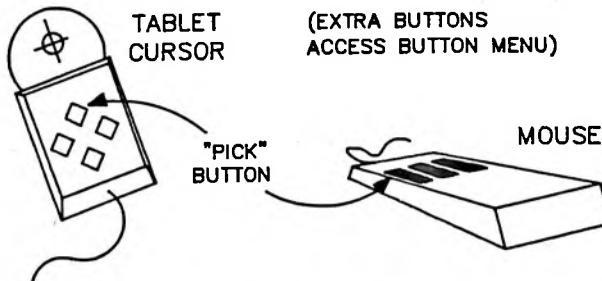


You can use the "TABLET CFG" command, described later in this chapter, to designate menu areas on the tablet. Once AutoCAD has learned the menu positions, you can select a menu item simply by pointing at the desired box and pressing the digitizer's "pick" button. The corresponding data from the tablet menu file is entered just as if you had typed it on the keyboard.

Appendix A and the AutoCAD Standard Menu User Guide describe the standard menu file, and Appendix B provides instructions on defining your own menu file. You can select a different menu file with the MENU command (Chapter 3).

12.2 Button Menu

One button on a tablet or mouse is known as the "pick" button. This button is always used to designate points, to select the tablet menu item on which the pointer is currently resting, or to select the screen menu item to which the screen crosshairs have been directed. If your tablet or mouse has more than one button, you can use the extra buttons to select items directly from a "button menu", no matter where the pointer or screen crosshairs are presently located. See the figure, below.



Consult your AutoCAD Installation Guide / User Guide Supplement to determine which button is the "pick" button, and which buttons are available for button menu selection.

Appendix A and the AutoCAD Standard Menu User Guide describe the standard menu file, and Appendix B provides instructions on defining your own menu file. You can select a different menu file with the MENU command (Chapter 3).

12.3 Copying Paper Drawings - Tablet Mode

When AutoCAD is used on a system equipped with a digitizing tablet, the tablet is normally used to "point" at items on the screen. That is, as the stylus is moved across the tablet, the crosshairs on the screen follow it. The actual coordinates on the tablet have no meaning: only the screen coordinates they point to relate to drawing coordinates.

This is usually the mode preferred when entering new drawings or editing drawings already stored with AutoCAD. When you zoom in on a drawing to work on a section in detail, the tablet automatically points with a finer degree of resolution because the screen now shows a smaller portion of the drawing.

Applications arise, however, when existing dimensioned material must be entered into AutoCAD. For example, you may have printed circuit artwork that was originally laid out by hand, which you now wish to store and edit with AutoCAD. When you digitize the drawing, you want the entities to be entered in the database with coordinates that relate to the original drawing coordinates, not numbers relating to AutoCAD's screen scale.

If the existing drawing is too big to fit on the tablet, you need to be able to enter the drawing in pieces, assuring registration between separately digitized parts.

AutoCAD accomplishes these goals with a special mode called Tablet. When in Tablet mode, AutoCAD treats the digitizing tablet as a true digitizer rather than a screen pointing device.

It maps the coordinate system of the original paper drawing directly into AutoCAD drawing coordinates, regardless of the scale, position, or rotation of the drawing with respect to the tablet, and regardless of the display window in effect.

WARNING: Do not attempt to copy a paper drawing without using Tablet Mode -- the aspect ratio of the drawing may be distorted if normal pointing mode is used.

12.3.1 Entity Pointing in Tablet Mode

The commands that allow you to select entities by pointing at them, such as ERASE, MOVE, and LIST, still work when you have placed the digitizer in Tablet mode. You can ERASE an entity simply by moving the tablet stylus until it points to the object on the paper drawing, or until the screen crosshairs are pointing to the desired entity on the graphics monitor. Then select this point, just as you would if Tablet mode was off.

Note that since entities drawn in Tablet mode may be off-screen, you may have to ZOOM or PAN to be able to point to them with the screen crosshairs. It is a good idea to do a ZOOM Window of the area being digitized to make sure all the digitized items are on the screen.

12.3.2 Tablet Mode and Snap Mode

The Snap setting remains in effect in Tablet mode. After the coordinates are transformed, they are forced to points on the Snap grid. If you don't want this, turn Snap mode off.

In Tablet mode, the digitizing tablet cannot be used to point to items on the screen menu; the right-hand edge of the tablet surface is available for drawing entry. However, the tablet menus remain active and can be used freely. Thus, all portions of the tablet outside tablet menu areas are available for coordinate digitization in Tablet mode.

12.4 TABLET Command

You can use the TABLET command to align (*calibrate*) the tablet with the coordinate system of a paper drawing, to turn Tablet mode on and off, and to designate the areas of the tablet reserved for menus and for screen pointing. When you enter the TABLET command, the first prompt is:

Option (ON/OFF/CAL/CFG):

Respond with one of the indicated options; their action is summarized in the following table.

- | | |
|-----|---|
| ON | Turns Tablet mode on. The "CAL" option must have been previously used to calibrate the tablet. |
| OFF | Turns Tablet mode off. The tablet again becomes a screen pointing device. |
| CAL | Use this option to calibrate the tablet with a paper drawing. |
| CFG | Select this option if you want to designate or realign the areas of the tablet used for tablet menus and for screen pointing. |

These options are described in detail in the following sections.

12.4.1 TABLET CAL - Calibration

The "TABLET CAL" command is used to turn on Tablet mode and calibrate the tablet. First, you should fasten the input drawing to the tablet so it won't shift during digitization. Then enter:

Command: TABLET Option (ON/OFF/CAL/CFG): CAL

Calibrate tablet for use . . .

Digitize first known point: (digitize)

Enter coordinates for first point: (x1,y1)

Digitize second known point: (digitize)

Enter coordinates for second point: (x2,y2)

After you enter "CAL", AutoCAD asks you to digitize a point and enter its coordinates in the system used for the original drawing. It then asks for a second known point. These points may be any points in the drawing, as long as they are not the same. Neither needs to be the origin or on either axis.

There is one restriction on the coordinate system of the paper drawing; it must be "right handed". (If you hold the paper so that its Y coordinates increase from bottom to top, its X coordinates must increase from left to right.) However, you can fasten the drawing on the tablet at any angle; AutoCAD figures out the proper coordinate transformation from the points and known coordinates.

Once you've done the calibration, you can draw objects by using the regular AutoCAD commands. Whenever you digitize a point, the tablet returns coordinates from your original drawing.

Remember that the drawing limits are still in effect. Use the LIMITS command (Chapter 3) to ensure that the drawing limits include all coordinates you'll generate from the tablet, or to turn limits checking off. The screen displays entities as they are drawn, but it may not show the portion of the drawing you're digitizing (of course, it's a lot more handy if it does, and the "ZOOM Window" command is perfect to accomplish this). The screen crosshairs follow the movement of the tablet cursor.

If the drawing shifts on the tablet, or if you move the drawing (for example to digitize another portion of a large drawing) you can recalibrate the tablet to the new position by giving another "TABLET CAL" command and entering two new known points.

12.4.2 TABLET OFF - Exit Tablet Mode

To return to normal screen pointing with the tablet, enter:

Command: TABLET

Option (ON/OFF/CAL/CFG): OFF

12.4.3 TABLET ON - Begin Tablet Mode

After turning Tablet mode off with the "OFF" option, you can turn it back on with the command:

Command: TABLET

Option (ON/OFF/CAL/CFG): ON

If the tablet has yet not been calibrated for this editing session, the calibration prompts are issued.

NOTE: You can also toggle Tablet mode on and off using a single control key, as described in Section 8.8.

12.4.4 TABLET CFG - Configuration

You can designate or realign your tablet menu areas, or define a small portion of a large tablet for use as a screen pointing area, with the "TABLET CFG" command sequence. AutoCAD prompts:

Enter number of tablet menus desired (0-4) <default>:

If you are already using tablet menus and you select the same number of menus, AutoCAD asks:

Do you want to realign tablet menu areas? <N>

Reply "Y" if you want to realign your tablet menu locations. Whether this is the initial alignment or a realignment, the prompts from this point on are identical. You can specify up to four separate menu areas; AutoCAD prompts for each one with:

Digitize upper left corner of menu area *n*;
Digitize lower left corner of menu area *n*;
Digitize lower right corner of menu area *n*;

The letter "*n*" is used here to represent the menu number (1-4). Affix your printed menu form to the tablet surface, and digitize the requested points. The set of three points must form a 90 degree angle. If it doesn't, AutoCAD prompts you to try again. You can enter tablet menu areas skewed at any angle. Note, however, that for maximum drawing area surface, menu area edges and tablet area edges should be as parallel as possible.

Once you have entered the location of the menu, AutoCAD prompts:

Enter the number of columns for menu area *n*;
Enter the number of rows for menu area *n*;

Enter positive integer values from the keyboard.

Tablet menu areas must not overlap.

Screen Pointing Area

A large digitizing tablet is very handy for copying large drawings in Tablet mode. However, use of the entire tablet surface for screen pointing can be awkward due to the large distances the tablet's cursor must be moved to effect a small movement of the screen crosshairs or to access the screen menu.

Therefore, you can use the "TABLET CFG" command to designate a small, convenient portion of the tablet's surface to be the screen pointing area. The entire area of the graphics monitor can be accessed from this area of the tablet.

After all interaction concerning tablet menus has been completed, AutoCAD asks:

Do you want to respecify the screen pointing area? <N>

If you reply "Y", the following prompts appear.

Digitize lower left corner of screen pointing area:

Digitize upper right corner of screen pointing area:

Respond using the tablet pointer. The screen pointing area must not overlap tablet menu areas.

12.5 SKETCH Command - Freehand Drawing (+1)

The SKETCH command permits freehand drawings to be entered as part of an AutoCAD drawing. Freehand drawings are distinguished from normal AutoCAD drawings in that they are automatically entered as the pointing device is moved, rather than explicitly built up of lines, arcs, etc., entered by pointing to endpoints or other geometric features. Freehand drawing is the best method to enter such material as map outlines, signatures, or other irregular material. It should not be used when normal AutoCAD data entry is sufficient.

The freehand drawing facility captures your sketching as a series of lines. You can perform limited editing on these lines before recording them in the AutoCAD drawing database. Once recorded, all the normal facilities of AutoCAD can be used on the freehand material--you can move it, delete all or part of it, make it part of a Block, and so forth.

Freehand drawings can be made only with a pointing device (digitizing tablet, mouse, pen, etc.). It is not practical to make freehand drawings with the keyboard's cursor movement keys; therefore, this facility is not provided.

You should be aware that freehand drawing, especially with very fine accuracy, generates a large number of lines. Although every effort is made to reduce the number of lines generated (by combining lines in the same direction), it is possible in twenty seconds of freehand sketching to create a drawing with as many lines as a normal drawing which took twenty hours to enter. Thus, a little of this goes a long way, and you should use it only when required and with no more accuracy than the application demands. On the other hand, AutoCAD imposes no limits on the size of your drawing other than those imposed by the physical hardware storing it, so if large capacity storage is available, you could enter, say, a high resolution map of the world's coastlines with AutoCAD.

Let's try a simple freehand sketch. Turn Ortho, Snap, and Tablet modes off and enter the SKETCH command:

Command: **SKETCH**

Record increment:

You should enter the distance, in drawing units, over which movement of the pointer justifies generating a new line. This establishes the resolution or accuracy of the sketch in the following way: starting at the first point, moving the pointer less than the record increment does not generate a new line. Once you move a distance equal to the record increment, a line is generated to that point; then you have to move another record increment (in any direction) to generate another line. Assuming that your drawing limits are from (0,0) to (10,10) and that the screen shows the entire drawing, a record increment of 0.1 would generate a reasonably high-resolution sketch (100 lines across the drawing), while a record increment of 1.0 would generate a very low-resolution sketch. (Only movement of 1/10 the drawing size would record a new line.)

Let's set the record increment to 0.1 and try some sketching. After you type "0.1" and RETURN, SKETCH displays the following message:

Sketch. Pen eXit Record Erase Connect .

to remind you that you're sketching and list some subcommands you can use while sketching (which we'll discuss shortly).

NOTE: Sketching should be performed only on a layer having the "CONTINUOUS" linetype.

12.5.1 The Sketching Pen

Sketching uses the concept of a "pen", which should not be confused with a physical pen you may be using as a pointer. When the pen is up, you are not drawing. When the pen is down, you are drawing. If you are using a physical pen as the pointer, whether it is on the screen or not has nothing to do with whether the sketching pen is down. Henceforth, "pen" refers to the sketching pen and "pointer" to the pointing device, whatever it may be.

The pen is initially up, and while it is up the crosshairs on the screen track movements of the pointer as usual in AutoCAD. To make a sketch, move to the start point of your sketch line and press the "P" key on the keyboard (or your pointing device's "pick" button). This key lowers the pen if it was up, and raises it if it was down. Once you lower the pen, a line is started at the point you lowered the pen, and as you move the pointer, new line segments are added. The rubber band cursor is used to show you lines not yet long enough to add to the sketched line. When you raise the pen by pressing "P" (or the pointer's "pick" button) again, the sketch line is extended to the point where the pen was raised (even if closer to the last point than the record increment), and the normal crosshairs return.

You may lower the pen again and sketch another line and continue to do so without leaving the SKETCH command. Sketched lines are not immediately added to the drawing; they are kept as temporary lines until you exit the SKETCH command or use one of its facilities to process them in some way. If you have a color display, the temporary lines you sketch are displayed in green, unless you are drawing in green, in which case red is used. When the lines are added permanently to the drawing, they are redrawn in the color associated with the current layer.

12.5.2 Using Sketched Lines in AutoCAD

The information you've sketched is entered as individual lines in the AutoCAD drawing. You can CHANGE, ERASE, MOVE, and otherwise work on these lines like any other lines. Remember that each line is independent of the rest, and that the sketched information is not a unit to AutoCAD. If you want to make it behave as a unit, you can do so by using the BLOCK command to make it into a Block.

If your copy of AutoCAD includes the ADE-3 package, you can also collect the sketched lines into one Polyline. The PEDIT command (Section 5.2) provides a "Join" option that you can use for this purpose.

12.5.3 SKETCH Subcommands

The "P" we used to raise and lower the pen is one of the "subcommands" to which SKETCH responds. Because freehand work is very different from structured data entry, SKETCH provides its own small set of commands for you to use while a sketch is in progress. These commands are all very simple compared to normal AutoCAD commands, and each can, in fact, be invoked by a single key. For convenience, if your pointer has more than one button, these subcommands can also be invoked by pressing the appropriate button on the pointer. Note that you cannot use the normal button menu while a SKETCH command is in progress; AutoCAD automatically redefines the buttons while you are using the SKETCH command. If your screen or tablet menus include SKETCH subcommands, you can enter these subcommands by the normal pointing and selecting process; be sure to raise the pen before using the pointing device to select menu items.

The following is a list of SKETCH subcommands, and the pointer buttons that correspond to them:

Command Character	Pointer Button	Function
P	"Pick"	Raise/lower pen
. (period)	1	Line to point
R	2	Record lines
X, Space, RETURN	3	Record lines and exit
Q, CTRL C	4	Discard lines and exit
E	5	Erase
C	6	Connect

In addition, the special keys that toggle Snap, Ortho, and Grid modes can be used while the SKETCH command is active.

Each of the SKETCH subcommands takes effect immediately on entry -- no RETURN is required after them.

12.5.3.1 P - Pen up/down

As described earlier, the "P" command lifts the sketching pen if it was down, and lowers it if it was up.

12.5.3.2 . (period) - Line to point

The "." command is used to draw a straight line from the endpoint of the last sketched line to the current location of the pointer. It is meaningful only when the pen is up. You might use it, for example, to draw a straight line which composes part of a political boundary on a map. After adding the straight line, the pen returns to the up position.

12.5.3.3 R - Record

The "R" command records all the temporary lines sketched so far as permanent lines, without changing the up/down state or the position of the sketching pen. Remember that once recorded, the lines cannot be edited with SKETCH subcommands, but only with the regular AutoCAD facilities. The number of lines added to the drawing by this command is displayed.

12.5.3.4 X - Record and Exit

The "X" command records all the temporary lines entered and returns to the AutoCAD command prompt. The space bar and RETURN key have the same effect, and the "X" is provided mainly for more convenient use in menus. The number of lines added to the drawing by this SKETCH command is displayed as:

nnn lines recorded.

12.5.3.5 Q - Quit

The "Q" command discards all temporary lines sketched (since the start of the SKETCH command or the last "R" command) and returns to the AutoCAD command prompt. CTRL C has the same effect, serving as the universal command terminator in AutoCAD.

12.5.3.6 E - Erase

The "E" command allows you to selectively erase from any portion of the line you have sketched to the end. Type "E" to activate the command, and you will receive the message:

Erase: Select end of delete.

If the pen was down, it is raised by this command. Move the crosshairs to the point at which you want to chop the line. As you do this, the line is displayed with the portion from the point closest to the crosshairs to the end blanked out. When you have selected the portion you wish to erase, press the "P" key (or the "pick" button on the pointer) to erase the indicated portion of the temporary line. Remember that you cannot erase lines by this method once you've recorded them. If you decide not to erase anything, just press "E" again (or any other command), and the message:

Erase aborted.

will appear. You are now back to the normal SKETCH prompt. Note: depending on the speed of your computer and display, and the complexity of the sketched material, the display of the portion of the line to be saved may be very fast or quite slow. If it is slow on your machine, move the pointer slowly, waiting for the crosshairs on the screen to move and the line to be redisplayed after each pointer move.

12.5.3.7 C - Connect

The "C" command lets you pick up a sketch after you've raised the pen to point to a menu item or after you've erased some material. When you enter the "C" command, you receive the prompt:

Connect: Move to endpoint of line.

and the crosshairs appear. Move them to the endpoint of the last line you've entered (you may only connect to the last endpoint, or the end of the last Erase). When the crosshairs are within one record increment of the endpoint, the pen is automatically lowered, the SKETCH prompt reappears, and you can extend that line without a gap. The "C" command makes sense only when the pen is up. If you use it while the pen is down, you'll receive the message:

Connect command meaningless when pen down.

If there's nothing to connect to (e.g., nothing drawn or all information recorded), you'll get the message:

No last point known.

You can terminate the "C" command without connecting by typing "C" again (or any other command). In that case you will receive the message:

Connect aborted.

12.5.4 Effects of Other Modes

Operation of the SKETCH command is affected in various ways by the current setting of Snap, Ortho, and Tablet modes.

12.5.4.1 Sketching in Tablet Mode

So far we have been sketching on the screen, equivalent to drawing on the screen. With a digitizer, it is possible to use Tablet mode to align a paper drawing and sketch information from it into the AutoCAD drawing. Coordinates are preserved just as with normal data entry from the digitizer in Tablet mode. To make more of the digitizer available in Tablet mode, you are not allowed to point to the screen menu from the digitizer while Tablet mode is in effect. The area normally used to do this can be used for your drawing. If you are using tablet menus, they will continue to work (although reducing the free area for the drawing, of course). During the SKETCH command, CTRL T cannot be used to toggle Tablet mode on and off.

If your system has both a tablet and another pointing device, such as a pen, which does not work in Tablet mode, you must not mix input to the SKETCH command from these two sources while Tablet mode is on. Once you use one, you must continue to use it for the duration of the SKETCH command. Attempting to mix input results in the message:

Screen and tablet mode input may not be mixed.

and further input from the second device is ignored for the rest of the SKETCH command. In order to sketch with the other device, you must end the current SKETCH command and issue another using the second device.

While sketching in Tablet mode, the crosshairs on the screen appear at the screen coordinates corresponding to the coordinates generated from the tablet. Thus, it is possible for the crosshairs and the lines you are sketching to be totally off screen. If you wish to refer to the screen while sketching, you should do a ZOOM Window to the desired work area before the SKETCH command.

Also, when sketching in Tablet mode, AutoCAD must do far more computation per point from the pointer than in screen mode, so when using very high resolution (small record increment values), you may have to trace more slowly to preserve accuracy. See Section 12.5.5 for details on how the SKETCH command can tell you when you should slow down.

12.5.4.2 Sketching and Snap Mode

When Snap mode is on, all coordinates received from the pointer are snapped before being tested by SKETCH for being outside the record increment. There is no point in setting the record increment smaller than the Snap value, as the minimum movement of the pointer is the Snap value.

12.5.4.3 Sketching and Ortho Mode

When Ortho mode is on, the SKETCH command draws only vertical and horizontal lines. If the movement between two adjacent points from the pointer is diagonal, an orthogonal line is drawn first in the longer direction between them, then over to the second point. This makes a "staircase" line which follows the pointer. Although this mode is not very useful for tracing, it is quite handy when deciding on the correct record increment, because it lets you see graphically on the screen the length of each line entered and the roundoff inherent in that length.

12.5.5 Protecting Sketch Accuracy

When the sketching pen is down, SKETCH examines each point from the pointer to see if its distance from the last endpoint is greater than the record increment. If so, a new line is generated. Each time the pointer sends a point, AutoCAD must do several calculations and possibly enter a temporary line.

When modes requiring considerable calculation are in effect, such as Tablet and Snap, this may result in a significant delay between points on slower computers. (No specifics can be given here, because performance of computers on which AutoCAD runs varies by more than a factor of ten). When the computing time per point is long enough to allow you to move the pointer more than the record increment while the computation is in progress, you will lose accuracy in your sketch: when the computer next looks at the pointer, it will have moved beyond the point where the next line would normally have been generated.

Consequently, when sketching with a small record increment on a slow computer, with complex modes in effect, you may have to move the pointer more slowly to avoid losing accuracy. The SKETCH command provides a quality control measure to protect the accuracy when precision is important (such as when tracing a map outline).

If you respond to the "Record increment:" prompt with a negative number, the SKETCH command uses the record increment as if it were positive, but it tests every point received from the pointer against twice the record increment. If the point is more than two record increments away, the computer "beeps" as a warning that you should slow down to avoid losing accuracy. Having the beeper beep from time to time is all right, especially when working with very fine accuracy or when tracing areas with little detail. Using this mode does not, in itself, measurably slow down the tracing speed.

On most machines running AutoCAD, the temporary lines generated while sketching fit in memory, and no delays occur due to disk operations while sketching. When working with very complex drawings, or tracing a large amount of material before recording it, or if your computer has a small amount of memory, a disk access may be required while tracing. Since this takes quite a while, you might lose a large number of points during this process. To prevent this loss, the SKETCH command constantly monitors the available memory. When a disk access will be required within the next hundred points or so, it displays the message:

Please raise the pen!

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and sounds a continuous beep. You should stop tracing (you don't have to stop instantly, SKETCH gives you reasonable warning time), and raise the pen with the "P" key or the pointer button. AutoCAD then performs the required disk operations while the pen is up and not tracing. The message:

Thank you. Lower the pen and continue.

appears when the disk operation is completed; you may then lower the pen and continue tracing. Most users of the SKETCH command never experience this, but the protection is there in case your computer or drawing needs it. On computers with just enough memory to barely run AutoCAD, you may see the message:

Warning -- low memory -- accuracy may be low.

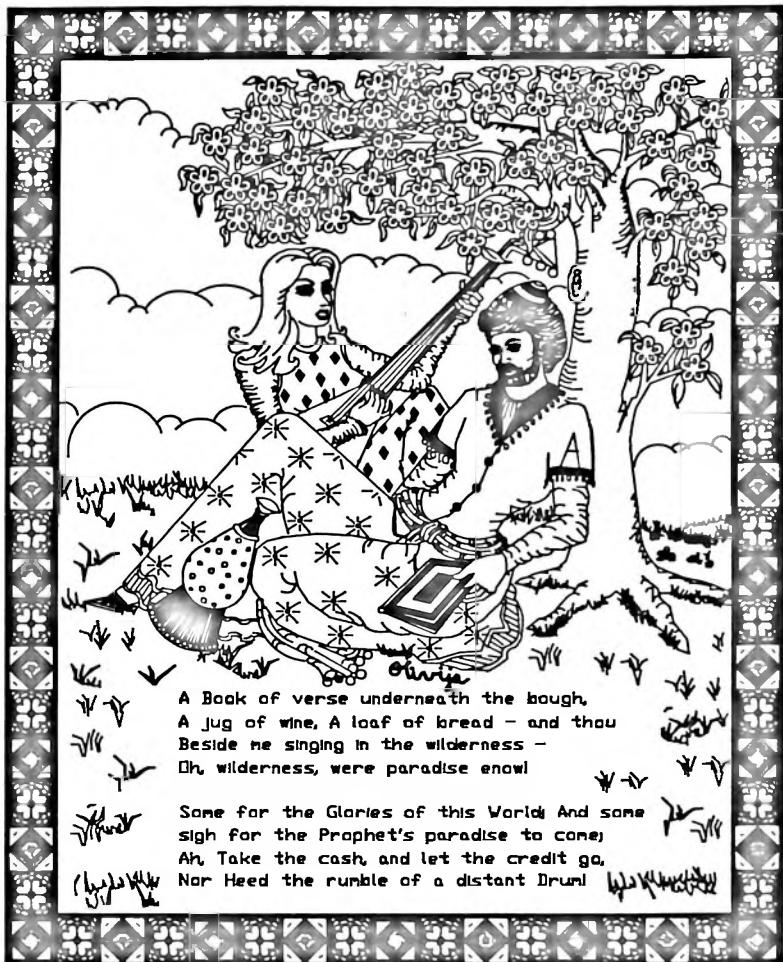
If this message appears, there is so little free memory that the SKETCH command cannot protect you against disk accesses, so if you hear the disk click, you'd better stop tracing right away. This message appears only if the physical memory available is marginal to run AutoCAD; it is not related to the drawing complexity or the amount of information sketched.

Disk Space Considerations

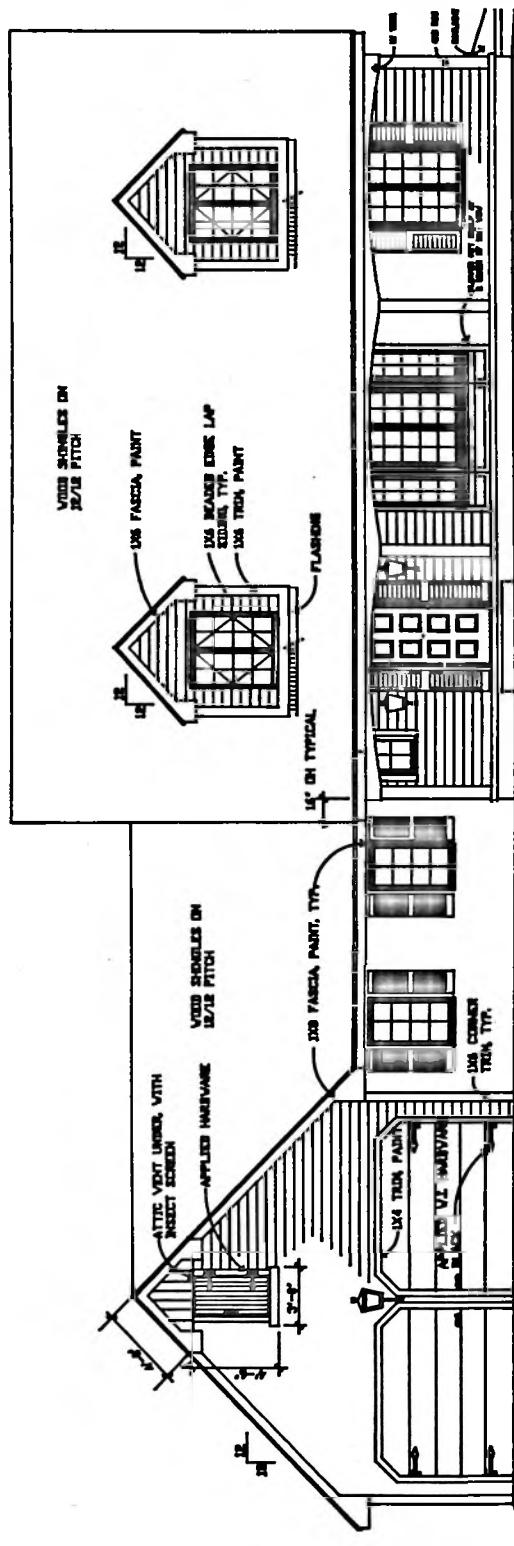
Remember again that SKETCHing with a small record increment may generate a phenomenal number of line entities, and this can rapidly consume free space on the drawing disk. When tracing a complex drawing, it is wise to leave the SKETCH command from time to time and check STATUS to make sure you're not about to run out.

12.5.6 Example

Portions of the drawing on the following page were created using SKETCH mode.



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Chapter 13

PLOTTING

You can plot your AutoCAD drawing on either a *pen plotter* or a printer with graphics capability (hereafter referred to as a *printer plotter*). You can configure AutoCAD for one of each of these devices and use whichever one seems more appropriate for a particular plot. Pen plotters are very accurate and many can plot on large paper sizes or in multiple colors. In general, printer plotters have limited resolution and smaller paper sizes, and most produce monochrome output. Printer plotters are usually faster than pen plotters.

In this chapter, the terms *plot* and *plotter* are used in a generic sense, referring either to a pen plotter or a printer plotter. When a distinction must be made, the type of device is indicated explicitly.

You can initiate a plot from AutoCAD's Main Menu or from the Drawing Editor. The task number or command indicates which type of device you wish to use:

- o For a pen plotter, use Main Menu Task 3 or the PLOT command.
- o For a printer plotter, use Main Menu Task 4 or the PRPLOT command.

The difference between plotting from the Main Menu and plotting from the Drawing Editor is that the Main Menu tasks ask for the name of a drawing to plot, whereas the PLOT and PRPLOT commands plot the current drawing. When the plot is finished, AutoCAD returns to where you were when the plot was initiated (either the Main Menu or the Drawing Editor's "Command:" prompt).

You can use CTRL C to abort the plot at any time. Some plotters have large data buffers, however, and do not stop drawing immediately when you enter the CTRL C.

The first thing you have to tell AutoCAD is which portion of the drawing to plot. When you begin a plot from the Main Menu, AutoCAD prompts for this information as follows:

Specify the part of the drawing to be plotted by entering:
Display, Extents, Limits, View, or Window <D>:

If you initiate the plot by means of a PLOT or PRPLOT command, the prompt is similar:

What to plot -- Display, Extents, Limits, View, or Window <D>:

Your response will specify a rectangular area of the drawing. The various options are described below. Each can be abbreviated to one letter. The option you select will be remembered and used as the default the next time you plot. The first time you plot, the default is D (for Display).

- | | |
|-------------|--|
| D (Display) | For the PLOT and PRPLOT commands, the Display option requests a plot of the view that is currently on the graphics monitor. When plotting a specified drawing from the Main Menu, the Display option plots the view that was displayed on the monitor just prior to the last SAVE, END, or ENDSV command for that drawing. |
|-------------|--|

- E (Extents)** The Extents option is similar to "ZOOM Extents". The plot will consist of that portion of the drawing which currently contains entities.
- L (Limits)** The "Limits" option will plot the entire drawing area as defined by the drawing limits.
- V (View)** The View option is used to plot a view that you previously saved using the Drawing Editor's VIEW command. This option is applicable only if the ADE-2 package is present and the drawing contains named views. When you choose this option, AutoCAD prompts:

View name:

Enter the name of the particular view you wish to plot.

- W (Window)** Using the Window option, you can plot any portion of your drawing by specifying a lower left corner and an upper right corner of the plot "window". AutoCAD prompts:

First point:

Second point:

If you are initiating the plot from the Drawing Editor (using the PLOT or PRPLOT command), your drawing is still on the monitor at this point and you can use your pointing device to designate the desired window if that area is totally visible on the screen. If the area you want is not on-screen or if you are plotting from the Main Menu, you can enter the two requested points from the keyboard, in drawing units.

Once you have chosen the portion of the drawing to be plotted, AutoCAD will display the basic plot specifications and ask if you want to change any of them. For example:

Sizes are in Inches
Plotting area is 15.75 wide by 11.20 high (MAX size)
Plot origin is at (0.00,0.00)
Pen width is 0.010
Area fill will be adjusted for pen width
Plot will be scaled to fit available area

Do you want to change anything? <N>

These specifications were initially set when you configured AutoCAD for this device (see Appendix D). You can use the stored values by responding "N" to the prompt or by pressing RETURN.

If you respond "Y" to the prompt, you can change any or all of the basic plot specifications. But if your plotter supports multiple pens, hardware line types, or software-controlled pen speeds, AutoCAD first asks if you want to modify the specifications of these things. A list of their current values is displayed, as shown below.

