Preface

The AutoCAD (tm) 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:

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

AutoCAD was designed for use by any facility with a microcomputer [1]. It can benefit both the professional and home computer user. 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, the features described in Chapter 9 are provided only with the optional Advanced Drafting Extensions package. In addition, some features differ depending on the characteristics of the computer system being used. Consult the accompanying AutoCAD-86 Installation Guide / User Guide Supplement for your computer for exceptions, additions, and other machine-dependent information.

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1 Versions of AutoCAD are currently available for CP/M-80 systems and for several 8086/8088-based systems using the PC-DOS, MS-DOS, or CP/M-86 operating systems.

This manual was prepared and published in November, 1983, and is based on Release 1.40 of the AutoCAD-86 (tm) drafting package.

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Chapter 1

INTRODUCTION TO AutoCAD

The AutoCAD (tm) drafting package is a powerful drawing aid. It follows your instructions to help quickly produce the exact drawing you want. It offers features that let you easily correct drawing errors and make even large revisions without redoing an entire drawing. It produces clean, precise final artwork. AutoCAD works for you. It doesn't put anything into your artwork "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.

1.1 General Operation

The following is a synopsis of how AutoCAD is used. Complete operating instructions begin in Chapter 2.

A **Graphics Monitor** is used to display your drawing. Everything done to the drawing is shown on this monitor so you can watch the progress each step of the way.

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

AutoCAD provides commands that let you modify the drawing; entities can be erased, moved, or copied to form repeated patterns. Other commands let you change the view of the drawing displayed on the graphics monitor, provide special information about the drawing, and offer drawing aids that help you position entities accurately. In some cases all you have to do is enter a command and the function will be performed. The most that's required is that you follow the command with some minimal, easily entered specifications.

The effect of every action you take is seen immediately on the graphics monitor.

A plotter is used to draw your artwork on paper. Like the other functions, plotting is accomplished with a simple command.

That's the essence of how AutoCAD is used. See Section 2.5 for a summary of the AutoCAD commands. Detailed command descriptions begin in Chapter 3.

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. On some systems, the text display screen can also be used for graphics, while on others the graphics monitor is a separate piece of equipment. A plotter may be connected to the system to produce hard copy of a drawing.

We suggest you add a pointing device such as a light pen, Touchpen (tm), "mouse", or digitizing tablet. Any one of these devices will provide the means for instant command and point entry. Keyboard entry is easy but requires a few keystrokes; pointing at the screen and pushing a button is even easier. In addition to locating points and entering commands, a digitizing tablet may be used to "copy" existing drawings. Further information on pointing devices and their use with AutoCAD may be found later in this chapter, and in Chapter 2. Additional information on digitizing tablets is presented in Chapter 8. Not all pointing devices are available on all systems. Supported configurations for your computer, and instructions for installing AutoCAD on it, may be found in the accompanying AutoCAD-86 Installation Guide / User Guide Supplement.

1.2 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 may be any size you desire, specified by any unit of measurement you want, and 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 **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 may be indicated by an X and Y coordinate pair. (Locations are measured from the Ø,Ø point, normally at the lower-left corner of the drawing.)

Drawing Units

As noted, entities in the drawing are positioned on coordinate points. For example, a line is drawn 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 corresponds to whatever form of measurement your drawing requires. It may be inches, feet, centimeters, angstroms, whatever. When the drawing is plotted you may specify a scale factor to plot each unit the exact size you want.

Display

The term display is used in two ways. First, it is used to indicate the graphics monitor screen upon which your picture is shown. More often in AutoCAD, the term display refers to that portion of the drawing which is currently being displayed.

Zooming

The display may be **zoomed** in or out to magnify or shrink the visible image of the drawing. When zoomed out you can see a large portion of the picture; in order to evaluate the drawing as a whole, for example. Zooming in shows a small portion of the picture in large size so you can examine detail.

The display is used as a **window** through which you may 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 screen will vary with different zoom levels. When zoomed out the distance between coordinates will appear 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 zoomed in the distance between coordinates will appear larger. (The one unit line may appear to be an inch or more long.) In both cases, the

absolute distance between the coordinates is constant. Only the screen display changes.

Resolution

Physical Resolution refers to the amount of detail that may 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 artwork. The resolution of your display device is specified by "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 may be specified far more accurately than the display screen would normally allow. The accuracy of the final plotted output will only be limited by the resolution of the plotter, not the resolution of the display screen.

Coordinates you enter may be aligned ("snapped") to the nearest point on an optionally visible grid. The spacing of the grid points is called the **Snap Resolution.** This is the resolution of the coordinates in the drawing data base, and 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 Ø.1 inch centres. In that case, AutoCAD would be set to a snap resolution of Ø.1. Snap resolution may be set to any "fineness" your artwork requires. It may be changed to any value at any time, or turned off entirely for "free style" drawings.

1.3 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.

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") which allows you to initiate various tasks, such as creating new drawings, modifying stored drawings, producing plots, and so on. The menu provides access to various parts of AutoCAD, such as the interactive Drawing Editor and the plotter interface.

Interactive Drawing Editor

The Drawing Editor displays your picture and provides commands to create, modify, view and plot drawings. AutoCAD automatically loads the Drawing Editor when certain tasks are selected from the main menu.

Entities

These are pre-defined elements that can be put into a drawing with a single command. The program offers the following built-in entity types:

Lines	Traces	Points	Circles
Arcs	Text	Solids	Shapes

Traces are solid lines of any width you specify. Shapes are small "objects" you can create and store in special Shape files. You may call them into the drawing, at a specified point, just like any of the other entities. Since Shapes are "special purpose" objects, there are some restrictions on the type of entities they may contain. (See Section 7.3 and Appendix A.)

Command Functions

A selection of easily entered AutoCAD commands provide the operations you need to work on drawings. The types of functions you can perform are briefly outlined on page 1. A summary of available commands appears in Section 2.5.

Screen Menu

A menu may be displayed on the graphics monitor while the Drawing Editor is active. This menu allows command entry simply by pointing to the command on the display screen. You can construct your own custom menus; see Section 8.1 and Appendix C.

Drawing Insertion

This powerful feature lets you insert an existing AutoCAD drawing (stored on disk) into the drawing currently being created. Thus you can interactively construct a drawing "part", store it in a regular AutoCAD drawing file, then insert as many copies of it as you like in subsequent drawings. Parts you create for insertion may contain any number and type of entities. Once inserted, the part is treated as an entity and may be moved or erased as a single unit. (See Section 7.1.)

Layers and Colours

Parts of the drawing may be assigned to any of 127 different layers. The layering concept is similar to the transparent overlays used in many drafting applications. Layering allows you to view and plot related drawings separately. For instance, a drawing file could contain the layout for a house on one layer, labeling for the floor plan on a second layer, and structural specifications on yet another layer. The drawing file can be considered to contain two different "types" of drawings. The floor plan and labeling can be plotted (or displayed) as one drawing; then the plan can be plotted again with the structural specifications added.

A colour can be associated with each layer. The colour is simply a number from 1 to 127; what it means depends on the output device. (Consult your AutoCAD-86 Installation Guide / User Guide Supplement.) For monochrome devices, it has no effect. For multi-pen plotters, however, a particular pen and line type can be associated with each colour number. You might want to make all traces of a given width the same colour, or assign a special colour to "hidden" lines.

Graphic Input (Pointing) Devices

A pointing device may be used for quick command entry and to locate points in the drawing. Several types of pointers may be used with AutoCAD. (Not all of these devices are available on all systems. See the accompanying AutoCAD-86 Installation Guide / User Guide Supplement for the pointing devices supported on your computer.)

Light pen — With a light pen, you can point directly at an area of the Graphics Monitor's screen to enter points or select commands from a "screen menu". Crosshairs appear on the screen and follow the pen until you select a point or menu item by releasing or deactivating the pen.

TouchPen (tm) — This device operates much like a light pen, but works by touching a special panel on the face of the Graphics Monitor. Selection of a point or menu item is accomplished by sliding the pen tip along the panel until the crosshairs are positioned at the desired point; then lift the pen tip from the screen.

Mouse — A "mouse" is operated by moving it around the tabletop while 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.

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; the tablet may be aligned with the coordinate system of an existing paper drawing so you can use AutoCAD to produce an exact copy of it, and an area of the tablet may be set aside for use with a "tablet menu" (see Chapter 8).

Point and Command Entry Flexibiity

Commands can be entered, and points in the drawing specified, in a variety of ways. From the keyboard, points can be specified by typing in absolute coordinates or coordinates relative to the last point specified; or by keyboard pointing, which utilizes keyboard control keys to move crosshairs around on the graphics monitor so you can visually position to the desired point. As described above, a graphic input device may also be used to designate points.

Tablet Menu

A menu of AutoCAD commands may be placed on the digitizing tablet, permitting a command to be entered simply by pointing to it with the stylus and pushing a button.

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.

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 and stored on disk.

Drawing Interchange Capability

An AutoCAD drawing created on your computer system may be saved in a form suitable for interchange with other computer systems. This capability can be used to move a drawing to another AutoCAD system, where it may be displayed and edited.

The drawing interchange file may also be processed by user-written programs. Many users will never have occasion to do this, but AutoCAD offers this special capability for those who may need it. A program to read this file might list the number of windows, of each size, in a house; or count the number of desks and chairs in an office; or calculate points on a curve.

Plotting

A hardcopy of the drawing can be plotted at any stage in its development. "Check-plots" may 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.

Arrays

Rectangular or circular patterns, or arrays, of objects may be easily constructed.

File Directory Access

Disk directories may be listed, and files may be deleted or renamed without exiting AutoCAD.

Advanced Drafting Extensions (ADEX)

A package of advanced features is available at additional cost. This package includes the following extensions to the basic AutoCAD software:

- The format of coordinates and distances displayed by AutoCAD or entered from the keyboard may be specified by the user. Two forms of "feet and inches" specifications are included in the choices. If one of these is selected, the "arbitrary" drawing units will be interpreted as inches.

- Semi-automatic dimensioning is provided for such applications as mechanical engineering and architectural drawing. It may be used to easily add dimension lines and arrows, and can even measure the distance and insert the appropriate text. The arrows and text are automatically placed outside the extension lines if they won't fit inside.

- An object may be cross-hatched, or filled with a pattern. The pattern may be defined "on the fly", read from a user-supplied disk file, or chosen from a library of patterns supplied with the AutoCAD software. Scaling and rotating the pattern is also possible.

- A freehand sketch facility is provided. This feature requires a pointing device, and allows you to draw a series of short connected lines, with a specified resolution, quickly and easily. This is very useful for tracing maps, drawing complex curves, signatures, and many other applications. The constructed lines can be edited before being stored in the drawing database.

- Smooth arcs, or fillets, may be drawn to connect two lines. The lines will be extended or trimmed, as necessary, so that they end precisely on the arc.

- Lines, traces, and arcs may be split or broken into two pieces, or one end may be cut off. A portion of a circle may be deleted, creating an arc.

- Axes, or ruler lines, may be displayed on the graphics monitor with user-specified spacing of the ruler ticks.

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- Lines, traces, and arcs may be split or broken into two pieces, or one end may be cut off. A portion of a circle may be deleted, creating an arc.

- Axes, or ruler lines, may be displayed on the graphics monitor with user-specified spacing of the ruler ticks.

Chapter 2

GETTING STARTED

This chapter describes how to use AutoCAD to create, view, modify, and make plots of drawings.

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, such 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-86 Installation Guide / User Guide Supplement or Quick Reference Card 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 enclosed in corner brackets, "<>", you should supply input of the indicated type, without the brackets. Command names and parameters are shown in upper case, but you may actually enter them in any combination of upper and lower case.

The space bar and RETURN key are used to terminate command and data fields entered from the keyboard, and may be used interchangeably. The only exception to this occurs when entering text strings; since spaces may be included in the text, RETURN must be used to terminate the string.

2.2 Loading AutoCAD

"Boot-up" your computer system by following directions supplied with it and put the working copy of your AutoCAD program diskette in Drive A. If your computer uses the CP/M-86 operating system, press CTRL C to inform the system that you have changed the disk. This is unnecessary for PC-DOS and MS-DOS systems.

Put a properly formatted diskette for storing your drawing files in Drive B. (Drawings may be stored on the program diskette, but since program files reduce disk space we suggest you store drawings separately.)

To load the AutoCAD program, type: ACAD followed by RETURN.

NOTE: AutoCAD-86 will execute from any drive, including a hard disk, as long as it is the operating system's current "logged-in" disk. This gives you additional flexibility in multiple drive systems.

2.3 The Main Menu

After the program is loaded, AutoCAD's Main Menu appears on the terminal 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.  Configure AutoCAD
    5.  Make drawing interchange file
    6.  Load drawing interchange file
    7.  File utilities

Enter Selection: _
```

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

Task Ø is used to terminate AutoCAD and return control to the operating system.

Tasks 1 and 2 are used to enter the interactive Drawing Editor. It is the Editor that displays your picture and provides the commands needed to work on it. To begin a new drawing, choose Task 1. To work on an existing drawing, choose Task 2. These choices elicit brief dialogues that lead you into the Drawing Editor, as described below. When you exit the Drawing Editor, AutoCAD redisplays the Main Menu. At that time you may use Task Ø to terminate the AutoCAD program entirely.

Main Menu Task 3 is used to produce a plot of a drawing. This may also be done with the PLOT command while working in the Drawing Editor. Plotting is fully described in Chapter 1Ø of this manual.

Before AutoCAD-86 can be used, it must be properly installed on your computer system. The "Configure AutoCAD" function, Task 4, 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, and may be used 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-86 Installation Guide / User Guide Supplement for instructions.)

Tasks 5 and 6 create and load Drawing Interchange files. Such files allow the database of an AutoCAD drawing to be read by other programs or displayed and edited on different computer systems. These functions are not used to create drawings and many AutoCAD users will never need them. They are fully described in Appendix B.

Menu task 7 will pass control to AutoCAD's disk file utility submenu, from which you can list the contents of a disk, delete selected files, or change the name of a file. This facility is described in Chapter 8.

2.3.1 Task Ø - Exit AutoCAD

Entering Ø at the Main Menu causes AutoCAD to terminate and return control to the operating system. After this entry the operating system prompt will reappear.

2.3.2 Task 1 - Create New Drawing

To start a new drawing, select Main Menu Task 1. AutoCAD will ask for the name of the drawing. This name may be from 1 to 8 characters long, and may consist of letters, numbers, and the special characters $ and -. Simply enter the name of the new drawing, followed by a carriage return. The name entered 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 will be saved on the currently logged disk, unless you precede the drawing name with a drive specifier (e.g., "B:"), which will cause the drawing file to be stored on that drive instead. For example:

```
Enter selection: 1
Enter NAME of drawing: HOUSE-3
```

AutoCAD will then load the Drawing Editor, with which you may create your new drawing. The initial size and modes for a new drawing can be selected when configuring AutoCAD (Task 4). See Appendix D.

2.3.3 Task 2 - Edit 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 will be requested, just as described above for creating a new drawing. For example:

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

If you've previously given AutoCAD a drawing name, it will be considered the "default drawing", and you may 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 may specify the default drawing name on the command line when you first load AutoCAD, as in:

```
A>ACAD B:PC
```

Once you've responded to the "Enter name..." prompt, AutoCAD will load the Drawing Editor and display the drawing, with the screen window and all modes set up exactly the way you last left the drawing.

2.4 Drawing Editor Usage

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

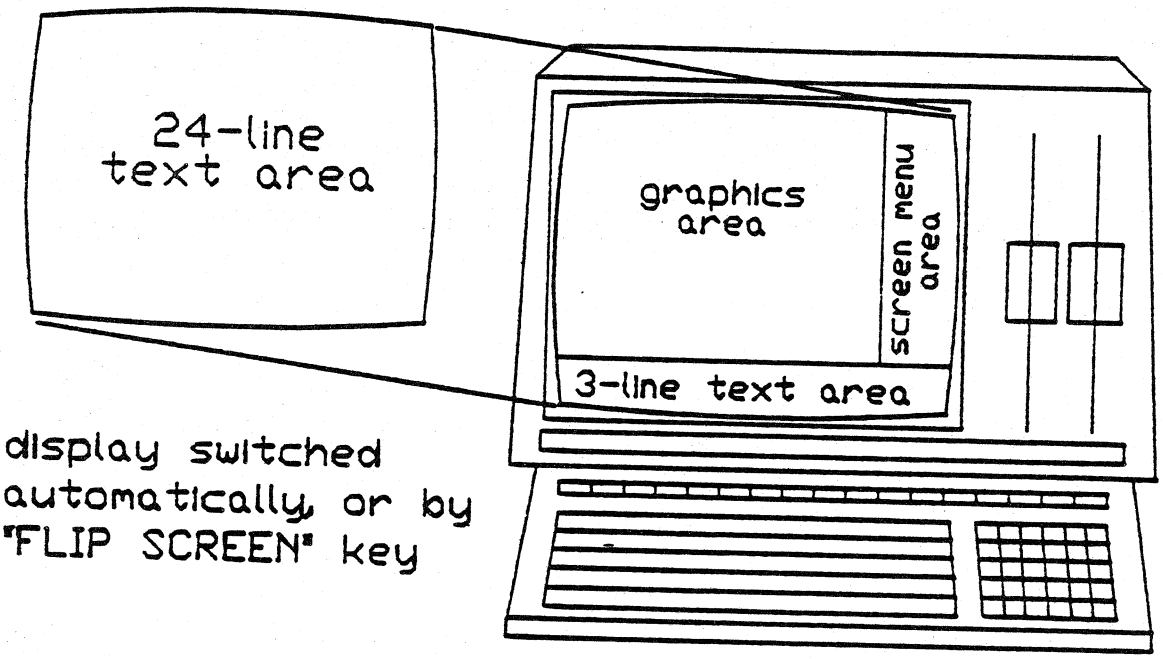
```
Command:
```

AutoCAD is now in Command mode and is ready to accept a command. You may 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. Giving a command puts the program into Data Entry mode. In this mode you are prompted to specify 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.

About the 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 will also display the screen menu along its right edge and a one-line prompt area across the bottom, as in the illustration below.

)n other systems, a single monitor may be used for both graphics ınd text purposes. In these cases, three lines at the bottom of :he screen are reserved for command entry and prompts, and the ight edge contains the screen menu. When AutoCAD is run on such single-monitor system, it remembers a full 24 lines (or more) f text, just like the regular text display. If you want to eview information which has scrolled off the three-line display, ust press the **FLIP SCREEN** key; press it again to return to the raphic display. AutoCAD will automatically switch to the 4-line text display when it puts out a large amount of nformation, as for a STATUS or LIST command. It will utomatically return to the graphic display when it draws nything. A single-screen system is illustrated below.



TYPICAL SINGLE-SCREEN CONFIGURATION

single- and dual-screen systems, the graphics display can clude a mode/coordinate status line, showing the current state various AutoCAD mode switches, and the X-Y location of the reen crosshairs. This status line is a configurable option, d it appears in different places on different computers.

5 AutoCAD Command Summary

e following is a synopsis of all valid AutoCAD commands, esented in alphabetical order. Use this list for reference. tailed descriptions of the commands may be found in the noted ctions of this manual. Commands marked with an asterisk are ailable only with the Advanced Drafting Extensions (ADEX) ckage.

AutoCAD Command Summary

ARC	Draws arcs of any size (Chapter 3).
AREA	Computes the area covered by a polygon (Section 6.3).
ARRAY	Makes multiple copies of selected objects in a rectangular or circular pattern (Section 6.2).
AXIS *	Displays a "ruler line" on the graphics monitor (Section 9.1).
BASE	Specifies origin for subsequent insertion into another drawing (Section 7.1).
BLOCK	Forms a new part drawing from a portion of the current drawing (Section 7.2).
BREAK *	Erases part of a line, trace, circle, or arc (Chapter 9).
CHANGE	Modifies properties of designated entities (Section 6.2).
CIRCLE	Draws circles of any size (Chapter 3).
COPY	Draws a copy of selected objects (Section 6.2).
DBLIST	Lists database information for every entity in the drawing (Section 6.3).
DELAY	Delays execution of the next command for a specified time (Command scripts - Section 8.4).
DIM *	Automatically adds dimension notations to a drawing (Section 9.6).
DIST	Computes the distance between two points (Section 6.3).
END	Exits the Drawing Editor after saving the updated drawing (Section 2.10).
ENDREP	Used with REPEAT command to define a repeat block (Section 6.2).
ERASE	Erases specified entities from the drawing (Section 6.2).
FILES	Provides access to disk directories (Section 8.3).
FILL	Controls whether Solids and Traces are automatically filled on the screen (Chapter 4).

FILLET * Constructs a smooth arc of specified radius between two lines (Chapter 9).

GRID Displays a grid of dots, at desired spacing, on the screen (Section 9.1).

HATCH * Performs cross-hatching and pattern-filling (Section 9.7).

HELP Displays a list of valid commands and data entry options, or obtains help for a specific command (Section 2.9).

ID Displays the coordinates of a specified point (Section 6.3).

INSERT Inserts a copy of a previously drawn part (object) into the current drawing (Section 7.1).

LAYER "Groups" entities in order to display (and plot) selected combinations. Can also associate a particular colour with each group (Chapter 5).

LIMITS Changes the position of the drawing boundaries (Chapter 4).

LINE Draws straight lines of any length (Chapter 3).

LIST Lists database information (location coordinates, layer number, etc.) of specified entities (Section 6.3).

LOAD Loads a file of user-defined Shapes or a Text Font to be used with the SHAPE and TEXT commands (Chapter 3 and Section 7.3).

MENU Loads a file of Drawing Editor commands into the screen (and tablet) menu area (Section 8.1).

MOVE Moves designated entities (objects) to another location (Section 6.2).

OOPS Restores accidentally erased entities (Section 6.2).

ORTHO Constrains LINE drawing so that only perfectly horizontal or vertical lines can be entered (Chapter 5).

PAN Moves the display "window" in any direction (Chapter 4).

PLOT Plots a drawing and returns to the Drawing Editor (Chapter 10).

POINT Draws single points (Chapter 3).

QPLOT Makes a quick plot of the screen display on a dot matrix printer (Machine dependent feature - Section 10.5).

QUIT Exits the Drawing Editor, discarding any changes to the drawing (Section 2.10).

REDRAW Redraws the screen to "clean-up" the display (Section 4.5).

REGEN Forces an explicit regeneration of the drawing (Section 4.7).

REPEAT Used with ENDREP to draw rectangular patterns of one or more entities (Section 6.2).

RESUME Resumes an interrupted command script (Section 8.4).

SHAPE Draws pre-defined shapes (Section 7.3).

SKETCH * Permits free-hand sketching (Section 9.8).

SNAP Specifies a "round-off" value for point entry so entities can be placed at precise locations easily (Chapter 5).

SOLID Draws filled-in polygons (Chapter 3).

STATUS Displays vital statistics about the drawing: status of display controls and drawing aids, number of entities, etc. (Chapter 5).

TABLET Aligns the digitizing tablet with coordinates of a "paper" drawing to accurately copy it with AutoCAD (Section 8.2).

TEXT Draws alphanumeric characters (Chapter 3).

TRACE Draws solid lines of specified width (Chapter 3).

UNITS * Selects coordinate display format and precision (Section 9.2).

WBLOCK Writes selected entities (or the entire drawing) out to a disk file (Section 7.2).

ZOOM Magnifies or shrinks the display of the drawing (Chapter 4).

2.6 Command Entry

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

2.6.1 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 many items, but most monitors can't display all of them at once. In this case, the last item on the menu will be **NEXT.** If you select this, another group of choices will appear on the screen.

A pointing device makes entry of a command from the screen menu very easy. Just move the pointer to the right edge of the screen, and up or down until the desired menu item is lighted. Then press the pointer's button (or lift the pen from the screen).

If you don't have a pointing device, you can access the screen menu via the keyboard. Pressing the **MENU CURSOR** key will cause 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 will be selected. You can escape without selecting anything by pressing the **ABORT CURSOR** key.

2.6.2 From the Tablet Menu

A tablet menu is a printed form occupying a portion of the digitizing tablet. To select a command from it, position the tablet stylus over the desired command on the tablet menu and press the button. For further details, see Chapter 8.

2.6.3 From the Keyboard

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

2.6.4 Repeated Commands

Whichever method was used to enter the last command, the space bar or RETURN may be pressed 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.

2.7 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 will prompt 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 will describe the data types commonly required by many of the AutoCAD commands. The specific command descriptions beginning in Chapter 3 will include any unique requirements.)

NOTE: There is sometimes 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.

2.7.1 Points

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. This marker will disappear when you next regenerate or redraw the display (see Chapter 4). If a point is improperly specified, or is outside the drawing limits, the message:

```
     *Invalid*
or
     ** Outside limits
```

will be displayed. All further action on the current command will be canceled, and AutoCAD will return to Command mode.

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

2.7.1.1 Absolute Coordinates

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

2.7.1.2 Relative Coordinates

You may specify a point as a distance from the last point specified. For example, if the last point entered was 10,2 and you enter "@+2,-1" as a relative specification, the result is to specify the point:

X=10+2=12 and Y=2-1=1 giving the point 12,1

The character "@" indicates a relative specification. You may also specify the relative coordinate as a distance and angle from the previous point, by using @distance<angle. For instance:

@2<45

specifies the point that is two units from the last specified point, at an angle of 45 degrees. For further information on how angles are defined, see Section 2.7.3.

2.7.1.3 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".

2.7.1.4 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 button (or lift the pen from the screen). The crosshairs will be replaced by a small marker (which will go away when the screen is redrawn), and the coordinates will be entered as if you typed them on the keyboard.

2.7.1.5 Keyboard Pointing

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 will 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, enter a space or RETURN. If you want to stop without selecting a point, use the **ABORT CURSOR** key.

If a pointing device other than a light pen is present, turning on the keyboard pointer will make 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.

2.7.2 Numeric Values

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

These prompts require entry of a number. You may use the following characters in the numbers you enter from the keyboard.

```
+ - Ø 1 2 3 4 5 6 7 8 9 E .
```

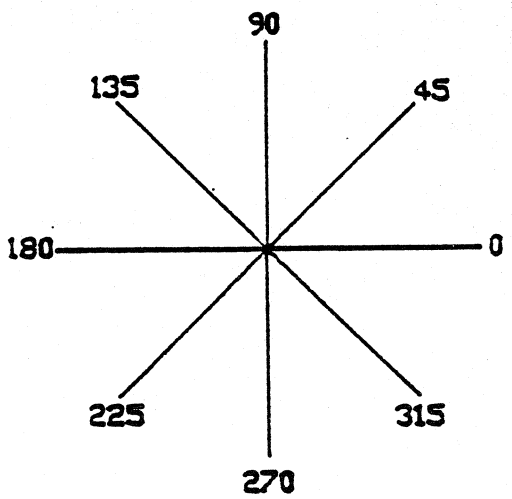
Examples are +35.7, -1Ø, 7.2E+6 (which is "scientific notation form" for the value 72ØØØØØ). For "columns" and "rows", the number must be an integer, however.

Whenever AutoCAD expects a distance, you may instead point to a location as explained previously. In some cases, the prompt "Second point:" will appear; when you have pointed to a second location, AutoCAD will use the distance between the points as the input. In most cases, though, AutoCAD will measure from some obvious base point to the point you gave. For instance, after you have given the centre of a circle, AutoCAD will ask for the radius. If you respond with a point, you surely want the radius to be the distance from the centre to that point, placing that point on the circumference.

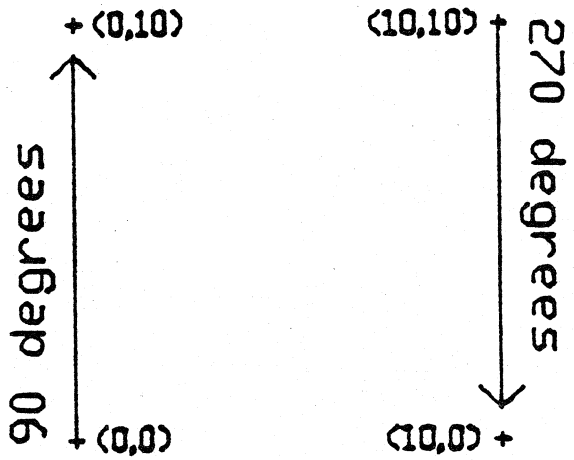
2.7.3 Angles

Prompt: Angle:

All angles in AutoCAD are specified in degrees. When the ang is used to specify an orientation (bearing) the followi standard applies: Angles increase in a counterclockwi direction, and zero degrees is directly to the right of t starting point.



An angle can be entered from the keyboard as a number. Foll the entry with space or RETURN. Alternatively, you can "sho AutoCAD the angle by pointing, indicating the start point and t end point of a line at the desired orientation. Note that t order in which these points are entered does matter. Pointing 0,0 then 0,10 specifies 90 degrees (straight up), while 0,10 th 0,0 specifies 270 degrees (straight down).



In some cases, the location of the start point is pretty obviou if you designate a point in response to the "Angle:" promp AutoCAD will assume that you are just specifying the end point.

2.7.4 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.

2.7.5 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 or tablet menu, just like a command.

2.7.6 Feet and Inches Input

If the Advanced Drafting Extensions (ADEX) package is present, AutoCAD may be instructed to interpret the drawing units as inches, and to allow input of coordinates, distances, and displacements in terms of feet and inches. For example, a distance of 15.5 drawing units could be input as:

1'3.5" (1 foot 3.5 inches in "engineering" format)

or

1'3-1/2" (1 foot 3.5 inches in "architectural" format)

For further information, see the description of the UNITS command in Chapter 9.

2.8 Error Correction

Errors made while typing in command names or required data may be corrected in three ways:

Backspace Deletes one character at a time. (Also **CTRL H**.)

CTRL X Deletes all characters on the line.

CTRL C Cancels the current command entry and restores the "Command:" prompt to the screen. A command may be entirely canceled at any time; that is, either while typing in the command name, or during data entry.

2.9 HELP Command - User Assistance

The HELP (or "?") command can be used to obtain a list of AutoCAD commands in case you've simply fogotten 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 any invalid command is entered, AutoCAD prints a message to remind you of the availability of the Help facility.

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

You can request more detailed information about a specific command by responding to the prompt with the desired command name, or you may press space or RETURN to obtain the list of valid AutoCAD commands.

The help text is stored in the disk file ACAD.HLP, which is supplied with the AutoCAD software. Using a text editor, you may revise this file to include any additional information you wish displayed when the HELP command is entered (such as names of INSERT, SHAPE, and TEXT font files you've created).

2.10 Drawing Editor Exit

Whether you're creating a new drawing or editing a previously stored drawing, you will be using AutoCAD's Drawing Editor. When you're finished editing the drawing, you must exit from the Drawing Editor and return to the Main Menu. Two commands, with different functions, are available to accomplish this.

2.10.1 END Command

Command: END

The END command returns to the Main Menu and updates the drawing. 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 will now be the ".DWG" file, so you can further edit it with AutoCAD if desired.

2.10.2 QUIT Command

Command: quit

The QUIT command returns to the Main Menu, but does not update the drawing. If you were just looking at a drawing, or have made some horrible error and wish to discard all the changes made in your editing session with the drawing editor, 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, and the ".DWG" file will be the same as before the editing session. Any previously existing ".BAK" file will be preserved unchanged also.

Chapter 3

ENTITY DRAW COMMANDS

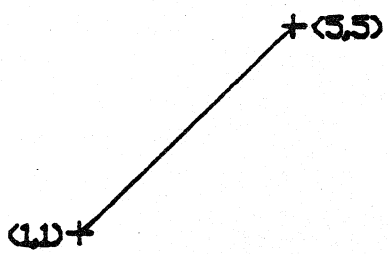
his chapter introduces AutoCAD's drawing entities and the ommands used to enter them. These commands all require that you pecify a point in the drawing where the entity should appear. ny of the Point Entry methods described in Section 2.7 may be sed.

here are two entity types not included in this chapter; Shapes nd Blocks. Shapes are special objects you can create yourself nd add to the drawing with a single command. Blocks are named ollections of entities (even entire other drawings), which can e manipulated as a whole and inserted wherever you like. Shapes nd Blocks are discussed in Chapter 7. All other commands which dd objects to a drawing (dimensions, for instance) actually form hese objects from the entities described in this chapter.

.1 LINE Command

he most fundamental drawing entity is the Line. To draw one, nter the LINE command. You will then be asked to identify both ndpoints of the line. An example would look like:

```
Command: LINE  From point: 1,1
  To point:  5,5
  To point:     (just press space or RETURN)
```



f available, a "rubber band" cursor will be displayed in ddition to the normal crosshairs as you point to each "to" oint. This helps you see where the resulting line will go.

any times, you will wish to enter a string of connected lines. ou could specify the start of one line at the same point as the nd of the previous line. This would waste time, so AutoCAD asks or more "To points". When you are finished entering a connected tring of lines, or you wish to use a different command, just ress RETURN when prompted for a "To point". There is no

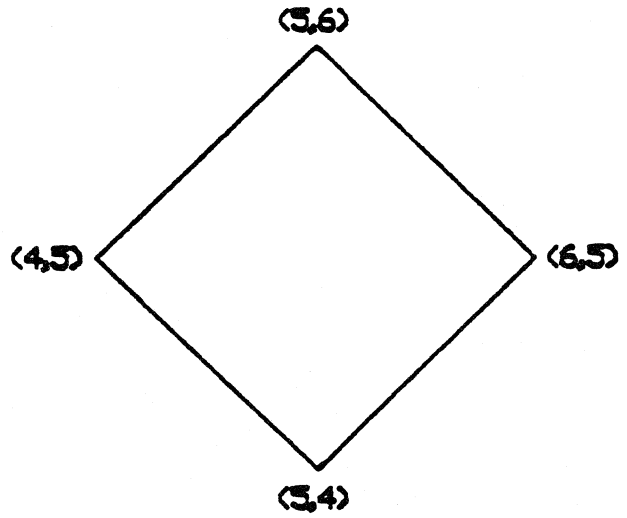
difference between a set of lines entered in this manner a lines entered with multiple LINE commands; each is a separa entity.

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

For example, either of the following command sequences could used 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
```



Line/Arc Continuation

One additional capability is provided for the LINE command. you respond to the "From point:" prompt with a space or RETUR the start of the line will be set to the end of the most rece Line or Arc which was drawn. This provides an easy means resuming a LINE command which was interrupted for some reaso and also simplifies the construction of tangentially connect lines and arcs.

```
Command: LINE
  From point: (just press space or RETURN)
```

The subsequent dialogue will depend 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 will appear as usual.

If, on the other hand, an Arc was more recently drawn, its end will define the starting point and the direction of the new line. Only the length of the new line needs to be specified, so AutoCAD will prompt with:

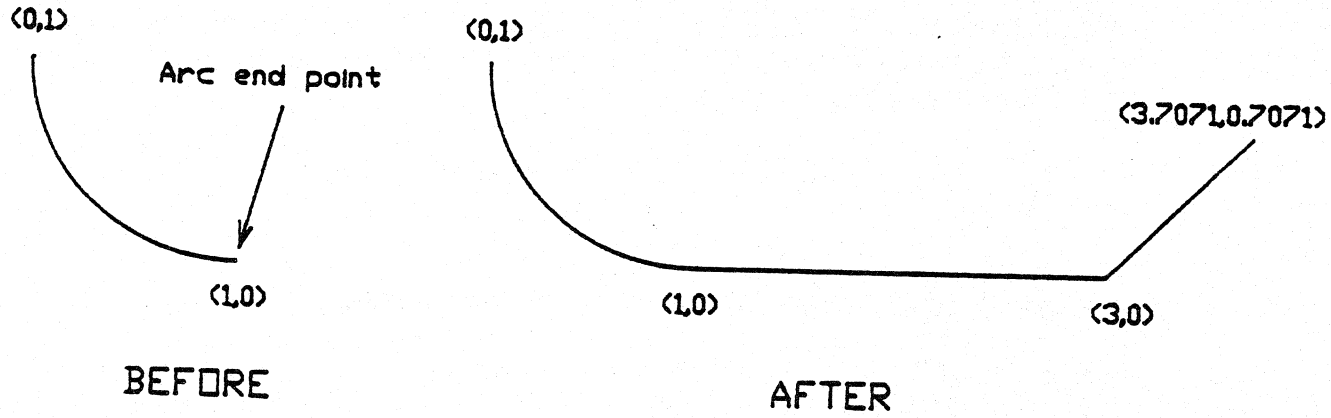
Length of line: <enter value>

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

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:

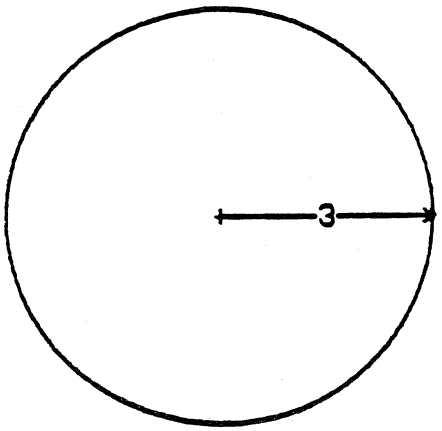


3.2 CIRCLE Command

A circle may be drawn in one of four ways using the CIRC command. You may enter its centre point and radius, as in:

```
Command: CIRCLE  Centre point (or 3P or 2P): 5,5
  Radius (or D): 3
```

The resulting circle will have a centre at (5,5) and a radius 3 drawing units.