Layer	Pen No.	Line Type	Pen Speed	Layer	Pen No.	Line Type	Pen Speed
Color				Color			
1 (red)	1	0	38	9	1	0	38
2 (yellow)	2	0	38	10	1	0	38
3 (green)	3	0	38	11	1	0	38
4 (cyan)	4	0	38	12	1	0	38
5 (blue)	5	0	38	13	1	0	38
6 (magenta)	6	0	38	14	1	0	38
7 (white)	7	0	38	15	1	0	38
8	8	0	38				

Line types 0 = continuous line
 1 =
 2 =
 3 = -----
 4 = - - - - -

Do you want to change any of these parameters? <N>

As described in Chapter 7, each layer of a drawing has a color associated with it. Each color may be plotted with a different plotter line type, pen, and pen speed. But note that all colors greater than 15 plot the same as color 1.

If your plotter supports multiple line types, the "Line types" display shows what patterns are available. Don't try to mix these plotter-generated line types with the linetypes associated with layers of the drawing. If a layer has anything but AutoCAD's CONTINUOUS line type, use only the plotter's normal (solid) line type for it.

For plotters with software-selectable pen speeds, AutoCAD normally uses the fastest speed. You can choose a slower speed on a pen-by-pen basis. This can be useful if, for example, you have one pen that skips if you move it too fast.

A feature of AutoCAD's Configurator makes it possible to assign different pen numbers to different layer colors even for single-pen plotters. If you choose this option when configuring the plotter (see Appendix D), AutoCAD will pause each time a new pen is needed and prompt you to change the pen.

13.1 Changing Pen and Line Type Parameters

If you answer "Y" to the question "Do you want to change any of these parameters?", AutoCAD will prompt:

Enter values. blank=Next value, Cn=Color n, S>Show current values, X=Exit

Layer	Pen No.	Line Type	Pen Speed
Color			

(current values for this color)

(parameter to change)

AutoCAD begins by showing the pen number, line type, and pen speed currently assigned to color 1 and asks first for a new pen number. After you enter a pen number, AutoCAD asks for a line type and then a pen speed for color 1. It then proceeds to color 2 and goes through the same sequence of pen number, line type, and pen speed. At every step, the "parameter to change" prompt shows the current value; you can retain that value and proceed to the next parameter by pressing space or RETURN. To change a parameter, enter the new value; the

"current values" fields to the left are then updated to include the new value. This dialogue continues cycling through the different colors until you end by entering an "X". In addition, you may skip from one color directly to another without answering all the pen number, line type, and pen speed questions. We'll describe how below.

As indicated in the display, there are several ways of responding to the "Pen number:", "Line type:", and "Pen speed:" prompts:

- Blank Retains the current value shown in corner brackets, "<>", and proceeds to the next parameter. RETURN does the same thing. When the last value for a particular color has been reached, AutoCAD proceeds to the next color. After color 15, AutoCAD cycles back to color 1.
- Cn Proceeds directly to the "Pen number" prompt for color number "n". If you enter a "C" without a number, you proceed directly to the next color. This is useful for selective changes. For example, to change the pen speed for color 5, enter "C5" to move to the "Pen number" prompt for color 5. Then enter two RETURNS to skip past the "Pen number" and "Line type" prompts.
- S Displays the updated table of assignments to all colors, then returns to the current prompt.
- X Exits from the dialogue. Enter an "X" when you are satisfied with the line types, pen numbers, and pen speeds you have selected.

When you change the value for a particular parameter (e.g., pen number), it normally changes only for the current color in the sequence. If you want to apply the change to the current color and all following colors, precede the new value with an asterisk ("*"). For example, if AutoCAD is presently asking you about the pen number for color 4, you can select pen number 2 for colors 4 through 15 by entering "*2".

Here's a sample dialogue:

Enter values. blank=Next value, Cn=Color n, S>Show current values, X=Exit

Layer Color	Pen No.	Line Type	Pen Speed	
1 (red)	1	0	32	Pen number <1>: <u>RETURN</u>
1 (red)	1	0	32	Line type <0>: <u>3</u>
1 (red)	1	3	32	Pen speed <32>: <u>*16</u>
2 (yellow)	2	0	16	Pen number <2>: <u>C6</u>
6 (magenta)	3	0	16	Pen number <3>: <u>5</u>
6 (magenta)	5	0	16	Line type <0>: <u>X</u>

Remember, if you tell AutoCAD that you want to change the pen and line type parameters, you must enter "X" to resume the normal plot dialogue.

13.2 Changing Basic Plot Specifications

When you have skipped or completed the pen and line type dialogue, AutoCAD will ask you about the basic plot specifications, such as "size units".

Size Units

AutoCAD allows you to choose either inches or millimeters as the units to be used for all plot size specifications. When the following prompt appears:

Size units (Inches or Millimeters) <current>:

enter "I" if you want to give plot sizes in inches, or "M" if you want to use millimeters. The current choice is shown in corner brackets; you can retain it by entering blank or RETURN.

Plot Origin

The next prompt is:

Plot origin in units <default X,Y>:

where "units" are "Inches" or "Millimeters". For a pen plotter the plot normally begins in the lower left corner of the paper (the plotter's "home" position). For a printer plotter the plot begins in the upper left corner of the paper. You can place the plot origin at another location on the paper by entering the plotter coordinates of the desired origin point, using the "size units" (inches or millimeters) that you selected above. For example, if you have selected inches as your plot size units, entering "2,3" would set the plot origin to the point 2 inches to the right of and 3 inches above the home position. For a printer plotter, the "2,3" origin specification would set the plot origin 2 inches to the right and 3 inches down from the upper left corner of the paper.

Plotting Size

Next, AutoCAD lists the plotting sizes that your plotter can accommodate. If any of these sizes match a standard ANSI (if Inches) or DIN (if Millimeters) size, the mnemonic for that size is listed. Also included in the list is a special "MAX" size. This is the maximum size that your plotter can handle; it may be larger than the largest of the standard sizes. If you have set the size to something other than a standard size or MAX, that size will appear in the table with the label "USER". For example:

Standard values for plotting size

Size	Width	Height
A	10.50	8.00
B	16.00	10.00
C	21.00	16.00
D	33.00	21.00
E	43.00	33.00
MAX	44.72	35.31
USER	8.00	11.00

Enter the Size or Width,Height (in units) <default>:

You can select one of the standard sizes from the list (or "MAX" or "USER" if shown) by typing in the mnemonic listed in the "Size" column. Alternatively, you can enter an explicit paper width and height separated by a comma (in Inches or Millimeters, whichever you have selected as the "size units"). For example:

Enter the Size or Width,Height (in Inches) <MAX>: 8.5,11

Special Notes

1. The plotting size is measured from the plot origin. Setting a plot origin other than (0,0) effectively creates a blank margin on the bottom and left edges of the paper, reducing the area left to plot. This reduction is reflected in the table of plotting sizes AutoCAD displays.
2. For a printer plotter, you cannot specify a size larger than MAX. For a pen plotter, you may specify a larger size; AutoCAD will print a warning that you have done so. You will have an opportunity to cancel the plot with CTRL C in case this larger size was entered by accident.
3. Occasionally, a plotter model is unknown to AutoCAD but is sufficiently similar to a known model that you can use it with AutoCAD. To do this, tell AutoCAD's Configurator that your plotter is actually the known model. If your plotter is capable of larger plots than the model AutoCAD thinks you are using, it may be useful to specify a plotting size larger than what AutoCAD thinks is the maximum.
4. There is an absolute maximum size that AutoCAD will not allow the plot to exceed (technically, 65535 plotter steps). If you try to exceed this size, AutoCAD will display a warning message and truncate the plot.

Plot Rotation

Next, AutoCAD prompts:

Rotate 2D plots 90 degrees clockwise? <N>

If you answer "Y" to this question, a 2D plot will be rotated 90 degrees on the paper. This will move the point that would have been at the lower left corner of the plot to the upper left corner, and the point that would have been at the upper left corner to the upper right corner.

Pen Width

AutoCAD also needs to know the width of the pen tip to be used. This governs how much work AutoCAD must do to "fill" a Solid, Trace, or wide Polyline. During the plot specifications dialogue AutoCAD will prompt:

Pen width <*default*>:

You can enter a new value in the size units previously specified (inches or millimeters). If you would prefer to retain the default value, reply with a blank or RETURN.

Area Fill Adjustment

The next prompt in the sequence is:

Adjust area fill boundaries for pen width? <N>

If you respond "Y", the plot routine will adjust for the pen width when plotting solid-filled Traces, Solids, and wide Polylines. That is, the boundaries of the filled region will be pulled inward one-half pen width. This option is only needed in applications where the plot must be accurate to half of the pen width, such as printed circuit artwork. For most applications an "N" response is more appropriate, so this is the default.

Plot Scale

Next, AutoCAD asks for the plot scale. The prompt is:

Specify scale by entering:

Plotted units=Drawing units or Fit or ? <default>:

where "units" are whatever you previously chose for the "size units" (Inches or Millimeters). As indicated, you can respond in one of three ways. If you enter "Fit" (or simply "F"), AutoCAD will scale the plot so the view you have chosen (the portion of the drawing to be plotted) is made as large as possible for the specified paper size.

If you want to set an explicit scale for the plot, you can do so by telling AutoCAD how many drawing units are to be plotted per inch (or per millimeter) on the paper. Just enter the number of plotter units and the number of drawing units, separated by an equal sign ("="). For example:

Specify scale by entering:

Plotted Inches=Drawing units or Fit or ? <F>: 1=1

would produce a plot at a scale of 1 drawing unit per inch on the plotter. Now suppose that your drawing units represent kilometers and your plotter "size units" are millimeters. Then:

Specify scale by entering:

Plotted Millimeters=Drawing units or Fit or ? <F>: 2.5=1

would produce a plot in which 2.5 millimeters represented a kilometer.

If you have used the ADE-I package's UNITS command to select "feet and inches" mode for your drawing and have selected inches as your plot "size units", you can enter the plot scale in terms of feet and inches. For example, suppose you want to produce a plot where one quarter inch on the paper is equivalent to 1 foot. You would enter:

1/4'=1'

If you do not have the feet and inches feature, the same plot could be produced by entering:

0.25=12

(assuming that your drawing units represent inches). A "?" response to the plot scale prompt results in a short description of these options.

Special Notes

The part of your drawing you selected to plot won't always be fitted exactly to the plot area on the paper, specified by the plotting size.

1. If the scale specification was by "Fit", only one of the dimensions of the drawing area, horizontal or vertical, will be fitted to the corresponding dimension of the plot area. Blank space will be left along the top or right edge of the plot area unless the drawing area has the same shape as the plot area.
2. If you specify an explicit plot scale, the selected drawing area may map into an area either larger or smaller than the plot area. If larger, it will be truncated at the bounds of the plot area. If smaller, some space in the plot area will be left unused.

13.3 Saving Plot Specifications

The current plot specifications are remembered in AutoCAD's configuration file, ACAD.CFG. Whenever you change plot parameters, the new values are stored so you don't have to re-enter them every time you plot something.

13.4 Readying the Plotter

When all the plot specifications are set to your satisfaction, AutoCAD displays the message:

Effective plotting area: *ww* wide by *hh* high

where "*ww*" and "*hh*" are measurements in the current "size units" (inches or millimeters). It then pauses to permit you to load paper and pens in the plotter, prompting with:

Position paper in plotter.

Press RETURN to continue or S to Stop for hardware setup.

If you respond to this prompt with RETURN, AutoCAD will send a "reset" function to the plotter and begin plotting.

Some plotters have additional features, such as selectable pen pressure and acceleration, that are not specifically addressed by AutoCAD's plot specifications. It is usually possible to enter the desired values at the plotter's control panel (following the instructions provided by the plotter manufacturer), but the "reset" function sent by AutoCAD often clears these settings. Therefore, if you reply to the above prompt with an "S", AutoCAD will pause after sending the "reset" function and prompt:

Do hardware setup now.

Press RETURN to continue:

Now you can make whatever adjustments are needed at the plotter before beginning the plot. To begin the plot, press RETURN on the computer's keyboard.

13.5 Multi-Pen Plotting with a Single-Pen Plotter

If you have a single-pen plotter and have selected the configuration option to plot different colors with different pens (as described in Appendix D), AutoCAD will pause when necessary during the plot and issue a prompt like:

Install pen number 2, color 3 (green)

Press RETURN to continue:

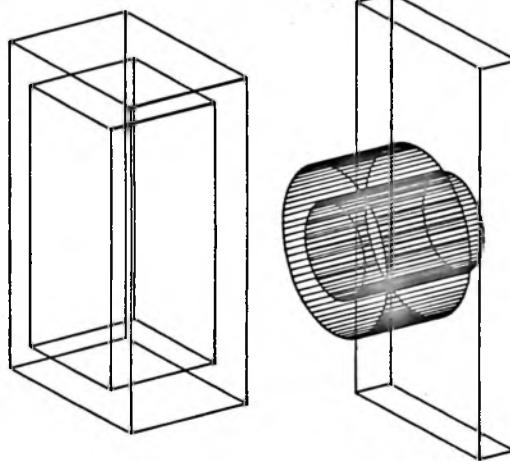
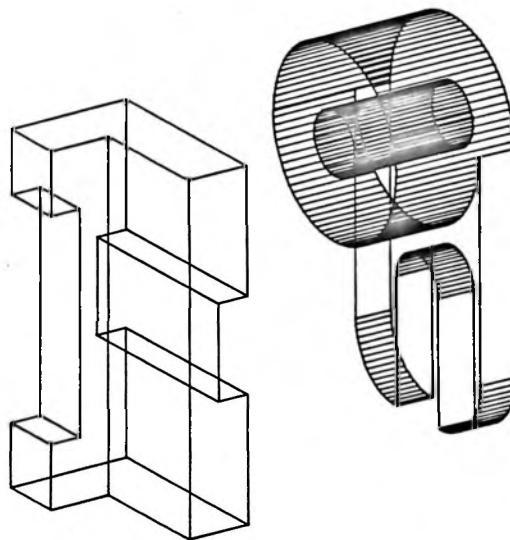
You should wait for the plotter to stop, and change the pen accordingly. Then press RETURN on the computer's keyboard to tell AutoCAD to resume plotting.

13.6 Single-Port Plotting

If your pointing device and pen plotter are connected to different communication ports on your computer, AutoCAD can communicate with either of them whenever needed. However, if your computer has only one port, that port can be used for both devices. Before plotting, you must attach the pen plotter to the port. When the plot is complete, AutoCAD pauses to allow you to reconnect the pointing device.

The single-port technique described above will work for a pen plotter, but cannot be used with a printer plotter. AutoCAD expects the printer plotter to be your system's print or list device, and simply sends output to that device via the operating system. The appropriate communication modes for the printer port are assumed to be set outside AutoCAD -- connection of a pointing device to the same port would probably garble communication with the printer plotter.

3D wireframe Example



Chapter 14

3D LEVEL 1 (ADE-3 FEATURE)

14.1 Introduction

AutoCAD's 3D Level 1™ facility (part of the optional ADE-3 package) provides the commands necessary to permit you to visualize your drawings from any view point in 3 space. From your selected view point, you can add new entities or delete and edit the entities you see. In addition, you can place entities on differing *Z planes* and assign *extrusion thicknesses* to them. These capabilities have been implemented in a way that doesn't interfere with AutoCAD's normal operation. Most of the regular AutoCAD commands support 3D visualizations while retaining their normal 2D prompt sequence. Only three commands are specific to 3D; these commands and the special effects of other commands when in 3D mode are described in the following sections.

14.2 Special 3D Commands

You can use the commands described in this section to assign a base *Z* plane and an extrusion thickness to objects as you draw them, to select a point from which to view your drawing, and to request suppression of hidden lines.

14.2.1 ELEV Command - Set Current Elevation

The ELEV command allows you to set the current *elevation* and *extrusion thickness* for subsequent entities you draw. The current elevation is also assumed any time a *Z* coordinate is required as input to a command. The prompt sequence is:

Command: **ELEV**
New current elevation <*current*>:
New current thickness <*current*>:

The elevation of an object is the *Z plane* on which its base is drawn. An elevation of zero indicates the "base" *Z* plane. Positive elevations are above the base, and negative elevations are below it.

An object's thickness is the distance that object is to be extruded above its elevation. Zero thickness means no extrusion. Negative thickness extrudes downward; an object with elevation 0.00 and a thickness of -1 unit will appear identical to an object with an elevation of -1.00 and a thickness of 1 unit. Extrusion is always performed uniformly on an object; a single entity cannot have different elevations and thicknesses for its various points.

For example, suppose we want to draw an upright cylinder at zero elevation with a radius of 0.5 units and a height of 3 units, and enclose it in a rectangular box (at zero elevation with a thickness of 1 unit). We could use the following command sequence:

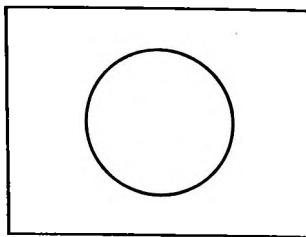
Command: **ELEV**
New current elevation <0.0000>: **RETURN** (*to use default*)
New current thickness <0.0000>: **3**

Command: CIRCLE
3P/2P/<Center point>: (designate center point)
Diameter/<Radius>: 0.5

Command: ELEV
New current elevation <0.0000>: RETURN (to use default)
New current thickness <3.0000>: 1

Command: LINE
From point: (draw rectangle around circle)
...
To point: C (to close the rectangle)

In the normal 2D "top" or "plan" view, this would appear on the screen as:



If you now use the VPOINT command (described below), you can view this figure from whatever angle you like.

To draw a vertical line in 3 space, apply an extrusion thickness to a Point entity.

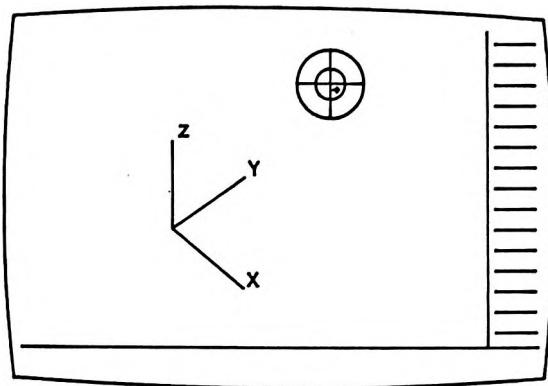
14.2.2 VPOINT Command - Select 3D View Point

The VPOINT command sets the view point from which you want to see your drawing. Once you specify the view point, AutoCAD regenerates the drawing, projecting the entities so they appear as you would see them from that view point. Your location is maintained until you use the VPOINT command again or use the "VIEW Restore" or "ZOOM Previous" commands, either of which will restore the view point from a previous view of the drawing.

To change your view point, enter the VPOINT command.

Command: VPOINT
Enter view point <current X, Y, Z view point>:

The current view point is displayed in corner brackets. You can enter new values for the X, Y, and Z components of the desired view point (separated by commas) or you can use a null response (RETURN) to request a *compass* and *axes tripod* to be displayed. The following diagram illustrates these.



The compass, in the upper right of the screen, is a two-dimensional representation of a globe. The center point is the north pole $(0, 0, 1)$; the inner ring is the equator $(\pi, \pi, 0)$; and the entire outer ring is the south pole $(0, 0, -1)$. A small crosshair is displayed on the compass. By moving your pointing device, you can direct this crosshair to any portion of the globe. As you do so, the axes tripod rotates to conform to the view point indicated on the compass.

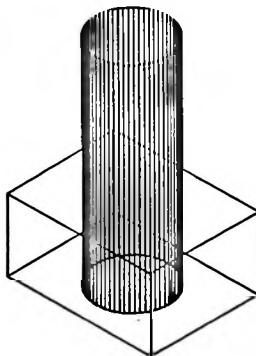
If you like, you can think of the pointing device as merely manipulating the axes tripod; some people prefer the compass, some like the axes tripod, and some use both. When you achieve the desired view point, press the "pick" button on your pointing device.

The compass and axes tripod in the above illustration indicate a "top, right, front" view. That is, the view point is from the top (northern hemisphere of the globe), right (eastern hemisphere), and front (bottom half of the compass). An X, Y, Z specification of " $1, -1, 1$ " would produce a similar view point.

If you press RETURN to obtain the compass and axes tripod but then decide to use the default view point that was displayed in corner brackets, simply press RETURN again.

Regardless of the view point you select, you are always "facing" the coordinate system's origin point, $(0, 0, 0)$. For purposes of specifying your view point, you can think of the origin as housing all the entities you have drawn. Note that you are only specifying a view direction. It is not possible to specify a view distance. You can ZOOM in on details, but the entities are always displayed using a *parallel projection*; perspective views are not generated.

With a view point of $(1, -1, 1)$ -- a "top, right, front" view -- our earlier example of a cylinder in a box would appear as shown below.



To restore the display to its normal 2D appearance (the "top" or "plan" view), specify a view point of $"0, 0, 1"$. For example:

Command: VPOINT
Enter view point <1.875, -4.227, 0.573>: 0.0.1

14.2.3 HIDE Command - Hidden Line Suppression

When you use the VPOINT command to generate a 3D visualization, a *wire-frame* display is produced. All lines are present, including those that would be hidden by other objects given the view point you have chosen. If you want to eliminate those "hidden" lines, you can use the HIDE command. This command regenerates your drawing with hidden lines suppressed.

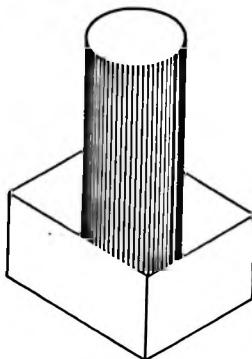
To use the HIDE command, simply enter:

Command: HIDE

There are no prompts to be answered. The screen will go blank for a period of time, depending on the complexity of your drawing. The drawing will then be redrawn with the hidden lines removed. The display will return to normal (wire-frame display) the next time the drawing is regenerated (ZOOM, PAN, REGEN, VPOINT, etc.).

The HIDE command treats some drawing entities specially. Circles, Solids, Traces, and wide Polyline segments are treated as solid objects, with top and bottom faces. If you want an open cylinder without a top or bottom, construct it from two Arcs.

Returning to our example of a cylinder in a box and retaining the view point of $(1, -1, 1)$ illustrated earlier, the HIDE command would produce the following display:



You can instruct the HIDE command to display the hidden lines in a different color rather than make them invisible. To do this, use the LAYER command to create layers with names identical to those of the layers in your drawing but with the prefix "HIDDEN" added. For example, if your drawing has layers named "A" and "B", create new layers named "HIDDENA" and "HIDDENB". When the HIDE command finds that all or part of an entity should be hidden, it tries to draw the hidden portion on layer "HIDDENxxx", where "xxx" is the layer on which the entity has been drawn. If no "HIDDENxxx" layer is found, the hidden portion of the entity will be invisible.

Since you can control the visibility and the color of each layer individually, this provides a great deal of flexibility in the handling of hidden lines.

14.3 Effects of 3D on Other Commands

This section outlines the effects of 3D visualizations on other AutoCAD commands and describes any additional capabilities of those commands while in 3D mode.

- | | |
|---------------|---|
| ATTDEF | Attribute Definitions are assigned a zero thickness, regardless of the current thickness set by the ELEV command. You can change their thickness afterward by means of the CHANGE command. |
| AXIS | The axis ruler lines are only displayed when the view point is the "top" or "plan" view, (0, 0, 1). |
| BASE | The BASE command uses the current elevation (set by the ELEV command) as the Z base coordinate for subsequent INSERTs of the drawing. |
| BLOCK | The current elevation (set by the ELEV command) is used as the Z base coordinate for the Block. |
| CHANGE | The "E" option on the CHANGE command allows you to change the elevation and thickness of existing entities. |
| GRID | The visible grid is projected onto a 3D view according to the view point and the current elevation. This is in addition to the standard/isometric, rotation, and aspect transformations permitted for the grid. |

ID	The ID command displays the current elevation. For all practical purposes, this is the Z component of the selected point.
INSERT	The Block is inserted at the current elevation. All entities associated with the Block are offset from that elevation by the appropriate Z amount, just like the other dimensions.
	By default, INSERT sets the Z scale for the inserted Block equal to the X scale, but a method is provided to let you enter an explicit Z scale. After you specify the insertion point for the Block, INSERT prompts:
	X scale factor <1> / Corner / XYZ:
	You can respond "XYZ", to let INSERT know that you want to specify all three scale factors. The prompt sequence then becomes:
	X scale factor <1> / Corner: Y scale factor <default=X>: Z scale factor <default=X>:
LIST	The LIST command displays the elevation and thickness of each entity. The same is true for the DBLIST command.
OSNAP	Objects must reside on the current elevation to become candidates for object snap point selection.
PLOT	See "3D Plotting", below.
PRPLOT	See "3D Plotting", below.
SKETCH	Line segments produced by the SKETCH command are not extruded until the "Record" option is selected.
STATUS	The current elevation and current thickness are included in the STATUS command's report.
TEXT	Text entities are assigned a zero thickness, regardless of the current thickness set by the ELEV command. You can change their thickness afterward by means of the CHANGE command.
VIEW	The VIEW command saves and restores the view point along with the other view parameters.
WBLOCK	The current elevation (set by the ELEV command) is used as the Z base coordinate for the Block.
ZOOM P	The "ZOOM Previous" command restores the previous view point as well as the previous display window.

In addition, solid-filling of Traces, Solids, and wide Polylines occurs only when the view point is "top" or "plan" (0, 0, 1), and hidden lines are NOT removed.

14.4 3D Plotting

You can plot a 3D visualization by using the normal PLOT and PRPLOT commands or by means of the Main Menu tasks devoted to plotting.

When AutoCAD asks for the portion of the drawing to be plotted (Display, Extents, Limits, View, or Window), respond in the usual manner. For the "View" option, the plot will be from the view point specified by the view you name. For the other options, the plot will be from the view point most recently established for this drawing in the Drawing Editor.

The "Limits" option is handled in a slightly different manner when the view point is not from the "top" (0, 0, 1). In this case, "Limits" plots exactly the same thing as the "Extents" option: all entities are scaled to fit the plotting area.

One additional prompt is displayed, both at plot time and during plotter configuration:

Remove hidden lines? <default>:

Reply with either "Y" or "N", or press RETURN to accept the default response shown in corner brackets. If you reply "Y", hidden lines will not be plotted; otherwise they will.

14.5 Tips on Use of the HIDE Command

There are a number of practical guidelines that you should follow to get realistic drawings with the HIDE command in AutoCAD 3D Level 1. Generally, you will use the HIDE command only occasionally during the creation or editing of a drawing. While working on a drawing, you will usually see the wire-frame representation and will only perform the HIDE command before plotting. The following suggestions and explanations are designed to help you make the best use of the HIDE command.

Wire-Frame versus Hidden Line Drawings

A wire-frame visualization of a drawing is frequently all that is needed to capture and convey a 3D image. Drawings that are composed of visually simple shapes, and drawings that do not have many object faces behind other faces often provide good representations without hidden line suppression. For such drawings, it may be easier to ignore the subtleties of generating correct hidden line representations.

On the other hand, many drawings appear much too cluttered or complex to convey useful information when viewed or plotted in wire-frame. Drawings of objects with many rounded edges, angles other than 90 degrees, or faces that in the real world would obscure other parts of the drawing, are best presented with hidden lines suppressed.

Tops and Bottoms

The main guideline in using the HIDE command is to create the objects in the drawing with entities that represent real, three-dimensional solids. As a simple example, a cube should be drawn as an extruded Solid, Trace, or wide Polyline segment. Although the wire-frame representation of a cube drawn with four extruded lines forming a square may look identical to a cube formed from an extruded Trace, the HIDE command would treat the two very differently.

A cube drawn by extruding four lines would be treated by the HIDE command as a square tube, or "milk carton", with no top or bottom, whereas a cube formed from an extruded Solid is treated as a solid object, with four side faces, a top face, and a bottom face.

The 3D Level 1 facility puts top and bottom surfaces on some extruded entities and not on others. In general, if the two dimensional entity being extruded encloses a planar region, then the extruded object has a top and a bottom surface and is treated as a solid object by the HIDE command. If the one or two dimensional entity is not closed, then its extrusion is considered to be a zero thickness vertical sheet by the HIDE command, and objects composed of such entities will lack tops and bottoms.

Solids, Traces, Circles and wide Polyline segments are the entities that have tops and bottoms when extruded. Each of these forms the perimeter of a two dimensional region and is treated as a three-dimensional solid object when extruded. A Solid or a Trace becomes a three- or four-faced prism. A Circle becomes a vertical pole (a solid cylinder).

Each segment of a wide Polyline is treated separately by 3D Level 1. Segments with a nonzero width will have tops and bottoms. Segments with zero width are treated exactly like ordinary lines and arcs. Note that no special treatment is given to closed Polylines. In particular, the entire region surrounded by a closed Polyline is *not* a top or a bottom surface to the HIDE command.

All other entities produce one- or two-dimensional surfaces in space. Where such an entity touches any other entity, HIDE may produce incorrect results.

Text, Attributes, and Attribute Definitions

Entities consisting entirely of textual information are not processed by the HIDE command. During the regeneration implicit in a HIDE command, Text, Attribute, and Attribute Definition entities are simply drawn without considering their visibility. It is generally advisable to put textual information on distinct layers. You can suppress the text by turning those layers off with the LAYER command.

Turned-Off and Frozen Layers

AutoCAD treats entities on turned-off layers just like any other entities, except that they are neither displayed nor plotted until the layer is turned on. It is therefore possible for entities that are invisible because they are on a turned-off layer to obscure other entities during a HIDE command. Since entities on frozen layers are ignored during entity generation, you should freeze a layer rather than turn it off when the effect of turned-off layers is undesirable.

"HIDDENxxx" Layers

Although you can make hidden lines visible by defining an appropriate "HIDDENxxx" layer prior to a HIDE command, this should only be done in very limited cases. Almost any drawing that has touching objects, extruded Polyline, extruded Arcs or extruded Circles will display poorly if a corresponding "HIDDENxxx" layer is defined.

Whenever two objects touch, there is a strong possibility that either of them might be drawn as if it were hidden by the other. Without a "HIDDENxxx" layer, this problem is usually not

noticeable. It is analogous to two objects touching in a two-dimensional drawing. On the other hand, defining a "HIDDENxxx" layer while editing a drawing can be a very useful visualization tool to help you see what has been drawn.

Objects that Touch or Intersect

The 3D Level 1 HIDE command is intended to produce credible renderings of real objects. Trying to HIDE lines of objects that cannot be realized in three-space is likely to produce marginal results. The two most common pitfalls are: drawing objects that intersect in space and drawing surfaces that exactly coincide.

It is very easy to inadvertently draw two objects whose boundary surfaces intersect. A trivial example would be two extruded lines that cross one another. To ensure accurate rendering of such objects by the HIDE command, you must remove a small piece of one of the objects where it would otherwise penetrate the second object. Generally, this means breaking one of two intersecting objects into two objects, one on either side of the surface of the intersecting object. This corresponds to cutting a penetrating object in half and attaching half to the outside and half to the inside.

Two objects that touch in space can also cause problems for the HIDE command. Objects can touch at a point, along a line, or along a plane. All three can result in the same problem, although the problem can usually be ignored for point contact and is usually irrelevant for line contact. When two objects touch, one of them will generally be considered to be hiding the other. The problem is that minute round-off errors in calculating the relative positions of two touching objects can result in an incorrect determination of which one covers the other.

Consider a drawing of a small extruded rectangular solid resting on the top surface of a much larger one, like a paperback book lying on a table. A very slight round-off error could make the bottom cover of the book appear to penetrate the table top. Since the HIDE command does not generate intersection lines, this error would result in hiding the bottom cover of the book.

If the intersection is a point, then at worst one point will be incorrectly displayed. In general, a single point not drawn or on the wrong layer is insignificant.

When the intersection is a line, it may or may not be significant. Consider a drawing of two walls meeting at a corner. If hidden lines are being suppressed and the two walls are on the same layer, it does not matter which of the two intersecting edges is hidden. On the other hand, if hidden lines are being drawn on a "HIDDENxxx" layer, then the boundary line where the two walls touch may be drawn as a hidden line, may be drawn as a visible line, or (most likely) both.

There are two possible solutions. Either leave small gaps between objects that would otherwise touch or use caution in defining "HIDDENxxx" layers.

Performance

The HIDE command effectively compares every pair of faces in a drawing to determine whether either wholly or partially obscures the other. The performance of the HIDE command depends on a number of factors. The number of entities, their complexity, their proximity in the flat projection, and their size on the flat projection all contribute to the time required to perform a HIDE command.