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

If you'd rather specify the diameter than the radius, simp enter "D" followed by space or RETURN in response to t "Radius:" prompt. AutoCAD will-then ask for the diameter. F instance, the command sequence:

```
Command: CIRCLE  Centre point (or 3P or 2P): 5,5
  Radius (or D): D
  Diameter: 6
```

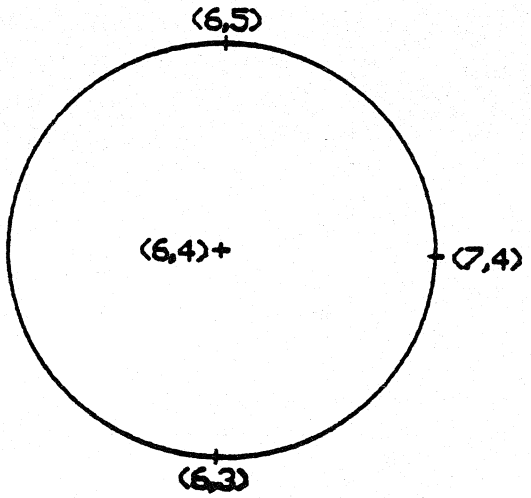
would draw the same circle as in the first example.

Three-point Circles

Another method of circle specification is to enter three points to be on the circumference of the circle. Respond "3P" to the "Centre point:" prompt to tell AutoCAD that this is what you wish to do. For example:

```
Command: CIRCLE  Centre point (or 3P or 2P): 3P
  First point: 6,5
  Second point: 7,4
  Third point: 6,3
```

The resulting circle has its centre at (6,4) and a radius of 1.



Two-point Circles

The last method of drawing a circle is to enter two endpoints of its diameter. You must tell AutoCAD that you wish to do this by responding "2P" to the "Centre point:" prompt, as in:

```
Command: CIRCLE  Centre point (or 3P or 2P): 2P
  First point: 6,5
  Second point: 6,3
```

The resulting circle is the same as in the previous example.

3.3 ARC Command

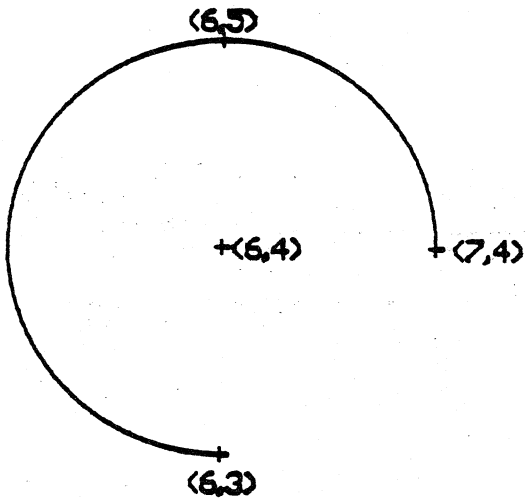
Arcs are partial circles, and are drawn using the ARC comman Eight different methods of specifying an arc are provide allowing for personal preferences and for the varying circu stances under which arcs are needed. You may specify:

1. Three points on arc
2. Start point, centre, end point
3. Start point, centre, included angle
4. Start point, centre, 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, "centre" refers to the centre point of the circ of which the arc is a part. The default method is "three poin on arc," which is similar to the "3P" method of specifying circle. The first and third points are the arc endpoints. F example:

```
Command: ARC
  Start point or C: 7,4
  Second point or C or E: 6,5
  End point or R or A or D: 6,3
```

will draw the arc:



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

The other arc specification methods are invoked by typing letter, followed by space or RETURN, to indicate whic

information you want to enter next. The ARC command option letters have the following meanings:

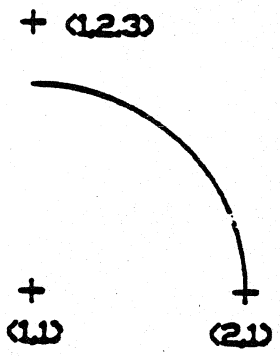
A - included Angle
C - Centre
D - starting Direction
E - End point
L - Length of chord
R - Radius

The prompts will indicate what options are available at each step. Each method will be discussed separately below.

Start point, centre, end point

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

```
Command: ARC  Start point or C: 2,1
  Second point or C or E: C
  Centre: 1,1
  End point or A or L: 1,2,3
```



As illustrated in this example, the end point you specify is used only to determine the angle at which the arc will end; the arc will not necessarily pass through this point.

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

```
Command: ARC  Start point or C: C
  Centre: 1,1
  Start point: @1<0
  End point or A or L: @1<90
```

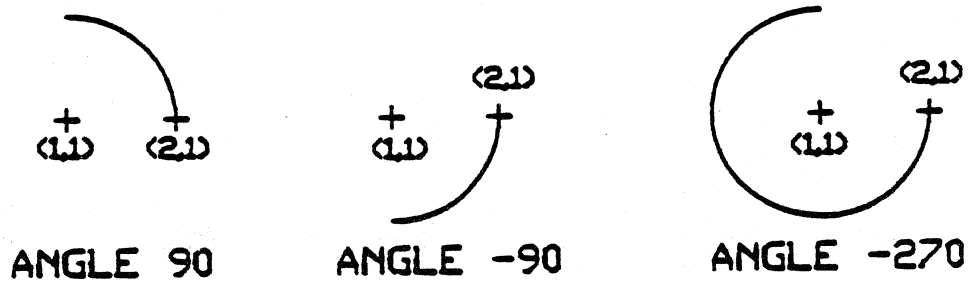
This would draw the same arc as in the previous example. Note that a relative coordinate for the end point will be based on the centre, even though the start point was entered later. This is what is usually needed.

Start point, centre, included angle

This draws an arc with the specified centre and start point, a spanning the indicated angle. Ordinarily, the arc will be dra counterclockwise from the start point. However, if the specifi angle is negative, the arc will be drawn clockwise. For example

```
Command: ARC  Start point or C: 2,1
  Second point or C or E: C
  Centre: 1,1
  End point or A or L: A
  Included angle: 90 or -90 or -270
```

The three different angle specifications would result in t three arcs shown below.

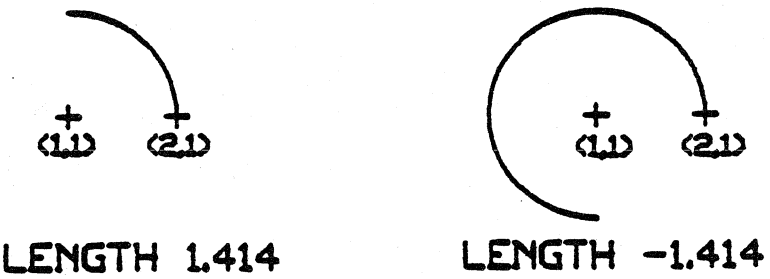


Start point, centre, length of chord

A chord is a straight line connecting an arc's start point a end point. For this type of specification, the chord length used to compute the ending angle. The same start, center, a chord length could apply to four different arcs; AutoCAD resolv this by always drawing this type of arc counterclockwise from t start point. By default, it draws the minor (less than 1 degree) arc, but a negative value for the chord length will cau the major arc to be drawn. For example:

```
Command: ARC  Start point or C: 2,1
  Second point or C or E: C
  Centre: 1,1
  End point or A or L: L
  Length of chord: 1.414 or -1.414
```

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

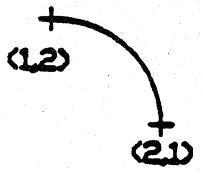


Start point, end point, radius

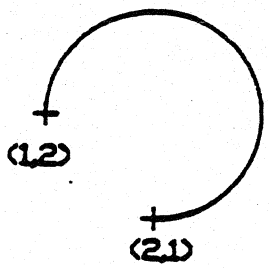
Since the same values for these three variables could apply equally to four different arcs, AutoCAD will always draw this type of arc counterclockwise from the start point, and will normally draw the minor arc. A negative value for the radius will instruct AutoCAD to draw the major arc instead. For instance:

```
Command: ARC  Start point or C: 2,1
  Second point or C or E: E
  End point: 1,2
  Centre point or R or A or D: R
  Radius: 1 or -1
```

The results of the two radius specifications are shown below.



RADIUS 1



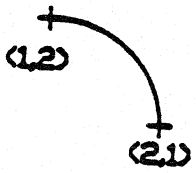
RADIUS -1

Start point, end point, included angle

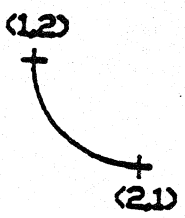
The type of arc is normally drawn counterclockwise from the start point to the end point. If you specify a negative value for the included angle, however, the arc will be drawn clockwise. For example:

```
Command: ARC  Start point or C: 2,1
  Second point or C or E: E
  End point: 1,2
  Centre point or R or A or D: A
  Included angle: 90 or -90
```

The results for the two angle specifications are shown below.



ANGLE 90



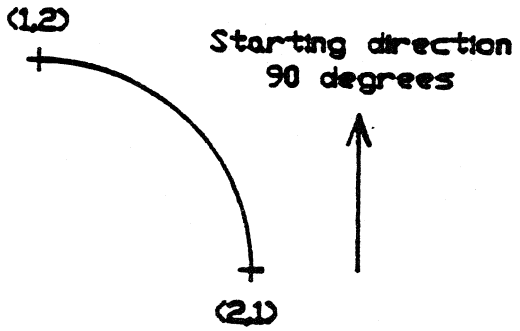
ANGLE -90

Start point, end point, starting direction

This begins the arc in a specified direction, and can be used draw an arc tangent to another entity. It will create any ar major or minor, clockwise or counterclockwise. For instance, t sequence:

```
Command: ARC  Start point or C: 2,1
  Second point or C or E: E
  End point: 1,2
  Centre point or R or A or D: D
  Direction from start point: 90
```

would draw the arc:



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

Continuation of previous line or arc

This is actually a special case of the "start point, end poin starting direction" method discussed above. If you respond t the first prompt with a space or RETURN, the start point ar direction of the arc will be taken from the end point and endi direction of the last arc or line drawn.

```
Command: ARC
  Start point or C: RETURN
  End point: 0,1
```

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

3.4 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 will be asked how wide the trace is. In repeat mode, all connected traces will 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
```

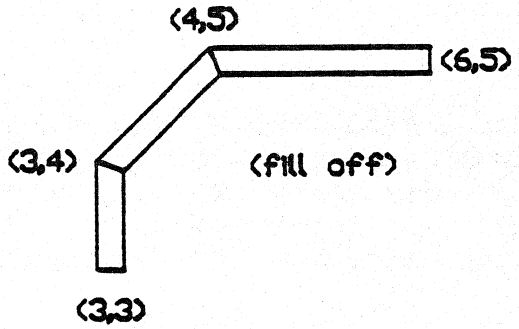
The end points specified are on the centre line of the trace. The TRACE command automatically calculates the correct ending angle for connection to the next segment of the trace. For this reason, the trace will not be drawn until either the next segment point is specified, or you press RETURN to stop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

Traces which are completely on the screen will be solid filled unless Fill mode is off (see Section 4.6). If Fill is off, only the trace outline will be drawn. When the final drawing is ready, Fill may be turned on for the slower but more appropriate solid fill.

Trace Example

```
Command: TRACE  Width: .25
  From point: 6,5
  To point: 4,5
  To point: 3,4
  To point: 3,3
  To point:     (just press RETURN)
```

would generate the trace shown below.



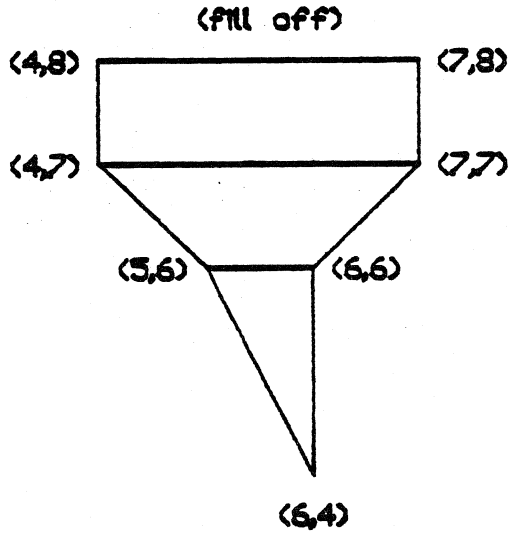
3.5 SOLID Command

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

First enter the two endpoints of a starting edge. Then ent either two points of the next edge, or one point and the RETU key to specify a triangular section. You may continue to ent edges to make up the solid. When complete, press RETURN wh prompted for the third point.

```
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:     (just RETURN - triangular section)
  Third point:      (just RETURN - end of the solid)
```

The above points will draw the solid shown below.



Solids which are completely on the screen will be solid fill unless Fill mode (Section 4.6) is off. When Fill is off, on the outlines will be drawn, permitting faster operation whi constructing the drawing. When complete, Fill may be turned and a REGEN operation performed to fill all Solids and Traces.

3.6 TEXT Command

Text may be added to a drawing via the TEXT command. AutoCAD will ask for the starting point (the start of the text base line), and then for a value specifying the height of the text. You may "show" AutoCAD the height of the text by designating a point; the height will be the distance between this and the starting point. If you simply press space or RETURN without supplying a value, the last height specified will be used.

Next, you will be asked to specify the angle of the text base line. The last angle specified will be used if you just press space or RETURN. You may "show" AutoCAD the angle by entering a point; the text baseline will run from the starting point to this point. If this is to the left of the starting point, the text will be upside down.

```
Command: TEXT  Starting point (or ACR): 2,1
  Height: .25
  Angle: 0
  Text: Master Bedroom
```

The text string is ended by pressing the RETURN key. Note that the height specifies how far above the baseline the capital letters extend. Lower case letters have "descenders" which will go below the baseline, and some special characters may extend further in each direction than ordinary characters.

Centred and Right Justified Text

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

```
Centre point:
```

Enter a point. The remainder of the command is as above, but the text will be centred vertically and horizontally at this point.

To right justify the text 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 text will be right justified so that it ends at the designated point.

Specifying Text Size by Width

If you prefer, you may specify the text size by defining t width of an area in which the text is to fit. This is done responding to the "Starting point:" prompt with "A" (f "aligned"). AutoCAD will ask for the two endpoints of the te area, and will select a text height such that the text will ju fit between those two points. Note that the two points al define the angle of the text base line, so no "Angle:" prom will appear. Text entered in this fashion is considered "le justified" for the purposes of continuation (see below), althou it entirely fills the width you specified.

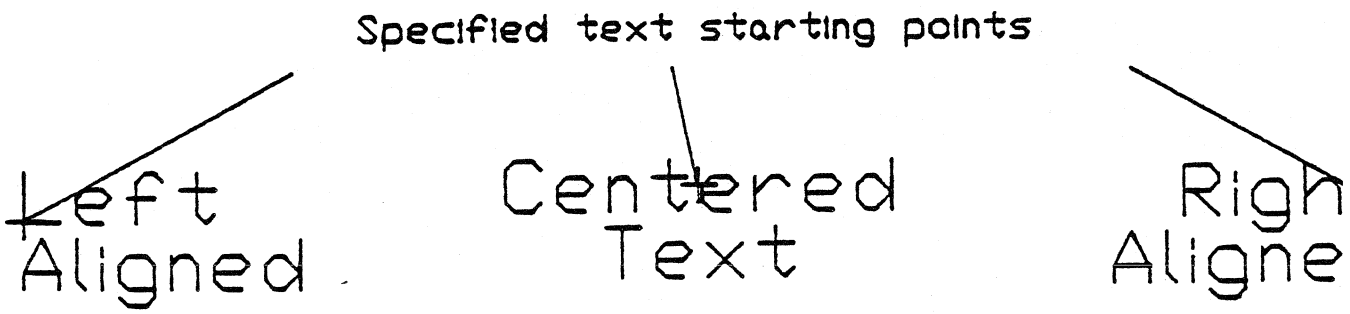
```
Command: TEXT  Starting point (or ACR): A
  First text line point: 1,1
  Second text line point: 16,1
  Text: Mona Lisa
```

Repeating TEXT Commands

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

```
Command: TEXT  Starting point (or ACR): C
  Centre point: 3,2.8
  Height: .25
  Angle: 0
  Text: Dining
Command:  (space or RETURN to cause command repeat)
  Text: Room
```

In this example, the second line ("Room") will appear below t first ("Dining"). Since the first line was centred, the secc line will also be centred. The illustration below shows each ty of text alignment.



3.7 LOAD of Text Fonts

By default, AutoCAD uses the simple text font defined in the file TXT.SHP. This file will be automatically loaded the first time a Text entity is drawn, if no other font has yet been loaded. You may make your own text font file (see Appendix A) and then use the LOAD command to tell AutoCAD to use your new definitions. For example:

Command: LOAD File name: MYTEXT

This causes AutoCAD to use the text font definition in the file MYTEXT.SHP. The file type ".SHP" is assumed for all font files, and should not be specified. LOAD is also used to enter Shape definitions as explained in Chapter 7.

AutoCAD is supplied with a few font files in addition to the simple one in TXT.SHP. These files are:

ROMAN-S.SHP - Roman Simplex: nicer than TXT.SHP
ROMAN-C.SHP - Roman Complex: multi-stroke high quality
ITALIC.SHP - Italic: multi-stroke high quality font

To use any of these alternate fonts, simply issue the appropriate LOAD command before you draw any Text entities. If you will always want to use one of these fonts instead of the standard one, you may rename the files such that the font you want is in file TXT.SHP. (But be sure to save the standard font somewhere!)

At the present time, only one font may be used per drawing.

3.8 POINT Command

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

Command: POINT Point: 5,6

will place a point in the drawing at location (5,6).

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

Chapter 4

DISPLAY CONTROLS

4.1 Drawing Limits

When working on a drawing using AutoCAD, there is always a coordinate system that locates entities in the drawing. The values of the "borders" of the drawing are called the drawing limits. As an example, if you are working on a drawing of a printed circuit board that is 8 inches high by 10 inches wide, and you choose to consider the lower left corner of the board as having coordinate 0,0, the drawing limits are:

```
Left border is at   X =  0
Right border is at  X = 10
Top border is at    Y =  8
Bottom border is at Y =  0
```

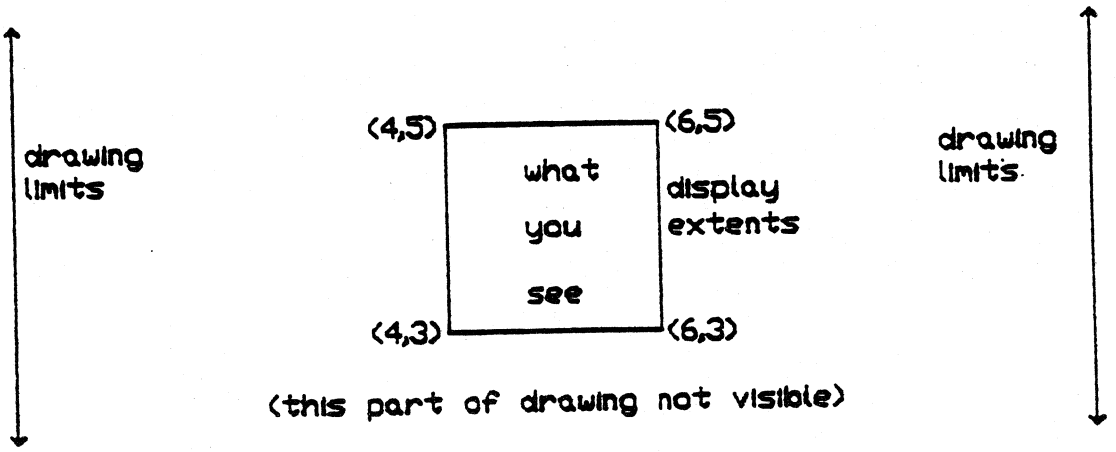
4.2 Display Extents

AutoCAD allows control of how much of the drawing you see on the screen by creating another set of borders, called the display extents. The display extents are the values of the borders of the display, in drawing coordinates. For example, to show a magnified view of the centre of an 8 inch by 10 inch printed circuit board, the display extents might be:

```
Left display border at   X = 4
Right display border at  X = 6
Top display border at    Y = 5
Bottom display border at Y = 3
```

See the example on the following page.

Display Extents Example



How AutoCAD Uses Extents

AutoCAD will use these border values, or extents, to allow you "zoom in" or "back off", and pan the display sideways a vertically. Thus, the display can be considered a "windo through which you may see part or all of your drawing different magnifications. This lets you draw intricate parts your drawing with exacting detail, and then "back off" and lo at the finished drawing.

The display control commands change the value of the displ extents by using the information supplied in the command and t current extent values to calculate the new display extents. T drawing is then regenerated (redrawn) to display only that pa of the drawing within the new display extents.

.3 ZOOM Command

he ZOOM command lets you magnify or shrink the display of the rawing. Several options let you specify what you want to see in arious ways. The ZOOM command prompts you with the options.

.3.1 ZOOM Magnification

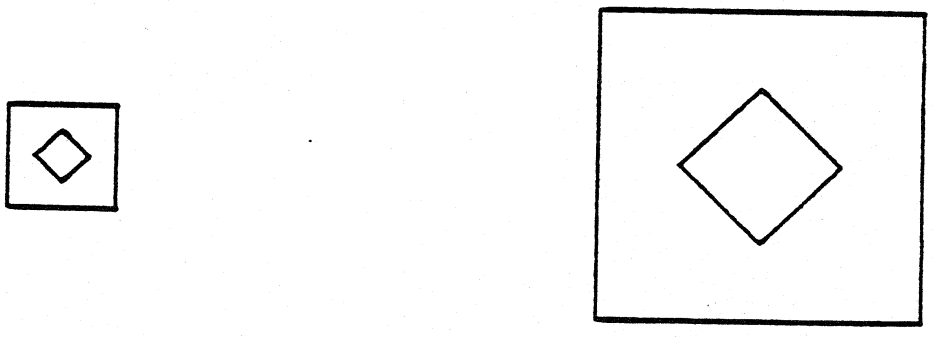
he simplest type of ZOOM command lets you blow up or shrink down he display by entering a magnification factor. Shrinking the rawing makes more of its surface visible. If you just enter a umber, the drawing will be magnified that amount from the full iew. A number followed by "X" will cause the magnification to e relative to the current display. Thus, entering "2X" will low the drawing up to twice its current size, while a agnification of ".5" will shrink the drawing to half its full ize no matter what the current display magnification is.

hen ZOOM blows up or shrinks the drawing, the display centre tays at the same location in the drawing. When shrinking, negative space" below the lower drawing limits may become isible. (There's nothing wrong with the drawing limits being egative numbers--the term "negative space" means coordinates off the page" [below the drawing limits].)

or instance, to blow up the drawing by a factor of 3 from its urrent size:

Command: ZOOM Magnification or type (ACELPW): 3X

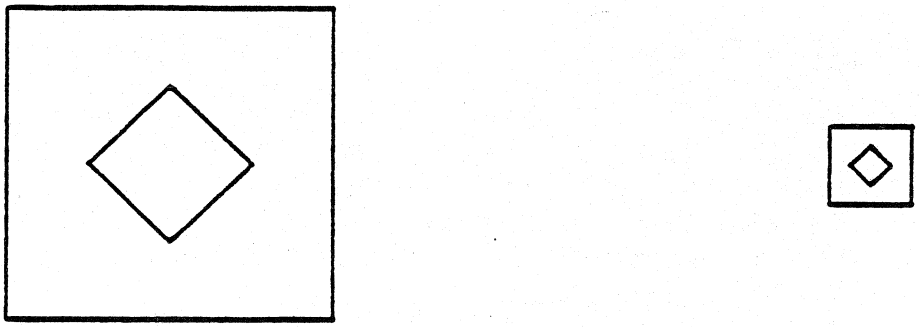
previous: new:



Similarly, to shrink the drawing to 1/4 its full size:

Command: ZOOM Magnification or type (ACELPW): .25

full: new:



4.3.2 ZOOM All

Whenever you wish, you may also enter the ZOOM All command:

Command: ZOOM Magnification or type (ACELPW): A

which will change the display so that you will see all of yo drawing on the screen (based on its limits or current extent whichever are greater). If the drawing has been extended outsi the drawing limits, the display will also extend outside t drawing limits so as to show all entities in the drawing in the entirety. Occasionally, ZOOM All will have to generate t drawing twice. When it does so, it will display the message:

** Second regeneration caused by change in drawing extents

Zoom All also resets the "Drawing uses" extents which appear the STATUS display (described in Chapter 5). When deletion of entity reduces the maximum extents used by the drawing, t display in the STATUS command will not be updated until the ne ZOOM All or ZOOM Extents (see below).

4.3.3 ZOOM Extents

As noted above, the ZOOM All command displays the entire drawi surface, even if only a small portion as yet contains a entities. Thus, the entities may be very small when displaye The ZOOM Extents command:

Command: ZOOM Magnification or type (ACELPW): E

uses only the current drawing extents, not its limits, a results in the largest possible display of the portion of yo

drawing which currently contains anything. Like ZOOM All, ZOOM Extents resets the "Drawing uses" extents for the STATUS display.

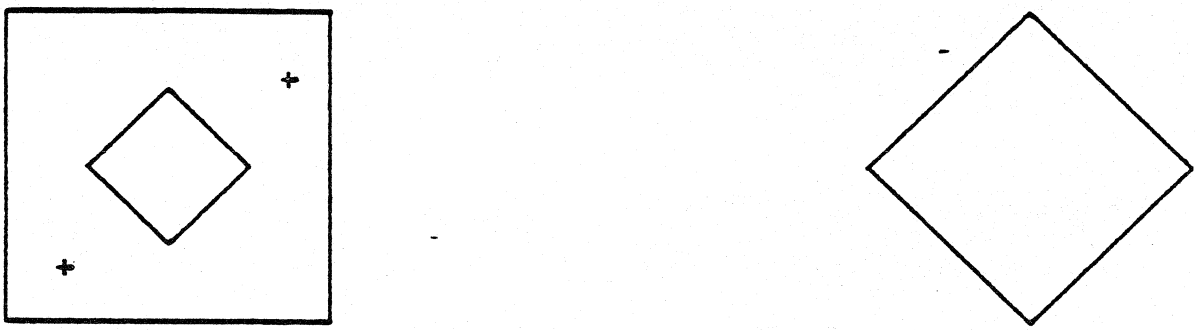
4.3.4 ZOOM Window

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 centre of that window will become the new display centre, and the area inside the window will fill the display as completely as possible. Of course, you can enter points by coordinates or with the pointing device.

The command format is:

```
Command: ZOOM  Magnification or type (ACELPW): W
  First point:  <enter point>
  Second point: <enter point>
```

This display becomes this display



after entering ZOOM W and entering the two indicated points.

If available, a "box" cursor will be displayed rather than the normal crosshairs so that you may see the window more clearly when pointing to the second corner.

4.3.5 ZOOM Centre

The ZOOM Centre command lets you specify your display window location by entering the desired centre point of the new display. You may also optionally specify the height of the window in drawing units. The old value of the height is displayed in the prompt to remind you of the current scale. If you do not specify a height value, then the magnification will be unchanged. If you specify a smaller value for the height, the magnification will be increased, and if you specify a larger value, the magnification will be reduced.

The command:

```
Command: ZOOM  Magnification or type (ACELPW): C
  Centre point: 7,4
  Magnification or Height <5>: 2
```

will place the display centre at (7,4) and make the display drawing units high. This example used the absolute value 2 fo the height; an "X" following the number indicates a change i magnification relative to the current value (e.g., "2X" woul make the drawing twice as large).

4.3.6 ZOOM Left Corner

The ZOOM Left corner command lets you specify your display windo location by entering the coordinates of the point you wish t appear at the lower left corner of the display. You may als optionally specify the height of the window in drawing units. Th old value of the height is displayed in the prompt to remind yo of the current scale. If you do not specify a height value, the the magnification will be unchanged. If you specify a smalle value for the height, the magnification will be increased, and i you specify a larger value, the magnification will be reduced.

The command:

```
Command: ZOOM  Magnification or type (ACELPW): L
  Lower left corner point: 2,3
  Magnification or Height <5>: 2
```

is identical to the ZOOM C command discussed above, except tha it specifies the lower left corner instead of the centre. Her again, a relative magnification may be specified by following th number with an "X".

4.3.7 ZOOM Previous

While editing or creating a drawing, you may find it useful t zoom in to a small area, then back out, and then zoom in t another small area. However, the ZOOM techniques described s far are not easily used to zoom back out to a prior view. Yo could use ZOOM All, of course, and then do repeated ZOOMs to ge back where you were. In a detailed drawing, this can be tedious

To make this operation more convenient, AutoCAD saves the curren view on a "stack" whenever it is being changed by any of the ZOO commands described so far, or by the PAN command (described i the next section). You can return to the previous view by usin the ZOOM Previous command:

```
Command: ZOOM  Magnification or type (ACELPW): P
```

his "pops" the prior view off the stack and displays it. Up to hree views may be saved on the stack; successive ZOOM P's can esurrect the previous three views.

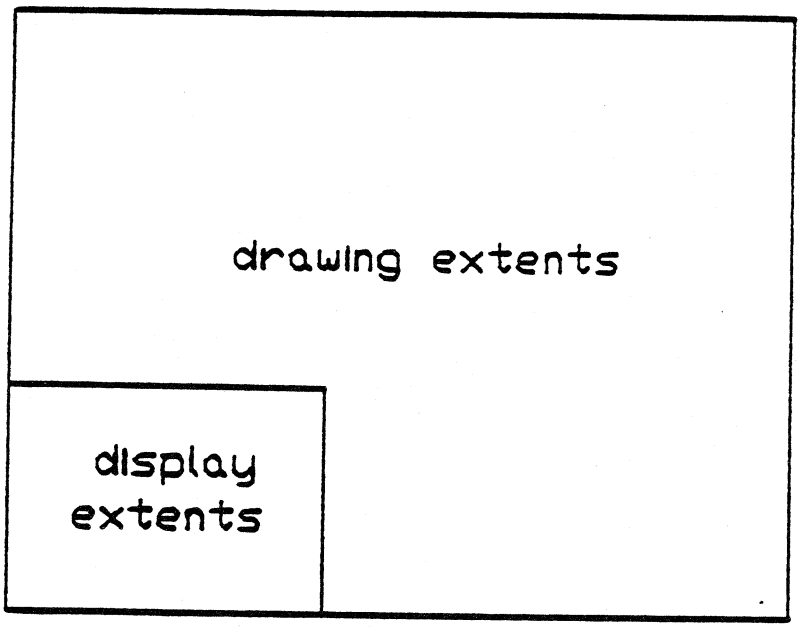
4.4 PAN Command

The PAN command lets you move the display window in any dir tion, without changing the magnification. This lets you details that were "off screen" before the PAN command.

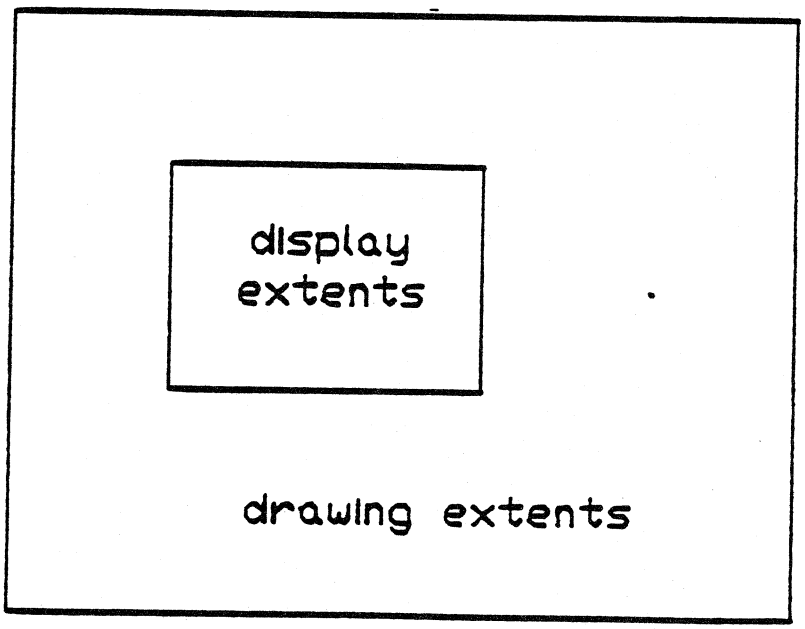
```
Command: PAN   From - to: 2,3
  Second point:   (just type RETURN)
```

This will move the window right 2 and up 3.

before:



after:



The "From-To" locations may also be pointed at to show t displacement you desire. When pointing at two points, think the first as the current window location and the second as whe you want the window "dragged" to.

4.5 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 marker blips, and is invoked with the command:

```
Command: REDRAW
```

A redraw can be aborted by pressing **CTRL C**. This can save time if you're about to issue another command that causes a redraw.

4.6 FILL Command

On some machines, it takes a long time to fill the interiors of Traces and Solids. The FILL command lets you turn this interior filling on or off.

```
Command: FILL  On/Off: ON   (interior filled)
Command: FILL  On/Off: OFF  (outline only)
```

The descriptions of Traces and Solids in Chapter 3 explain exactly what will be drawn when Fill mode is off. Changing the Fill mode will not affect the display of existing objects until the drawing is regenerated (see REGEN command, below).

4.7 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 is seldom necessary. The command format is:

```
Command: REGEN
```

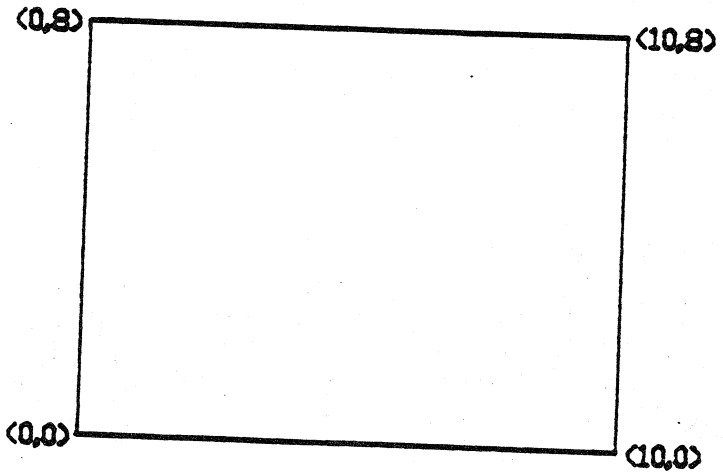
A REGEN may be useful if you have changed or moved one instance of a REPEAT block (see Section 6.2); the other instances won't be updated until the drawing has been regenerated. A regeneration can be aborted by pressing **CTRL C**. This can save time if you're about to issue another command that causes a regen. (NOTE: The ZOOM and PAN commands automatically regenerate the drawing.)

4.8 LIMITS Command

When creating a new drawing, AutoCAD starts out with the defau drawing limits specified via the "Configure AutoCAD" main me task. Attempts to enter points outside these limits will rejected by AutoCAD (although an entity, such as a circle, m start on-screen but extend outside the limits). If you wish change the drawing limits while working on a drawing, the LIMI command may be used. For example, if the old limits were Ø to in both X and Y dimensions, and you wished to change them to 4 7 in X and Ø to 14 in Y, the following command would be used:

```
Command: LIMITS
Old lower left corner   X =    Ø.ØØØØ  Y =    Ø.ØØØØ
Enter new lower left corner or return: 4,Ø
Old upper right corner  X =   1Ø.ØØØØ  Y =   1Ø.ØØØØ
Enter new upper right corner or return: 7,14
```

Note that although the lower left corner limits default to Ø,Ø they may be changed to any desired value, including negativ numbers. In fact, it would be possible to do an entire drawin in negative coordinates.



Chapter 5

DRAWING AIDS

5.1 SNAP Command

Points entered via a pointing device may be forced into alignment with an imaginary grid by the "snap" mechanism. The snap resolution defines the spacing of this grid; the crosshairs and all input coordinates are rounded ("snapped") to the nearest point on the grid if Snap mode is on. This lets you enter points quickly, letting AutoCAD ensure that they line up precisely.

When a new drawing is started, its snap resolution is set to the default chosen during the "Configure AutoCAD" main menu task. However, the snap resolution may be changed at any time by using the SNAP command. For example:

Command: SNAP On/Off/Value: Ø.Ø5

changes the snap resolution to Ø.Ø5 drawing units. Snap mode may be turned off by specifying a snap resolution of zero, or by using the command:

Command: SNAP On/Off/Value: OFF

Snap mode may be turned back on (using the previous value) with:

Command: SNAP On/Off/Value: ON

(If this form is used while the snap resolution is set to zero, AutoCAD will prompt you to supply a new value.) See Section 5.6 for another method of controlling Snap mode.

Note that setting or changing the snap resolution only affects the coordinates of points entered subsequently. Points already in the drawing will retain their coordinates, even if they do not line up with the snap resolution now in effect.

Also note that the imaginary snap grid and the GRID command's visible grid are separate, and need not have the same spacing. However, setting the spacing of the two grids to equal or related values is often helpful. See the GRID command, below, for details on accomplishing this.

5.2 GRID Command

The GRID command will display a reference grid of dots, with a desired spacing. This allows you to have a "feel" for the siz of drawing entities and their relationships. The grid may turned on and off at will, and the dot spacing may be chang easily. The grid is not considered part of the drawing - it for visual reference only. A grid will never be plotted on a p plotter. However, if a grid is in effect, it will be plotted a quick plot is sent to a dot-matrix printer. (See Secti 10.5.)

A grid is specified by entering the command:

Command: GRID On/Off/Value(X): <grid value>

where <grid value> is the desired dot-to-dot spacing. If th value is set too small, you might get a display having such tight grid that you could not see the drawing clearly. Th could also occur if you later zoomed out to a smaller sca (shrinking the drawing). In such cases, AutoCAD will display t following message, and will not draw the grid.

Grid too dense to display.

While not necessary, it is often useful to set the grid spaci equal to the snap resolution, or to a multiple of it. If t grid value is zero (the default case), the grid spacing wi equal the snap value automatically. To specify the grid spaci as a multiple of the snap value, put an "X" after the value. F example, if the snap value is 0.1, and you want a visible gr point at every fifth snap point, use a grid value of "5X".

Even if the display shows space outside the drawing limits, t grid will appear only within the limits. This clearly defin the drawing limits on the screen.

The command:

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

will turn the grid off, and:

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

will turn it back on with the previous spacing. GRID will perfo an automatic redraw operation if needed.

See Section 5.6 for another method of turning the grid on a off.

5.3 ORTHO Command

The ORTHO command allows you to ensure that all Lines and Traces drawn using a pointing device will be orthogonal with respect to the coordinate system. This means that all Lines and Traces will be either vertical or horizontal, but never diagonal. If a "rubber band" crosshair is being used while Ortho mode is on, its rubber band line will be orthogonal.

Begin Ortho mode by entering the command:

Command: ORTHO On/Off: ON

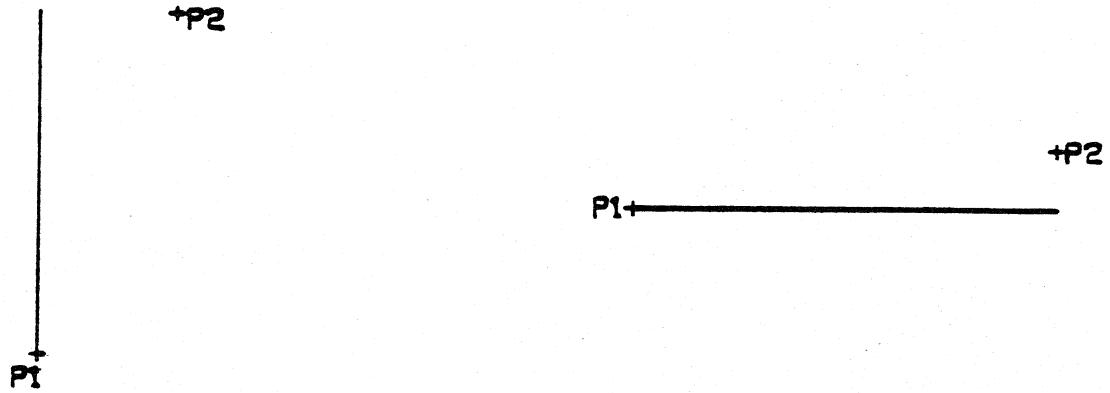
You can return to normal mode with the command:

Command: ORTHO On/Off: OFF

See Section 5.6 for another method of controlling Ortho mode. While in Ortho mode, any Line or Trace is drawn from the first point entered to a point determined by the vertical and horizontal distance to the second point entered.

If the vertical distance is greater than the horizontal, then the line will be drawn vertically with a length equal to the vertical distance to the second specified point. If the horizontal distance is greater, then the line will be drawn horizontally with a length equal to the horizontal distance between the two specified points.

P1 is the first point specified, and P2 the second in the following examples:



Ortho mode applies when the second point of a Line or Trace segment is entered using a pointing device, and when an angle or distance is specified via two points. Keyboard point entry, by its nature very accurate, is not affected by Ortho mode.

5.4 LAYER Command

AutoCAD allows you to group the entities in your drawing on a of 127 drawing "layers". You can control the colour of ea layer, and select combinations of layers for display. It may helpful to think of layers as sheets of transparent draw overlays, although layers are not limited to such uses.

In the Drawing Editor, a "current layer" is always in effe This is the layer upon which newly created entities will drawn. When you begin a new drawing, the current layer initially 1. The LAYER command may be used to change the curr layer; any new entities you then draw will go on the new layer.

Any layer may be made visible or invisible at will, while st being part of the drawing. When beginning a new drawing, o layer 1 is visible. As you select new layers for use, they w automatically be made visible until such time as you explici turn them off with the LAYER command. The command format is:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?:
```

As you can see, several options are available at this point; will discuss them one at a time. After a LAYER option has b processed, the options prompt will reappear. To end the LA command, simply respond to this prompt with a space or RETURN.

If you reply to the options prompt with a layer number (a va between 1 and 127), the current layer will be changed to t layer number. Subsequently drawn entities will appear on the layer.

An "ON" or "OFF" reply to the options prompt will result another prompt for:

```
Layer number(s):
```

You should then supply a list of layer numbers, separated commas, which you want turned on or off. For example, "LAYER 3,4,5" would turn layers 3, 4, and 5 on, while "LAYER OFF would simply turn layer 2 off. Note that the keywords "ON" "OFF" must precede the layer numbers to which they apply.

The "COLOUR" and "?" LAYER options are discussed below.

5.4.1 Colours

As well as a current layer, the Drawing Editor maintain "current colour". Turning a layer on for the first time assi this colour to it. Turning a layer off makes it transpare AutoCAD remembers what colour it should be if it is turned

again later. To change the current colour, use a command such as:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?: COLOUR
  New current colour: 6
```

This actually has two effects; it changes the "current colour" to 6 (affecting layers subsequently turned on for the first time), and it immediately changes the colour of the current layer to 6. Note that the "current colour" is not necessarily the same as the colour of the current layer, since simply changing the current layer, as in "LAYER 5", does not affect the "current colour". (We understand that a lot of this is confusing, and we are working on a new command structure. For now, we suggest some experimentation with the various forms of the LAYER command.)

Colours 1 to 127 are available for use. To facilitate exchange of drawings between different computer systems, the first seven colour 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 hardware used with AutoCAD is not capable of displaying all these colours in this order, and other devices can display many more colours. About all we can state with assurance in this guide is that any layer which is "on" will display on the monitor, and each display driver adheres as closely as possible to the above colour sequence. See your AutoCAD-86 Installation Guide / User Guide Supplement for exceptions, and for information on what will happen if you use colour numbers between 8 and 127.

It is sometimes useful to treat each layer as a different colour; the "LAYER COLOUR *" command accomplishes this easily:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?: COLOUR
  New current colour: *
```

will set the colour of each layer equal to its layer number (e.g., layer 1 will have colour 1, layer 2 will have colour 2, etc.).

Colour Query

The colours assigned to the layers in the drawing may be listed by the "LAYER ?" command:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?: ?
```

The layers will be listed ten per line. Layers which are turn on will have their current assigned colour listed. Layers turn off will show their assigned colour preceded by a minus sig Layers never used will be printed as "..". Lines of all unus layers are not printed.

5.4.2 Mixing LAYER Options

The various options listed can be mixed in any order on o command. After each option has been entered, AutoCAD will retu to the LAYER options prompt, and will perform all the specifi actions when you finally terminate the command by replying to t options prompt with a space or RETURN. For example, suppose entered:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?: 5
  Layer no./ON/OFF/COLOUR/?: RETURN
```

This specifies a single action; make layer 5 the current lay (the one on which newly created entities will be drawn). If th is the first time layer 5 has been selected, the "current colou will be assigned to it. Otherwise, the layer will retain t colour previously set for it. Suppose we continued with:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?: OFF
  Layer number(s): 5
  Layer no./ON/OFF/COLOUR/?: 6
  Layer no./ON/OFF/COLOUR/?: COLOUR
  New current colour: 2
  New current colour: RETURN
```

This would make layer 5 transparent, set the current layer to then assign it colour number 2. (The "current colour" would th also be changed to 2.) Continuing the example:

```
Command: LAYER
  Layer no./ON/OFF/COLOUR/?: OFF
  Layer number(s): 6
  Layer no./ON/OFF/COLOUR/?: ON
  Layer number(s): 5
  Layer number(s): RETURN
```

would turn layer 6 off and layer 5 back on, with its origin colour.

LAYER performs an automatic REDRAW at the end of the command changes have been made which affect the visibility or colour entities.

Layering Example

As an example of the use of layers, a printed circuit board layout might be organized on four layers:

Layer 1: Component side traces (red)
Layer 2: Pads and drilling locations (white)
Layer 3: Solder side traces (blue)
Layer 4: Component side silkscreen legend (yellow)

You could then select a display that showed pads and solder traces, pads and top traces, or top and bottom traces to check continuity and signal routing.

Plotting Layers

AutoCAD only plots the layers that were "on" the last time you worked on the drawing. Consequently, layering allows you to produce separate plots of different layer combinations. By plotting one layer at a time and changing pens between plots, this capability may be used to produce multi-colour plots on a single-pen plotter.

5.5 STATUS Command

There are many defaults, modes, and extents used by AutoCAD. Th STATUS command:

Command: STATUS

will display their current values, as in:

```
49 entities in SAMPLE
Limits are              X:     0.0000    11.0000
                        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
Snap resolution:     0.2500  Grid value:  0.5000
       Grid too dense to display
Axis value:  0.2500
       Axis ticks too close to display
Dimension arrow size:     0.0156
Current layer:   1  Current colour: 15

Axis: Off   Fill: On   Grid: On   Ortho: Off   Snap: Off
Tablet: Off

Free RAM space: 13283 bytes  Free disk space: 67584 bytes
```

The notation "** Over" means that the drawing extends outside th drawing limits. Items relating to optional features (e.g., "Axi value") will be listed only if that feature is present in you copy of AutoCAD. The Dimension arrow size will be listed only i you are actually using the optional Dimensioning feature.

The free RAM space reported relates to the amount of space lef in which to load Shape definitions, and can normally be ignored The free disk space refers to space left on the drive tha contains your drawing file. AutoCAD will terminate execution o the Drawing Editor if it runs out of disk space, after firs saving the work you've done so far.

If your copy of AutoCAD includes the Advanced Drafting Extension package, all distances and coordinates will be displayed in th format specified by the most recent UNITS command (see Sectio 9.2).

5.6 Mode Toggle Control Keys

The SNAP, GRID, and ORTHO commands described earlier in this chapter, and the TABLET command (Section 8.2) may 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'd have to terminate the LINE command, issue the ORTHO OFF command, use LINE to draw your diagonal lines, issue ORTHO ON, and then LINE again. While all this is possible, it is not very convenient.

Therefore, control keys are provided for Snap, Grid, Ortho, 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 keys are:

CTRL B - Snap mode on/off (unfortunately, CTRL S is used by the operating system)
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 will be displayed confirming the action.

There is also a control key for the continuously updated coordinate display (ADEX feature - Chapter 9).

CTRL D - Continuously updated coordinates on/off

On some computers, special function keys may be assigned to these functions in addition to the standard control keys listed above. See your AutoCAD-86 Installation Guide / User Guide Supplement for information. It is also possible to construct screen or tablet menu items to invoke these control functions.

Chapter 6

EDIT AND INQUIRY COMMANDS

6.1 Entity Selection

Most of the commands described in this chapter ask you to select a group of entities for processing. The prompt for this is:

    Select objects or Window or Last:

There are three ways to select a group of objects for processing:

6.1.1 Object Pointing

You can enter a sequence of points, one after another, each on or very near one of the objects you wish to select. Any of the methods of point specification may be used, although use of the pointing device is the easiest. Terminate the sequence of points by pressing space or RETURN. The message:

    nnn selected, nnn found.

will tell you how many entities were found by the object selection process. If you point to an object by mistake, entering **CTRL** C will abort the command and allow you to start over.

NOTE: 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.

6.1.2 Windowing

You may select all the objects contained inside a rectangular area defined by a lower left corner and an upper right corner. Enter **"W"** followed by space or RETURN, and AutoCAD will prompt you with:

    Lower left corner:

Enter a point, and you will get the prompt:

    Upper right corner:

Enter another point. AutoCAD will respond with the message:

    nnn found.

telling you how many entities it found within the window y specified.

Only objects currently visible on the screen will be consider for selection. If only part of an object is visible, it will selected if every part of it that's visible is within the windo

If available, a "box" cursor will be displayed rather than t normal crosshairs so that you may see the window more clear when pointing to the second corner.

6.1.3 Last Entity

To select the last entity you created, simply enter "L" follow by space or RETURN. AutoCAD will respond with:

```
1 found.
```

The entity selected will be the most recently created enti which is currently visible on the screen.

6.2 Edit Commands

Use these commands to modify your drawings.

6.2.1 ERASE Command

The ERASE command lets you specify entities that are to be removed from the drawing permanently. The command 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. Repeated ERASE Last commands may be used 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
Command: ERASE  Select objects or Window or Last: L
```

would erase the two circles.

6.2.2 OOPS Command

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

```
Command: OOPS
```

will restore 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.

6.2.3 MOVE Command

The MOVE command lets you move one or more entities from the present location on a drawing to a new location. The comma first asks you to enter a displacement vector, which is done designating two points, giving the move-from point and then t move-to point. This directed distance will be applied to a selected entities. The command format is:

```
Command: MOVE  Displacement: <1st point or x,y distance>
  Second point: <2nd point or just RETURN>
  Select objects or Window or Last: <show what to move>
```

6.2.4 COPY Command

To make a copy of one or more entities, enter the COPY comman The prompts that appear are identical to those described abo for MOVE. After you "select objects" AutoCAD will draw a copy them at the position specified by the displacement.

```
Command: COPY  Displacement: <1st point or x,y distance>
  Second point: <2nd point or just RETURN>
  Select objects or Window or Last: <show what to copy>
```

6.2.5 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 freely 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 will be discussed separately.

6.2.5.1 Rectangular Arrays

If a rectangular array is selected, AutoCAD will ask 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 will be rejected.

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

```
This command will repeat the selected items <n> 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 will then ask 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 will commence construction of the array. You may terminate this process before completion by entering CTRL C.

6.2.5.2 Circular Arrays

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

    Centre point of array:

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

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

will appear. You should reply with the desired angular spaci between items, in degrees. If the angle is positive, successi items will be inserted in the counterclockwise direction; negative, clockwise. Angles of zero degrees, and those great than or equal to 360 degrees will be 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 elements in the array (counting the original "cornerston element). A negative response is used to indicate the number degrees the array is to occupy. For convenience, a zero reply this prompt defaults to a full circle, and is equivalent to reply of "-360".

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

Point	- insertion point
Circle, Arc	- centre 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 t array's centre, after rotating the designated angle betwe items.

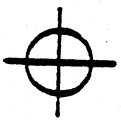
Note that a different reference point is used for each obje being replicated. Thus, the objects may shift with respect each other in each replication. The solution to this is to pla the desired objects in a Block (see Section 7.2) and replica that.

If only one object is selected for the array, and that object a Block, one additional prompt will appear:

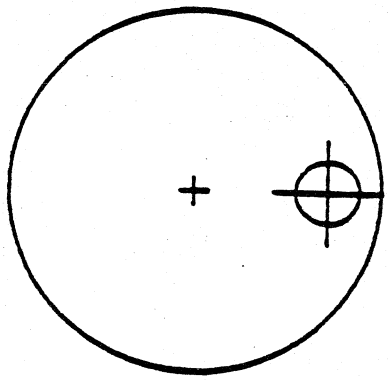
    Rotate block as it is copied?

A "Y" response will cause the replicated block to be rotated around its insertion point in accordance with the array's rotation around its centre point.

An example might make this a bit more clear. Suppose we have defined a Block which looks like:



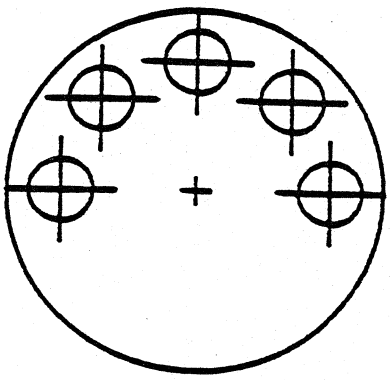
and that we've inserted one of them in the drawing:



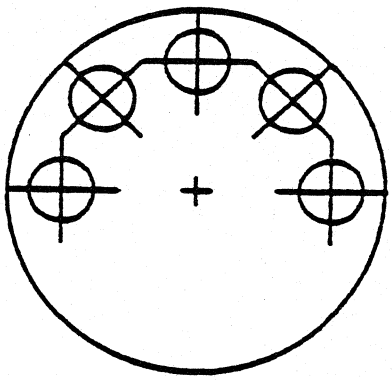
Then we enter the command sequence:

```
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 block as it is copied?
```

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



BLOCK NOT ROTATED



BLOCK ROTATED AROUND ITS BASE

6.2.6 REPEAT and ENDREP Commands

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

To begin a Repeat block, enter:

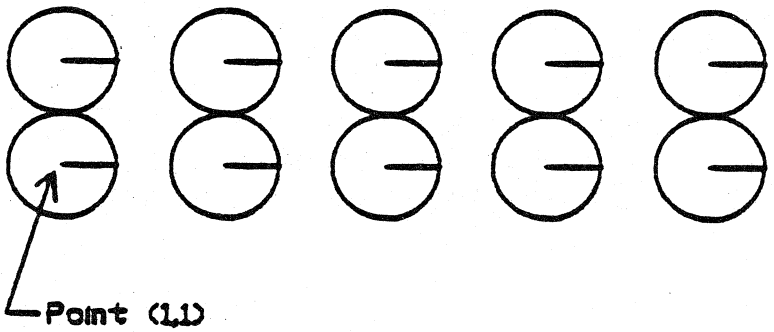
```
Command: REPEAT
```

Next use other AutoCAD commands to draw the basic pattern (o occurrence of the pattern). You may use as many commands desired, as in:

```
Command: CIRCLE  Center point: 1,1
  Radius (or D): Ø.25
Command: LINE  From point: 1,1
  To point: 1.25,1
  To point: <RETURN>
```

When you have drawn the basic pattern, enter the command ENDRE AutoCAD will ask for the number of columns and rows and for t spacing between them; then it will draw the repeated items.