To obtain optimal performance from the HIDE command, follow these guidelines:

1. Avoid drawing details that would not be visible at the scale at which the drawing will be displayed or plotted.
2. ZOOM in to the part of the drawing that you really want to see before performing the HIDE command. This removes unwanted entities from consideration by the HIDE command and reduces the proximity of those entities that are visible on the flat projection plane.
3. Avoid large surface entities such as the floor of a facilities planning drawing or the substrate of an integrated circuit.

Appendix A

STANDARD LIBRARIES

This appendix describes the standard menu and prototype drawing supplied with AutoCAD and illustrates the contents of the standard text font, linetype, and hatch pattern (+1) libraries.

A.1 Standard Prototype Drawing

AutoCAD is supplied with a standard prototype drawing named ACAD. This is the default prototype used to establish the environment for each new drawing you create via Main Menu Task 1.

It is helpful to know just what the initial environment established by the ACAD prototype looks like. The following initial modes are set by the ACAD prototype drawing as supplied with the AutoCAD program.

APERTURE	10 pixels
ATT DISP	Normal (controlled individually)
AXIS	Off, spacing (0.0, 0.0)
BASE	Insertion base point (0.0, 0.0)
BLIP MODE	On
CHAMFER	Distance 0.0
DRAG MODE	On
ELEV	Elevation 0.0, thickness 0.0
FILLET radius	0.0
FILL	On
GRID	Off, spacing (0.0, 0.0)
ISOPLANE	Left
LAYER	Current/only layer is "0", On, with color 7 (white) and linetype "CONTINUOUS"
LIMITS	Off, drawing limits (0.0, 0.0) to (12.0, 9.0)
LINETYPE	Only loaded linetype is "CONTINUOUS"
LTS SCALE	1.0
MENU	ACAD
ORTHO	Off
OSNAP	None
PLINE	Line-width 0.0
QTEXT	Off
REGENAUTO	On
SKETCH	Record increment 0.0
SNAP	Off, spacing (1.0, 1.0)
SNAP/GRID	Standard style, base point (0.0, 0.0), rotation 0.0 degrees
STYLE	Only defined text style is "STANDARD", using font file "TXT", with variable height, width factor 1.0, and no special modes.
TABLET	Off
TEXT	Style "STANDARD", height 0.20, rotation 0.0 degrees
TRACE	Width 0.05

AutoCAD -- (A) STANDARD LIBRARIES

UNITS (linear)	Decimal, 4 decimal places.
UNITS (angular)	Decimal degrees, 0 decimal places.
ZOOM	To drawing limits

Of course, you can modify the ACAD prototype drawing to achieve whatever initial conditions you like. To do this, simply edit the ACAD drawing, set the modes you prefer, and save your updated version via the END command.

ACAD is just the *default* prototype drawing. When configuring AutoCAD (Appendix D), you can choose a different prototype drawing to be the default. As described in Chapter 2, you can also specify an explicit prototype when creating a new drawing via Main Menu Task 1. To do this, enter:

Enter NAME of drawing: drawing=prototype

It is also possible to create a new drawing without any prototype:

Enter NAME of drawing: drawing=

In this case, AutoCAD uses default values for all environment settings. As supplied with AutoCAD, the ACAD prototype drawing also sets the default values. In fact, we created the ACAD prototype drawing by responding:

ACAD=

to the "Enter NAME of drawing" prompt and then issuing the END command.

A.2 Standard Menu

AutoCAD is supplied with a standard menu file named ACAD.MNU. The menu file contains sections designed for the screen, tablet, and button menus. Each section is described briefly in the following table.

Screen Menu	The screen menu is comprehensive, including special submenus for most AutoCAD commands.
Tablet Menu	The tablet menu contains one item for each of the basic AutoCAD commands, and is designed for menu area 1 (up to four tablet menu areas may be defined; see Chapter 12).
Button Menu	The button menu contains items for use with the extra buttons, if any, on your pointing device.

You can find further information on the standard menu file in the accompanying AutoCAD Standard Menu User Guide.

A.3 Standard Linetypes

A standard library of linetypes is supplied with AutoCAD, in a file named ACAD.LIN. The linetypes in it are illustrated below.

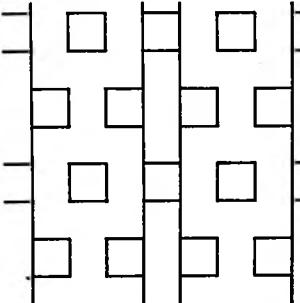
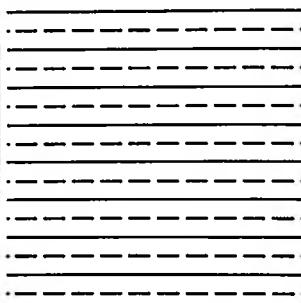
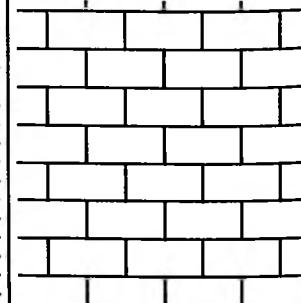
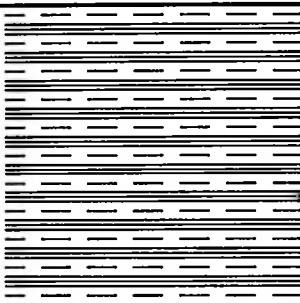
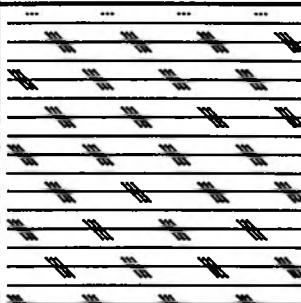
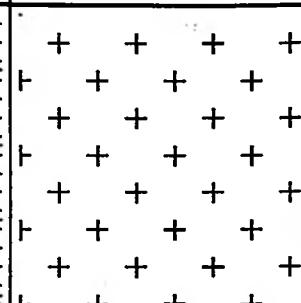
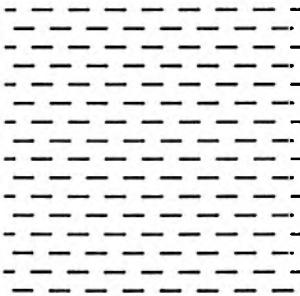
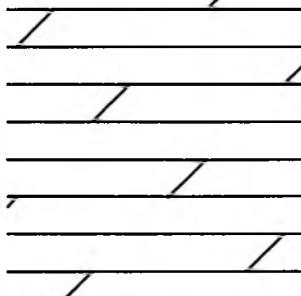
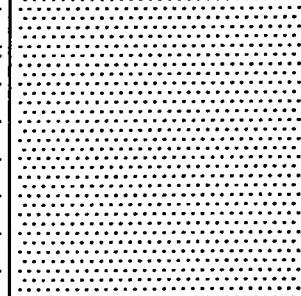
<u>NAME</u>	<u>SAMPLE</u>
Dashed	-----
Hidden	-----
Center	— - - - -
Phantom	— - - - -
Dot
Dashdot	- - - - -
Border	- - - - -
Divide	- - - - -

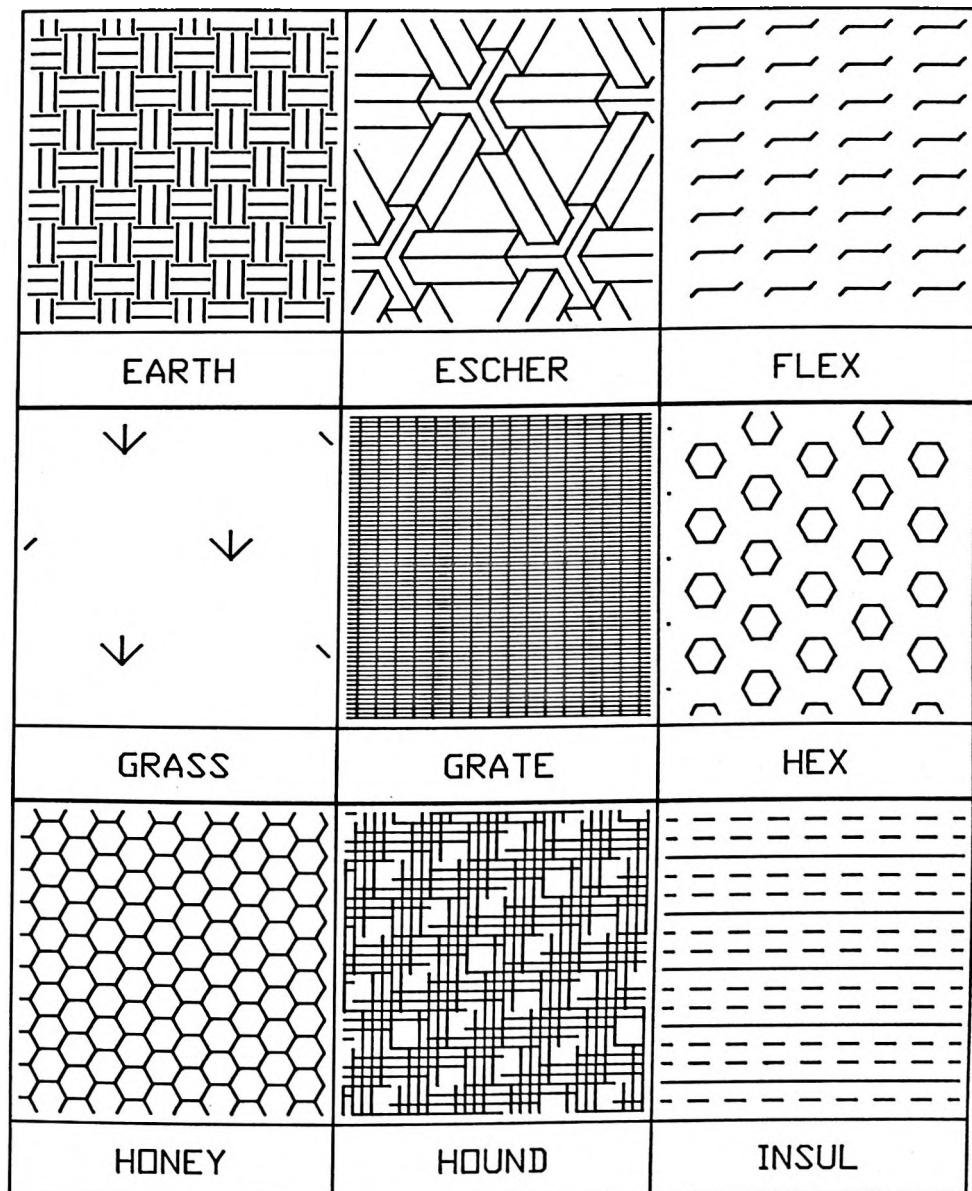
A.4 Standard Hatch Patterns (+1)

The following pages illustrate the standard hatch patterns supplied in file ACAD.PAT for systems with the ADE-1 package.

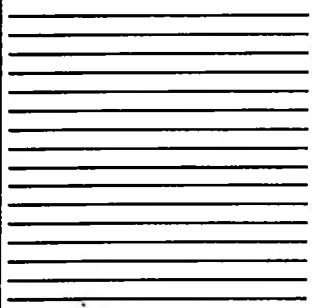
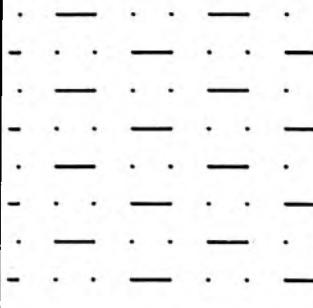
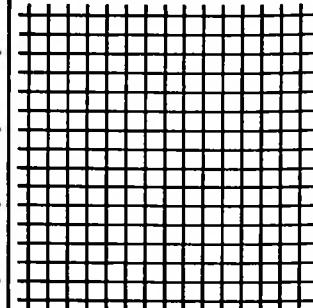
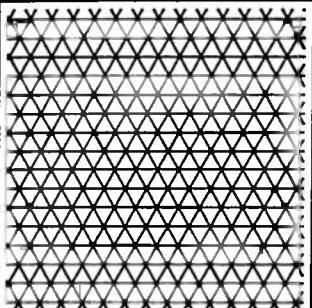
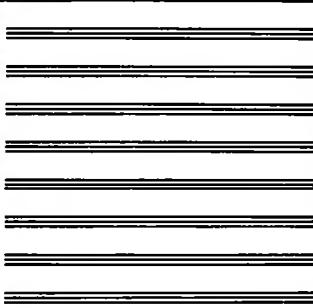
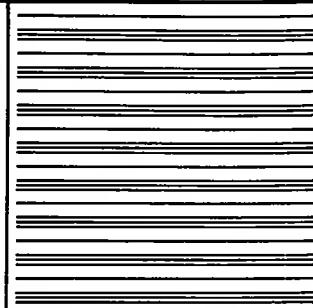
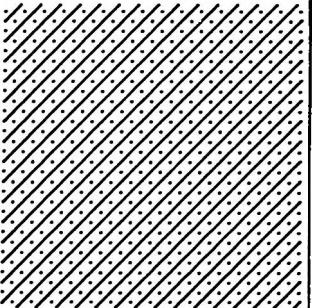
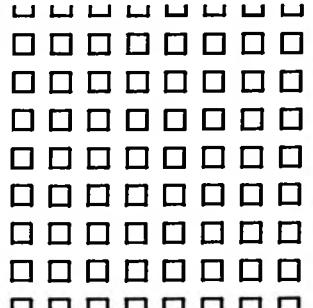
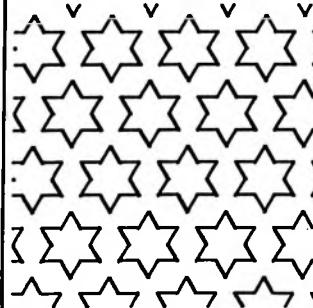
ANGLE	ANSI31	ANSI32
ANSI33	ANSI34	ANSI35
ANSI36	ANSI37	ANSI38

AutoCAD -- (A) STANDARD LIBRARIES

		
BOX	BRASS	BRICK
		
CLAY	CORK	CROSS
		
DASH	DOLMIT	DOTS



AutoCAD -- (A) STANDARD LIBRARIES

		
LINE	MUDST	NET
		
NET3	PLAST	PLASTI
		
SACNCR	SQUARE	STARS

AutoCAD -- (A) STANDARD LIBRARIES

STEEL	SWAMP	TRANS
TRIANG	ZIGZAG	

A.5 Standard Text Fonts

AutoCAD is supplied with several text fonts. You can use the STYLE command to apply expansion, compression, or obliquing to any of these fonts, thereby tailoring the characters to your needs. You can draw characters of any desired height using any of the fonts.

The fonts supplied with AutoCAD are:

- | | |
|----------|--|
| TXT | This is the standard AutoCAD text font. It is very simple and can be drawn very quickly. |
| SIMPLEX | This is a "simplex" Roman font drawn by means of many short line segments, and produces smoother-looking characters than those of the TXT font. |
| COMPLEX | This is a "complex" Roman font with short line segments and multiple strokes, forming smooth characters with varying thickness. |
| ITALIC | This is a complex font, not simply a slanted version of COMPLEX, but a true Italic font. |
| VERTICAL | This font contains the same characters as the simple TXT font, but each character has been "turned on its side" and adjusted so that, if drawn with a rotation angle of 270 degrees, the characters will appear in a vertical column, with each character centered below the previous character. |

Samples of these fonts appear on the following page. (The VERTICAL font is not shown; as noted above, its characters are the same as those in the TXT font.)

Each font resides in a separate disk file with the name .SHX. This is the "compiled" form of the font, for direct use by AutoCAD. Another file named .SHP is supplied as well; this file contains the symbolic description of the font's characters, and is not normally needed by AutoCAD. The ".SHP" files are provided as examples for those users who might want to define their own text fonts. If you wish to do this, see Appendix B for further information.

!"#\$%&'()*+,-./01234567
 89:;<=>?@ABCDEFGHIJKLMNO
 PQRSTU VWXYZ[\]^_`abcdefg
 hijklmnopqrstuvwxyz{|}~°±∅

TXT font

!"#\$%&'()*+,-./01234567
 89:;<=>?@ABCDEFGHIJKLMNO
 PQRSTU VWXYZ[\]^_`abcdefg
 hijklmnopqrstuvwxyz{|}~°±∅

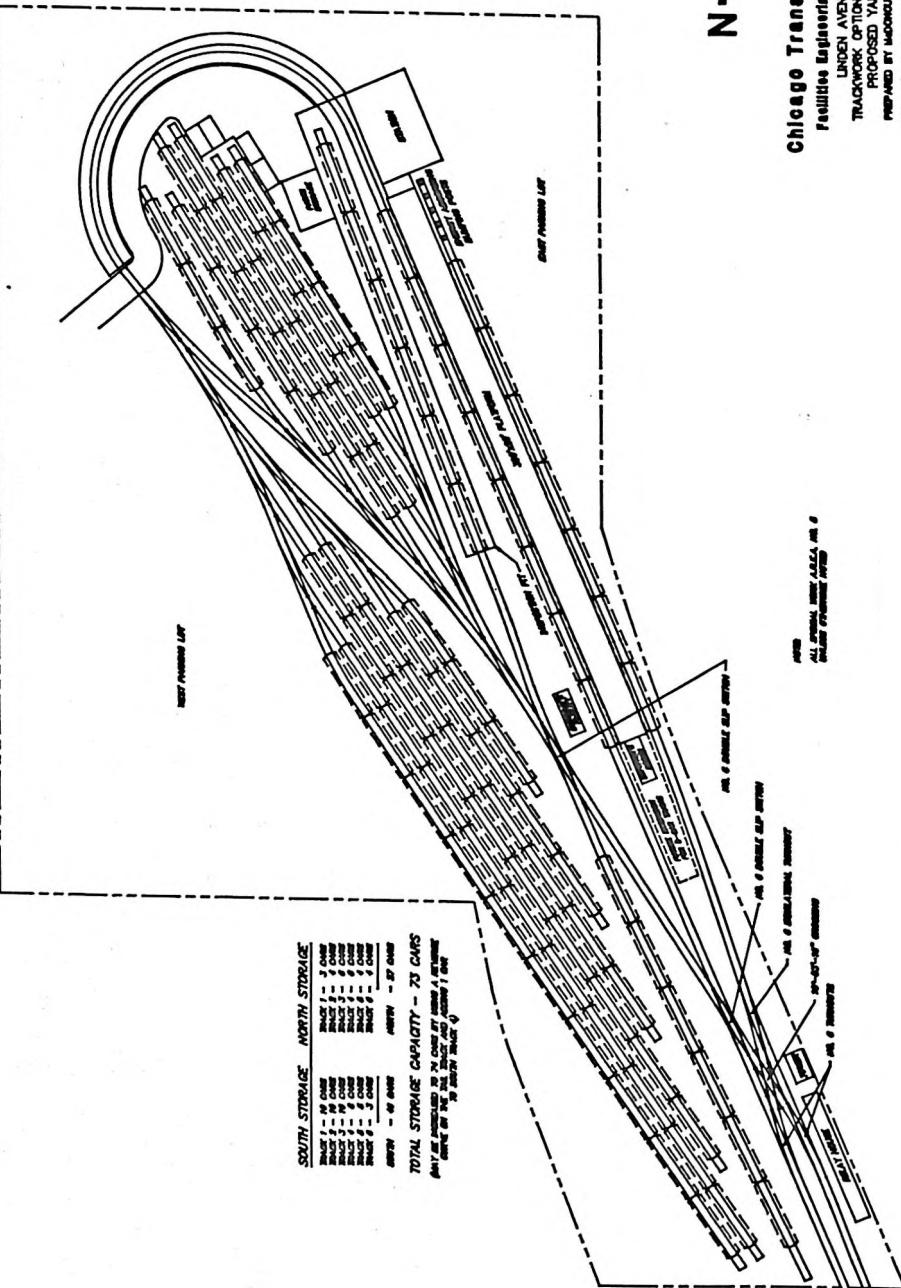
SIMPLEX font

!"#\$%&'()*+,-./01234567
 89:;<=>?@ABCDEFGHIJKLMNO
 PQRSTU VWXYZ[\]^_`abcdefg
 hijklmnopqrstuvwxyz{|}~°±∅

COMPLEX font

!"#\$%&'()*+,-./01234567
 89:;<=>?@ABCDEFGHIJKLMNO
 PQRSTU VWXYZ[\]^_`abcdefg
 hijklmnopqrstuvwxyz{|}~°±∅

ITALIC font



Appendix B

CUSTOMIZING AutoCAD

You can extend AutoCAD's capabilities and customize it for your application by designing your own menus and libraries to complement those supplied with the program. The techniques used to create custom AutoCAD menus and libraries are described in this appendix. Use of multiple file directories, and the format of the HELP text file are also described herein.

Creation of some of the custom libraries requires a fair amount of technical skill. Use the standard libraries (Appendix A) as examples; once you understand how they work, you can devise your own to achieve the results you want.

B.1 Directory Usage

AutoCAD makes full use of the tree-structured file directories available under the MS-DOS or PC-DOS operating systems, level 2.0 or higher. If you aren't familiar with the use of tree-structured directories, you can simply place all AutoCAD program files and auxiliary files (menus, text font files, HELP file, etc.) in the current directory of the current drive, and keep your drawing files there as well.

B.1.1 Maintaining Several Drawing Directories

If you would like to maintain several directories of drawings but only one copy of the AutoCAD program and its auxiliary files, that is also possible. Simply place the entire contents of the AutoCAD release disks (except for the sample drawings disk) in a directory that is on the search path used on your system, defined by the MS-DOS/PC-DOS "PATH" command. Then, whether or not you make that directory the current directory when you execute AutoCAD, all the necessary program and auxiliary files will be available.

For instance, if you have used the following DOS command:

```
PATH C:\BIN;C:\PROGS
```

then you can place the AutoCAD program files in either of two directories, C:\BIN or C:\PROGS, but don't divide the files between the two. Since both directories are on the system's search path, the system and AutoCAD will be able to find the necessary files for proper operation of AutoCAD.

Whenever AutoCAD prompts you for a file name, you can reply with a simple name, a name preceded with a drive letter, or a full path name including directory names, like C:ACADDIR\PARTS\FAUCET. Either forward or backward slashes may be used to separate directory names. If the file name has a drive and/or directory prefix, those prefixes specify explicitly where AutoCAD will look for or place the file.

If you enter a file name with neither drive nor directory prefixes, this usually means a file in the current directory of the currently logged disk drive. This is always true if you are

naming a file AutoCAD is to create (like a drawing file created by the "Create a NEW drawing" task of the Main Menu, or by the WBLOCK or SAVE commands). But if you are specifying an existing file for AutoCAD to use (such as a drawing file for the INSERT command, a menu file for the MENU command, or a text font file for the STYLE command), then AutoCAD will also look in the directory where the AutoCAD program files are located (which we will refer to as the "System Directory") if it doesn't find the file in the current directory. And, if you like, you can set up another directory for AutoCAD to search, after the current directory but before the System Directory. To make AutoCAD look in this other directory, you must use the MS-DOS/PC-DOS "SET" command to create a variable called ACAD, whose value is the name of the directory. For example:

```
SET ACAD=C:\ACaddir\EXTRAS
```

B.1.2 Maintaining Several AutoCAD Configurations

When you run the AutoCAD Configurator (Appendix D), the information you supply (on such matters as device selection and operating parameters) is recorded in the configuration file, ACAD.CFG. When AutoCAD starts execution, it first determines the System Directory (see above), then it determines a Configuration Directory. Normally, these two directories are the same, but you may specify a different Configuration Directory by using the MS-DOS/PC-DOS "SET" command to create a variable called ACADCFG, whose value is the name of the directory. For example:

```
SET ACADCFG=C:\SPECIAL\CFG
```

If AutoCAD doesn't find file ACAD.CFG in the Configuration Directory, it performs an initial configuration, creating ACAD.CFG and placing it in the Configuration Directory. If you execute AutoCAD several times, each time with a different value for the variable ACADCFG, you can create several different configurations of AutoCAD, each in its own directory. For an example of how this facility might be used, suppose you have both a mouse and a tablet you can use with AutoCAD. Rather than reconfiguring every time you want to switch from one to the other, you could use the ACADCFG mechanism to create a mouse configuration in one directory and a digitizer configuration in another. Then in the future you can direct AutoCAD to use one or the other just by doing a "SET" command before executing AutoCAD.

B.2 Custom Menus**B.2.1 General Information**

A menu file is simply a text file with the type ".MNU" containing AutoCAD command strings. Sections of the file can be associated with different menu devices, such as the screen and tablet menus. (See Subsection 12.4.4 for information on allocating menu areas on a tablet). To construct a custom menu, use any text editor or a word processor in "programmer" mode; you don't want the editor to put in extra characters, indentation, etc.

Each menu item may consist of a command, a parameter, or a sequence of commands and parameters. Normally, each menu item resides on one line of the file.

NOTE: If you intend to supply command parameters as part of a menu item, you must be very familiar with the sequence in which that command expects its parameters. Every character in a menu item is significant, even the blank spaces. From time to time as AutoCAD is revised and enhanced, the sequence of prompts for various commands (and sometimes even the command names) may change; your custom menus may require minor changes when you upgrade to a new version of AutoCAD.

A simple command menu might contain the lines:

```
line
ZOOM A
ZOOM W
GRID
ON
GRID ON
GRID .1
SNAP 0.001
```

When you select any of these items from the menu, AutoCAD treats it as though you had typed it directly from the keyboard. For instance, selecting "GRID", then "ON", is the same as typing "GRID ON" or selecting the item that says "GRID ON".

B.2.2 Screen Menu - Item Titles

Only the first eight characters of a menu item can appear on the screen menu; therefore, the "SNAP 0.001" command above would display as "SNAP 0.0". You can give a short title, *at most eight characters*, to an item by enclosing the title in square brackets, [], at the start of the line. If such a title is given, it is displayed in the appropriate screen menu box; the string immediately following the closing bracket is issued whenever that menu item is selected. For instance, consider the menu:

```
[FINE]snap 0.001
[COARSE]snap 0.1
[DONE]end
[GIVE UP]quit
```

Here, "DONE" and "GIVE UP" are simply alternate names for the "END" and "QUIT" commands. "GIVE UP" still asks whether you really want to discard all changes to the drawing; you could circumvent this safety measure by saying "quit y" in the menu (although we do not recommend doing so).

B.2.3 Menu File Section Labels

A menu file can be logically broken into seven sections, identified by section labels. Each section belongs to a different menu device and contains command strings targeted for that device. The seven section labels are:

***SCREEN	- Screen menu area
***BUTTONS	- Pointer device button menu
***TABLET1	- First tablet menu area
***TABLET2	- Second tablet menu area
***TABLET3	- Third tablet menu area
***TABLET4	- Fourth tablet menu area
***AUX1	- Auxiliary "function box" button menu

These labels specify that subsequent menu items, up to the next section label or the end of file, belong to a specific menu device. The following is a short menu file with section labels.

```
***SCREEN
[Help]help
[Bye]end
***TABLET1
line
circle
***BUTTONS
erase
oops
```

In this example, items "Help" and "Bye" belong to the screen menu, items "line" and "circle" belong to the first tablet menu, and "erase" and "oops" belong to the button menu. If the label "****SCREEN" does not appear in the file, AutoCAD acts as if that label preceded the first menu item in the file. If a section label is missing for a particular menu device, that device's items will be the same as those of the screen menu.

B.2.4 Submenus

A menu section can be very large, containing many more items than there are boxes (on the screen or tablet) with which to select them. *Submenus* are smaller groups of menu items within a menu section, which can be activated and made available for selection by the user. For instance, selecting an item called "ZOOM" from the main screen menu might activate a submenu containing the options for the ZOOM command. The submenu items temporarily replace all or part of the current menu. Usually, when you finish using the submenu, you return to the previous menu. Submenus can be nested; we will explain this shortly.

Submenu Definition

A *submenu label* marks the start of a submenu. It has the format:

**name

The "name" is a string up to 31 characters long containing letters, digits, and the special characters "\$" (dollar), "-" (hyphen), and "_" (underscore). It specifies the name of a submenu. The submenu label must reside on a menu file line by itself and must not contain embedded blanks. All submenu names within a file must be unique. Menu items that immediately

follow a submenu label, up to the next submenu label, section label, or end of file, belong to that submenu label. Submenu names defined on consecutive lines all refer to the next set of menu items. You can use this to create submenus with aliases.

A submenu can contain any number of items, but the menu device may limit the number of accessible items. For instance, if a submenu of the screen menu section has 21 items, but the screen can display only 20 items at a time, the last item in the submenu is inaccessible.

When a submenu is activated, its items normally replace those of the previous menu starting at the beginning (for the screen menu, the top box) and continuing through all items of the submenu. Thus, a submenu may replace only a portion of the previous menu. You can add an item number after the section or submenu label to specify replacement starting with an item number other than 1. For example:

**SAMPLE 3

When the SAMPLE submenu is activated, the first two menu boxes are left unchanged and submenu replacement begins with the third menu box. If you specify a negative item number, replacement starts that number of items from the end of the menu (bottom of the screen).

Submenu Reference

You use the following construct within a menu item to activate or deactivate a submenu:

\$section=submenu

where:

section Specifies the menu section. Valid names are:

S	(for the SCREEN menu)
B	(for the BUTTONS menu)
T1 - T4	(for TABLET menus 1 through 4)
A1	(for the AUX1 menu)

submenu Specifies which submenu to activate. The name must be either one of the submenu labels (without the "***") in the currently loaded menu file or a menu section name as defined above.

Examples:

```
$S=PARTS
$T1=EDITCMDS
$T2=SCREEN
```

Before a submenu is activated, the currently activated menu items in the section are copied (pushed) onto a stack. The menu items of the named submenu are then activated. For example, if a menu item issues the command:

```
$S=PARTS
```

the active screen items are pushed onto the screen stack, and the items of the submenu labeled "***PARTS" are activated. To restore the previous screen items, a menu item must issue:

```
$S=
```

without a submenu label. This pops the last pushed items off the stack, reactivating them. The number of nested submenu calls allowed is 8. If you exceed 8, the first menus you pushed are forgotten.

The submenu mechanism can be activated in the middle of a command, without interrupting that command. Command strings like the following are thus possible.

```
ARC $S=ARCSTUFF
```

This starts the ARC command, switches to the "ARCSTUFF" screen submenu, and awaits entry of arc parameters.

Expanding on the previous example, the following is a menu file using the submenu facility.

```
***SCREEN
[HELP      ]$S=Help_Root
[Bye      ]end
**Help_Root 1
[General ]help ;
[Entities]$S=Entity_Help
[Display  ]$S=Display_Help
[-MAIN-   ]$S=SCREEN
**Entity_Help 4
[Line     ]help line
[Circle   ]help circle
[Arc      ]help arc
[*Cancel*]^C
[-PREV-   ]$S=
**Display_Help 4
[Zoom    ]help zoom
[Pan     ]help pan
[View    ]help view
[*Cancel*]^C
[-PREV-   ]$S=
***TABLET1
line
circle
***BUTTONS
erase
oops
```

In this example, there are three submenus. "Help_Root" is called from the main screen menu when you select the HELP menu item; it in turn can call submenus "Entity_Help" and "Display_Help". If a submenu has more items than there are boxes or pointer buttons available, the excess items are ignored.

Note that the blanks in the menu item titles are not required; they are included here to improve the readability of the file.

All menu labels, control statements, and parameters can be entered in upper, lower or mixed case. They are converted to upper case when read from the file.

B.2.5 Commands Requiring Input

Sometimes it's useful to accept input from the keyboard or pointing device in the middle of a menu item. To do this, place a backslash character ("\") at the point where you want input. For instance:

```
[CIRCLE-1]circle \1
[ERASE 1]erase \;
```

"CIRCLE-1" asks for the center point, then reads a radius of 1 from the menu. Note that there is no space after the "\". "ERASE 1" lets you select one object by means of pointing, and immediately erases that object (a semi-colon acts as a RETURN when you select the menu item). The normal ERASE command requires that you enter a blank or RETURN after the last object has been designated.

B.2.6 Item Termination

When a menu item is selected, AutoCAD automatically places a blank after it, before processing the command sequence. If the menu item is:

LINE

AutoCAD will process it as though you typed "L I N E <blank>". There are times when this is undesirable; for instance, the text string supplied to a TEXT or DIM command must be terminated by RETURN, not blank. Also, it is sometimes necessary to supply more than one space (or RETURN) to complete a command, but some text editors don't let you create a line with trailing blanks. There are two special conventions that get around these problems.

- o Anywhere a semicolon (;) appears in a menu item, AutoCAD substitutes a RETURN.
- o If a line ends with a control character, a backslash (\), a plus sign (+), or a semicolon (;), AutoCAD does not add a blank after it.

For instance, look again at the "ERASE 1" menu item used in an earlier example:

```
[ERASE 1]erase \;
```

If this item simply ended with the backslash, it would fail to complete the ERASE operation, because AutoCAD does not add a blank after the backslash. Therefore, this menu item uses a semicolon to force a RETURN after the user input. Further examples are:

```
HELP
[HELP]HELP ;
[Address]text \.4 0 DRAFT Inc;;53 1st St.;;City, State;
```

The first line acts as "HELP <blank>" and results in a prompt for a command name. The second acts as "HELP <blank> <RETURN>", and displays the general Help information. In this example, no difference between these items would be evident on the screen; naturally, you wouldn't put both on the same menu. The second line has a title in brackets instead of just saying "HELP ;", because the semicolon is converted to a RETURN before the item is displayed on the screen, and the result would look odd on the screen.

The third item above prompts for a starting point, and then draws the address on three lines. Again, the semicolons are changed to RETURNS by AutoCAD. Note that the

double-semicolon (;;) construct ends the text string and causes repetition of the TEXT command. A menu item may contain any sequence of commands.

B.2.7 Long Menu Items

If an item in the menu file will not fit on one line, use the plus "+" character to continue it on the next line. Do this by entering "+" as the last character of the line to continue. For example:

```
[SETUP]layer set ground-floor;;grid on; ... ;fill off;+
limits 0,0 12,9;status
```

This item, which you might use to set initial conditions for a new drawing, is continued onto a second line. Menu items may be continued on as many lines as necessary.

B.2.8 Control Characters in Menu Items

You can place ASCII control characters in the command string portion of a menu item by entering the up-arrow character ^ followed by another character. For example, ^C will be converted to a single character, CTRL C. The non-alphabetic control characters are specified as follows:

^@	(ASCII code 0)
^[(ASCII code 27)
^`	(ASCII code 28)
^]	(ASCII code 29)
^^	(ASCII code 30)
^_	(ASCII code 31)

The ^ character itself may be entered using ^ followed by a blank.

You can use this technique to construct menu items that do such things as toggle the grid on and off (using ^G), or cancel a command (^C). Examples of these uses are shown below.

```
[*Cancel*]^C
[GridFlip]^G
```

B.2.9 Special Handling for HELP Command

Ordinarily, if you select a menu item that has a title in brackets ("[]"), the command string following the closing bracket is submitted. However, special handling is provided for the HELP command; if you issue a HELP command and then respond to its "Command name" prompt by selecting a screen menu item, the item's title is used as the command name, and the command string following the title is not processed. For example, given the following menu items:

```
HELP
[OSNAP]osnap endpoint,midpoint
```

selection of the "HELP" item followed by the "OSNAP" item issues the "HELP OSNAP" command sequence, obtaining information about the OSNAP command. But selection of the "OSNAP" item by itself sets particular parameters for object snap mode.

B.2.10 Special Handling for Button Menu

When you select a menu item by means of one of the menu buttons on a multi-button pointing device, AutoCAD receives not only the button number, but also the coordinates of the screen crosshairs at the time you press the button. By careful construction of the items in the BUTTONS portion of the menu file, you can choose to ignore these coordinates or to use them in the command activated by the button.

As described earlier, a backslash character (\) can be included in a menu item to pause for user input. For the BUTTONS menu, the coordinates of the screen crosshair are automatically supplied as the "user input" when the button is pressed. This occurs only for the first backslash in the menu item; if the item contains no backslashes, the pointer coordinates are not used.

For example, consider the following menu items.

```
***BUTTONS
line
line \
```

The first menu button issues an ordinary LINE command and solicits the "From" point from the user in the normal fashion. The second menu button also issues a LINE command, but AutoCAD automatically reads the current pointer location and uses it as the "From" point.

B.2.11 Use of Variables and Expressions (+3)

If your copy of AutoCAD includes the optional ADE-3 package, you can use its "variables and expressions" feature, described in Section 10.5, to create menu items that perform complex tasks. For example, the following menu item prompts the user for two points. Then it draws a rectangular Polyline with the specified points as its corners.

```
[BOX](setq a (getpoint "Enter first corner: "));\+
(setq b (getpoint "Enter second corner: "));\+
pline !a (list (car a) (cadr a)) !b (list (car b) (cadr a)) c;
```

This is a fairly advanced use of AutoCAD's custom menu facility. If you want to create such menu items, study the information in Section 10.5, and this example, carefully. Experimentation and practice will then help you make effective use of this feature.

B.3 Creating and Modifying Linetypes

The AutoCAD package includes a library of standard linetypes from which you can select those you wish to use. You can modify the supplied linetypes or create your own; these operations are described in this section. You need to understand this information only if you plan to create or modify linetype definitions.

There are two methods available for creating your own linetype definitions. You can create them outside AutoCAD using a text editor or within AutoCAD using the LINETYPE command's "Create" option. Regardless of the method used, in no way is the created linetype definition associated with a particular drawing. The definition is simply written to a library file.

To create or modify a linetype definition within AutoCAD, first make sure you are in the Drawing Editor. Then enter the LINETYPE command and choose its CREATE option as in:

Command: **LINETYPE**
?/Load/Create: **CREATE**

AutoCAD then prompts:

Name of linetype to create:
File for storage of linetype <default>:

Respond with the name or the linetype you wish to create and the name of the library file in which you plan to store its definition. Do not include a file type in the file name; type ".LIN" is assumed. If you give a null response, the default file is used. If the linetype name is found to already exist in the file, the prompt:

Name already exists in this file. Current definition is:
*linetype-name [,description]
alignment, dash-1, dash-2, . . .
Overwrite (Y/N) <N>?

appears. For reference purposes the current definition is displayed. If you answer "N", AutoCAD will prompt for a new linetype name and file name. If you answer "Y", the current definition will be overwritten with the new definition.

After entering the linetype name, you are prompted for its definition:

Descriptive text:
Enter pattern (on next line):
A,

You should respond to these prompts with the fields of your new linetype definition. Each field is described below.

All linetype definitions are stored externally in disk files. These are ordinary text files; you can create or modify them using a text editor if you wish. Each file can contain many linetype definitions; each definition has a header line of the form:

*linetype-name [,description]

followed by one pattern line of the form:

alignment, dash-1, dash-2, . . .

For example, a linetype called "DD1" with the repeating pattern:

- o Dash, 0.5 drawing units long
- o Space, 0.25 drawing units long
- o Dot
- o Space, 0.25 drawing units long

would be defined as follows:

```
*DD1, . . . . .  
A,.5,-.25,0,-.25
```

The "DD1" is the name by which the linetype is known, and the "*description*" field is the prose description of the linetype which is displayed by the "LAYER Ltype ?" command sequences. In this case, the description is a simple representation of the dash-dot pattern. The description may be omitted, in which case no comma should follow the linetype name. If you do enter a description, it should be no more than 47 characters long.

Postponing the discussion of the "*alignment*" field for a moment, we will now describe the "*dash-n*" fields. Each of these fields specifies the length of a segment making up the linetype. If the length is positive, this is a "pen down" segment, which will be drawn. A negative length denotes a "pen up" (blank) segment. A dash length of zero draws a "dot". You can specify up to 12 dash length specifications per linetype, provided that they fit on one 80-character line.

The "*alignment*" field specifies the action for pattern alignment at the ends of individual lines, circles and arcs. Currently, there is only one alignment action supported by AutoCAD. It is specified by entering "A" in the field. Since this is the only alignment action supported, it is automatically entered into the definition when you use the LINETYPE CREATE option. AutoCAD rejects any other character found in the alignment field.

With "A"-type alignment, AutoCAD guarantees that the endpoints of lines and arcs start and stop with a dash. For example, suppose you create a linetype called "CENTER1" which displays the repeating dash-dot sequence commonly used as a center line. AutoCAD automatically adjusts the dash-dot sequence, on an individual line basis, so that dashes and line-endpoints coincide. The pattern is fit to the line in such a way that at least half of the first dash specification begins and ends the line. If necessary, the first and last dashes drawn are lengthened to accomplish this. If a line is too short to hold even one dash-dot sequence, AutoCAD draws a continuous line between the endpoints. For Arcs the pattern is also adjusted so that dashes are drawn at the endpoints. Circles do not have endpoints, but AutoCAD automatically adjusts the dash-dot sequence to provide reasonable displays.

The "A" alignment action requires that the first dash length specification be a value of 0 or greater (a "dot" or "pen down" segment). The second dash length should be a value less than 0 (a "pen up" segment). The minimum number of dash specifications for this alignment action is two. Between the starting and ending dashes of the line, the pattern dash specifications are sequentially drawn, starting with the second dash specification and restarting the pattern with the first dash specification when required.

B.4 Creating Hatch Patterns (+1)

Developing a hatch pattern definition for AutoCAD requires some knowledge, some practice, some patience, and a text editor. A pattern may be added to the library file ACAD.PAT or stored in a file by itself. Regardless of where it is stored, it has the same format: a header line which looks like:

**pattern-name [, description]*

and one or more line descriptors of the form:

angle, x-origin, y-origin, delta-x, delta-y [, dash-1, dash-2, ...]

For example, a pattern called "L45" which hatches with 45 degree lines, separated by a spacing of .5, would be:

```
*L45,45 degree lines
45,0,0,0,.5
```

This simple pattern specifies that the line is to be drawn at an angle of 45 degrees, that the first line of the family of hatch lines passes through the (0,0) drawing origin, and that the spacing between hatch lines of the family is .5 drawing units. The L45 is the name by which the pattern is chosen, and the "*description*" field is the description of the pattern which is displayed by the "HATCH ?" command. The description may be omitted, in which case no comma should follow the pattern name.

Terminology

Before we undertake to explain patterns including dashed lines, let's define some terms and look at what the *delta-x* and *delta-y* parameters mean.

A pattern is made up of one or more pattern lines (there is no upper limit on the number of lines in a pattern). Each pattern line is considered the first member of a line family which is generated by taking the first member and applying the delta offsets in both directions to generate an infinite family of lines parallel to it. The *delta-y* value gives the spacing between members of the family (i.e., it is measured perpendicular to the lines). *Delta-x* gives the displacement in the direction of the line between members of the family; it is meaningful only for dashed lines. (A line is considered to be of infinite length; if there is a dash pattern, it is superimposed on the line.)

The process of hatching consists of taking each pattern line in the pattern definition and expanding it to its infinite family of parallel lines. All selected entities are then checked for intersections with any of these lines; any intersections found cause the hatch lines to be turned on and off as governed by the chosen hatching style. If the hatch line is dashed, it is drawn with the dash pattern in those areas in which it is on.

Since each family of hatch lines is generated by parallel transport from an initial line which has an absolute origin, hatching of adjacent areas is guaranteed to align properly.

Thus, all patterns are made up of one or more families of parallel lines. You might think this to be constraining, but almost any imaginable pattern can be constructed.

Patterns with Dashed Lines

Now let's consider patterns which involve dashed lines. Dashed line patterns are defined by appending dash length items to the end of the line definition item. Each dash length item specifies the length of a segment making up the line. If the length is positive, this is a "pen down" segment, which will be drawn. If negative the segment is "pen up", and it will not be drawn. The pattern will start at the origin point with the first segment, and cycle through the segments supplied in circular fashion. A dash length of zero draws a dot. A maximum of 6 dash length specifications may be supplied per pattern line.

Let's modify our pattern for 45 degree lines to draw dashed lines with a dash length of .5 and a spacing between dashes of .5. Such a pattern would be:

```
*DASH45,Dashed lines at 45 degrees
45, 0,0, 0,.5, .5,-.5
```

(This is the same as the original 45 degree pattern, but we have added a dash specification to the end.) The pen down length is .5 and the pen up length is .5, meeting the stated objectives. Suppose we wanted to draw a .5 long dash, a .25 space, a dot, and a .25 space before the next dash. This would be:

```
*DDOT45,Dash dot dash pattern: 45 degrees
45, 0,0, 0,.5, .5,-.25,0,-.25
```

Now let us consider the effect of *delta-x* specifications on dashed line families. First, consider the definition:

```
*GOSTAK
0, 0,0, 0,.5, .5,-.5
```

This will draw a family of lines separated by .5, each equally broken into dashes and spaces. Since *delta-x* is zero, the dashes in each member of the family will line up; an area hatched with this pattern would look like:



Now let's change the pattern to be:

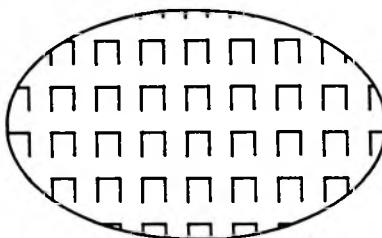
```
*SKEWED
0, 0,0, .5,.5, .5,-.5
```

This is the same, except we've set *delta-x* to .5. This will offset each successive member of the family by .5 in the direction of the line (in this case, parallel to the X axis). Now since the lines are, of course, infinite, this will have the effect of sliding the dash pattern down the selected amount. In this case, our hatched area would become:



So far, all the patterns we have examined use origin points of (0,0), and thus one member of the line family passes through the origin, with its dash pattern starting at that point. In composing more complex patterns, it is often necessary to specify the starting point, offsets, and dash pattern of each line family carefully so that the hatch pattern is formed correctly.

Suppose we want to draw a pattern which is a squared-off, inverted "U". That is, one line up, one over, and one down: a box with the bottom missing. Let's define a pattern of these that repeats every 1 unit. Each will be .5 high and wide, as illustrated below:



This pattern would be defined as:

```
*IUS,Inverted U's
90, 0,0, 0,1, .5,-.5
0, 0,.5 0,1, .5,-.5
270, .5,.5, 0,1, .5,-.5
```

The first line (up bar) is a simple dashed line with (0,0) origin. The second line (top bar) should begin at the end of the up bar, so we have set its origin to (0,.5). This will offset this line by .5 up so it aligns with the end of the up bar. The third line (down bar) must start at the end of the top bar, or at (.5,.5) for the first instance of the pattern, so we have set its origin at this point. We could have defined the third line of the pattern as:

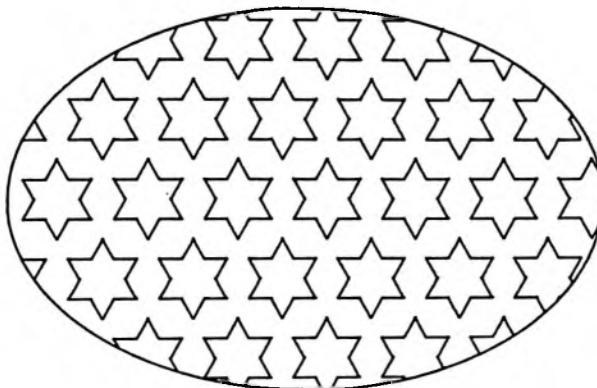
90, .5,0, 0,1, .5,-.5

or as:

270, .5,1, 0,1, -.5,.5

Do you understand why? Remember that the dashed pattern starts at the origin points and goes in the vector direction given by the angle specification. Therefore two dashed line families opposed 180 degrees are not alike. Two solid line families are, obviously.

Finally, consider the following pattern of six-pointed stars.



Here is AutoCAD's definition of this pattern. This is a good example to pick apart to hone your skills at pattern definition. (Hint: .866 is the sine of 60 degrees.)

```
*STARS,Star of David
0, 0,0, 0,.866, .5,-.5
60, 0,0, 0,.866, .5,-.5
120, .25,.433, 0,.866, .5,-.5
```

B.5 Defining Text Fonts and Shapes

This section explains how to define Shape/Font files for drawing symbols and text fonts, and how to compile these definitions for use by AutoCAD. See Section 4.10 for an introductory discussion of Shapes. You only need to read the following information if you are going to create your own shapes or fonts.

B.5.1 Compiling Shape/Font Files

Shapes are described using specially-formatted text in a disk file with a file type of ".SHP". Such a file may be created using a text editor or word processor in "programmer" mode; the required format is described later in this appendix.

Task 7 of the Main Menu is provided to "compile" a shape description file into a format that is more rapidly digested by the LOAD or STYLE command. When you select Task 7, AutoCAD prompts:

Enter NAME of shape file:

and you should reply with the name of the shape description file you wish to compile. AutoCAD assumes the ".SHP" file type; do not include a file type in the name you enter. If the name is valid and the file exists, the compilation process begins. It may take several seconds. If any errors are encountered in the shape descriptions, a message is displayed to tell you the type of error and the line number within the shape description file on which the error was discovered. If all goes well, the shape compiler displays:

Compilation successful.

Output file *name.SHX* contains *nnn* bytes.

and returns to the Main Menu. The compiled file has the same name as the shape description file, but with a file type of ".SHX". This is the file that will be read by the LOAD command.

B.5.2 Shape Descriptions

As noted in the previous section, shape description files have a file type of ".SHP" and contain text in a special format. To create or modify such a file, use a text editor or word processor in "programmer" mode. This section explains what goes into the file.

AutoCAD is supplied with two sample Shape files, PC.SHX and ES.SHX. One is for printed circuit layout, and the other is for electronic schematics. Examining the text versions of these files (PC.SHP and ES.SHP, respectively) and playing with their shape descriptions will help you to master the definition of AutoCAD shapes.

Every shape in a particular shape file must have a unique number between 1 and 255. Text fonts (which are simply files containing shape definitions for each character) require specific numbers corresponding to the value of each character in the ASCII code; ordinary shapes can be assigned whatever numbers you like.

Each shape definition has a header line of the form:

**shapenumber, defbytes, shapename*

followed by one or more lines containing specification bytes separated by commas and terminated by a 0. All the numbers in the shape definition file are scanned with the

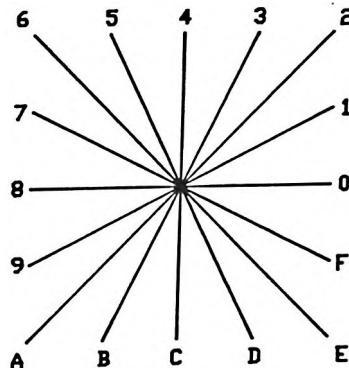
convention that a leading zero signifies hexadecimal. Hence, 10 is decimal 10, but 010 is hexadecimal with decimal value 16.

The shape name is ignored if it contains any lower case characters. This allows text fonts to name each letter for documentation but not waste space in memory storing names.

"Defbytes" is the number of data bytes required to describe the shape, including the terminating zero. There is a limit of 1000 bytes per shape.

The shape specification bytes contain vector length and direction encoded into one byte. The high-order hex nibble (4 bits) is the number of vector lengths (relative to the specified "height" of the shape) and the low-order nibble is the vector direction code. (See diagram, below.)

Vector Length and Direction Encoding



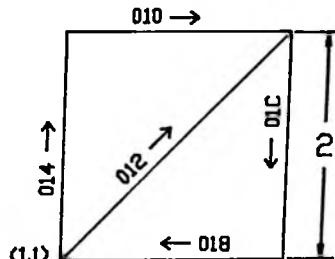
All the vectors in this figure were drawn with the same length specification. Diagonal vectors are "stretched" to match the X or Y displacement of the closest orthogonal vector. As a working example, let's construct a shape named DBOX with an arbitrarily assigned shape number of 230.

```
*230,6,DBOX
014,010,01C,018,012,0
```

The above sequence of bytes defines a box one unit high by one unit wide, with a diagonal line running from the lower left corner to the upper right corner. If we LOADED the Shape file containing this definition and then entered:

```
Command: SHAPE Name (or ?): DBOX
Starting point: 1,1
Height <default>: 2
Angle <default>: 0
```

the shape shown on the right would be drawn.



Special codes

In addition to drawing vectors, several special codes are defined. They all have the high-nybble = 0. They are:

- | | | |
|--------|------|---|
| Codes: | 000 | - End of shape definition |
| | 001 | - Activate draw mode (pen down) |
| | 002 | - De-activate draw mode (pen up) |
| | 003 | - Divide vector lengths by next byte |
| | 004 | - Multiply vector lengths by next byte |
| | 005 | - Push current location onto stack |
| | 006 | - Pop current location from stack |
| | 007 | - Draw subshape number given by next byte |
| | 008 | - X-Y displacement given by next two bytes |
| | 009 | - Multiple X-Y displacements, terminated by (0,0) |
| | 00A | - Octant arc defined by next two bytes |
| | .00B | - Fractional arc defined by next five bytes |

Code 0: End of Shape

This code simply marks the end of the shape definition.

Codes 1 & 2: Draw Mode Control

These codes control DRAW mode. DRAW is on at the start of each shape. When DRAW mode is on, vectors cause lines to be drawn. When DRAW mode is off, vectors move to a new location without drawing.

Codes 3 & 4: Size Control

Codes 3 and 4 control the relative size of each vector. The height specified when a SHAPE command is entered is initially considered the length of a single orthogonal vector (directions 0, 4, 8 or C). Codes 3 and 4 are followed by a byte containing an integer scale factor (1 to 255). If you want the SHAPE height to specify the size of the entire shape, and 10 vector lengths are used to draw it, then you can use 3,10 to scale the height specification. The scale factor is cumulative within a shape. That is, multiplying by 2 and again by 6 results in a scale factor of 12. The scale factor is reset at the end of a shape, but not at the end of a subshape.

Codes 5 & 6: Location Save/Restore

Codes 5 and 6 push (save) and pop (restore) the current coordinate while drawing a shape so that you can return to that location at a later point in the shape. *You must pop everything you push!* This stack is only 4 locations deep. For instance, subscripts and superscripts might use codes 5 and 6.

Code 7: Subshape

Code 7 is a subshape reference. The byte following the code 7 is a shape number from 1 to 255. The shape with that number (which must be in the same shape file) is drawn at this

time. (Note that DRAW mode is not reset for the new shape). When the subshape is completed, drawing of the current shape resumes.

Codes 8 & 9: X-Y Displacements

The normal vector bytes are only capable of drawing in the 16 predefined directions, and the longest length is 15. These restrictions help make Shape definitions efficient, but are sometimes very limiting. Therefore, codes 8 and 9 are available to permit "nonstandard" vectors to be drawn using X-Y displacements. Code 8 must be followed by two bytes in the format:

8, X-displacement, Y-displacement

The *X* and *Y* displacements may range from -128 to +127. The leading "+" is optional, and parentheses may be used to improve readability. For example:

8, (-10, 3)

results in a vector which draws (or moves) 10 units to the left and three units up. Following the two displacement bytes, the shape returns to normal vector mode.

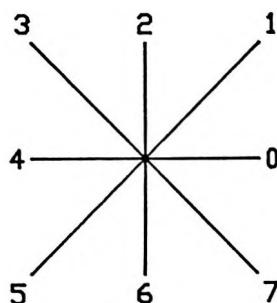
Code 9 can be used if a sequence of "nonstandard" vectors is to be drawn. This code may be followed by any number of *X-Y* displacement pairs, and it is terminated by a (0,0) pair. For instance:

9, (3,1), (3,2), (2,-3), (0,0)

draws three nonstandard vectors and then returns to normal vector mode. Note that you must terminate the sequence of *X-Y* displacement pairs with a (0,0) pair in order for AutoCAD to recognize any normal vectors or special codes which follow.

Code 00A: Octant arc

Special code 00A (or 10) uses the next two bytes to define an arc. This type of arc is called an "octant" arc because it spans one or more 45-degree octants, starting and ending on an octant boundary. The octants are numbered counterclockwise from the 3 o'clock position, as shown below:



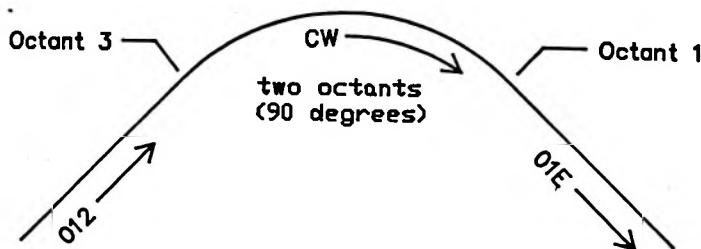
The arc specification is:

10, radius, (-)OSC

The radius may be any value from 1 to 255. The second byte indicates the direction of the arc (counterclockwise if positive, clockwise if negative), its starting octant ("S", a value from 0 to 7), and the number of octants it spans ("C", a value from 0 to 7 with zero meaning eight octants or a full circle). Parentheses may be used to improve readability. For example, consider the following fragment of a shape definition:

...012,10,(1,-032),01E,...

This would draw a 1-unit vector up and to the right, a clockwise arc from octant 3 with a radius of 1 unit for 2 octants, and then a 1-unit vector down and to the right, as illustrated below.



Code 00B: Fractional arc

Special code 00B (11) is used to draw an arc which does not necessarily start and end on an octant boundary. The definition uses five bytes:

11, start-offset, end-offset, high-radius, low-radius, (-)OSC

The start and end offsets represent how far from an octant boundary the arc begins or ends. The <high radius> is the most significant 8 bits of the radius; it will be zero unless the radius is greater than 255 units. Aside from that, the radius and control byte are the same as for the octant arc specification (code 00A, above). The octant count ("C") is the number of octants containing any portion of the arc. Again, 0 means 8.

You determine the start offset by calculating the difference in degrees between the starting octant's boundary (a multiple of 45 degrees) and the start of the arc. Then multiply this difference by 256, and divide by 45. If the arc starts on an octant boundary, its start offset is zero.

The end offset is calculated in a fashion similar to that of the start offset, but using the number of degrees from the last octant boundary crossed to the end of the arc. If the arc ends on an octant boundary, its end offset is zero.

For example, a fractional arc from 55 degrees to 95 degrees with a 3-unit radius would be coded:

11, (56,28,0,3,012)

Explanation:

starting octant	= 1	= 45 degrees
ending octant	= 2	= 90 degrees
start offset	= 56	= $((55 - 45) * 256 / 45)$
end offset	= 28	= $((95 - 90) * 256 / 45)$

B.5.3 Text Fonts

Text fonts are files of shape definitions, with shape numbers corresponding to the ASCII code for each character. The ASCII (American National Standard Code for Information Interchange) code table is listed below for your convenience.

32	space	56	8	80	P	104	b
33	!	57	9	81	Q	105	i
34	" double quote	58	:	82	R	106	j
35	#	59	;	83	S	107	k
36	\$	60	<	84	T	108	l
37	%	61	=	85	U	109	m
38	&	62	>	86	V	110	n
39	' apostrophe	63	?	87	W	111	o
40	(64	0	88	X	112	p
41)	65	A	89	Y	113	q
42	*	66	B	90	Z	114	r
43	+	67	C	91	[115	s
44	,	68	D	92	\ backslash	116	t
45	-	69	E	93]	117	u
46	.	70	F	94	caret	118	v
47	/	71	G	95	underscore	119	w
48	0	72	H	96	' reverse apost	120	x
49	1	73	I	97	a	121	y
50	2	74	J	98	b	122	z
51	3	75	K	99	c	123	{
52	4	76	L	100	d	124	vertical bar
53	5	77	M	101	e	125	}
54	6	78	N	102	f	126	- tilde
55	7	79	O	103	g		

Codes 1 to 31 are for control characters, only three of which are used in AutoCAD text fonts. These three are:

01 - SOT The Start of Text (SOT) character must push the current location onto the stack. For example:

```
*1,2,sot
5,0
```

10 - LF The Line Feed (LF) must drop down one line without drawing. This is used for repeated TEXT commands, to place succeeding lines below the first one. For example:

```
*10,5,lf
2,8,(0,-10),0
```

The spacing of the lines may be modified by adjusting the amount of downward movement specified by the LF shape definition.

13 - CR

The Carriage Return (CR) must restore ("pop") the starting position from the stack. For example:

```
*13,2,cr
6,0
```

Text fonts must include a special shape number 0 that conveys information about the font itself. The format is:

```
*0,4,Font name
above, below, flag-bits, 0
```

"*Above*" specifies the number of vector lengths that the upper case letters extend above the baseline, and "*below*" indicates how far the lower case letters descend below the baseline. These values define the basic character size, and they are used as scale factors for the height specified in the TEXT command. The "*flag-bits*" byte should be zero for a normal font, and 1 for a vertically-oriented font such as the one supplied in VERTICAL.SHX.

The standard fonts supplied with AutoCAD include a few additional characters required for the ADE-1 package's dimensioning feature:

- 127 - "Degrees" symbol
- 128 - "Plus/minus" tolerance symbol
- 129 - "Circle diameter" dimensioning symbol

You can use them in text by means of the "%%" control sequence described in Section 4.8.

If you intend to devise your own text fonts, we suggest that you study TXT.SHP and the other supplied fonts for working examples of the topics discussed here. Note that AutoCAD draws text characters by means of their ASCII codes (shape numbers), and never by name. To save memory, therefore, you should specify the "shape name" portion of each text shape definition in lower-case letters; lower-case names are not saved in memory. For example:

```
*65,11,uca
024,043,04d,02c,2,047,1,040,2,02e,0
```

The shape name "uca" contains lower-case letters, so AutoCAD does not save the name in memory. However, you can use the name for reference when editing the font definition file; here, "uca" stands for "upper-case A".

B.6 Customizing the HELP File

The text displayed by AutoCAD's HELP command is kept in the file ACAD.HLP. This is a standard text file that you can edit or revise using a text editor or a word processor in "programmer" or "non-document" mode.

The help text is organized into numerous sections. Lines beginning with a backslash ("\") and a name, as in "\ZOOM", serve to divide and label the sections. The first portion of the file contains the general information displayed when you enter "HELP" followed by RETURN, and has no section name. The named sections are displayed when you supply the section name on the "HELP" command. For instance, "HELP BLOCK" displays the section of the help file that follows the "\BLOCK" section label. Any section of the file may have several names. A line consisting of just a backslash causes the HELP command to pause, and to resume after the user presses RETURN.

An outline of the help file is shown below.

```
---
--- This text (up to the first line that starts with a
--- "\name") is displayed when you enter "HELP (RETURN)".
---
\BLOCK
---
--- This text (up to the next "\name") is displayed when
--- you enter "HELP BLOCK". AutoCAD will pause at the next
--- line because it consists of just a "\".
\
--- Help will resume here after you press RETURN.
---
\COLOR
\COLORS
---
--- This section of the help file has two section labels.
--- The text (up to the next label) is displayed when
--- you enter either "HELP COLOR" or "HELP COLORS".
---
(end of file)
```

When a HELP command selects a particular section, the text in that section is displayed until the next "\name" (or the end of the file) is encountered. For a working example, list out the supplied version of ACAD.HLP.

NOTE: AutoCAD also maintains an index file, ACAD.HDX, in order to make the HELP facility fast. If you change the contents of the ACAD.HLP text file in any way, delete the index file. The next time you use AutoCAD, it will rebuild ACAD.HDX based on your revised text.

B.7 External Commands (+3)

If the ADE-3 package is present, you can invoke other programs while remaining in AutoCAD's Drawing Editor. Programs you might execute in this manner include:

- o CAD/cameraTM
- o DOS internal commands, such as TYPE or DIR
- o Other DOS utilities, such as CHKDSK or FORMAT
- o Text editors and word processors
- o Database managers
- o Communications programs
- o User-supplied programs
- o Etc.

In order to run another program from within AutoCAD, you must first inform AutoCAD of the name of the program, how much system memory (RAM) it requires, and various other details. You must supply this information in the "program parameters" file, ACAD.PGP; this file must be present when you begin executing AutoCAD.

ACAD.PGP is an ASCII text file wherein each line describes a program that can be executed from within AutoCAD. You can consider this to be a list of custom AutoCAD commands; any time you enter a command, AutoCAD looks for that command in ACAD.PGP if it does not find it in the regular AutoCAD command list.

Each line in ACAD.PGP has five fields, delimited by commas. The fields are:

Command name	The AutoCAD external command to be added. Must not be an internal AutoCAD command or it will be ignored. This name is case-insensitive.
File command	This constant string will be sent to the operating system for execution when the command name is entered. It may be any valid command which you might execute when at the DOS command prompt. The string may include switches, parameters, etc. Note that even internal DOS commands such as "DIR" and "SET" may be used here.
Memory reserve	AutoCAD normally reserves all of the computer's memory (RAM) for itself. Before a program may be executed, AutoCAD must release sufficient memory to load the program (and for any allocation that program might do from free space while executing). This parameter, which is required, specifies the number of bytes which must be released before the command is executed. Your program will be run under a copy of DOS's COMMAND interpreter, so allow for the 17K or so occupied by the command interpreter when you calculate the memory reserve.
Prompt	This field, if supplied, specifies a prompt to be displayed to the user. The response to this prompt will be appended to the constant command string supplied by the file command field (see above). If the first character of the prompt field is an asterisk ("*"), the response may contain spaces, and the user must press RETURN to terminate it. Otherwise, the response is terminated by either space or RETURN and must be a single field. If a Block is being defined by the results of the

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command (see below), this field specifies the Block name and must be a valid name for a Block. No previously defined Block by this name may exist. If no prompt is specified, no input will be requested. If no prompt is desired and fields follow, just use two commas for the prompt field.