```
Command: ENDREP
  Columns: 5  Rows: 2
  Column distance: Ø.75
  Row distance: Ø.5
```



The column and row distances may be entered by "showing" AutoC two opposite corners of a rectangle defining one cell of t pattern. You may use REPEAT and ENDREP within oth REPEAT/ENDREP blocks without limit.

6.2.7 CHANGE Command

The CHANGE command allows you to modify the properties of some types of entities. Currently, only Lines, Circles, Text, and INSERTed blocks may be changed.

```
Command: CHANGE
  Select objects or Window or Last: <Desired objects>
  Intersection point/L: <enter point>
```

The operation performed depends on the type of object selected. (In the following descriptions, "intersection point" will be abbreviated "IP".)

Line The endpoint closest to the IP will be changed to the designated IP. If ORTHO mode is on, only orthogonal lines will result from a CHANGE.

Circle The radius will be changed so that the circumference of the circle passes through the specified IP.

Text A new location, angle, height, and text may be defined. The height and angle may be specified by entering a point, which will be relative to the new text location, or to the original location if no IP was specified. Note that to retain the old text, you must press RETURN when the "New text:" prompt appears.

Block A new location and angle may be specified. The angle may be specified by entering a point, in which case it will be relative to the new IP, or to the block's origin if no IP was specified.

Changing Layers

A variation of the CHANGE command permits changing the layers on which selected entities reside. Respond to the "Intersection point/L:" prompt with an "L" followed by space or RETURN, as in:

```
Command: CHANGE
  Select objects or Window or Last: <Desired objects>
  Intersection point/L: L
  New layer: <layer number>
```

Each selected object will have its layer changed to the new value. (This form of the CHANGE command works for all entity types.)

6.3 Inquiry Commands

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

6.3.1 LIST Command

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

```
Command: LIST
  Select objects or Window or Last: <Objects to list>
```

The information that is listed depends on the type of entity. For a line, the coordinates of the two endpoints and the laye the entity is on are displayed.

6.3.2 DBLIST Command

The DBLIST command will list data on every entity in th drawing's data base and is primarily used during trainin exercises and debugging. The command 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 DBLIS command and return to the command prompt by typing **CTRL C. CTRL S** may be used to momentarily stop the listing; typing any character will cause the listing to resume.

If you have created "blocks" using the INSERT or BLOCK commands (see Chapter 7), the DBLIST output will include the block definitions as well as all existing references to them. Even if you have erased all such references, the block definitions will still be present in the drawing database, and will therefore be listed.

6.3.3 DIST Command

The DIST command measures the distance between two designated points, and displays the distance in drawing units. If your copy of AutoCAD includes the Advanced Drafting Extensions (ADEX) package, the result will be printed using the current display format (see the UNITS command in Section 9.2).

```
Command: DIST
  First point:  <point>
  Second point: <point>
  Distance = <actual calculated distance>
```

As an added convenience, if you supply a single decimal number to the DIST command, it will display that number in the current display format.

6.3.4 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>
```

Alternatively, you can specify the point with numeric coordinates, and AutoCAD will draw a little blip to show you where that point is on the screen. (The blip will vanish when the screen is next redrawn.)

6.3.5 AREA Command

The AREA command allows you to specify any number of points enclosing a space on the drawing, and AutoCAD will calculate the area of the enclosed space. The command is:

```
Command: AREA
  First point: <point>
  Next point:  <point>
  Next point:  <point>  and so on, finally
  Next point:  (just type RETURN
  Area = <calculated area>
```

The enclosed space will be 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.

Chapter 7

INSERTIONS, BLOCKS, AND SHAPES

7.1 Inserting Drawings

You can create your own library of "parts" interactively, and insert them into your drawing at any specified location. A part is nothing special; any drawing created by the Drawing Editor may be used as a part. The first time you insert a particular part, the part is copied into the drawing you are working on and becomes a Block with the special characteristics described in Section 7.2. Later insertions of the same part use this Block without requiring a reference to the drawing file the part came from. The part will be known by its file name, or by a block name assigned when the part is first used.

It is also possible to create a Block dynamically from a collection of entities in the drawing you are currently working on, using the BLOCK command. Once created, this Block can be inserted in exactly the same way as a part which started out as a separate drawing.

7.1.1 - INSERT Command

The INSERT command is used when, in the process of making a drawing, you wish to insert a previously defined part. The command interaction looks like:

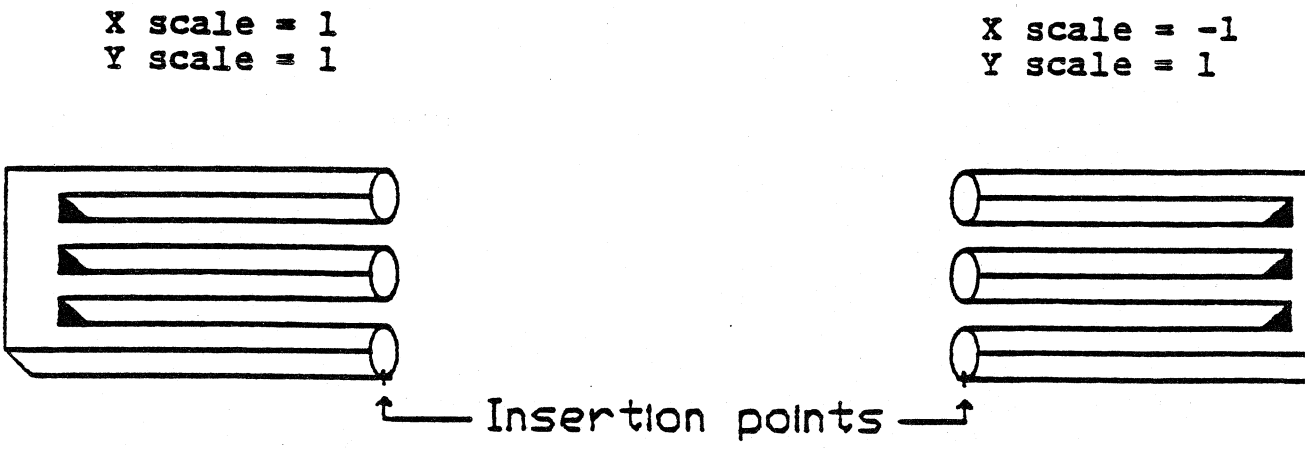
```
Command: INSERT  File name: <enter the name of the part>
  Insertion point: <indicate where it is to be placed>
  X scale factor (default=1) or Corner: <number or point>
  Y scale factor (default=X): <number>
  Rotation angle (default=Ø): <number or point>
```

The part will be copied into your drawing with its base point (see the next section) placed at the insertion point you specified. First, AutoCAD will multiply all X and Y dimensions by the X and Y scale factors, then rotate the part by the rotation angle, using the insertion point as a centre. If you hit RETURN in response to a prompt, the default value is used; thus, you can get the part inserted at its original scale and rotation just by hitting RETURN after all three prompts. Note that the Y scale equals the X scale by default.

Negative Scale Factors

You can specify negative values for the X and Y scale factors, which has the effect of inserting a mirror image of the part. For

example, the part on the left was INSERTed with X and Y sca factors of 1, while the right-hand drawing shows the same par INSERTed with an X scale factor of -1.



Corner Specification

It is possible to define the X and Y scales at the same tim using the insertion point and another point as the corners of box. The X and Y dimensions of the box will become the X and scale factors. If the original definition of the part fit in one by one box, the inserted object will fit in the box you ha defined.

In order to use this feature, simply enter a point in response t the "X scale factor:" prompt. Or, you can reply to this promp with "C", and AutoCAD will prompt for the "Corner point:".

The second point should be above and to the right of th insertion point; otherwise you will get a negative scale facto and a mirror image of the original part.

Angle Specification via Point

If you enter a point for the rotation angle, AutoCAD will use th angle of the line from the insertion point to that point. I Ortho mode is on, the rotation angle will be forced orthogonal.

Retaining Part Entities

No matter how complex the INSERTed drawing part may be, it wil be treated as a single entity by AutoCAD; if you point to on line of it and use the MOVE command, for instance, the entir object will move, not just the line you pointed to.

If you'd prefer to simply transcribe the drawing part, retaining its separate entities, you can do so by preceding the file name with an asterisk on the INSERT command. For instance:

```
Command: INSERT  File name: *NAME
  Insertion point: 10,12
```

Note that scale factors and rotation may not be applied to this type of insertion, and the associated prompts will not appear.

1 x 1 Parts

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 will simply become the actual dimensions in drawing units. This is particularly handy when using the "corner" method of specifying the X and Y scales.

Suppose we created a 1 x 1 drawing called "WALL", with one horizontal line from (0,0) to (1,0) and another above it, from (0,1) to (1,1). Then, whenever we wanted a double line, we could "INSERT WALL". The X scale factor would become the length of the lines, and the Y scale factor would be the distance between them!

7.1.2 BASE Command

When you insert a part in a drawing, you specify a point in that drawing to indicate where the inserted part is to be placed. When creating the part, you also need to specify an origin, or base point for subsequent insertions. This "insertion base" is the location in the part file that will occupy the coordinates you specify in the INSERT command, and is the centre around which the part will be rotated if rotation is used when the part is inserted.

The insertion base is specified while making the part, and is stored in the part drawing file for later use by the INSERT command. If you do not use the BASE command, the point 0,0 in the part becomes its insertion base.

```
Command: BASE  Base point: <point>
```

7.2 Blocks

7.2.1 Block Names

Once a part has been inserted in a drawing, it is known by "block name". If just a file name is entered on the INSER command, that name (less any drive specification) will become th block name. If you wish to enter the file with another bloc name, you can specify "<block name>=<file name>" instead. Fo example:

Command: INSERT File name: MUSTANG=B:EDSEL

This will load the file B:EDSEL.DWG (the ".DWG" is assumed ar should not be specified) and call the block MUSTANG. Block name may be up to 31 characters long.

To list all the block names used in a drawing, use the "INSERT ? form of the command:

Command: INSERT File name: ?

Redefining Blocks

Once INSERTed, a named block may be redefined by using th "INSERT <block name>=<file name>" form of the command. If th block name and file name are identical, the <file name> may omitted, but the "=" must be specified if redefinition of th block is desired. For instance:

Command: INSERT File name: MUSTANG=

would load block MUSTANG from file MUSTANG.DWG and replace th previous definition of block MUSTANG. The drawing will automatically regenerated.

7.2.2 Blocks as Entities

An insertion of a block is considered as an entity by AutoCA As a result, you can MOVE, ERASE, or LIST a block simply pointing to any point on any entity within it. This means th the internal structure of a block is irrelevant once it has be inserted in a drawing; it is considered a primitive entity ju like a line, and can be manipulated as such by the Edit a Inquiry commands.

When a block is inserted, it is physically transcribed into t drawing. This means that you don't have to worry about havi the file you originally inserted available when you later work the drawing into which it was inserted. The block is copied in

the drawing only the first time it is used. Subsequent references to the block simply use the first copy, with different position, scale, and rotation. This makes drawings with many uses of a single block very compact and efficient.

7.2.3 Blocks and Layers

When a block is inserted, its original layer information is also preserved. Thus, an inserted part may contain entities on several layers. However, entities within the block that are on layer 127 will be generated on the layer in effect when the block was inserted. This allows construction of blocks that are inserted on the current layer just like lines, circles, and other built-in entities. Fixed (1-126) and variable (127) layers may be freely mixed within a block.

7.2.4 Nested Blocks

A block inserted within a drawing may itself contain other blocks and block references. For example, one might have a "canned" memory array which was inserted on various printed circuit boards. That memory array may contain blocks which define the various IC pads used within the array. The IC pad itself may be a block. There is no limit to the complexity of nesting blocks. Once a block has been inserted in a drawing, all blocks within it are also available for use in the drawing.

If an inner block includes entities on layer number 127, those entities will "float" up through the nested block structure until an outer block is inserted on a "fixed" (1 to 126) layer; if this occurs, these "layer 127" entities will be placed on that layer. If no "fixed" layer is encountered in the outer layers, the entities will remain on layer 127. For example, consider the following:

```
Block A contains:  Circle on layer 127
Block B contains:  Block A on layer 127
Block C contains:  Block B on layer 3
```

When Block B is inserted, its circle (Block A) will be drawn on the current layer. However, no matter what layer Block C is inserted on, its circle (Block A of Block B) will always be drawn on layer 3.

7.2.5 BLOCK Command - Dynamic Blocks

The BLOCK command lets you create new blocks "on the fly" fr
parts of an existing drawing. This allows more spontaneity whi
drawing, as you don't have to define all your parts ahead of ti
so you can insert them from files.

When you enter the BLOCK command, you will first be prompted f
the block name. Simply enter the name you want the new block
have (as no file name is relevant here, no equal sign shou
appear in the reply). Next, AutoCAD will ask for the insertio
base. Supply the point which is to become the insertion base
the new part (point to it on the screen). This has exactly t
same effect as the BASE command when making an INSERT file. Ne
the standard "Select objects" prompt will appear, and you may u
any of the entity designation options to choose the entitie
which are to make up the new part. To confirm that the entitie
you've selected are the right ones, they will disappear from t
screen.

AutoCAD will then construct a block of the specified na
containing the designated entities with the specified insertio
base. You may then immediately INSERT that block just as y
would a file. The entities which were placed into the block a
deleted from the drawing. If you don't want them to be delete
just say "OOPS" after the BLOCK command finishes, and they wi
be restored to the drawing.

For example:

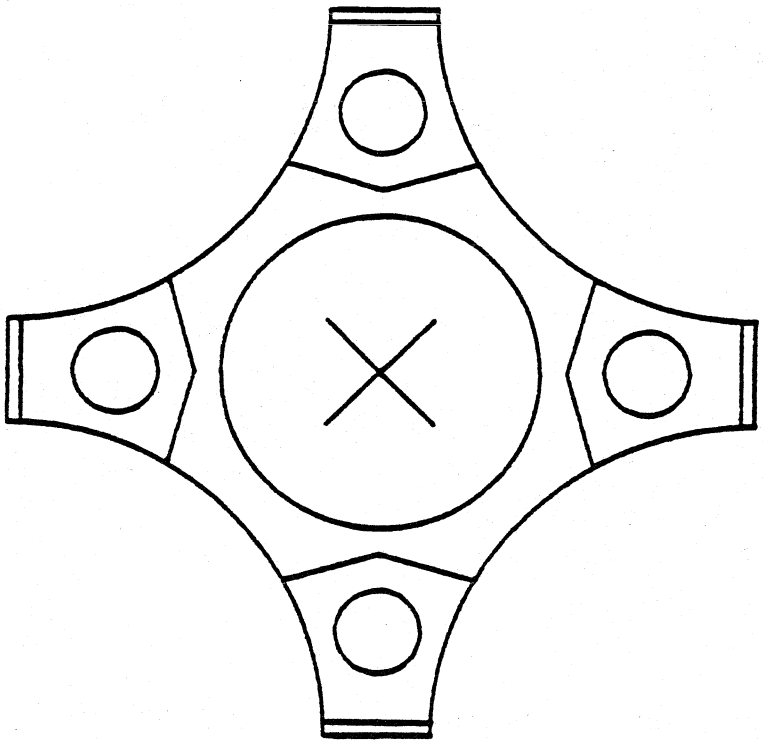
```
Command: BLOCK  Block name: MYPART
  Insertion base point: 2,5
  Select objects or Window or Last: <do so>
```

would create a block called "MYPART" from the selected object
using point (2,5) as the insertion base. This new block may
referenced freely in subsequent INSERT commands, just as thou
it had been loaded from a file using INSERT. However, this blo
will not be stored separately on disk, and therefore will not
available for insertion in other drawings.

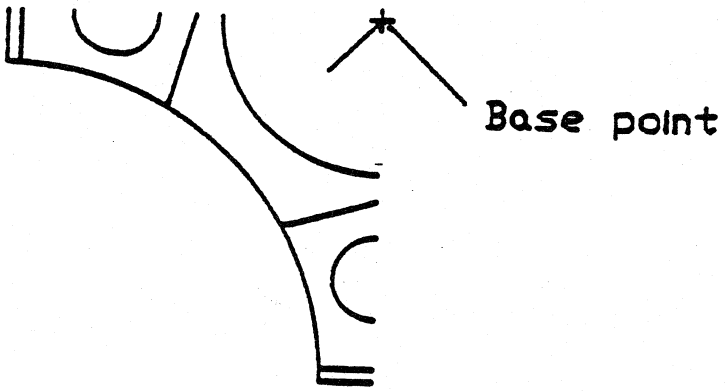
If the specified block name already exists, AutoCAD will a
whether you want to replace the old block definition.

Typical Usage

Often, a figure is composed of a basic group of entitie
duplicated to form a symmetrical object. The BLOCK and INSE
commands may be used to ease the construction of such figure
but this requires some planning. For example, the figure:



ɔuld be created by drawing the simpler object:



ıe BLOCK command could then be used to form a Block from this ɔject, with the indicated base point. Then INSERT could be used ɔ quickly draw the four instances at the same insertion point, .th angles of 45, 135, 225, and 315 degrees.

ıother method of creating this figure would be to INSERT one ɔpy of the Block, and then use the ARRAY command (Section 6.2) ɔ construct a circular pattern. The center of the pattern would ı the Block insertion point, and the pattern would contain four :ems spaced at 90 degrees.

7.2.6 WBLOCK Command - Output to disk

The WBLOCK command may be used to write all or part of a drawi out to a disk file. The format is:

```
Command: WBLOCK  File name: <Output file name>
  Block name:
```

Do not include a file type when entering the file name; a type ".DWG" is assumed.

Four different responses to the "Block name:" prompt a possible:

Name The entities comprising the definition of the specifi block will be written.

= Same as above, but the block has the same name as t specified file.

***** The entire file is written. Whenever you exit from t Drawing Editor via the END command, your drawing automatically saved on disk. This form of the WBLO command can be used to do this periodically if you wis or to write the drawing to a different file, whi remaining in the Drawing Editor.