Return code

This is an optional bit-coded integer parameter. Its bits are defined as follows:

1: Load DXB file

A DXB file named \$CMD.DXB will be loaded into the drawing when this command terminates. That file will then be automatically deleted. DXB files are described in Appendix C.

2: Construct block from DXB file

If this bit is used, bit 1 must also be specified. The response to the Prompt field is taken as a Block name. A Block will be added to the drawing with that name, consisting of entities read from the \$CMD.DXB file written by the file command. The \$CMD.DXB file will be automatically deleted. The default name for subsequent INSERT commands will be set to the newly defined Block's name. This mode may not redefine a previously-defined Block.

4: Restore text/graphics mode

Each command in ACAD.PGP switches to text mode (on single-screen systems) during the execution of the command. If this bit is set, the previous mode will be restored when the command terminates. If not set, the screen will be left in text mode.

The following sample ACAD.PGP file illustrates some of the things you can do with it. Note carefully the definition of the very useful "SHELL" command.

```
SHELL,,131000,*DOS Command: ,0
DIR,DIR,64000,File specification: ,0
catalog,DIR /W,120000,*Files: ,0
type,type,30000,File to list: ,0
EDIT,EDLIN,40000,File to edit: ,0
NULL,REM Hello,32000,,0
LISTSET,SET,30000,,4
scan,CAMERA,60000,,1
Symscan,CAMERBLK,40000,Symbol name: ,7
```

A default ACAD.PGP file containing some of the items shown above is supplied with the AutoCAD software if the ADE-3 package is purchased.

IMPORTANT NOTES:

1. Since AutoCAD is still running, numerous files are open, and disk check programs such as CHKDSK may incorrectly report these open files as errors or lost blocks. *Do not attempt to correct these apparent errors while AutoCAD is running!* For instance, do not use the /F option on CHKDSK.
2. For some display devices, AutoCAD keeps the image of your drawing in a graphics memory area. If an external program writes in that graphics memory area, AutoCAD may restore the drawing incorrectly when your program has completed. Frequently, a REDRAW operation will correct the image.

Appendix C

DRAWING INTERCHANGE FILES

AutoCAD can be used by itself as a complete drawing editor. In some applications, however, other programs must examine drawings created by AutoCAD or generate drawings to be viewed, modified, or plotted with AutoCAD.

For example, if you have made an architectural drawing with AutoCAD, using INSERTed parts to represent windows, doors, and so on, you can process the drawing file and produce a bill of materials of all the items used in the drawing, or even make energy use calculations based on the area and the number and type of windows used.

Another possible application is to use AutoCAD to describe structures which are then submitted to a large computer for finite element structural analysis. You can calculate stresses and displacements and transmit back information to display the deformed structure as an AutoCAD drawing.

Since the AutoCAD drawing database is stored in a very compact format, it is hard for user programs to read directly. In addition, different machine implementations of AutoCAD may use different internal formats for the database, tuned for maximum performance on the machine on which AutoCAD is running. To assist in interchanging drawings between different implementations of AutoCAD on different machines, and between AutoCAD and other programs, a "Drawing Interchange" file format has been defined. All implementations of AutoCAD accept this format and are able to convert it to and from their internal drawing file representation.

C.1 DXFOUT Command - Writing a DXF File

You can generate a drawing interchange file from an existing drawing by means of the Drawing Editor's DXFOUT command. The command format is:

Command: **DXFOUT** File name: *(name or RETURN)*

The default name for the output file is the same as that of the current drawing, but with a file type of ".DXF". If you specify an explicit file name, do not include a file type; ".DXF" is assumed. If a file with the same name already exists, it is deleted.

NOTE: The output file produced by DXFOUT is in the new DXF format introduced in AutoCAD Version 2.1 and described later in this appendix. If you are upgrading from a previous version of AutoCAD and need to produce an old style DXF file, respond to the "File name" prompt with "name,OLD". This is a temporary conversion aid and will be removed in a future version.

For the new output format only, DXFOUT also asks what precision you want for floating point numbers.

Enter decimal places of accuracy (0 to 16) <6>:

C.2 DXFIN Command - Loading a DXF File

A drawing interchange file can be converted into an AutoCAD drawing by means of the DXFIN command. First enter the Drawing Editor using the "Create new drawing" task from the Main Menu. Then issue the DXFIN command.

Command: DXFIN File name: (name)

Enter the name of the drawing interchange file to be loaded.

The DXFIN command can only be used in a newly created drawing, before any entities have been drawn. Upon completion, if any errors were detected, the drawing is discarded. If all input is valid, an automatic ZOOM All is performed to set the drawing extents.

C.3 Drawing Interchange Format

This section describes the format of a DXF file in detail. It contains a great deal of technical information that you need only if you are writing your own program to process DXF files. Otherwise, you can skip this section.

It would probably be helpful to produce a DXF file from a small drawing, print it out, and refer to it occasionally while reading the information presented below.

C.3.1 General File Structure

A Drawing Interchange File is simply an ASCII text file with a file type of ".DXF" and specially-formatted text. The overall organization of a DXF file is as follows:

1. HEADER section -- General information about the drawing is found in this section of the DXF file. Each parameter has a *variable name* and an associated value.
2. TABLES section -- This section contains definitions of named items.
 - o Linetype (LTYPE) table
 - o Layer table
 - o Style table
 - o View table
3. BLOCKS section -- This section contains Block Definition entities describing the entities comprising each Block in the drawing.
4. ENTITIES section -- This section contains the drawing entities, including any Block References.
5. END OF FILE

A DXF file is composed of a multiplicity of *groups*, each of which occupies two lines in the DXF file. The first line of a group is a *group code*, which is a nonnegative integer output in FORTRAN "I3" format (that is, right justified and blank filled in a three character field). The second line of the group is the *group value*, in a format which depends on the type of the group as specified by the group code.

The specific assignment of group codes depends upon the item being described in the file. However, the type of the value this group supplies is derived from the group code in the following way:

Group code range	Following value
0 - 9	String
10 - 59	Floating point
60 - 79	Integer

Thus a program can easily read the value following a group code without knowing the particular use of this group in an item in the file. The appearance of values in the DXF file is not affected by the setting of the UNITS command: coordinates are always represented as decimal (or possibly scientific notation if very large) numbers, and angles are always represented in decimal degrees with zero degrees to the east of origin.

Variables, table entries, and entities are described by a group that introduces the item, giving its type and/or name, followed by multiple groups that supply the values associated with the item. In addition, special groups are used for file separators such as markers for the beginning and end of sections, tables, and the file itself.

Entities, table entries, and file separators are always introduced with a 0 group code that is followed by a name describing the item.

C.3.2 Group Codes

Group codes are used both to indicate the type of the value of the group, as explained above, and to indicate the general use of the group. The specific function of the group code depends on the actual variable, table item, or entity description. This section indicates the general use of groups, noting as "(fixed)" any that always have the same function.

- 0 Identifies the start of an entity, table entry, or file separator. The text value that follows indicates which.
- 1 The primary text value for an entity.
- 2 A name; Attribute tag, Block name, etc.
- 3-5 Other textual or name values.
- 6 Line type name (fixed).
- 7 Text style name (fixed).
- 8 Layer name (fixed).
- 9 Variable name identifier (used only in HEADER section of the DXF file).
- 10 Primary X coordinate (start point of a Line or Text entity, center of a Circle, etc.).
- 11-18 Other X coordinates.
- 20 Primary Y coordinate. $2n$ values always correspond to $1n$ values and immediately follow them in the file.

- 21-28 Other Y coordinates.
- 30 Primary Z coordinate. 3n values always correspond to 1n and 2n values and immediately follow them in the file.
- 31-36 Other Z coordinates (future).
- 38 This entity's elevation, if nonzero (fixed).
- 39 This entity's thickness, if nonzero (fixed).
- 40-48 Floating point values (text height, scale factors, etc.).
- 49 Repeated value -- multiple 49 groups may appear in one entity for variable length tables (such as the dash lengths in the LTYPE table). A 7x group always appears before the first 49 group to specify the table length.
- 50-58 Angles.
- 62 Color number (fixed).
- 66 "Entities follow" flag (fixed).
- 70-78 Integer values, such as repeat counts, flag bits, or modes.

The DXF file is subdivided into four sections. File separator groups are used to delimit these file sections. The following is an example of a void DXF file with only the section markers and table headers present.

```

0                               (Begin HEADER section)
SECTION
2
HEADER
    <<<Header variable items go here>>>
0                               (End HEADER section)
ENDSEC                         (Begin TABLES section)
0
SECTION
2
TABLES
0
TABLE
2
LTYPE
70
(Line type table maximum item count)
    <<<Line type table items go here>>>
0
ENDTAB
0
TABLE
2
LAYER
70
(Layer table maximum item count)
    <<<Layer table items go here>>>

```

```

0
ENDTAB
0
TABLE
2
STYLE
70
(Text style table maximum item count)
    <<<Text style table items go here>>>
0
ENDTAB
0
TABLE
2
VIEW
70
(View table maximum item count)
    <<<View table items go here>>>
0
ENDTAB
0
ENDSEC          (End TABLES section)
0
SECTION
2
BLOCKS          (Begin BLOCKS section)
    <<<Block definition entities go here>>>
0
ENDSEC          (End BLOCKS section)
0
SECTION          (Begin ENTITIES section)
2
ENTITIES
    <<<Drawing entities go here>>>
0
ENDSEC          (End ENTITIES section)
0
EOF              (End of file)

```

C.3.3 HEADER Section

The HEADER section of the DXF file contains settings of variables associated with the drawing. These variables are set with various commands and are the type of information displayed by the STATUS command. Each variable is specified in the header section by a 9 group giving its name, followed by groups that supply its value. The header variables, the groups that follow, and their meanings are given in the following table.

\$ACADVER	1 (the AutoCAD version number).
\$ATTMODE	70 (Attribute visibility: 0=none, 1=normal, 2=all).
\$AUNITS	70 (UNITS format for angles).
\$AUPREC	70 (UNITS precision for angles).
\$AXISMODE	70 (axis on if nonzero).
\$AXISUNIT	10 and 20 (axis X and Y tick spacing).
\$BLIPMODE	70 (blip mode on if nonzero).
\$CHAMFERA	40 (first chamfer distance).

\$CHAMFERB	40 (second chamfer distance).
\$CLAYER	8 (current layer name).
\$DIMASZ	40 (dimensioning arrow size).
\$DIMCEN	40 (size of center mark/lines).
\$DIMDLI	40 (dimension line increment).
\$DIMEXE	40 (extension line extension).
\$DIMEXO	40 (extension line offset).
\$DIMLIM	70 (dimension limits generated if nonzero).
\$DIMSCALE	40 (overall dimensioning scale factor).
\$DIMSE1	70 (first extension line suppressed if nonzero).
\$DIMSE2	70 (second extension line suppressed if nonzero).
\$DIMTAD	70 (text above dimension line if nonzero)..
\$DIMTIH	70 (text inside horizontal if nonzero).
\$DIMTM	40 (minus tolerance).
\$DIMTOH	70 (text outside horizontal if nonzero).
\$DIMTOL	70 (dimension tolerances generated if nonzero).
\$DIMTP	40 (plus tolerance).
\$DIMTSZ	40 (dimensioning tick size: 0=no ticks).
\$DIMTXT	40 (dimensioning text height).
\$DRAGMODE	70 (DRAGMODE on if nonzero).
\$ELEVATION	40 (current elevation set by ELEV command).
\$EXTMAX	10 and 20 (drawing extents upper right corner).
\$EXTMIN	10 and 20 (drawing extents lower left corner).
\$FILLETRAD	40 (fillet radius).
\$FILLMODE	70 (FILL mode on if nonzero).
\$GRIDMODE	70 (GRID mode on if nonzero).
\$GRIDUNIT	10 and 20 (grid X and Y spacing).
\$INSBASE	10 and 20 (insertion base set by BASE command).
\$LIMCHECK	70 (nonzero if limits checking is on).
\$LIMMAX	10 and 20 (drawing limits upper right corner).
\$LIMMIN	10 and 20 (drawing limits lower left corner).
\$LTSCALE	40 (global linetype scale).
\$LUNITS	70 (UNITS format for coordinates and distances).
\$LUPREC	70 (UNITS precision for coordinates and distances).
\$MENU	1 (name of menu file)
\$ORTHOMODE	70 (ORTHO mode on if nonzero).
\$OSMODE	70 (running object snap modes).
\$QTEXTMODE	70 (quick text mode on if nonzero).
\$REGENMODE	70 (REGENAUTO mode on if nonzero).
\$SKETCHINC	40 (sketch record increment).
\$SNAPANG	50 (snap grid rotation angle).
\$SNAPBASE	10 and 20 (snap grid base point).
\$SNAPISOPAIR	70 (isometric plane: 0=left, 1=top, 2=right).
\$SNAPMODE	70 (snap mode on if nonzero).
\$SNAPSTYLE	70 (snap style: 0=standard, 1=isometric).
\$SNAPUNIT	10 and 20 (snap grid X and Y spacing).
\$TEXTSIZE	40 (default text height).
\$TEXTSTYLE	7 (current text style name).
\$THICKNESS	40 (current thickness set by ELEV command).
\$TRACEWID	40 (default Trace width).
\$VIEWCTR	10 and 20 (center of current view on screen).
\$VIEWDIR	10, 20, and 30 (current view point set by VPOINT command).
\$VIEWSIZE	40 (height of current view on screen).

C.3.4 TABLES Section

The TABLES section of the DXF file contains four tables each of which in turn contains a variable number of table entries. The tables always appear in the order given in the sample file above. Each table in the TABLES section is introduced with a 0 group with the label "TABLE". This is followed by a 2 group naming the table ("LTYPE", "LAYER", "STYLE", or "VIEW"), and a 70 group that specifies the maximum number of table entries that may follow. The tables in a drawing may contain deleted items, but these are not written to the DXF file. Thus, the actual number of items that follow the table header may be less than the number given in the 70 group, so do not use the count in the 70 group as an index to read in the table. It is provided so that your program to read DXF files can allocate an array in advance to hold all the table items that follow.

Following this header for each table are the table entries. Each table item consists of a 0 group identifying the item type (same as table name, e.g., "LTYPE" or "LAYER"), a 2 group giving the name of the table entry, a 70 group specifying flags relevant to the table entry (defined for each table below), and additional groups that give the value of the table entry. The end of each table is indicated by a 0 group with the value "ENDTAB".

The following are the groups used for each type of table item. All groups are present for each table item.

LTYPE	3 (descriptive text for linetype), 72 (alignment code), 73 (number of dash length items), 40 (total pattern length), 49 (dash length 1), 49 (dash length 2), ...
LAYER	62 (color number, negative if layer is off), 6 (linetype name). The 1 bit is set in the 70 group flags if the layer is frozen.
STYLE	40 (fixed text height; 0 if not fixed), 50 (obliquing angle), 71 (text generation flags), 42 (last height used), 3 (text font or shape file name). A STYLE table item is used to record shape file LOAD requests also. In this case the 1 bit is set in the 70 group flags and only the 3 group is meaningful (all the other groups are output, however).
VIEW	40 and 41 (view height and width), 10 and 20 (view center point), 11, 21, and 31 (view direction from origin).

C.3.5 BLOCKS Section

The BLOCKS section of the DXF file contains all the Block Definitions. This section contains the entities that make up the Blocks used in the drawing. The format of the entities in this section is identical to those in the ENTITIES section described below, so refer to that section for details. All entities in the BLOCKS section appear between BLOCK and ENDBLK entities. BLOCK and ENDBLK entities appear only in the BLOCKS section. Block definitions are never nested (that is, no BLOCK or ENDBLK entity ever appears within another BLOCK-ENDBLK pair).

C.3.6 ENTITIES Section

Entity items appear in both the BLOCK and ENTITIES sections of the DXF file. The appearance of entities in the two sections is identical. The following gives the format of each

entity as it appears in the file. Some groups that define an entity always appear, and some are optional and appear only if they differ from their default values. In the following discussion, groups that always occur are given by their group number and function, while optional groups are indicated by "-optional *N*" following the group description. "*N*" is the default value if the group is omitted.

Programs that read DXF files should not assume that the groups describing an entity occur in the order given here. The end of the groups that make up an entity is indicated by the next 0 group, beginning the next entity or indicating the end of the section.

Remember that a DXF file is a complete representation of the drawing database, and that as AutoCAD is further enhanced, new groups will be added to entities to accommodate additional features. Writing your DXF processing program in a table-driven way, making no assumptions about the order of groups in an entity, and ignoring any groups not presently defined, will make it much easier to accommodate DXF files from future releases of AutoCAD.

Each entity begins with a 0 group identifying the entity type. The names used for the entities are given in the table that follows. Every entity contains an 8 group that gives the name of the layer on which the entity resides. If the entity has a nonzero elevation or thickness, a 38 or 39 group (specifying the elevation or thickness) will be included as well. The rest of the groups that make up an entity item are as follows:

LINE	10 and 20 (start point), 11 and 21 (end point).
POINT	10 and 20.
CIRCLE	10 and 20 (center), 40 (radius).
ARC	10 and 20 (center), 40 (radius), 50 (start angle), 51 (end angle).
TRACE	Four points defining the corners of the trace: 10 and 20, 11 and 21, 12 and 22, 13 and 23.
SOLID	Four points defining the corners of the solid: 10 and 20, 11 and 21, 12 and 22, 13 and 23. If the solid only has 3 sides, the coordinates specified by the 12 and 22, and the 13 and 23 groups will be the same (e.g., 12 and 13 groups the same, 22 and 23 groups the same).
REPEAT	No groups.
ENDREP	70 (number of columns), 71 (number of rows), 40 (column spacing), 41, row spacing.
TEXT	10 and 20 (insertion point), 40 (height), 1 (text value), 50 (rotation angle -optional 0), 41 (relative X scale factor -optional 1), 51 (obliquing angle -optional 0), 7 (text style name -optional "STANDARD"), 71 (generation flags -optional 0), 72 (justification type -optional 0), 11 and 21 (alignment point -optional, appears only if 72 group is present and nonzero).
SHAPE	10 and 20 (insertion point), 40 (size), 2 (shape name), 50 (rotation angle -optional 0), 41 (relative X scale factor -optional 1), 51 (obliquing angle -optional 0).

BLOCK	2 (Block name), 70 (Block type flags), 10 and 20 (Block base point). Appears only in BLOCKS section.
ENDBLK	No groups. Appears only in BLOCKS section.
INSERT	66 ("Attributes follow" flag -optional 0), 2 (Block name), 10 and 20 (insertion point), 41 (X scale factor -optional 1), 42 (Y scale factor -optional 1), 50 (rotation angle -optional 0).
ATTDEF	10 and 20 (text start), 40 (text height), 1 (default value), 3 (prompt string), 2 (tag string), 70 (Attribute flags), 73 (field length -optional 0), 50 (text rotation -optional 0), 41 (relative X scale factor -optional 1), 51 (obliquing angle -optional 0), 7 (text style name -optional "STANDARD"), 71 (text generation flags -optional 0), 72 (text justification type -optional 0), 11 and 21 (alignment point -optional, appears only if 72 group is present and nonzero).
ATTRIB	10 and 20 (text start), 40 (text height), 1 (value), 2 (Attribute tag), 70 (Attribute flags), 73 (field length -optional 0), 50 (text rotation -optional 0), 41 (relative X scale factor -optional 1), 51 (obliquing angle -optional 0), 7 (text style name -optional "STANDARD"), 71 (text generation flags -optional 0), 72 (text justification type -optional 0), 11 and 21 (alignment point -optional, appears only if 72 group is present and nonzero).
POLYLINE	70 (Polyline flags), 40 (default starting width), 41 (default ending width). The flags currently have the bit values 1 (closed Polyline) and 2 (curve-fit information has been added). The default widths apply to any vertex that does not supply widths (see below).
VERTEX	10 and 20 (location), 40 (starting width -optional, see above), 41 (ending width -optional, see above), 42 (bulge), 70 (vertex flags), 50 (curve fit tangent direction -optional). The bulge is the tangent of 1/4 the included angle for an arc segment, made negative if the arc goes clockwise from the start point to the end point; a bulge of 0 indicates a straight segment, and a bulge of 1 is a semicircle. Currently, the vertex flags are 1 (extra vertex created by curve fitting) and 2 (curve fit tangent defined). A curve fit tangent direction of 0 may be omitted from the DXF output, but is significant if the "tangent defined" bit is set in the vertex flags.
SEQEND	No fields. This entity marks the end of vertices (VERTEX type name) for a Polyline, or the end of Attribute entities (ATTRIB type name) for an INSERT entity that has Attributes (indicated by 66 group present and nonzero in INSERT entity).

C.3.7 Entity Flag Definitions

The entity items listed above use various "flag" values. These are integer codes (6x or 7x groups) that encode various pieces of information regarding the entity. The following paragraphs describe the meaning of the various flag groups used in entities. In this discussion, the term "bit coded" means that the flag contains various true/false values coded as the sum of the bit values given. Any bits not defined in the following section should be ignored in these fields and set to zero when constructing a DXF file.

"Attributes Follow" Flag

This is an optional 66 group presently used only in an INSERT entity. If the value that follows is 1, Attributes (ATTRIB) entities are expected to follow the INSERT entity.

Attribute Flags

This is a 70 group that appears in ATTDEF and ATTRIB entities. This is a bit coded field in which 1 means the Attribute is invisible (does not display), 2 means the Attribute is constant, and 4 means that verification is required on input of this Attribute.

Text Generation Flags

This is an optional 71 group that appears in TEXT, ATTDEF, and ATTRIB entities. It is bit coded with 2 meaning that the text is mirrored in the X direction (e.g. regular mirror imaged), and 4 meaning that the text is generated upside down.

Text Justification Type

This is an optional 72 group that appears in TEXT, ATTDEF, and ATTRIB entities. Its value (NOT bit coded) indicates the text justification style used on this entity. 0 means left justified, 1 indicates centered text, 2 means right justified, and 3 specifies "aligned" text. If this group appears with a nonzero value, 11 and 21 groups will also appear in the entity and specify the alignment point of the text (center, rightmost, or second alignment point).

Block Type Flags

This is a 70 group that appears in BLKDEF entities. It is bit coded, with 1 indicating that this is an "anonymous" block generated by hatching or other internal operations, and 2 indicating that this block has Attributes.

C.4 Writing DXF Interface Programs

Writing a program that communicates with AutoCAD via the DXF mechanism often appears far more difficult than it really is. The DXF file contains a seemingly overwhelming amount of information, and examining a DXF file manually may lead to the conclusion that the task is hopeless.

However, the DXF file has been designed to be easy to process by program, not manually. The format was constructed with the deliberate intention of making it easy to ignore information you don't care about while easily reading the information you need. Just remember to handle the groups in any order and ignore any group you don't care about, and you'll be home free.

As an example, the following is a Microsoft BASIC program that reads a DXF file and extracts all the LINE entities from the drawing (ignoring lines that appear inside Blocks). It prints the endpoints of these lines on the screen. As an exercise you might try entering this program into your computer, running it on a DXF file from one of your drawings, then enhancing it to print the center point and radius of any circles it encounters. This program is not put forward as an example of clean programming technique nor the way a general DXF

processor should be written; it is presented as an example of just how simple a DXF-reading program can be.

```

1000 REM
1010 REM Extract lines from DXF file
1020 REM
1030 LINE INPUT "DXF file name: "; A$
1040 A$=A$+".DXF"
1050 OPEN "1",1,A$
1060 REM
1070 REM Ignore until section start encountered
1080 REM
1090 GOSUB 1320
1100 IF G% <> 0 THEN 1090
1110 IF S$ <> "SECTION" THEN 1090
1120 GOSUB 1320
1130 REM
1140 REM Skip unless ENTITIES section
1150 REM
1160 IF S$ <> "ENTITIES" THEN 1090
1170 REM
1180 REM Scan until end of section processing LINEs
1190 REM
1200 GOSUB 1320
1210 IF G% = 0 AND S$="ENDSEC" THEN STOP
1220 IF G%=0 AND S$="LINE" THEN GOSUB 1270 : GOTO 1210
1230 GOTO 1200
1240 REM
1250 REM Accumulate LINE entity groups
1260 REM
1270 GOSUB 1320
1280 IF G%=10 THEN X1=X : Y1=Y
1290 IF G%=11 THEN X2=X : Y2=Y
1300 IF G%<0 THEN PRINT "Line from (";X1;",";Y1;
") to (";X2;",";Y2;")" : RETURN
1310 GOTO 1270
1320 REM
1330 REM Read group code and following value
1340 REM
1350 INPUT #1, G%
1360 IF G% < 10 THEN LINE INPUT #1, S$ : RETURN
1370 IF G% >= 30 AND G% <= 49 THEN INPUT #1, V : RETURN
1380 IF G% >= 50 AND G% <= 59 THEN INPUT #1, A : RETURN
1390 IF G% >= 60 AND G% <= 69 THEN INPUT #1, P% : RETURN
1400 IF G% >= 70 AND G% <= 79 THEN INPUT #1, F% : RETURN
1410 IF G% >=20 THEN PRINT "Invalid group code ";G% : STOP
1420 INPUT #1,X
1430 INPUT #1,G1%
1440 IF G1% <> (G%+10) THEN PRINT "Invalid Y coord code ";
G1% : STOP
1450 INPUT #1,Y
1460 RETURN

```

Writing a program that constructs a DXF file is more difficult, because you must maintain consistency within the drawing in order for AutoCAD to find it acceptable. AutoCAD allows

you to omit many items in a DXF file and still obtain a usable drawing. The entire HEADER section can be omitted if you don't need to set any header variables. Any of the TABLES in the TABLES section can be omitted if you don't need to make any entries, and in fact the entire TABLES section can be dropped if nothing in it is required. If you define any linetypes in the LTYPE table, this table must appear before the LAYER table. If no Block Definitions are used in the drawing, the BLOCKS section can be omitted. If present, however, it must appear before the ENTITIES section. Within the ENTITIES section, you can reference layer names even though you have not defined them in the LAYER table. Such layers will be automatically created with color 7 and the CONTINUOUS linetype. The EOF item must be present at the end of file.

The following Microsoft BASIC program constructs a DXF file representing a polygon with a specified number of sides, leftmost origin point, and side length. This program supplies only the ENTITIES section of the DXF file, and places all entities generated on the default layer "0". This may be taken as an example of a minimum DXF generation program. Since this program does not create the drawing header, the drawing limits, extents, and current view will be invalid after performing a DXFIN on the drawing generated by this program. You can do a "ZOOM E" to fill the screen with the drawing generated. Then adjust the limits manually.

```

1000 REM
1010 REM Polygon generator
1020 REM
1030 LINE INPUT "Drawing (DXF) file name: "; A$
1040 OPEN "o",1,A$+".DXF"
1050 PRINT #1,0
1060 PRINT #1,"SECTION"
1070 PRINT #1,2
1080 PRINT #1,"ENTITIES"
1090 PI=ATN(1)*4
1100 INPUT "Number of sides for polygon: ";S$
1110 INPUT "Starting point (X,Y): ";X,Y
1120 INPUT "Polygon side: ";D
1130 A1=(2*PI)/S$
1140 A=PI/2
1150 FOR I%=1 TO S%
1160 PRINT #1,0
1170 PRINT #1,"LINE"
1180 PRINT #1,8
1190 PRINT #1,"0"
1200 PRINT #1,10
1210 PRINT #1,X
1220 PRINT #1,20
1230 PRINT #1,Y
1240 NX=D*COS(A)+X
1250 NY=D*SIN(A)+Y
1260 PRINT #1,11
1270 PRINT #1,NX
1280 PRINT #1,21
1290 PRINT #1,NY
1300 X=NX
1310 Y=NY
1320 A=A+A1
1330 NEXT I%
1340 PRINT #1,0
1350 PRINT #1,"ENDSEC"
```

```

1360 PRINT #1,0
1370 PRINT #1,"EOF"
1380 CLOSE 1

```

The DXFIN command is relatively forgiving with respect to the format of data items. As long as a properly formatted item appears on the line on which the data is expected, DXFIN will accept it (of course, string items should not have leading spaces unless these are intended to be part of the string). The above program takes advantage of this flexibility in input format, and does not go to great effort to generate a file appearing exactly like one generated by AutoCAD.

In the case of error loading a DXF file using DXFIN, AutoCAD reports the error with a message indicating the nature of the error detected and the last line processed in the DXF file before the error was detected. This may not be the line on which the error occurred, especially in the case of such errors as omission of required groups.

C.5 Binary Drawing Interchange Files (+3)

The DXF file format described earlier in this appendix is a complete representation of an AutoCAD drawing, in an ASCII text form easily processed by other programs. However, for use by CAD/camera and other programs executed via the "external commands" facility (an ADE-3 feature; see Appendix B), a much more compact file format called "DXB" (for "drawing interchange binary") is supported.

C.5.1 DXBIN Command

To load a DXB file produced by a program such as CAD/camera, enter the DXBIN command:

Command: DXBIN
DXB file:

enter the name of the file you wish to load. Don't include a file type; ".DXB" is assumed.

C.5.2 DXB File Format

This information is for experienced programmers, and is subject to change without notice.

The format of a DXB file is as follows:

Header:	"AutoCAD DXB 1.0" CR LF ^Z 0	(19 bytes)
Data:	... Zero or more data records ...	
Terminator:	0	(1 byte)

Each data record begins with a single byte giving its type, followed by data items. In the format of items given below, "W-" before an item indicates that the item is a 16-bit integer, byte reversed in the standard 8086 style. The code "L-" indicates a 32-bit integer, also with the bytes reversed. The code "F-" before a value indicates that it is an IEEE 64-bit floating point value.

LINE:	1 W-fromx W-fromy W-tox W-toy	(9 bytes)
POINT:	2 W-x W-y	(5 bytes)
TRACE:	9 W-x1 W-y1 W-x2 W-y2 W-x3 W-y3 W-x4 W-y4	(17 bytes)
SOLID:	11 W-x1 W-y1 W-x2 W-y2 W-x3 W-y3 W-x4 W-y4	(17 bytes)
SEQEND:	17	(1 byte)
POLYLINE:	19 W-closureflag	(3 bytes)
VERTEX:	20 W-x W-y	(5 bytes)
SCALE FACTOR:	128 F-scalefac	(9 bytes)
NEW LAYER:	129 "layername" 0	(strlen("layername") + 2)
LINE EXTENSION:	130 W-tox W-toy	(5 bytes)
TRACE EXTENSION:	131 W-x3 W-y3 W-x4 W-y4	(9 bytes)
BLOCK BASE:	132 W-bx W-by	(5 bytes)
BULGE:	133 L-2h/d	(5 bytes)
WIDTH:	134 W-startw W-endw	(5 bytes)

The LINE EXTENSION item extends the last line or line extension from its "to" point to a new "to point". The trace extension item similarly extends the last trace solid, or trace extension from its x3,y3-x4,y4 ending line to a new x3,y3-x4,y4 line.

The SCALE FACTOR is a floating point value which is multiplied by all integer coordinates to obtain the floating point coordinates used by the actual entities. The initial scale factor when a file is read is 1.0. The NEW LAYER item will create a layer if none exists, giving it the same defaults as the "LAYER NEW" command, and will set that layer as the current layer for subsequent entities. At the end of the DXB file load, the layer in effect before the command will be restored.

The BLOCK BASE item specifies the base (origin) point of a Block being created. The Block base must be defined before the first entity record is encountered. If DXB is not defining a Block, this specification will be ignored.

A Polyline consists of straight segments of fixed width connecting the vertices, except as overridden by the BULGE and WIDTH items described below. The closure flag should be 0 or 1; if it is 1, then there is an implicit segment from the last vertex (immediately before the SEQEND) to the first vertex.

A BULGE item, encountered between two VERTEX items (or after the last VERTEX of a closed Polyline), indicates that the two vertices are connected by an arc, not a straight segment. If the line segment connecting the vertices would have length d , and the perpendicular distance from the midpoint of that segment to the arc is h , then the magnitude of this number is $(2 * h / d)$. The sign is negative if the arc from the first vertex to the second is clockwise. A semicircle thus has a bulge of 1 (or -1). Additionally, BULGE items are scaled by 2^{16} .

The WIDTH item indicates the starting and ending widths of the segment (straight or curved) connecting two vertices. This width stays in effect until the next width item or the SEQEND. If there is a WIDTH item between the POLYLINE item and the first VERTEX, it is stored as a default width for the Polyline; this will save considerable database space if the Polyline has several segments of this width.

Appendix D

CONFIGURING AutoCAD

Before you can use AutoCAD, you must tailor it to the graphics devices present on your computer system. Displays, digitizers, pen plotters, and printer plotters made by different manufacturers (and sometimes even by the same manufacturer) must often be handled differently; for each device, you must choose the appropriate handler (or "driver") and provide various information for its use. This process, known as "configuration", is accomplished using the "Configure AutoCAD" task on the Main Menu. If AutoCAD detects that it has not yet been configured, it automatically invokes this task.

The configuration task can also be used to set various defaults and operating parameters to suit your particular needs. Once AutoCAD has been installed, you may occasionally wish to change some of these parameters, to modify a device driver, or to change to a new device driver (if you buy a new plotter, for example). These changes are also accomplished using the "Configure AutoCAD" Main Menu task.

D.1 Configuration Menu

When the "Configure AutoCAD" task is selected from the Main Menu, the Configurator gains control and displays its menu:

Configuration menu

0. Exit to Main Menu
1. Show current configuration
2. Allow I/O port configuration

3. Configure video display
4. Configure digitizer
5. Configure plotter
6. Configure printer plotter
7. Configure system console
8. Configure operating parameters

Enter selection:

To select a configuration task, simply enter the task number, followed by RETURN. When the task is complete, AutoCAD returns to the Configuration Menu and asks you to select another task. Continue selecting tasks until you are satisfied with the configuration as it stands; then select Task 0 to exit from the Configurator.

In each task, reasonable defaults are provided for all options. The defaults are displayed within corner brackets, "< >"; you can choose the default simply by pressing space or RETURN. (Space and RETURN are equivalent throughout the Configurator.)

Task 0. Exit to Main Menu

This task exits from the Configurator and returns to AutoCAD's Main Menu. Before doing so, however, it asks whether you really want to change the old configuration.

Task 1. Show current configuration

Select this task to display AutoCAD's current configuration.

Task 2. Allow I/O port configuration

AutoCAD must be told the address and characteristics of the input/output (I/O) port to which your digitizer, pen plotter, printer plotter, and (perhaps) display are connected. Reasonable defaults are provided, however, and the ports are often of no concern to you. Therefore, the Configurator usually does not ask about I/O ports or include them in the "Show current configuration" display. However, Task 2 lets you indicate that you want to specify I/O port characteristics. It asks you

Do you wish to do I/O port configuration? <N>

If you answer "yes", it displays the current configuration, including I/O ports, and instructs AutoCAD to allow you to deal with I/O ports in subsequent tasks. Specifically, Task 1 will include I/O ports in its display, and Tasks 4 and 5 (and sometimes Task 3) will allow you to configure I/O ports for their respective devices. You do not actually configure the ports from within Task 2.

If your computer has standard names for its serial and parallel I/O ports, you can use them during port configuration.

The digitizer and the pen plotter (or even the video display and the pen plotter) can share the same I/O port, since AutoCAD does not use these devices at the same time. A switch or some other means of swapping cables must be provided if this type of connection is necessary. However, the video display and digitizer can never be connected to the same I/O port. If such a condition is detected, the Configurator displays the message:

I/O conflict: video display and digitizer are on the same port

and does not permit this configuration to be used. The "allow I/O port configuration" switch is turned on automatically, so that you can see and reconfigure the I/O ports in question. Also, the printer plotter must not share its port with any other device.

Task 3. Configure video display

On some computers, AutoCAD can operate with a variety of graphics displays. This task lists the available displays and asks you to indicate which one you wish to use. You may also be prompted to set various parameters associated with the display. For instance, if a display has a TouchPen (tm) capability, you can choose whether or not to use it. The particular displays with which AutoCAD can operate on your computer, and more details on their configuration and installation, may be found in your AutoCAD Installation Guide / User Guide Supplement.

The ADE-1 and ADE-2 packages include features that may be used with most video displays; the "mode status line" (ADE-1; see Section 8.7), and the ability to disable the screen menu and

prompt areas (ADE-2). If your copy of AutoCAD includes the appropriate package, the Configurator allows you to customize these features to your liking.

Status Line You can choose to enable or disable the mode status line. The default for most displays is "enabled".

Menu/Prompt Areas On some displays, the screen menu can be disabled to make more room for graphics. Likewise, the command prompt area can be disabled to enlarge the drawing area still further. On single-screen systems, turning off the prompt area is not recommended.

Dot Aspect Ratio Correction

On many video displays, the physical resolution (dot density) is different in the horizontal and vertical directions. The ratio of horizontal to vertical dots-per-inch is called the *dot aspect ratio*. In order to draw round circles and square squares, AutoCAD must take this aspect ratio into account.

A default correction factor is built into the display driver for each video display supported by AutoCAD, but variations from monitor to monitor do occur. Therefore, provisions are made for you to adjust AutoCAD's aspect ratio correction factor. The Configurator prompts:

If you have previously measured the height and width of a "square" on your graphics screen, you may use these measurements to correct the aspect ratio.

Would you like to do so? <N>

The first time you configure a display device, respond "N" (to use the default correction factor). If you later find that circles drawn with AutoCAD appear oblong on the monitor or that squares do not appear square, you can reconfigure and adjust the aspect ratio. To do this, instruct AutoCAD to draw a square, and then measure the lengths of the resulting horizontal and vertical lines. For instance, a square 5 units high and 5 units wide might be 2.0 inches high and 2.2 inches wide on the monitor.

After measuring the dimensions, exit from the Drawing Editor and return to Configurator Task 3. This time, answer "yes" when the Configurator asks if you want to adjust the aspect ratio. AutoCAD then asks for the two lengths you measured; it uses these lengths internally to adjust the aspect ratio. For instance, you might enter 2.0 inches for the vertical length and 2.2 inches for the horizontal length. If your original measurements were not sufficiently accurate, you can repeat this process until you are satisfied with the appearance of the display.

If for some reason you want to return to the default aspect ratio initially configured for your display device, simply reconfigure and tell AutoCAD that you want to choose a new display device; then select the same one. This will reset the aspect ratio correction factor to its initial value.

Task 4. Configure digitizer

Several digitizers (tablets and mice) are supported by AutoCAD, and the list is still growing. Few of these devices work exactly the same (as seen by the program), so this task is provided

to let you designate which device you have. A list of the available digitizers is displayed. When you choose one, additional information may be requested. (If you have no digitizer, you should so indicate by selecting "None".)

Your AutoCAD Installation Guide / User Guide Supplement has the latest list of supported digitizers, as well as their configuration and installation requirements.

Task 5. Configure plotter

AutoCAD supports several pen plotters, and more are being added all the time. Few of these plotters work exactly the same (from the software standpoint), so this task is provided to let you designate which plotter you have. A list of the available plotters is displayed. When you choose one, additional information may be requested. (If you have no plotter, you should so indicate by selecting "None".)

Your AutoCAD Installation Guide / User Guide Supplement has the latest list of supported plotters and their configuration and installation requirements.

If you select a plotter other than "None", the Configurator will ask you to specify the initial values for various plot specifications and parameters. These are described in Chapter 13.

One additional question will be asked if the plotter you choose supports just one pen:

Do you want to change pens while plotting? <N>

If you reply "Y", AutoCAD will pause during a plot each time a new pen is needed, and will prompt you to insert the new pen. You should then wait for the plotter to stop, and change the pen accordingly (see Chapter 13).

AutoCAD's plot routine normally assumes that your plotter meets the manufacturer's specifications. For instance, a 10-inch line in a drawing, plotted at a scale of 1:1, should be exactly 10 inches long on the paper. The Configurator allows you to adjust for any scaling discrepancy of your plotter. It displays:

If you have previously measured the lengths of a horizontal
and a vertical line that were plotted to a specific scale,
you may use these measurements to calibrate your plotter.

Would you like to calibrate your plotter? <N>

The first time you configure a particular plotter, respond "N" (to use the default calibration). If you later find that plotted output does not "measure up", you can reconfigure and change the calibration. To do this, instruct AutoCAD to plot a 1-inch by 1-inch square, and then measure the lengths of the resulting horizontal and vertical lines on the paper.

After measuring the dimensions, return to Configurator Task 3. This time, answer "yes" when the Configurator asks if you want to calibrate the plotter. AutoCAD then asks for the measured and correct lengths of the lines.

Enter measured length of horizontal line <1.0>

Enter correct length of horizontal line <1.0>

Enter measured length of vertical line <measured-value-above>

Enter correct length of vertical line <correct-value-above>

For instance, you might enter 1.1 inches for the measured vertical length and 1.2 inches for the measured horizontal length. If your original measurements were not sufficiently accurate, you can repeat this process until you are satisfied with the appearance of the plot.

If for some reason you want to return to the default calibration initially configured for your plotter, simply reconfigure and tell AutoCAD that you want to choose a new plotter; then select the same one. This will reset the calibration to its initial value.

Task 6. Configure printer plotter

This task is similar to Configuration Task 5, but allows you to select any one of several printer plotters for use with AutoCAD. You can configure one printer plotter *and* one pen plotter. As for Task 5, "None" is a valid choice.

You can adjust the calibration of the printer plotter as described above for pen plotters.

Task 7. Configure system console

On some computers, AutoCAD must be told about certain characteristics of the system console. If this is necessary on your computer, appropriate self-explanatory questions are asked.

Task 8. Configure operating parameters

This task displays a submenu:

Operating parameters menu

0. Exit to configuration menu
1. Alarm on error
2. Initial drawing setup

Enter selection:

The operating parameter subtasks are:

Subtask 0 This subtask exits from Task 7, and returns to the Configuration Menu.

Subtask 1 If you wish to hear an audible alarm whenever AutoCAD detects an invalid entry, you can enable it using this subtask.

Subtask 2 This subtask allows you to specify the default prototype drawing for creation of new drawings. The Configurator prompts:

Enter name of default prototype file for new drawings
or . for none <current>:

You can always override the default prototype file during the "Create new drawing" Main Menu task. If you want to set the default to no prototype drawing at all, enter a single period in response to the prompt.

D.2 Error Recovery

Although AutoCAD attempts to validate the choices you make and provides reasonable defaults for everything, mistakes can occur. Therefore, three levels of error recovery are provided.

- o Entering a CTRL C during any configuration task discards all changes made since you selected that task, and returns to the most recent menu.
- o Upon exiting from the Configurator, you are asked whether you really want to save the new configuration.
- o Configuration data is kept in a disk file named ACAD.CFG. If you are updating an existing configuration, the old file is retained, with its name changed to ACAD.BAK. If you produce a bad configuration, you can delete its ACAD.CFG file, rename ACAD.BAK as ACAD.CFG, and start over. Note that if you have changed device drivers, you must go through the device selection process again, but you should be able to use the defaults from your old configuration file simply by entering RETURN for most of the prompts.

Once you have a working version of AutoCAD, make a backup copy of the disk, and keep it in a safe place. Be sure to include your AutoCAD serial number and the Autodesk Inc. copyright notice on the label. You can recover using this disk if all good copies of ACAD.CFG are lost.

Of course, if you have saved the original AutoCAD release disks as recommended, you can start fresh by using them.

Appendix E

UPGRADING TO VERSION 2.1

This appendix contains important information for users upgrading to AutoCAD Version 2.1 from an earlier version.

E.1 Compatibility

Version 2.1 of AutoCAD can be used with drawings, slides, and DXF files that were originally produced on an older version of AutoCAD. In some cases, conversion of an old drawing to the new format is required; Main Menu task 8 is provided for this purpose. Note, however, that drawings, slides, and DXF files produced or edited by AutoCAD Version 2.1 *cannot* be used with older versions of AutoCAD. That is, drawings, slides, and DXF files are *upward compatible*, but not *downward compatible*.

E.2 Upgrading From Version 2.0

If you are upgrading from Version 2.0, you will find changes in the prompt sequences for several commands, and in the numbers assigned to some menu items. No drawing conversion is necessary, but, as noted above, once a drawing has been edited by Version 2.1, it can no longer be used with an earlier version.

Command Changes in Version 2.1

The prompt sequences and actions of several commands have been changed in Version 2.1. If you have custom menus or command scripts, they may need revisions.

The introduction of the "interactive selection" mechanism described in Section 5.1 caused changes in most of the edit and inquiry commands (ARRAY, BREAK, CHANGE, COPY, ERASE, FILLET, LIST, MIRROR, and MOVE), and in other commands that ask you to select objects (BLOCK, HATCH, and WBLOCK). If you are selecting only the Last object, or the objects in just one Window, an extra space or RETURN is now needed to inform the command that the selection set is complete.

The prototype drawing is now used to establish the initial values for nearly all editing modes, limits, and so forth. Normal creation of a new drawing always uses a prototype drawing as input; if you want to create a new drawing without benefit of a prototype, you must respond with a trailing "=" when Main Menu Task 1 asks for the name of the drawing to be created. See Chapter 2.

Other assorted improvements were made, as listed below.

ARC The prompts have been changed to eliminate single-letter abbreviations, but the responses remain the same.

ATTEXT (+2) For CDF and SDF format extracts, the template file name is now requested before the extract (output) file name.

CHANGE	The prompts have changed significantly. See Section 5.2.
CIRCLE	The prompts have been changed to eliminate single-letter abbreviations, but the responses remain the same.
COPY	Now asks you to select objects first, before requesting the displacement.
DELAY	The duration of the delay has been calibrated to be about one millisecond per increment on all computers. There may still be differences, but they will be less severe.
FILES	A new "Copy file" selection has been added to the File Utilities Menu.
FILLET (+1)	For ADE-3 users, a new "Polyline" option is provided, to fillet an entire Polyline.
INSERT	For ADE-3 users, a new option permits specification of the Z scale for inserting 3D objects.
LAYER	For ADE-3 users, new "Freeze" and "Thaw" options are provided.
MOVE	Now asks you to select objects first, before requesting the displacement.
PLOT	The prompts and options have changed significantly, to reduce confusion and add capabilities. See Chapter 13.
QTEXT	When quick text mode is on, Text items are drawn as rectangles, not just as two parallel lines.
UNITS (+1)	Additional selections have been added for the format of angles.

Command Menu Changes in Version 2.1

The standard menu file supplied with AutoCAD has been updated to include all the new commands, and to agree with the command changes listed above.

Drawing Interchange Files

New items were added to the DXF format for Polylines, 3D elevation and thickness, BLIPMODE status, and various other new features. See Appendix C.

Miscellaneous Changes in Version 2.1

- o A new "Printer plot" item has been added to the Main Menu, and a new "Configure printer plotter" item has been added to the Configurator's menu.
- o The "Configure operating parameters" subtask of AutoCAD's Configurator has been substantially trimmed. Most of the parameters that you used to specify using that subtask (and many others that were not previously configurable) are now obtained from the prototype drawing.

- o The option to configure the axis (+1) outside the drawing area has been removed. The axis is now always inside the drawing area.

E.3 Upgrading From a Version Older Than 2.0

If you are upgrading to AutoCAD Version 2.1 from a version older than 2.0, the items listed below apply in addition to those mentioned above.

Drawing Conversion

Drawings created by AutoCAD versions older than 2.0 must be converted to the new format if they are to be edited or plotted by Version 2.1. Main Menu Task 8 is provided for this purpose. Conversion is invoked automatically if you attempt to edit (Task 2) or plot (Task 3 or 4) an old-format drawing, but drawings you expect to insert into other drawings must be converted explicitly because the INSERT command does not perform automatic conversion.

One of the major enhancements in Version 2.0 was the change from 127 layers, with numbers 1-127, to an unlimited number of layers, with names of up to 31 characters. Also, the variable layer for Block insertions has changed from 127 to "0". When an old-format drawing is converted, the layers in the new drawing are given names equal to the old drawing's layer numbers, with the exception of layer 127. Entities on old layer 127 move to new layer "0" of the converted drawing.

Another significant change in Version 2.0 was the addition of full support for multiple text fonts and styles. Older versions of AutoCAD supported multiple text fonts, but only one at a time per drawing; the most recently loaded font was used for all text. To avoid changing the appearance of the drawing when it is converted to the new format, the font file associated with the "STANDARD" Text style is set to the last font used in the old drawing. Text style entries are also created for all the fonts that were loaded, but all Text entities in the converted drawing use the "STANDARD" style.

The text fonts that had been named "ROMAN-S" and "ROMAN-C" are now called "SIMPLEX" and "COMPLEX". During drawing conversion, any references to the old font names are changed accordingly.

When conversion of a drawing is complete, some manual editing may be desirable to apply the finishing touches that AutoCAD cannot do automatically. For instance, you can:

- o use the RENAME command to give meaningful names to the drawing's layers,
- o use the STYLE command to change the font file for the "STANDARD" Text style,
- o CHANGE selected Text entities to different styles, or
- o use the MENU command to associate a special menu file with the drawing.

Command Changes in Version 2.0

The prompt sequences and actions of several commands changed in Version 2.0; if you have custom menus or command scripts, they may need revisions.

DIM (+1) Dimensioning was enhanced significantly; the operation of the DIM command changed considerably. See Section 10.1.

END	In previous versions, the END command saved the vector file <i>drawing-name.RF</i> if possible. To conserve disk space, this feature was removed from the END command. If you do want to save the vector file, use the new ENDSV command. See Section 3.2.
FILES	The File Utility Menu items for listing menu, shape, and pattern files were eliminated. You can use the "user specified files" item to list these files.
HATCH (+1)	The HATCH command now has a "repeat" capability, bypassing all prompts for pattern name, etc. It proceeds directly to selection of the hatch area boundary. Also, the Snap base point (ADE-2 feature) is now used as the reference point for all hatch lines. See Section 10.2.
INSERT	The variable layer for Block insertion was changed from 127 to "0". Blocks drawn on layer "0" are inserted on the current layer, acquiring that layer's color and linetype. See Chapter 9.
LAYER	Everyone's favorite command was extensively modified, and it should now be easier to use. "LAYER SET x" is now required to change the current layer to "x", and "LAYER NEW y" is used to create a new layer named "y". Since layers now have names, "LAYER COLOR *" no longer makes sense and has been dropped. The concept of the "current color" has also been dropped; new layers are simply assigned color number 7 (white). See Chapter 7.
LINE	Invalid responses to the "To point:" prompt no longer cause the LINE command to terminate; the message "*Invalid*" is printed and the input is ignored. To terminate the LINE command, give a null response (space or RETURN) to the "To point:" prompt, or use CTRL C. See Chapter 4.
LOAD	The LOAD command is no longer used to load Text fonts; the STYLE command now performs this function. LOAD is still used for Shape files, however. See Section 4.9 and Section 4.10.
PAN	In response to suggestions from numerous users, the meaning of the PAN displacement was reversed. Now the displacement specifies how the drawing should shift with respect to the screen.
UNITS (+1)	After you have specified the format and precision of distances and coordinates, the UNITS command now asks if you'd like angles to be in degrees, minutes, and seconds. See Section 3.6.

Command Menu Changes in Version 2.0

In Version 2.0, several changes were made regarding command menus:

- o Separate screen, tablet, and pointer device button menus are now supported. The tablet can have up to four menu areas. See Chapter 12 and Appendix B.
- o The format of menu files was changed. See Appendix B. (Old-style menu files are no longer documented, but they are still supported.)

- o A comprehensive new standard menu file was supplied with Version 2.0 of AutoCAD, with separate sections for the screen, button, and tablet menus. The screen menu section uses multiple submenus; some practice may be needed to use it effectively.
- o Versions of AutoCAD prior to 2.0 searched for a menu file with the same name as the drawing and automatically loaded that menu. The default menu was used if no menu file was found with the same name as the drawing.

Version 2.0 used the default menu for new drawings, but you can use the MENU command (Section 3.9) to associate a different menu with a drawing. The file specified on the most recent MENU command is remembered along with your drawing, and is loaded whenever that drawing is subsequently edited.

Drawing Interchange Files

Drawing Interchange (DXF) files can no longer be created or loaded from the Main Menu. New drawing editor commands DXFIN and DXFOUT perform these functions.

The DXF format has undergone major changes in order to make it extensible and easy to process in any imaginable programming language. Creation of the older DXF format is possible, as is loading of old-format DXF files, but we encourage you to revise your DXF analysis programs, at your earliest opportunity, to honor the new format. The old format is no longer documented in this manual; keep your old AutoCAD manual for information on the old DXF format.

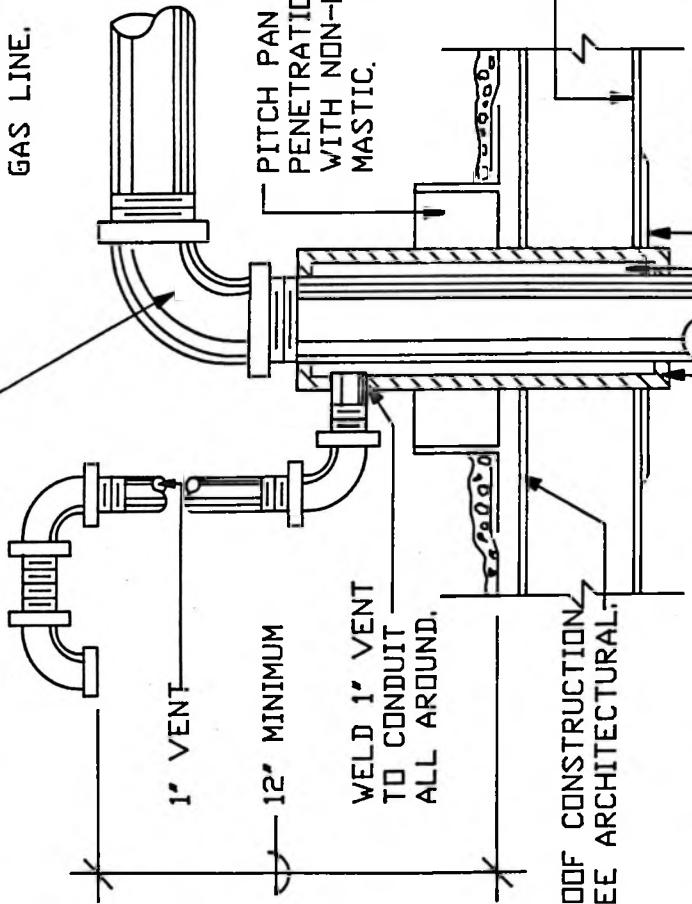
NOTE: If Version 2.0 or higher is used to load or produce an old-format DXF file and the drawing contains Shapes, the Shape's name follows its insertion parameters (on a separate line). Some editing or program changes may be required; this is the only incompatibility with the older DXF format.

Miscellaneous Changes in Version 2.0

- o If you have defined your own Shapes or Fonts, they must be compiled using Main Menu Task 7 before they can be used with AutoCAD Version 2.0 or higher. See Appendix B.
- o AutoCAD now maintains an index file, ACAD.HDX, in order to make the HELP facility fast. If you change the contents of the ACAD.HLP text file, delete the index file; the next time you use AutoCAD, it will rebuild the index file based on your revised text.
- o The method used to fill Solids and Traces was changed to improve performance. Solids and Traces can no longer be selected by pointing to the solid-filled region; you must now point to an edge of the object.
- o For ADE-1 users, the Configurator no longer asks whether you want a continuously updated coordinate display. The display is now static until CTRL D is used to change it. See Section 8.7.
- o DBLIST no longer lists Block Definitions.

NOTE: CONDUIT AND VENT
SHALL BE OF THE
SAME MATERIAL AS
GAS LINE.

GAS LINE, SEE
PLANS FOR SIZE.



WELD CAP TO GAS LINE
AND CONDUIT ALL AROUND
BOTH ENDS.

CONDUIT 2 PIPE SIZES
LARGER THAN GAS LINE.

VENTILATED CONDUIT DETAIL

SCALE: NONE

Appendix F

ERRORS AND PROBLEM REPORTS

AutoCAD is a rather complex software package and is used for a wide variety of drafting tasks with numerous computer / pointing device / plotter combinations. Although any new version of AutoCAD must pass a barrage of quality assurance tests before it is released for sale, unforeseen uses or configurations occasionally result in problems. We are committed to solving any such problems, but we need your help to do so. This appendix describes AutoCAD's error handling, and suggests steps you can use to assist us in solving any problem you encounter.

F.1 Invalid Input

The error message you are most likely to see when using AutoCAD is:

Invalid

This message appears if AutoCAD has asked you for one type of information, but you have entered something else. For instance, if AutoCAD has prompted you to enter a point, it needs an *X,Y* coordinate pair and will not accept a simple number. Likewise, if the prompt indicates that an "On/Off" modifier is needed, a point would be invalid input.

F.2 Disk-Full Handling

When AutoCAD senses that there is insufficient disk space on the drive containing the drawing file, it displays the message:

<Disk almost full!>

If you see this message, you should either exit the Drawing Editor using the END or QUIT command, as desired, or use the FILES command to delete some unneeded files from the drive containing the drawing file.

F.3 Disaster Handling

AutoCAD updates the drawing database after each command that adds, deletes, or changes anything in the drawing. The drawing database is thus always up-to-date when you begin a new command. If, for some reason, AutoCAD encounters a problem during execution of a command and cannot continue, one of the following messages will be displayed:

INTERNAL ERROR (followed by a bunch of numbers)

or

FATAL ERROR

Usually, an additional short message will be displayed giving an error code. Following this, AutoCAD will display:

AutoCAD cannot continue, but all changes to your drawing made up to the start of the last command can be saved.

Do you want to save you drawing changes? <Y>

If you respond "Y" (the default), AutoCAD will write the drawing database to disk before exiting. If this operation is successful, the message:

DRAWING FILE SUCCESSFULLY SAVED

is displayed just before AutoCAD exits to the operating system.

F.4 END Command Error Handling

AutoCAD actually uses a temporary name for the drawing file while a drawing is being edited. When you issue the END (or ENDSV) command, the drawing file is renamed, and is given the name you specified (after first changing the file type of the existing drawing file, if any, to ".BAK"). If the END (or ENDSV) command cannot rename the drawing file as you specified, it will try another name, DWG.\$\$. If that also fails, the temporary name EF.\$AC is retained. Appropriate messages will be displayed to let you know what's happening. In any case, your drawing will be saved.

F.5 Reporting Problems

If you encounter problems when using AutoCAD, contact your dealer. You can assist in solving the problem if you follow the guidelines listed below.

- o Supply the serial number from your AutoCAD release disks. This information is required whenever you need assistance.
- o List out the ACADn.MID files (where "n" is a number) from each of your AutoCAD release disks. These files contain text identifying the master diskettes that were used to produce your release diskettes, as in:

Number: 000019
Type: IBM 2.1 ADE-3
Date: 10/31/84
Time: 15:00

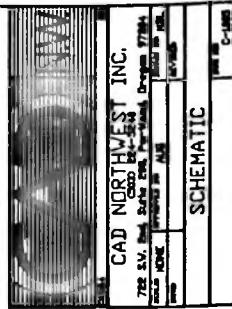
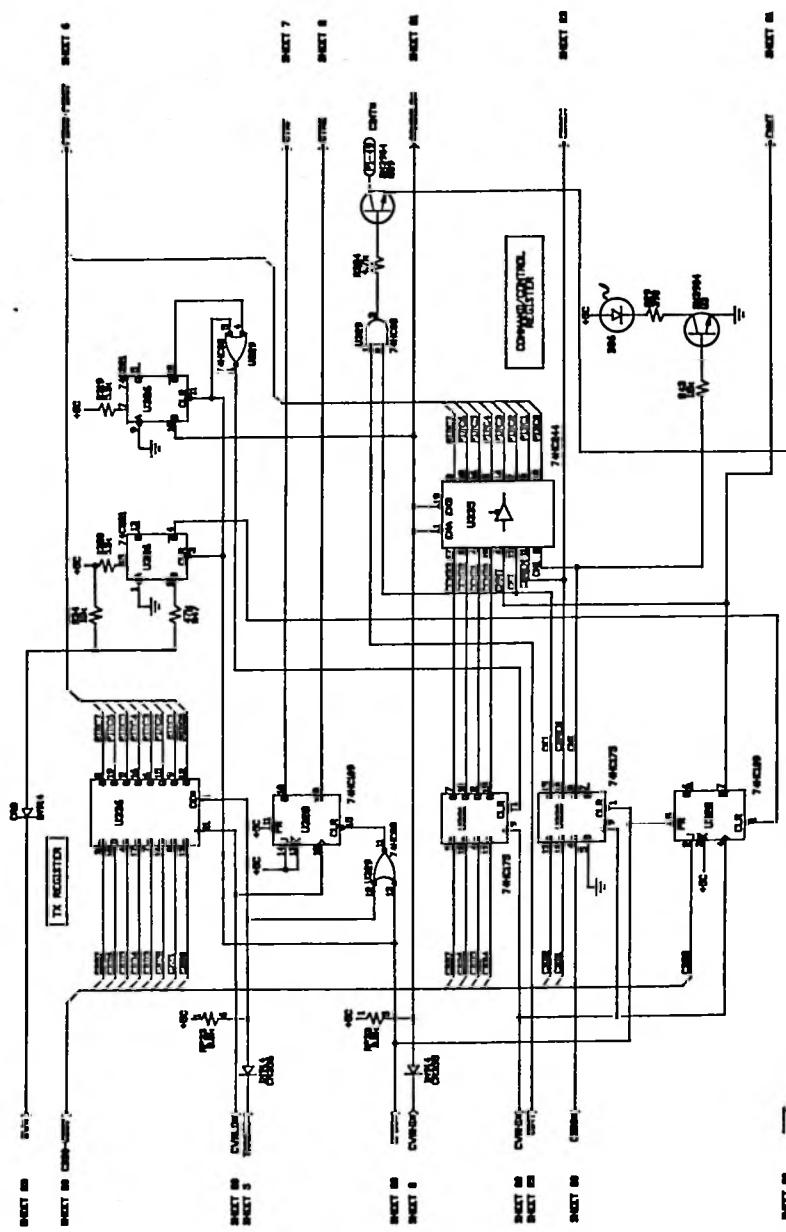
Carefully write down the information from these files and include it with your problem report.

- o Provide a complete description of the hardware environment in which you are using AutoCAD (computer, memory size, math co-processor, peripheral devices, etc.).
- o Specify the operating system and version being used.
- o Describe the circumstances, as best you can determine them, under which the problem occurs.

AutoCAD -- (F) ERRORS AND PROBLEM REPORTS

- o Supply the exact text of any messages accompanying the error. If AutoCAD terminates with an "INTERNAL ERROR" and displays a few lines of hexadecimal numbers, carefully write these down and include them with the problem report.
- o If the problem occurs when working with a particular drawing, provide a diskette containing a copy of the ".DWG" drawing file.
- o If the problem appears to be configuration-related, provide a diskette containing a copy of your ACAD.CFG configuration file
- o Supply any other supporting documentation (plotter output, listings, file dumps, screen dumps, etc.) that might help in diagnosis of the problem.

AutoCAD -- (F) ERRORS AND PROBLEM REPORTS



SCHEMATIC

Appendix G

REVISION HISTORY

As a service to users who are upgrading from an older version of AutoCAD, this appendix contains a brief list of the changes incorporated into recent versions.