Blank You will be prompted to select objects and an insert point, just as for the BLOCK command. The select objects will be written to the specified file, and wi be erased from the current drawing. OOPS can be used retrieve them if desired.

Any unreferenced block definitions in the selected group entities will not be written to the file. This provides a mea of deleting block definitions once all references to them ha been deleted.

7.3 Shapes

AutoCAD provides for special types of parts, called "shapes", that are very memory-efficient and easy to use. The configuration of a shape is defined by the user. Once defined, a shape can be put into a drawing, at a desired scale and rotation, with the SHAPE command. This subsection briefly describes shapes and explains how to use the LOAD and SHAPE commands to enter a shape into your drawing.

There are restrictions placed on shapes: they must be capable of being completely specified with one origin point, a scale factor, and a rotation, and must consist exclusively of lines (no circles or arcs). For instructions on defining shapes, see Appendix A.

NOTE: The INSERT command (Section 7.1) provides the primary means of defining and using a library of drawing parts. It is more versatile in every way, and far easier to learn and apply than shape definition. Shapes, however, are more efficient to store and draw, and are 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 speed is of the essence. Most users, though, will not need the information in this section.

Shapes are defined by special files called "shape definition" files. These files have a file type of ".SHP" (see Appendix A). Shape definitions must be LOADed before they are used.

7.3.1 LOAD of Shape Files

Before a set of shapes may be used in a drawing, it must be LOADed into the drawing. This is accomplished by the command:

Command: LOAD File name: <shape file name>

where <shape file name> is the name of the shape definition file (minus the ".SHP", which is assumed). Once the shape file has been loaded into a drawing, it will be automatically loaded every time the drawing is later edited or displayed (that is, you don't need to use the LOAD command every time you use AutoCAD on the drawing, just the first time you need the shapes). Since the shapes are loaded from the .SHP file every time you edit the drawing, you must be sure the file is on the system when you edit the drawing. (As described in Chapter 3, the LOAD command is also used to load Text fonts.)

7.3.2 SHAPE Command

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

```
Command: SHAPE  Name: <shape name>
  Starting point: <enter shape origin>
  Height: <enter size of the shape>
  Angle: <enter angle of the shape, or RETURN for Ø>
```

The shape name is specified in the file loaded into memory AutoCAD decides which definition to use by matching the name yc enter with names specified in the definition file. The messag "**Invalid" will be displayed if no shape can be found with matching name.

The values specified for height and angle are used to control th way the shape is drawn. Their exact effect varies for eac shape. All shapes should have documentation available tha instructs the operator in their use. The height and angle for th shape may be entered either as numbers, or as two points tha "show" AutoCAD the height (as the distance between the points) c the angle (as the direction from the first point to the second).

NOTE: AutoCAD is supplied with two sample Shape files. One i for printed circuit layout and the other is for electroni schematics. Examining these files (PC.SHP and ES.SH respectively) with a text editor and playing with the definitio will help you to master the use of AutoCAD shapes.

Chapter 8

MISCELLANEOUS TOPICS

8.1 Command Menus

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.

When the Drawing Editor is entered to work on a particular drawing, AutoCAD will look for a menu file, "XXX.MNU", where "XXX" is the same name as the drawing file being edited. For example, if you are editing a drawing called "OFFICE", the menu file "OFFICE.MNU" will be searched for. If found, this menu file will be automatically loaded for use.

AutoCAD is distributed with a simple menu file named ACAD.MNU, which has the commands most often used. You can customize this menu or create specific ones for your application (see Appendix C). The MENU command, described later in this section, permits you to load a new menu at any time. The "Configure AutoCAD" task on the Main Menu permits selection of a default menu file.

8.1.1 Screen Menu

A **Screen Menu** is displayed along the right-hand edge of the graphics monitor while you are using the Drawing Editor. The pointing device or keyboard may be used to select commands and modifiers from the menu, as described in Section 2.6.

On some monitors, it may not be possible to display all the menu items on the screen at once. In this case, a special **"NEXT"** item will always be the last one on the screen; selecting it will cause another group of menu items to be displayed.

8.1.2 Tablet Menu

If you have a digitizing tablet, you may elect to use AutoCAD's **Tablet Menu** feature. This menu is a rectangular area in the upper portion of the tablet surface. It is divided into 4 rows of 10 columns each, forming 40 "boxes". Each box is assigned a number from left to right, top to bottom, starting with one (see diagram). A tablet menu master is supplied with AutoCAD. You may copy it and write in the commands you assign to the menu.

Once AutoCAD has learned the position of the menu on the table you can select an item from it simply by pointing at the desir box and pressing the digitizer button. The data on t corresponding line (1 through 40) of the menu file will entered exactly as if you had just typed it on the keyboard.

1	2	3	4	5	6	7	8	9	10
11	12	13	14	15	16	17	18	19	20
21	22	23	24	25	26	27	28	29	30
31	32	33	34	35	36	37	38	39	40
(digitizing in this area ignored)									

(main drawing coordinate digitizing area)

8.1.3 **MENU** Command - Menu File Load

The MENU command is used to load a new menu definition from a disk file. The command format is:

Command: MENU File name: <file name>

The file name should not include a file type; ".MNU" will be assumed. When the command is complete, the contents of the new file will appear in the screen menu area. If you have configured a tablet menu, the newly loaded menu will also be available for use from the tablet menu area.

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

You can change the default menu or realign the tablet menu using the "Configure AutoCAD" task from the Main Menu.

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

8.2 Copying Paper Drawings - Tablet Mode

Use of a digitizing tablet as a pointing device was described Chapter 2, and the Tablet Menu feature was described in Secti 8.1. This section introduces AutoCAD's "Tablet Mode" used f copying existing paper artwork.

When AutoCAD is used on a system equipped with a digitizi tablet, the tablet is normally used to "point" at items on t screen. That is, as the stylus is moved across the tablet, t crosshairs on the screen will follow it. The actual coordinat on the tablet have no meaning: only the screen coordinates th point to relate to drawing coordinates.

This is usually the mode preferred when entering new drawings editing drawings already stored with AutoCAD. When you zoom on a drawing to work on a section in detail, the tabl automatically points with a finer degree of resolution becau the screen now shows a smaller portion of the drawing.

Applications arise, however, when existing dimensioned materi must be entered into AutoCAD. For example, printed circu artwork originally laid out by hand, which you now wish to sto and edit with AutoCAD. When you digitize the artwork, you wa the entities to be entered in the database with coordinates th relate to the original artwork coordinates, not numbers relati to AutoCAD's screen scale.

Perhaps the existing drawing is too big to fit on the tablet. need to be able to enter the drawing in pieces, assur registration between separately digitized parts.

AutoCAD accomplishes these goals with a special mode cal Tablet. When in Tablet mode, AutoCAD treats the digitiz tablet as a true digitizer rather than a screen pointing devi It maps the coordinate system of the original paper draw directly into AutoCAD drawing coordinates, regardless of scale, position, or rotation of the drawing with respect to tablet, and regardless of the display window in effect.

WARNING: Do not attempt to copy a paper drawing without us Tablet Mode -- the aspect ratio of the drawing may be distor if normal pointing mode is used.

8.2.1 TABLET Command

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): 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 will ask you to digitize a point, then enter its coordinates in the system used for the original drawing. It will then ask for a second known point. These points can 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, AutoCAD doesn't care about rotation of the drawing with respect to the tablet. You can put it on straight, at a 45 degree angle, or upside down; AutoCAD will figure this out from the points and known coordinates, and transform the coordinates properly.

Once you've done the calibration, you can use the regular entity entry commands to draw lines, points, circles, shapes, and so on. Whenever a set of coordinates is asked for and you digitize a point, the tablet will return coordinates from your original drawing.

Remember that the drawing limits are still in effect. You should make sure that the drawing limits include all coordinates you'll generate from the tablet. The screen displays entities as they are inserted in the drawing, but 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 will follow the movement of the tablet cursor.

The Snap setting remains in effect in Tablet mode. After the coordinates are transformed, they are forced to points on the Snap boundary. 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 men remains active in Tablet mode and may be used freely. Thus, onl the portion of the tablet below the menu is available fo coordinate digitization in Tablet mode.

If the drawing shifts on the tablet, or if you move the drawin (for example to digitize another portion of a large drawing) yo can recalibrate the tablet to the new position by giving anothe "TABLET CAL" command and entering two new known points.

TABLET Off

You can return to normal screen pointing with the tablet b issuing the command:

Command: TABLET Option (ON/OFF/CAL): OFF

TABLET On

After having turned Tablet mode off with the "OFF" option, yo can turn it back on with the coordinates previously set in thi editing session with the command:

Command: TABLET Option (ON/OFF/CAL): ON

If the tablet has not been previously calibrated in this session the calibration prompts will be issued.

NOTE: Tablet mode also may be turned on and off using a singl control key, as described in Section 5.6.

8.2.2 Entity Pointing in Tablet Mode

The commands that allow you to select entities by pointing a them, such as ERASE, MOVE, and LIST, will still work even thouq you have placed the digitizer in Tablet mode. If you wish t ERASE an entity, you can do this simply by moving the table stylus until it points to the object on the paper drawing, c until the screen crosshair is pointing to the desired entity 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 th screen crosshairs. This is another good reason to do a ZOO Window of the area being digitized to make sure all the digitize items are on the screen.

8.3 FILES Command - Directory Access

It is sometimes useful to list a disk directory without exiting AutoCAD, or even while still using the Drawing Editor. Main Menu task 7 is provided for this purpose, as is the FILES command.

```
Command: FILES
```

Whichever method you use to invoke it, AutoCAD's file utility menu will be displayed:

```
File Utility Menu

  Ø.   Exit File Utility Menu
  1.   List Drawing files
  2.   List Menu files
  3.   List Shape files
  4.   List Pattern files

  5.   List User specified files
  6.   Delete files
  7.   Rename files

Enter selection:
```

8.3.1 Listing File Names

Selections 1 through 4 search for and list the names of selected types of files used by AutoCAD; drawing (".DWG"), menu (".MNU"), shape (".SHP"), and pattern (".PAT") files. AutoCAD needs to know which disk drive to search, so it will ask:

```
Enter drive <X>
```

where X is the current default disk drive. You can reply with another drive letter, or press RETURN to use the default.

Menu selection 5 allows you to supply an explicit file name to be searched for. It will prompt:

```
Enter file search specification:
```

Supply a file name in the format accepted by your computer's operating system. If you wish to scan a disk drive other than the default, include the drive letter in your response, as in "B:WIDGET.BAK". You may use the "wild card" characters "?" and "*" if desired. The "?" will match any character in that position, while "*" matches all characters up to a period, or to the end of the file name.

8.3.2 Deleting Files

File Utility Menu selection 6 allows you to delete specifi files. This can be very handy. If you are running out of di space, for example, you can make some room by deleting unneec files (perhaps some ".BAK" backup files). The prompt is:

Enter file deletion specification:

As with selection 5, you should supply an explicit file nam optionally containing the "?" and "*" wild card characters. no wild cards are used (the name is unambiguous and refers tc single file), the specified file will simply be deleted. Howeve if wild card characters are included, AutoCAD will prompt y with each matching file name, and ask whether you want to dele it. For example, if you specified "B:?X.*", the followi dialogue might ensue:

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

A "Y" response will delete the file, whereas "N" will retain on the disk. As indicated, retention of the file is the defaul

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

8.3.3 Renaming Files

Selection 7 on the File Utility Menu permits you to change t name of an existing file. A sample dialogue would be:

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

This would change the name of file WIDGET.DWG (on drive B) THINGO.DWG.

8.4 Command Scripts

AutoCAD provides a script facility which allows commands to be read from a text file. This feature can be used to execute a predetermined sequence of commands as soon as you begin running AutoCAD. It can also provide a continuously running display for product demonstrations and trade shows. Script usage is invoked when AutoCAD is first loaded, using the command form:

A>ACAD <default drawing> <script file>

The script file must be the second file named on the ACAD call line, and is assumed to have a file type of ".SCR". Commands will be read from the script file starting with the Main Menu prompts, so the first line of the file should select the task number.

For example, suppose that every time you began creating a new drawing, you immediately turned the grid on, loaded a fancy text font, and set layer 3 as your current layer, with colour 5. All this could be accomplished by the following script stored in a file called, say, SETUP.SCR:

```
1                       (create new drawing)
                        (blank: use default name)
grid on                 (turn on grid)
load fancy              (load text font)
layer 3 colour 5        (set layer options)
                        (blank to end LAYER command)
```

Now, if you wanted to create a new drawing called B:WIDGET, you might invoke AutoCAD as follows:

A>ACAD B:WIDGET SETUP

This would set the default drawing name to B:WIDGET and begin reading commands from SETUP.SCR. These commands would create a new drawing with the default name (B:WIDGET), and proceed to issue your usual sequence of setup commands. When the end of the script file is encountered, the "Command:" prompt will appear and AutoCAD will await 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 these prompts may change in new releases of AutoCAD.

8.4.1 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 (sort of an electronic flipchart). This can be accomplished easily, using a variation of the script

mechanism. If the last command in the script file is "END" o "QUIT Y", the main menu will reappear and the script file will b restarted from the beginning.

Two additional commands are available to make continuous script more flexible.

8.4.2 DELAY Command

Some AutoCAD operations happen rather quickly, making i difficult for people to see what's going on. For instance, i your script draws a line and then erases it, your audience migh not see the action. The DELAY command is provided to cause sufficient pause between such operations. The format is:

Command: DELAY <number>

where <number> specifies the length of the pause. The larger th number, the longer the delay. Due to the wide range o processing speeds for the computer systems used with AutoCAD, i is impossible to be specific about the delay time, but you ca experiment on your machine by entering "DELAY 1000", fo instance, from the keyboard. See how long it takes for th "Command:" prompt to reappear.

8.4.3 RESUME Command

Pressing any key on the keyboard will interrupt a running scrip at the end of the current command, and let you issue command using the normal methods. (The character you enter will becom the first character of a command; we recommend interruptin scripts with the **Backspace** key to avoid confusion.)

If you later wish to return to the script, simply enter:

Command: RESUME

Continuous Script Example

The following is a simple example of a continuous script:

```
1                          (new drawing)
demo                       (drawing name)
line 1,1 2,1 2,2 1,2 c     (draw a box)
delay 500                  (pause)
move 3,0  w 0,0 3,3        (move box to right)
delay 1000                 (pause)
quit y                     (cycle)
```

If we stored this script in a file named BOX.SCR, we could invok 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 will be ignored. The script will create drawing "demo", draw a box one unit square, and then move the box 3 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,Ø" and "w" for the MOVE command. The first space is needed to input the "3,Ø" and the second space indicates to the MOVE command that "3,Ø" was not a point, but a displacement. The characters in the script file are processed just as if they had been entered from the keyboard, and every character counts.

An error encountered while processing a command from a script file will cause the script to be terminated. If this occurs while the Drawing Editor is active, the RESUME command may be used to continue the script.

Chapter 9

ADVANCED DRAFTING EXTENSIONS

The features described in this chapter are contained in the extra-cost Advanced Drafting Extensions (ADEX) package. If this package was not purchased with your AutoCAD system, an attempt to use any of these features will simply result in the message:

    Advanced drafting extension, not present.

9.1 AXIS Command - Ruler Lines

The AXIS command instructs AutoCAD to display a "ruler line" on the graphics monitor, with specified tick spacing. The command format is similar to that of the GRID command:

    Command: AXIS On/Off/Tick spacing(X):

Responding "On" or "Off" will change the status of the axis line accordingly. The tick spacing is specified in terms of drawing units. Alternatively, you may follow the number with an "X" to set the tick spacing relative to the Snap resolution.

As with the GRID command, a small tick spacing (or ZOOMing out) may result in ticks so dense that you cannot discern them on the screen. If this occurs, AutoCAD will display the following message:

    Axis ticks too close to display.

To display the axis, issue another AXIS command, specifying a larger tick spacing.

Wnen a "feet and inches" display format is in effect (see the UNITS command in the next section), some of the tick marks may be double size, to indicate whole inches or whole feet. The large tick marks will appear only if the specified tick spacing is an exact fraction of an inch or a foot.

9.2 UNITS Command - Format Control

In this user guide, all examples of distances and coordinat have used ordinary decimal notation. However, in so disciplines, other forms of notation are preferred. T precision of displayed numbers may also be subject to person preference. In order to accomodate these varied requirement AutoCAD's Advanced Drafting Extensions package includes t "units" capability.

You may select a default display format and precision when y configure the program (Main Menu Task 4). Also, you can use t UNITS command to change formats while drawing. The UNITS comma is invoked by entering:

Command: UNITS

The following menu will be displayed:

System of units:

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

Enter choice, 1 to 4 <default>:

The default shown is the format currently in effect. You m retain this format by simply pressing RETURN. To illustrate t various output formats, we will show how a distance of 15 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 "fe and inches" displays; these formats assume that each drawing un represents one inch.

Once you have selected the format, AutoCAD will ask for t precision. The prompt depends on which display format you ha 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 u 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.

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 is slightly different 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.

9.3 Mode/Coordinate Display

An area of the graphics monitor may be used to display t current status of Fill, Ortho, Snap, and Tablet modes. T current layer number is displayed, as well. On some comput systems, the current coordinates of the screen crosshair may al be displayed. Thus, the status line might look something li this:

```
foSt L:1                          X=1.1234 Y=2.3456
```

The "foSt" are letters denoting Fill, Ortho, Snap, and Tabl modes; a capital letter will be displayed if the associated mo is on, while a lower-case letter signifies that the mode is of In this example, Snap mode is on and the others are off. "L: indicates that layer 1 is the current insertion layer.

You can decide at configuration time whether or not to displ the status line (see Appendix D). In fact, you may select one three alternatives:

1) No status line

   On some displays, suppression of the status line wi make additional room available for drawing.

2) Status line with continuously updated coordinates

   This would be the default on many systems. As t pointing device is moved, the drawing location to whi it points is displayed. **CTRL D** may be used to fl between this display mode and the one described below

3) Status line without continuously updated coordinates

   Choose this option if the continuous updating of t pointer location hampers your concentration, or if results in a noticeable slowdown in crosshair movemer This option will cause the coordinates in the stat line to be updated only when you actually select point.

9.4 BREAK Command - Partial Erase

The BREAK command erases part of a line, trace, circle, or arc. To use it, select the object to be broken, and then 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 will respond with:

```
Enter second point (or F):
```

In this case, the "Enter first point:" prompt is skipped; it is assumed 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 will find the nearest point on the object. If you simply 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.

The exact effect of the BREAK command depends on the type of object being broken.

Line If both specified points are within the ends of the line, the line will be 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 will simply be cut off.