G.1 Version 2.0 (October, 1984)

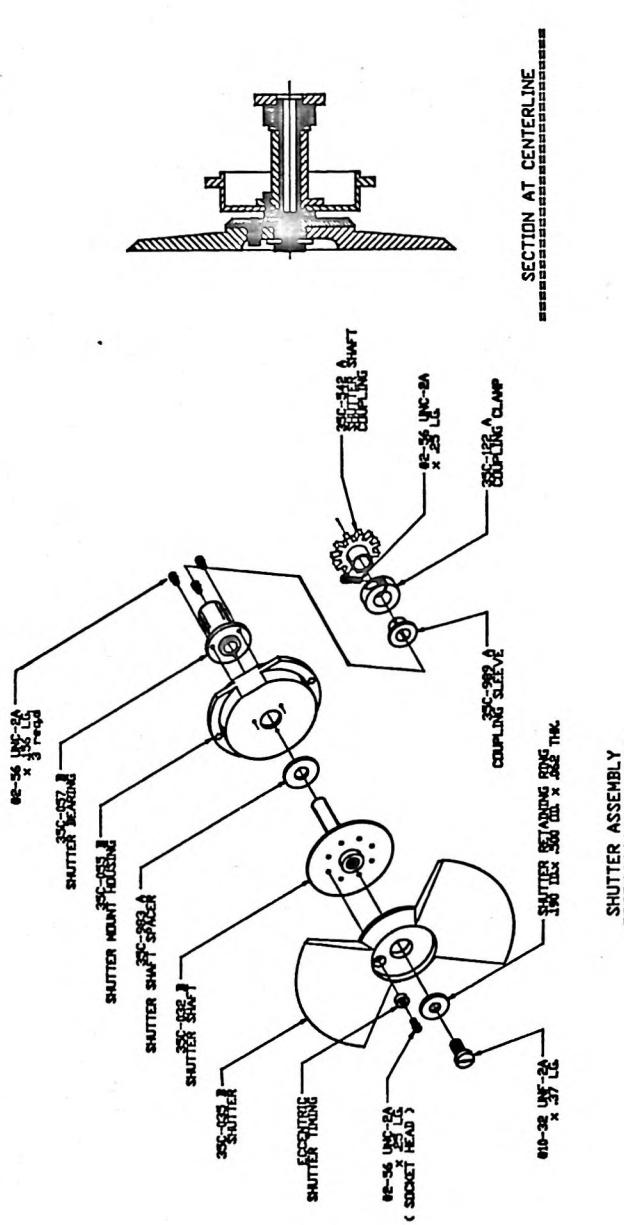
- o Added dot/dash linetype capability (LINETYPE, LTSCALE, and LAYER LTYPE commands).
- o Layers now have user-chosen names, and there is no limit to the number of layers in a drawing.
- o Layer "0" is now the variable layer for Block insertions, and is the initial current layer when a new drawing is begun.
- o Multiple text fonts can be used in a drawing.
- o Text styles: obliquing, mirroring, and expansion/compression may be applied to a text font (STYLE command).
- o Text may be underscored or overscored.
- o Text alignment type is remembered, for CHANGE command.
- o Added "quick text" mode (QTEXT command).
- o Up to four tablet menu areas can be defined.
- o A separate menu is provided for the buttons on multi-button pointing devices.
- o Menus may be segmented into submenus that can be invoked to display the options for particular commands.
- o Standard colors can be specified by number or by name.
- o Traces and Solids are now filled even if partially off-screen, and filling is faster.
- o Faster access to help.
- o Faster entity selection for editing.
- o Faster execution in general.
- o Faster loading of shapes and fonts due to shape/font compiler.
- o Multiple Shape files can be used at once.
- o The area of a digitizing tablet used for screen-pointing is now configurable.
- o "Undo" feature added to LINE command.
- o The LINE command now ignores invalid points, rather than terminating.
- o The meaning of the PAN command's displacement has been reversed.
- o New DXFIN and DXFOUT commands have replaced the Main Menu tasks to load and create drawing interchange files.
- o Seldom-used options have been removed from the FILES command.
- o A dense grid can now be canceled by means of CTRL C.
- o In previous versions, item rotation for a circular array worked only if the only item in the array was a Block. This restriction has been lifted.
- o Enhanced script mechanism (SCRIPT and RSCRIPT commands).
- o The AREA command now also displays the perimeter.
- o The DIST command now displays the angle and delta-X/Y between two points.
- o The LIST and DBLIST commands display length, angle, and delta-X/Y for lines; area and circumference for circles.
- o Added ENDSV command to save vector file upon exit.
- o New SAVE command to save changes without exiting the Drawing Editor.

AutoCAD -- (G) REVISION HISTORY

- o New slide show viewing facility (VSLIDE command). Slide creation is a new feature of the ADE-2 package.
- o The Advanced Drafting Extensions package was renamed ADE-1 and the following features were added:
 - Dimensioning has been enhanced to provide angular dimensioning, arc/circle diameter and radius, dimension lines at any desired angle, and several other requested features.
 - Angles in degrees, minutes, and seconds (UNITS command enhancement).
 - Length and angle display of rubber-band in status line.
 - The parameters used on the HATCH command are remembered and presented as defaults on the next HATCH command. A "repeat" capability has also been added; it asks for another area to hatch using the same pattern.
 - Hatching of figures bounded by arcs has been improved.
- o A new ADE-2 optional package was added. It requires ADE-1 and includes the following features:
 - Dynamic specification (dragging) for ARC, CIRCLE, SHAPE, INSERT, CHANGE, COPY, and MOVE commands.
 - Object snap (OSNAP command) for snapping to reference points of existing objects.
 - Object mirroring (MIRROR command).
 - Named views (VIEW command).
 - SNAP, GRID, and AXIS may be rotated, offset, or given differing X and Y spacings. The snap offset can be used to vary the alignment of hatch patterns.
 - Isometric grid/snap capability ("SNAP STYLE ISO" and ISOPLANE commands).
 - New MSLIDE command makes slides for slide show facility.
 - For some display devices, the screen menu and the text prompt area can be disabled, by means of the AutoCAD configuration process, to provide a larger area for graphics.
 - For some display devices, a SNAPSHOT command is provided to erase menus and prompts from the screen temporarily and center the graphics image for photographic purposes.
 - New Attributes feature. Attributes associate textual information with Block Definitions, and prompt for appropriate values at insertion time. The text may be visible or invisible.
 - Global or individual editing of Attributes is available.
 - Attributes can be extracted from the drawing for transfer to a database program.

G.2 Version 2.1 (May. 1985)

- o Interactive object selection. You can select objects in multiple windows, and add or remove objects from the selection-set before submitting the set to any edit/inquiry command.
- o Directory path names are now honored on all file names, and you can maintain multiple drawing directories or multiple AutoCAD configurations.
- o Revised and enhanced the PLOT routine.
 - Plots can now be rotated 90 degrees.
 - Optimized the changing of plotter pens and further optimized pen movement.
 - Multiple pens can now be used with single-pen plotters; AutoCAD will pause and prompt you to change the pen manually.
 - Added printer plotter output (PRPLOT command).
- o New LIMITS ON/OFF options allow you to turn the limits check off entirely.
- o A new BLIPMODE command permits control over the drawing of marker blips.
- o A prototype drawings is now used to establish the initial environment (limits, modes, etc.) for a new drawing. You can create as many prototype drawings as you wish.
- o CHANGE of multiple entities is now easier.
- o The HELP command can now pause during long help displays.
- o QTEXT now produces a rectangle rather than two parallel lines.
- o You can now copy files from the File Utility Menu (FILES command).
- o The DELAY command has been calibrated to delay about 1 millisecond per increment.
- o New GRAPHSCR and TEXTSCR commands permit scripts and menu items to flip between the text and graphics displays on single-screen systems.
- o CTRL Q can now be used to toggle printer echo on/off.
- o General speed improvements were made.
- o ADE-1 enhancements:
 - The CHAMFER command was added.
 - You can now use the UNITS command to select display and input of angles in grads or radians.
 - BREAK now allows a zero-length break, splitting an object into two entities.
 - FILLET was extended to handle Polylines (see below).
- o ADE-2 enhancements:
 - The VIEW command has a new "Window" option, permitting you to define a named view without first ZOOMing in to that view.
- o A new ADE-3 optional package was introduced. It includes the following new features:
 - "3D Level 1" for 3D visualizations with optional removal of hidden lines (ELEV, VPOINT, and HIDE commands). 3D visualizations can be edited and plotted just like 2D AutoCAD drawings.
 - Polylines (PLINE and PEDIT commands).
 - Curve fitting (PEDIT command).
 - New "Freeze" and "Thaw" options for the LAYER command.
 - User-supplied programs and operating system utilities can be executed as "external commands" while running AutoCAD.
 - CAD/camera support (DXBIN command).
 - You can define variables to hold integers, reals, strings, and points, and can use these variables (along with predefined system variables) in arithmetic expressions. These expressions can be entered in response to AutoCAD data prompts.
 - Objects can be highlighted during the selection process.



ULTRACAM
SHUTTER
ASSEMBLY

Paul Jenkins 1998

Drawn By : Paul B. Jenkins

Appendix H

AutoCAD COMMAND REFERENCE

Commands

APERTURE +2	DELAY	HELP / ?	OSNAP +2	SCRIPT
ARC	DIM +1	HIDE +3	PAN	SHAPE
AREA	DIST	ID	PEDIT +3	SHELL +3
ARRAY	DRAGMODE +2	INSERT	PLINE +3	SKETCH +1
ATTDEF +2	DXBIN +3	ISOPLANE +2	PLOT	SNAP
ATTDISP +2	DXFIN	LAYER	POINT	SOLID
ATTEDIT +2	DXFOUT	LIMITS	PRPLOT	STATUS
ATTEXT +2	ELEV +3	LINE	PURGE	STYLE
AXIS +1	END	LINETYPE	QTEXT	TABLET
BASE	ENDREP	LIST	QUIT	TEXT
BLIPMODE	ENDSV	LOAD	REDRAW	TEXTSCR
BLOCK	ERASE	LTSCALE	REGEN	TRACE
BREAK +1	FILES	MENU	REGENAUTO	UNITS +1
CHAMFER +1	FILL	MIRROR +2	RENAME	VIEW +2
CHANGE	FILLET +1	MOVE	REPEAT	VPOINT +3
CIRCLE	GRAPHSCR	MSLIDE +2	RESUME	VSLIDE
COPY	GRID	OOPS	RSCRIPT	WBLOCK
DBLIST	HATCH +1	ORTHO	SAVE	ZOOM

+1 = ADE-1 feature

+2 = ADE-2 feature

+3 = ADE-3 feature

Dimensioning Commands (+1)

ALIGNED	CENTER	EXIT	RADIUS	STATUS
ANGULAR	CONTINUE	HORIZONTAL	REDRAW	UNDO
BASELINE	DIAMETER	LEADER	ROTATED	VERTICAL

Point Entry Formats

Absolute:	x, y
Relative:	$@dx, dy$
Angle, distance:	$@dist < angle$

Repeating a Command

To repeat the last command, respond to the "Command:" prompt with a space or RETURN.

AutoCAD -- (H) COMMAND REFERENCE

Object selection:	point M L W A R U	= One object = Multiple objects = Last visible object drawn = Objects within window = Add mode: add following objects = Remove mode: remove following objects = Undo (remove objects last added)
ARC	A C D E L R blank	= Included angle = Center point = Starting direction = End point = Length of chord = Radius = (as reply to Start point) sets start point and direction as end of last Line or Arc
ARRAY	C R	= Circular array = Rectangular array
ATTDISP +2	ON OFF N	= Make all Attributes visible = Make all Attributes invisible = Normal: visibility set individually
ATTEXT +2	C D S	= CDF comma-delimited format extract = DXF format extract = SDF format extract
AXIS +1	ON OFF A <i>number</i> <i>numberX</i>	= Turn axis (ruler line) on = Turn axis off = Set aspect (differing X-Y spacings) = Set tick spacing (0 = use snap spacing) = Set spacing to multiple of snap spacing
BLIPMODE	ON OFF	= Enable temporary marker blips = Disable temporary marker blips
BLOCK	?	= List names of defined Blocks
BREAK +1	F	= Re-specify first point
CHAMFER	D P (+3)	= Set chamfer distances = Chamfer an entire Polyline
CHANGE	E (+3) L	= (as reply to Change point) change elevation = (as reply to Change point) change layer
CIRCLE	2P 3P D	= Specify by 2 endpoints of diameter = Specify by 3 points on circumference = To enter diameter instead of radius
DRAGMODE +2	ON OFF	= Dynamic specification enabled for SHAPE, ARC, CIRCLE, INSERT, MOVE, COPY, CHANGE = Dynamic specification disabled

AutoCAD -- (H) COMMAND REFERENCE

FILL	ON	= Solids, Traces, and wide Polylines filled
	OFF	= Solids, Traces, and wide Polylines outlined
FILLET +1	P (+3)	= Fillet an entire Polyline
	R	= Set fillet radius
GRID	ON	= Turn grid on
	OFF	= Turn grid off
	A (+2)	= Set grid aspect (differing X-Y spacings)
	number	= Set grid spacing (0 = use snap spacing)
	numberX	= Set spacing to multiple of snap spacing
HATCH +1	I	= Ignore internal structure
	N	= Normal style: turn hatch lines off and on as internal structure is encountered
	O	= Hatch outermost portion only
INSERT	name	= Load file "name" as a Block
	name=f	= Create Block "name" from file "f"
	*name	= Retain individual part entities
	?	= List names of defined Blocks
	C	= (as reply to X scale prompt) specifies scale via two points (Corner specification of scale)
	XYZ (+3)	= (as reply to X scale prompt) readies INSERT for X, Y, and Z scales
ISOPLANE +2	L	= Left plane
	R	= Right plane
	T	= Top plane
	blank	= Toggle to next plane
LAYER	SET a	= Set current layer to "a"
	NEW a,b	= Create new layers "a" and "b"
	ON a,b	= Turn on layers "a" and "b"
	OFF a,b	= Turn off layers "a" and "b"
	FREEZE a,b (+3)	= Freeze layers "a" and "b"
	THAW a,b (+3)	= Thaw layers "a" and "b"
	COLOR c	= Set specified layers to color "c"
	LTYPE t	= Set specified layers to linetype "t"
	?	= List layers, colors, and linetypes
LIMITS	point	= Set lower left drawing limit, prompt for upper right
	ON	= Enable limits checking
	OFF	= Disable limits checking
LINE	blank	= (as reply to From point) start at end of previous Line or Arc
	C	= (as reply to To point) close polygon
	U	= (as reply to To point) undo segment
LINETYPE	C	= Create linetype definition
	L	= Load linetype definition
	?	= List linetype library
ORTHO	ON	= Force lines to horizontal or vertical
	OFF	= Do not constrain lines

AutoCAD -- (H) COMMAND REFERENCE

OSNAP +2	CEN = Center of Arc or Circle END = Closest endpoint of Arc or Line INS = Insertion point of Text/Block/Shape INT = Intersection of Line/Arc/Circle MID = Midpoint of Arc or Line NEA = Nearest point of Arc/Circle/Line/Point NOD = Node (point) NON = None (off) PER = Perpendicular to Arc/Line/Circle QUA = Quadrant point of Arc or Circle QUI = Quick mode (first find, not closest) TAN = Tangent to Arc or Circle
----------	--

PEDIT +3	C = Close an open Polyline E = Edit vertex (see below for suboptions) F = Fit curve to Polyline J = Join to Polyline O = Open a closed Polyline U = Uncurve Polyline W = Set uniform width for Polyline X = Exit PEDIT command
----------	---

During Polyline vertex editing:

B	= Set first vertex for Break
G	= Go (perform Break or Straighten operation)
I	= Insert new vertex after current one
M	= Move current vertex
N	= Make next vertex current
P	= Make previous vertex current
R	= Regenerate the Polyline
S	= Set first vertex for Straighten
T	= Set tangent direction for current vertex
W	= Set new width for segment following current vertex
X	= Exit vertex editing, or cancel Break/Straighten request

PLINE +3	H = Set new half-width U = Undo previous segment W = Set new line width RETURN = Exit PLINE command
----------	--

In line mode:

A	= Switch to arc mode
C	= Close with straight segment
L	= Specify segment length (continue previous segment)

In arc mode:

A	= Included angle
CE	= Center point
CL	= Close with arc segment
D	= Starting direction
L	= Chord length, or switch to line mode
R	= Radius
S	= Second point of three-point arc

AutoCAD -- (H) COMMAND REFERENCE

PURGE	A B LA LT SH ST	= Purge all unused named objects = Purge unused Blocks = Purge unused layers = Purge unused linetypes = Purge unused Shape files = Purge unused Text styles
QTEXT	ON OFF	= Quick text mode on = Quick text mode off
REGENAUTO	ON OFF	= Allow automatic regens = Prevent automatic regens
RENAME	B LA LT S V	= Rename Block = Rename layer = Rename linetype = Rename Text style = Rename named view
SHAPE	?	= List available Shape names
SKETCH +1	C E P Q R X .	= Connect: restart sketch at end point = Erase (back up over) temporary lines = Raise/lower sketching pen = Discard temporary lines, exit Sketch = Record temporary lines, remain in Sketch
SNAP	number ON OFF A (+2) R (+2) S (+2)	= Set snap alignment resolution = Align designated points = Do not align designated points = Set aspect (differing X-Y spacings) = Rotate snap grid = Select style, standard or isometric
TABLET	ON OFF CAL CFG	= Turn tablet mode on = Turn tablet mode off = Calibrate tablet = Configure tablet menus, pointing area
TEXT	A C R S	= Fit Text between two points, choose appropriate height = Center Text = Right justify Text = Select Text style
VIEW +2	D R S W ?	= Delete named view = Restore named view to screen = Save current display as named view = Save specified window as named view = List named views
VPOINT +3	RETURN <i>x,y,z</i>	= Select view point via compass and axes tripod = Specifies view point

AutoCAD -- (H) COMMAND REFERENCE

VSLIDE	<i>file</i>	= View slide
	* <i>file</i>	= Preload slide, next VSLIDE will view
WBLOCK	<i>name</i>	= Write specified Block Definition
	=	= Block name same as file name
	*	= Write entire drawing
	blank	= Write selected objects
ZOOM	<i>number</i>	= Multiplier from original scale
	<i>numberX</i>	= Multiplier from current scale
	A	= All
	C	= Center
	E	= Extents ("drawing uses")
	L	= Lower left corner
	P	= Previous
	W	= Window

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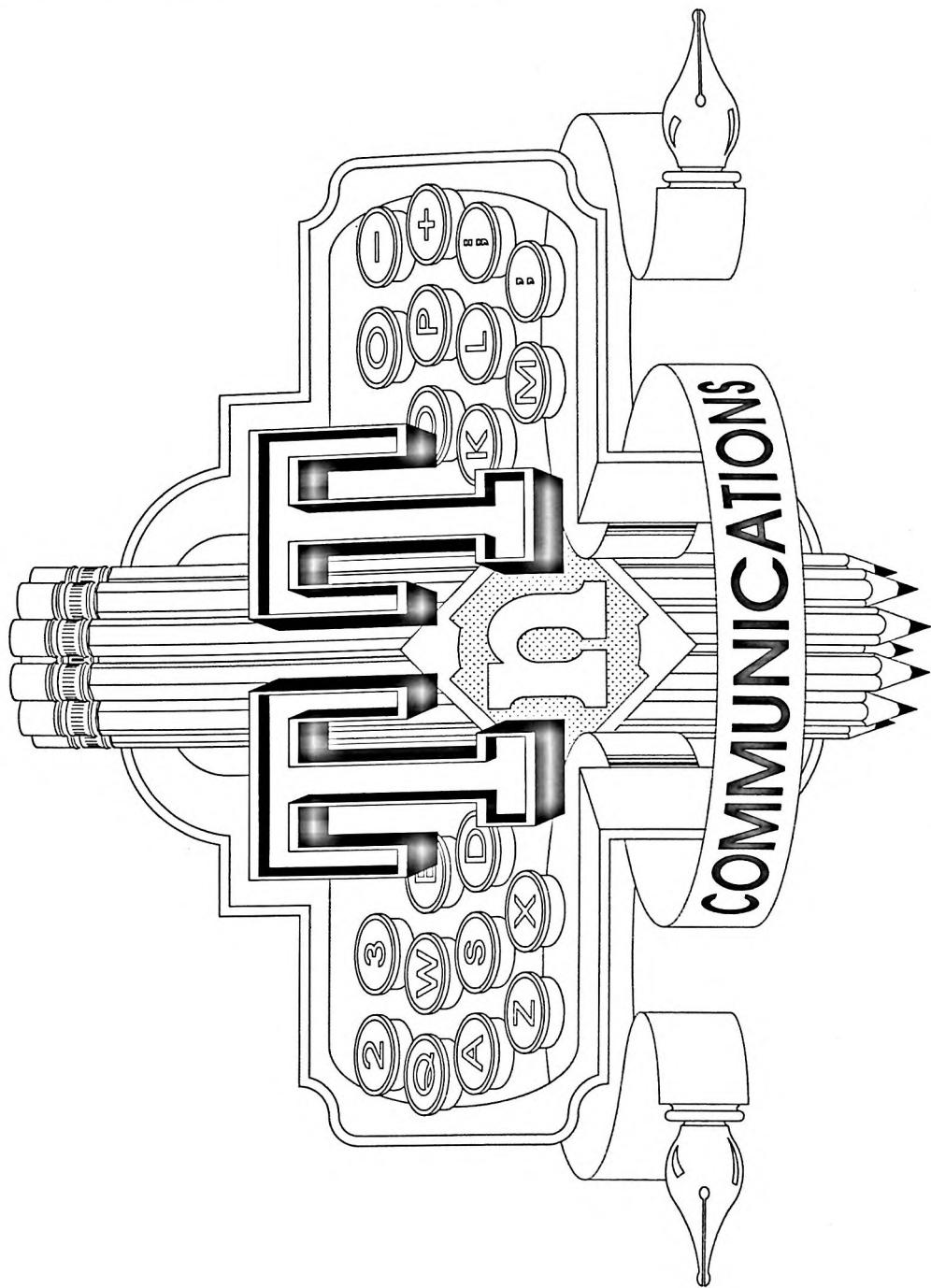
 Previous 120

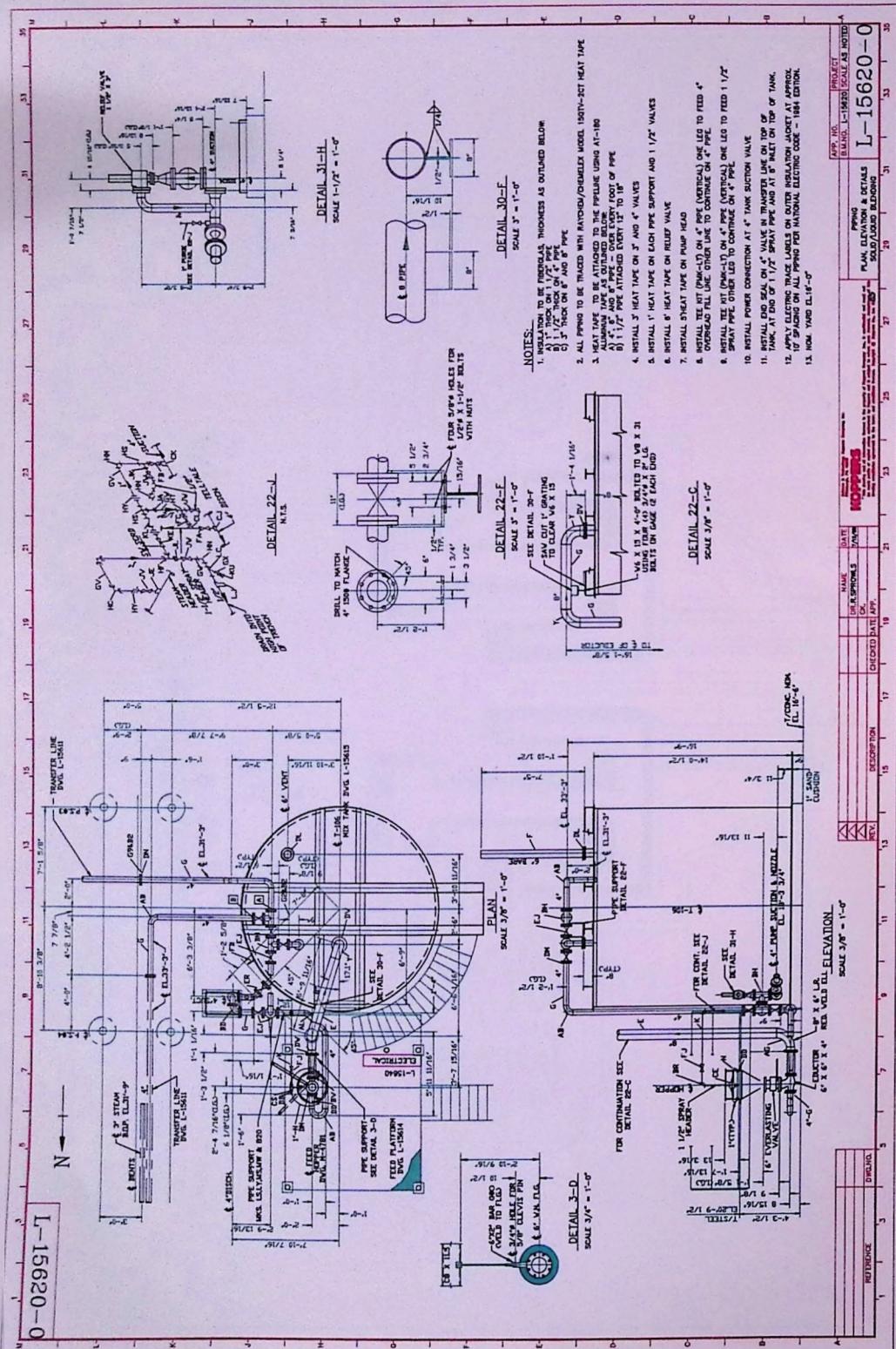
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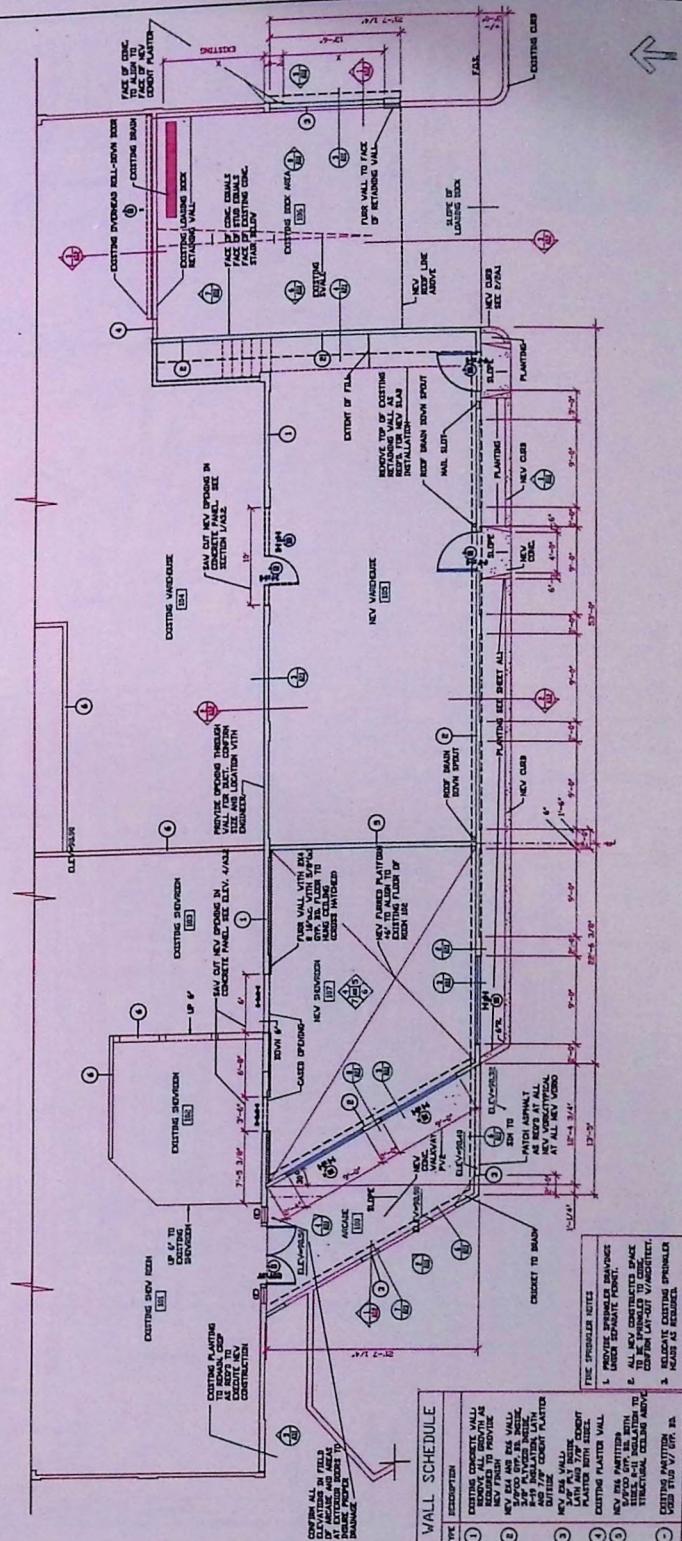
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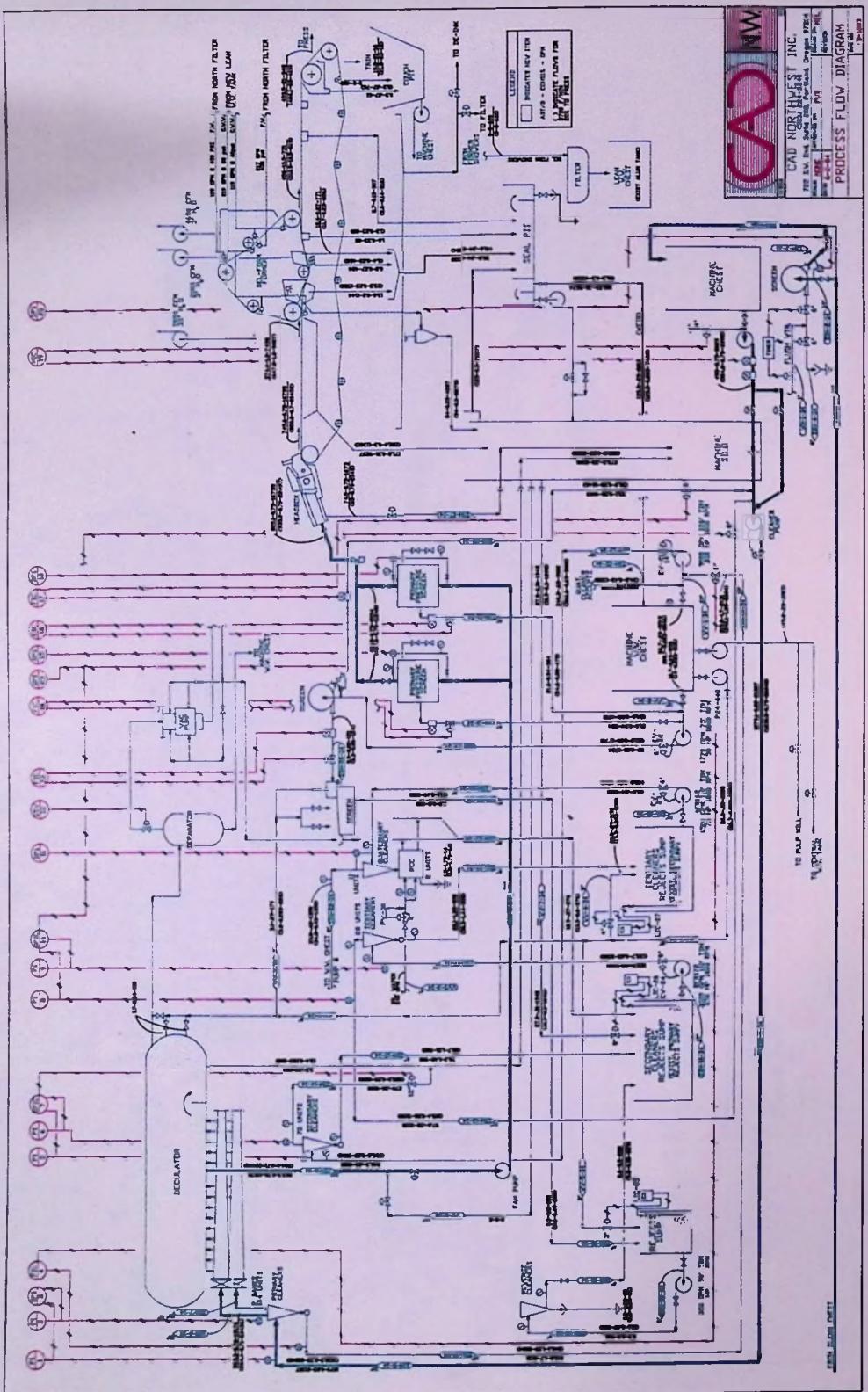






WALL SCHEDULE

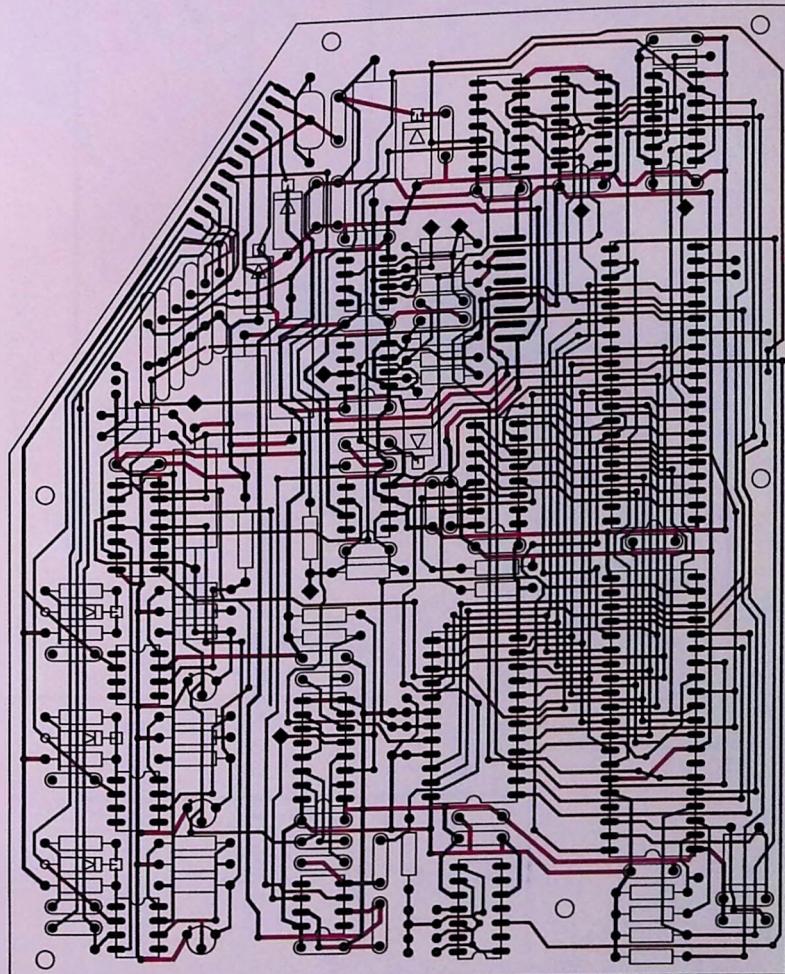
DESCRIPTION		CRICKET TO BAG
(1)	CREATING CONCRETE WALL REINFORCED ALONG ALL SIX FRESH CUTS	TYPE SPIDERLETS 1. PROTECT SPIDERLETS WITH SEPARATE FOAM
(2)	CREATING CONCRETE PLASTER ON THE INSIDE OF THE PLANTER	2. ALL NEW CONCRETE SPACE SHOULD BE SPIDERLED WITH SPIDERLETS OR SPIDERLETS
(3)	NEW CONCRETE WALL, LAY CONCRETE PLASTER WITH SIZE	3. SELECT CREATING SPIDERLET AS REQUIRED
(4)	CREATING PLASTER WALL ON THE INSIDE OF THE PLANTER	
(5)	CREATING CONCRETE CEILING ON THE INSIDE OF THE PLANTER	
(6)	CREATING PLASTER WALL ON THE INSIDE OF THE PLANTER	

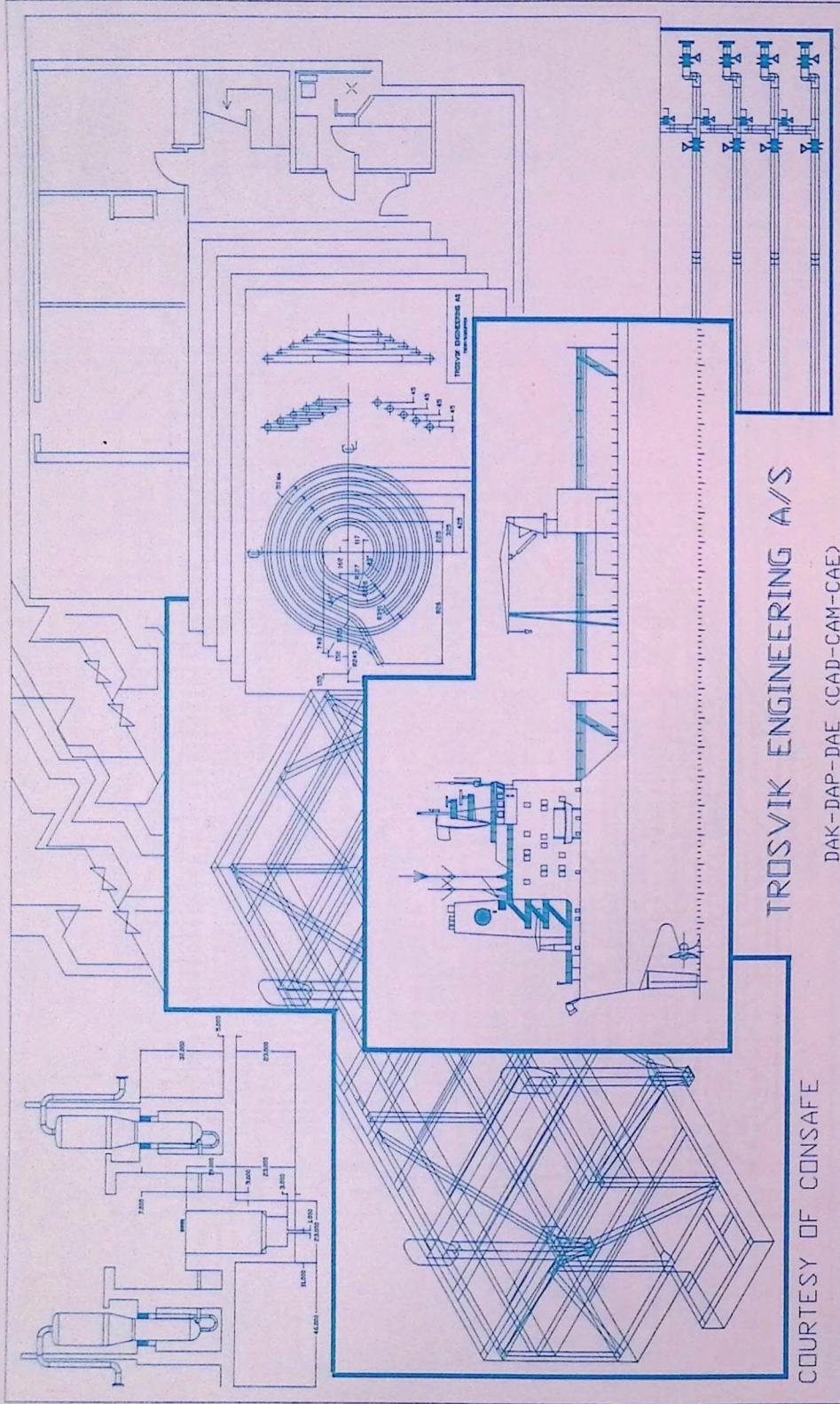


CAD NW

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CIRCUIT BOARD LAYOUT

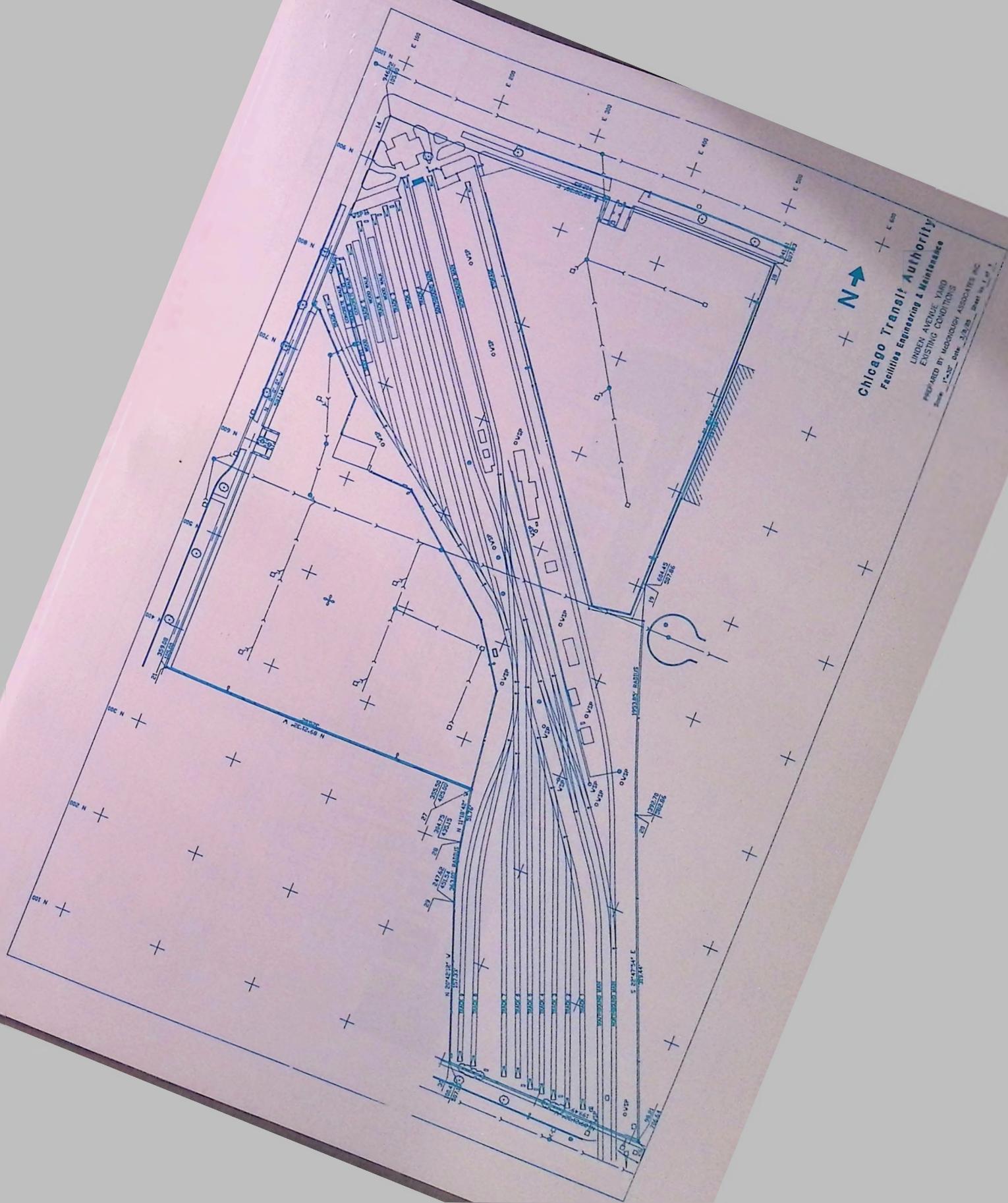




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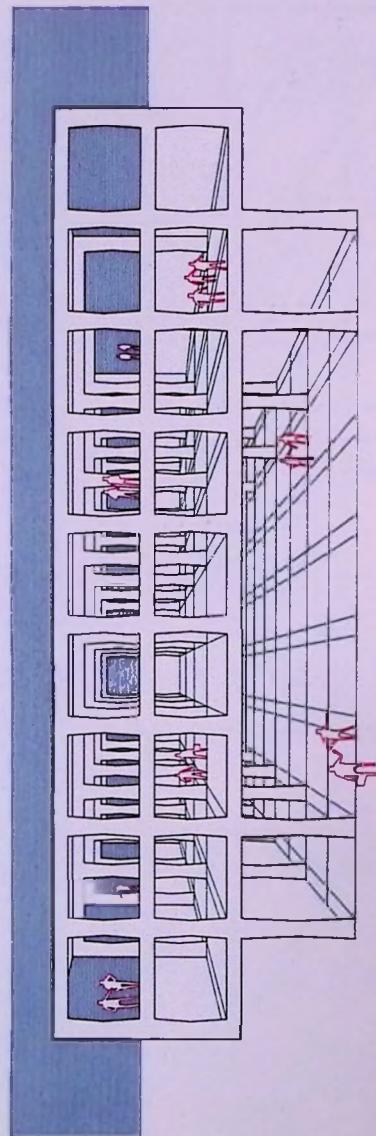
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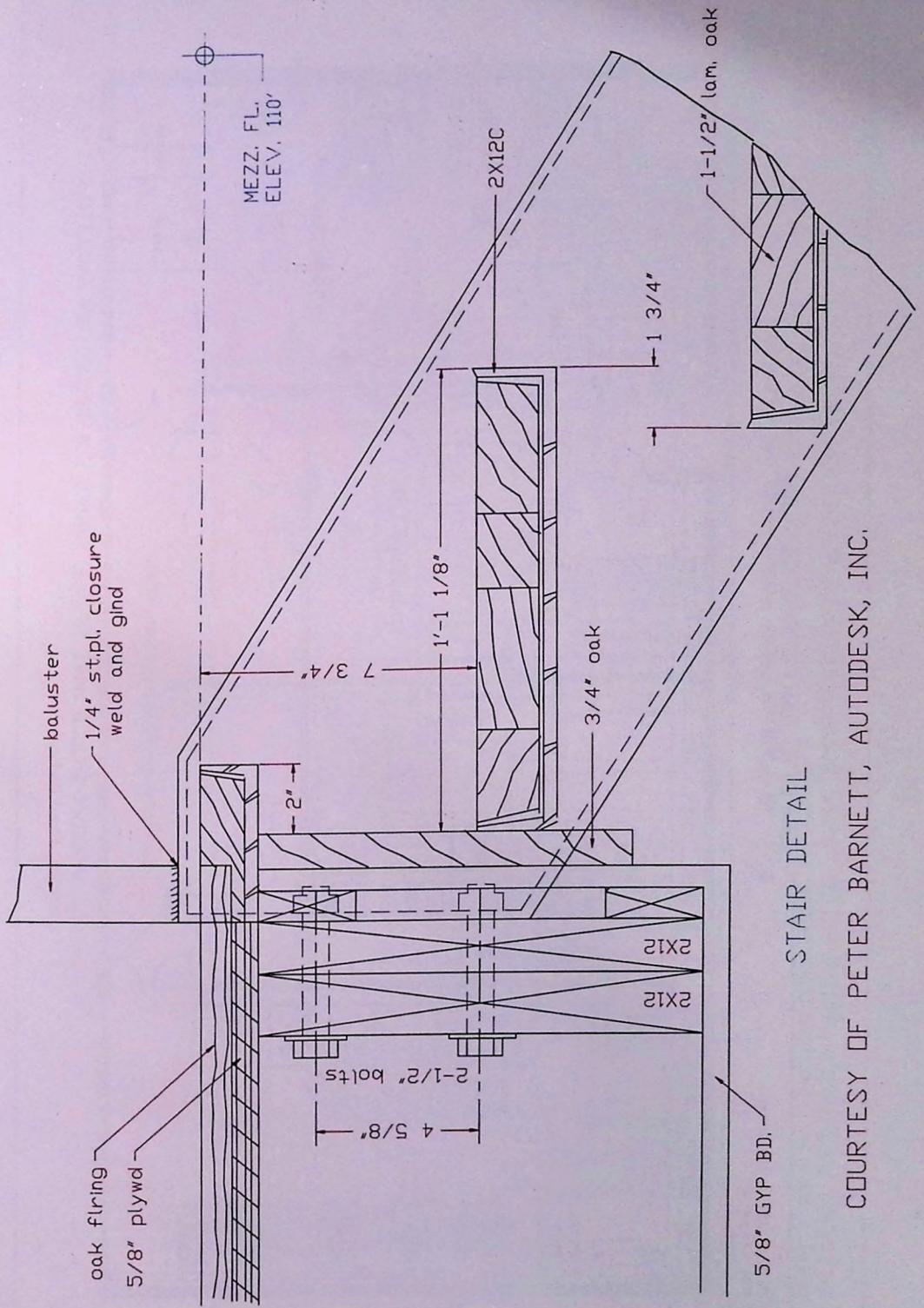
This drawing is copied from
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by Heinrich Engel
(Praeger, New York 1967)



REVISIONS		
Rev	Subject	Date

Project	AutoCAD Demos		
Multi-panel Frame	Drawn	Checked	Owner
Structure	No Scale	T Armour	ACD-007

4/5/85

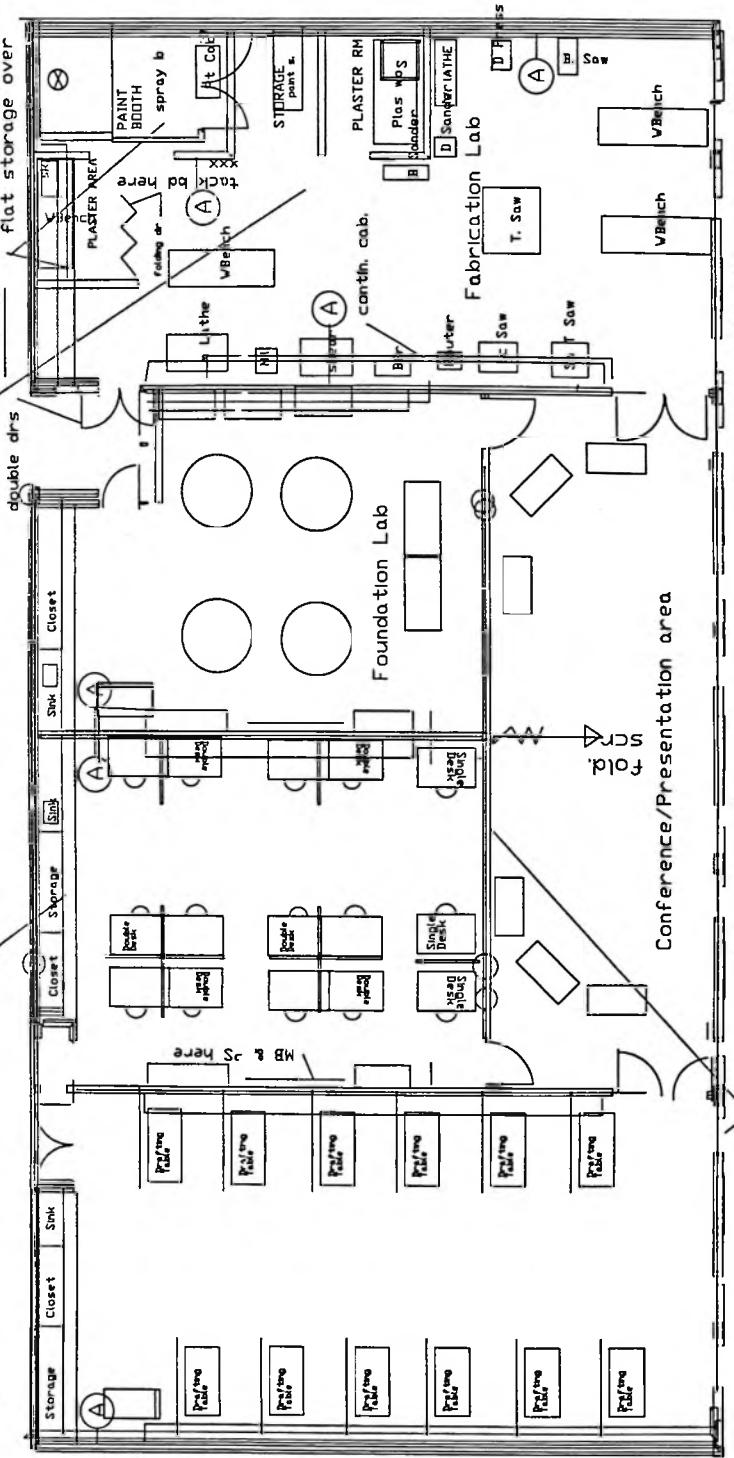


**WESTERN WASHINGTON UNIVERSITY
GENERAL FLOOR PLAN INDUSTRIAL DESIGN PROGRAM**

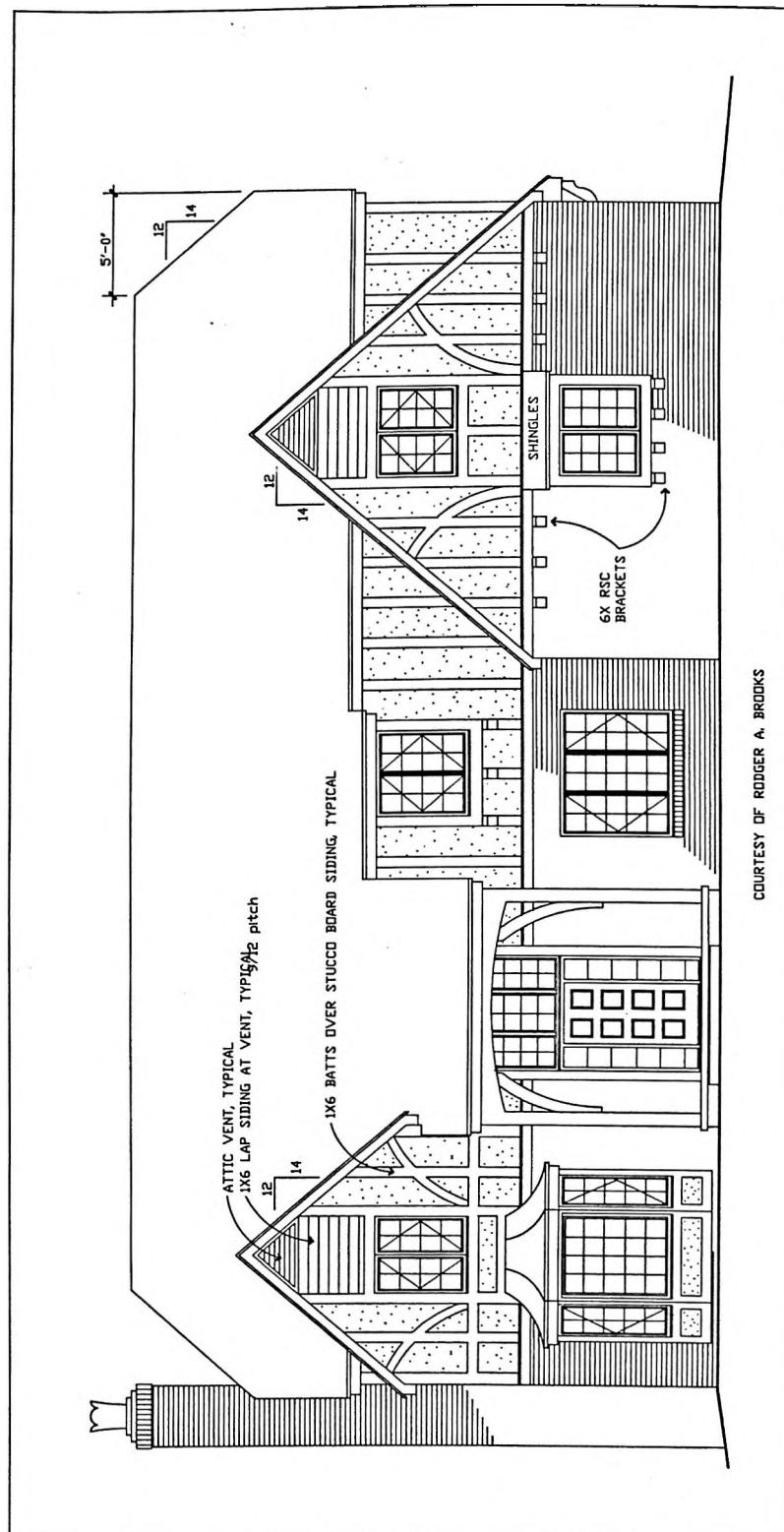
Provide AIR to all areas
See circled A

New Building INDUSTRIAL DESIGN AREA

TRANSFORM ALL FURRED IN SPACES INTO STORAGE UNITS
ALL WALLS PREFERRED TO BE TACKABLE RATHER THAN GYPSUM VB



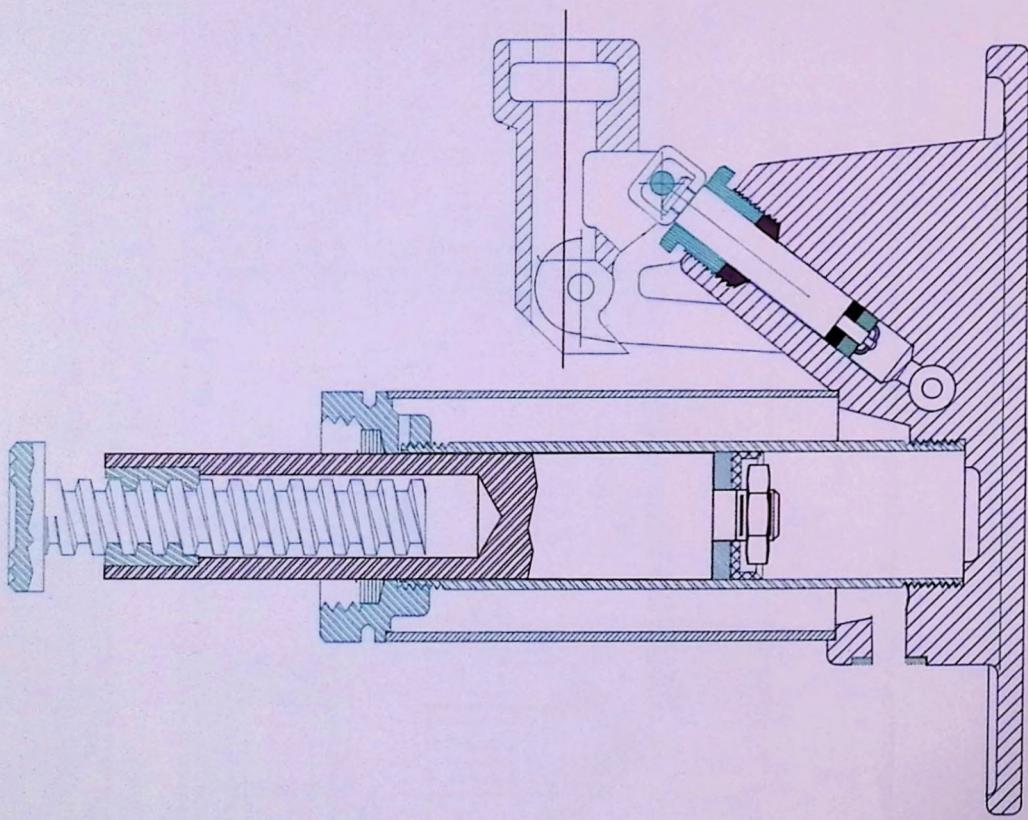
ALL WINDOWS SHOULD HAVE OPAQUE SHADES FOR PROJECTION FACILITIES
except fabrication lab



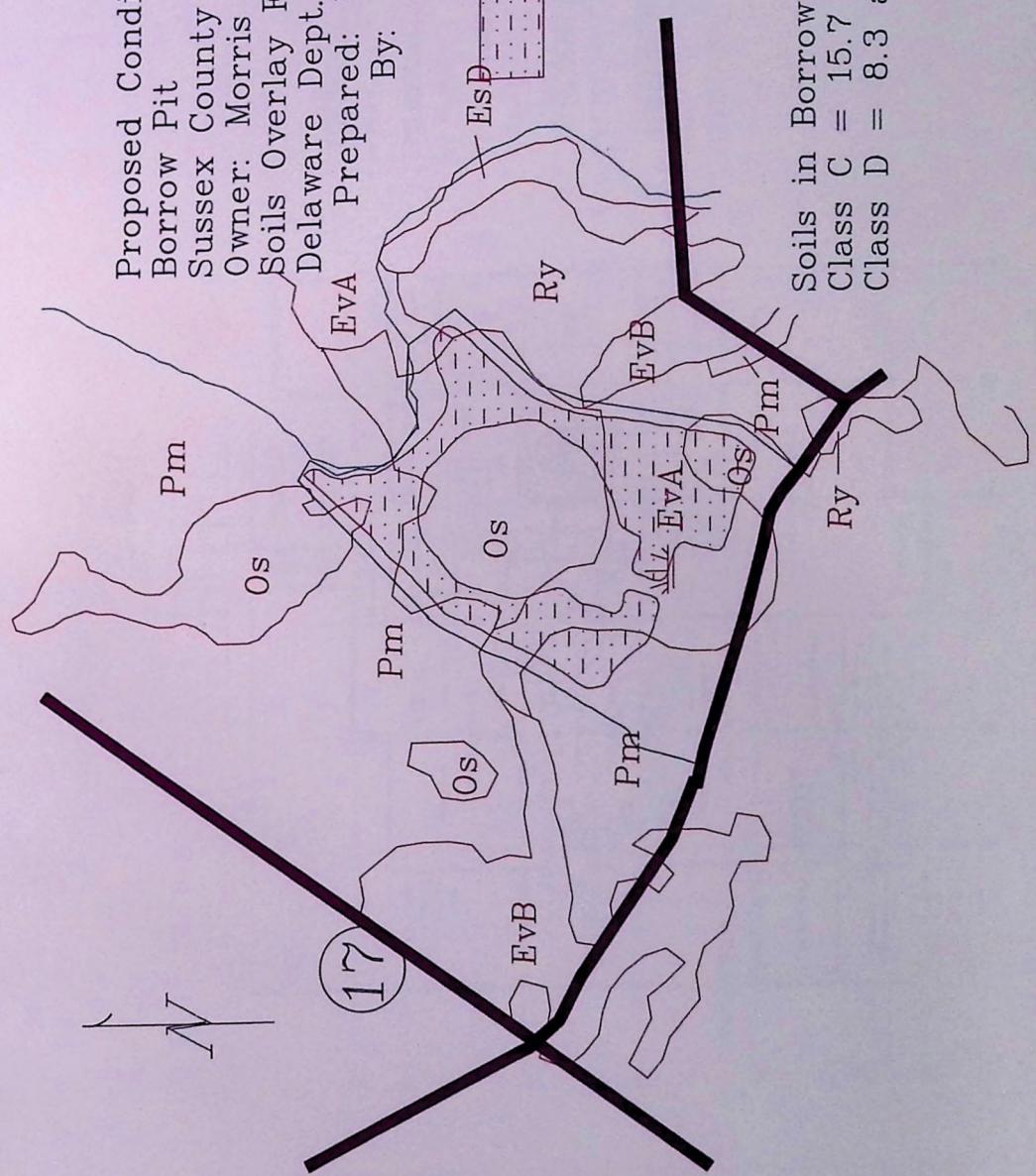
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JACK ASSEMBLY

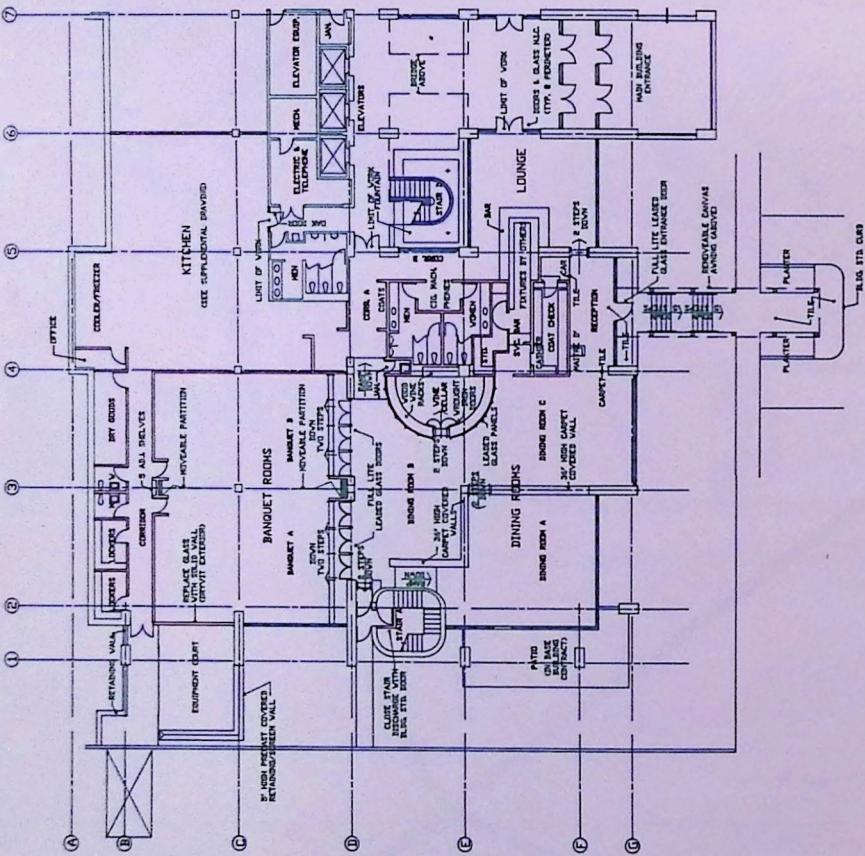


Proposed Conditional Use
Borrow Pit
Sussex County
Owner: Morris Justice
Soils Overlay F/SCS Survey
Delaware Dept. of Agriculture
EVA Prepared: May 24, 1985
By: M. McGrath



Soils in Borrow Pit Area
Class C = 15.7 ac.
Class D = 8.3 ac.

ARCHITECTURAL PLAN	
dc1000 Restaurant FOSTER PLAZA BUILDING SEVEN	A-1 MAY 14, 1993 W/W W/ LAMINATE SLATE W/W W/ LAMINATE WHITE/GRANITE



PARTIAL FIRST FLOOR PLAN
LW/P-107

NOTES



AUTODESK, INC.

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SAUSALITO, CA 94965