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

Circle A circle will be changed to an arc by removing a piece going counter-clockwise from the first to the second point.

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

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

9.5 FILLET Command

The FILLET command will connect two lines by means of a smooth fitted arc of a specified radius. It will adjust the lengths the lines so that they end exactly on the arc.

```
Command: FILLET
  Select two lines or R: <Point to two lines>
```

After pointing to the two lines, do not hit the space bar. T FILLET command knows that you will select only two points. Y can select the lines with a Window specification, but this risky; if there are more than two lines in the window, two lin will be selected arbitrarily (usually the last two lines drawn)

The radius to be used for fillets is remembered as part of t drawing file. It is initially zero. To change the fill radius, reply "R" to the prompt.

```
Command: FILLET
  Select two lines or R: R
  Enter fillet radius: <enter value>
```

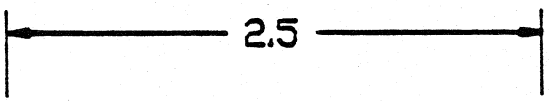
FILLET works by extending the two selected lines, if necessar until they intersect. The lines are then trimmed and the fill arc created. (A fillet radius of zero can be used to adjust t lines so that they end precisely at the same point.) If bo lines are on the same drawing layer, the fillet will be placed that layer. If the lines are on different layers, the fill will be placed on the current layer. If no intersection point found within the drawing limits, the command will be rejected.

Given a fillet radius of Ø.5, the effect of a FILLET command shown below.

```
Command: FILLET
  Select two lines or R: <Point to the two lines>
```

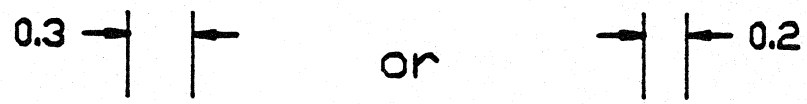
.6 Semi-automatic Dimensioning

n AutoCAD, a dimension consists of two extension lines (referred o as "witness lines" in some texts) which lead from the points eing dimensioned to the specification of the dimension, a imension line which has arrows on the end and which points to he extension lines, and the dimension text, which gives the ctual distance between the two extension lines. If there is ufficient room between the extension lines, the dimension line nd text will be within them, as in:



f there isn't room between the extension lines, the dimension ine will be split and point to the extension lines from outside, s in:

utoCAD dimension lines always run either horizontally or ertically. You can dimension a diagonal by running the xtension lines to the ends of the diagonal line, but note that utoCAD will only measure the vertical or horizontal aspect of he line, not its true diagonal length.



9.6.1 DIM Command

To create a dimension, enter the DIM command:

Command: DIM

The first prompt to appear will be:

First extension line origin or (ABCT):

You should designate the point near the item being dimensioned where you want the first extension line to appear. Next the prompt:

Dimension line intersection:

will appear. You should then designate where the dimension line should be placed relative to the first extension line point (i.e., where the arrow will hit the first extension line). If the line between the first extension line origin and the dimension line intersection is more horizontal than vertical, the dimension line will run vertically, if more vertical, the dimension line will run horizontally. (Draw a couple and you'll understand.) Finally, you will be asked

Second extension line origin:

which requests you to designate the point near the object where the other extension line is to go. Before moving on to the rest of the command and its action, let's look at how these three points are used to define the dimension.

The first extension line will always start at the point designated for the first extension line origin. It will run either vertically or horizontally depending on the predominant direction between the origin point and the dimension line intersection. Any displacement of the dimension line intersection point from a true vertical or horizontal is ignored--it supplies only the direction and distance between the extension line end and the dimension line (or in other words, the dimension is taken from the extension lines, not the dimension line, which is what you want). The second extension line is then drawn from the second extension line origin point, in the same direction as the first extension line, as far out as the first extension line went (note that the ends of the extension line at the dimension line always line up, but the ends nearest the item being dimensioned may appear anywhere, to accomodate strangely shaped objects). The dimension line will then be run either vertically or horizontally from the first extension line to the second.

9.6.2 Dimension Text

AutoCAD will request the dimension text with the prompt:

```
Dimension text:
```

If you enter text, that text will be used as-is. Since the text is terminated by a RETURN, spaces may be used within it. If you just reply to the prompt with RETURN, AutoCAD will measure the (orthogonal) distance between the extension lines and edit it as specified by the latest UNITS command (Section 9.2). Trailing zeroes and decimal points will be suppressed, as will zero feet or zero inches if "feet and inches" units are in effect.

The ANSI drafting standard specifies "unidirectional" dimensioning as the preferable technique, so AutoCAD's default is to draw the dimension text horizontally, even for vertical dimension lines. You may force the text to run in the direction of the dimension line by using the **DIM T** form of the command:

```
Command: DIM
  First extension line origin or (ABCT): T
  Text within the dimension can be drawn along the dimen-
      sion or horizontally.  Do you want it horizontal? <Y>:
  Text outside the dimension can be drawn along the dimen-
      sion or horizontally.  Do you want it horizontal? <Y>:
```

Note that the orientation may be specified separately for the case where the dimension lines and text will fit within the extension lines, and for the case where they will not. For the purposes of this discussion, however, we will assume that the dimension text is horizontal.

9.6.3 Arrow Size

The ANSI drafting standard specifies that drawn arrows should be 1/8 inch in size. Since AutoCAD doesn't know the scale of the final drawing, it doesn't initially know how big to make the arrows. Thus AutoCAD has a variable called **arrow size** which controls this. If no arrow size is specified, AutoCAD will use 1/64th of the drawing size (horizontal or vertical drawing limit, whichever is smaller). The arrow size controls the size of the arrows themselves, which are drawn with solid fill at the ANSI-specified aspect ratio of 1 to 3. The arrow size also controls how far the extension lines extend past the dimension line (1 arrow length), how much spacing will be placed between the dimension text and the dimension lines (1 arrow length), and the height of the dimension text (1.5 times arrow length). Further, if the arrows must be moved outside the extension lines, their shafts will have a length of one arrow length. The arrow size can be explicitly specified by the **DIM A** command:

```
Command: DIM
  First extension line origin or (ABCT): A
  Dimension arrow size: <enter size>
```

and will be remembered as part of the drawing. ANSI specifi that all arrows in a drawing shall have the same size.

The Result

Now that all of the dimension components have been acquired, t dimension is drawn on the screen. First, AutoCAD determin whether the dimension lines, arrows, and text will fit within t extension lines, or whether they must be drawn outside. possible, they will be drawn inside, with the text centred in t dimension line.

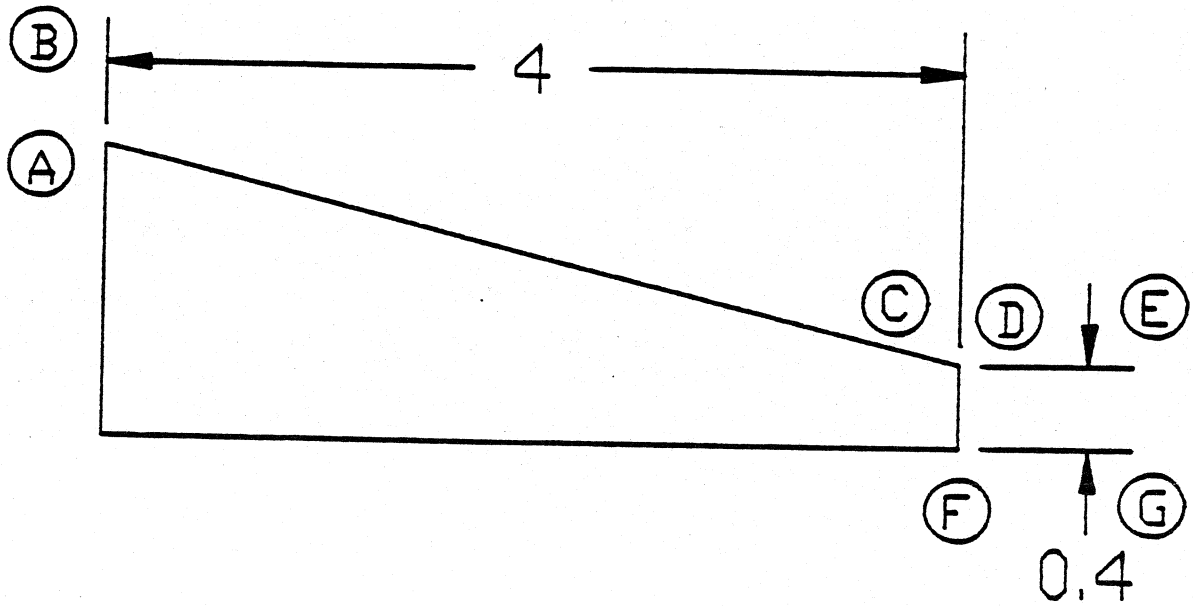
In the case of a horizontal dimension line, the length of t text is an important consideration in composing the dimension. the vertical case, only the text height matters. Howeve ignoring the length might overlay the object being dimension with the text. Consequently, for both the horizontal a vertical dimension line cases, AutoCAD computes the text leng using a height of (1.5 times arrow size). If there is not enou room between the extension lines for the dimension lines, arrow and text length (or height), these will be moved outside t extension lines. For a vertical dimension line, text whi extends past the end of either extension line will be shifted guarantee that the text won't overlap a straight figure. If t figure is convex, it may still overlap, and you will have to mo the dimension line further out.

The extension lines and dimension line may have any geometric relationship you desire; you may draw vertical dimensions top bottom or bottom to top, and with the dimension line to the le or right of the extension line origins, and of course t equivalent for horizontal dimensions. The order in which y specify the ends matters only when the arrows are drawn outsi the extension lines, as the text will be placed on the end of t second-specified dimension line (giving you control placement), and when continuing dimensions (see below).

All the items that make up the dimension (lines, solids, text are drawn as primitive entities, so they may be manipulate individually.

The current arrow size is printed in the STATUS display if it nonzero (and thus only if a DIM command has been used in t drawing).

n example would now be in order.



he dimensions in this drawing were created with the commands:

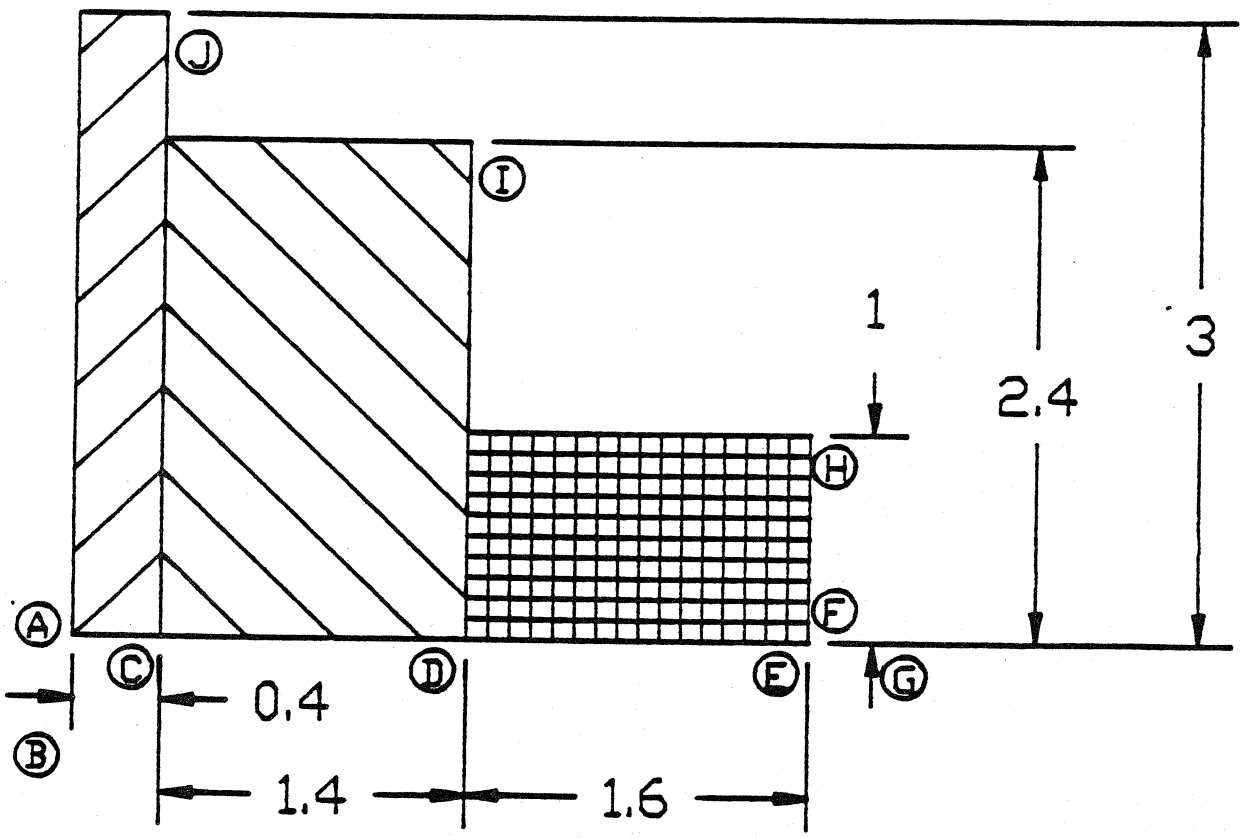
```
Command: DIM
  First extension line origin or (ABCT): A
  Dimension arrow size: 0.2

Command: DIM
  First extension line origin or (ABCT): <point "A">
  Dimension line intersection: <point "B">
  Second extension line origin: <point "C">
  Text: <RETURN to measure distance>

Command: DIM
  First extension line origin or (ABCT): <point "D">
  Dimension line intersection: <point "E">
  Second extension line origin: <point "F">
  Text: RETURN
```

ote that although points "A" and "C" are at the ends of a iagonal, the first dimension has been taken and drawn orizontally. The second dimension has been drawn vertically, but ie distance between the extension lines was too small to hold ie arrows, dimension lines, and text. The arrows were therefore awn outside the extension lines, and the text was placed near oint "F" because it defined the second extension line. (If we id defined the second dimension by designating points "F", "G", id "D", the text would have been drawn above the extension nes.)

9.6.4 Continuing Dimensions

At the point the "First extension line origin:" prompt appea you may enter "B" or "C" to continue the previous dimensi Either reply immediately advances you to the "Second extens line origin:" prompt, and draws a new dimension based on the l one. "B" adds a dimension relative to the first point of last dimension, drawing the dimension above (or whatever) last one. This is what you do when you want a set of dimensi all relative to the same base line. Multiple "B" DIM comma will all reference the same base line. The "C" response draw dimension relative to the second extension line of the l dimension, with the dimension line colinear with the last drawn (unless the last one was outside the extension lines, which case the new dimension line will be moved to clear i Thus multiple "C" DIM commands align.

For example:



In this drawing, the dimensions were created with the commands:

```
Command: DIM
  First extension line origin or (ABCT): <point "A">
  Dimension line intersection: <point "B">
  Second extension line origin: <point "C">
  Text: RETURN
Command: DIM
  First extension line origin or (ABCT): C
  Second extension line origin: <point "D">
  Text: RETURN
Command: DIM
  First extension line origin or (ABCT): C
  Second extension line origin: <point "E">
  Text: RETURN

Command: DIM
  First extension line origin or (ABCT): <point "F">
  Dimension line intersection: <point "G">
  Second extension line origin: <point "H">
  Text: RETURN
Command: DIM
  First extension line origin or (ABCT): B
  Second extension line origin: <point "I">
  Text: RETURN
Command: DIM
  First extension line origin or (ABCT): B
  Second extension line origin: <point "J">
  Text: RETURN
```

The first horizontal dimension was specified normally, designating points "A", "B", and "C". Then the "C" response was given to continue that dimension, creating a new dimension from points "C" to "D". This method was used again to add a dimension from points "D" to "E". All three dimensions would have "lined up" horizontally if the first one had fit within its extension lines.

The vertical dimensions were drawn using the "B" method to make the second and third dimensions start at the same point as the first one.

9.7 Crosshatching and Pattern Filling

In many drafting applications, it is common practice to fill area with a pattern of some sort. The pattern can he differentiate between components of a three-dimensional objec or can signify the material composing an object. This proces called "crosshatching" or "pattern filling", may be accomplish using AutoCAD's **HATCH** command, and will be referred to simply "hatching" in this discussion.

AutoCAD is provided with a library of hatching patterns used normal drawing. This library is in the file ACAD.PAT, whi should be on your AutoCAD execution disk. You may hatch with o of these patterns, with a simple pattern defined "on the fly" the HATCH command, or with a user-defined pattern in your o file. Pattern definition will be described later.

What are hatch lines?

Hatching generates Line entities for the chosen pattern and ac them to the drawing. Since a pattern may be made up of ma lines, these lines are normally grouped into an internal generated block. Thus, if you have hatched an area and deci you don't like the hatching, you can just point to any line of with the ERASE command and all the hatching will go away. ERA Last may be used when appropriate. If you wish to edit t output of the hatching process manually, or if you're debuggin pattern and want to be able to examine the individual genera lines, you can suppress this block grouping.

The block generated for hatch lines will not appear in t "INSERT ?" display, and will be automatically deleted if a references to it are deleted.

Defining the Boundary

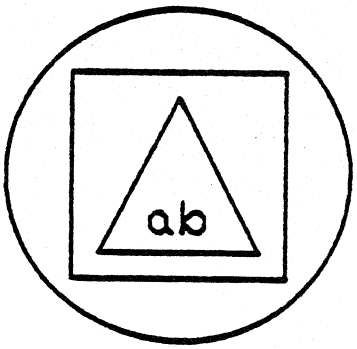
Hatching works on areas of the drawing enclosed by a bounda made up of Line, Arc, Circle, and Trace entities. When hatch an area, you must select the entities which define the bounda via the normal "Select objects" mechanism; of course, Window Last selection may be used.

Hatching of the enclosed area may be affected by the presence other entities inside the boundary. However, hatching only kn about entities it has been told about via the object select mechanism -- a Window selection will guarantee that all inter structure is correctly seen.

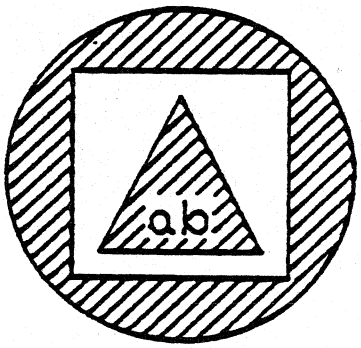
.7.1 Hatching Styles

f the space inside the boundary is empty (or if none of the nternal objects have been selected), the area simply is filled ith the chosen pattern. If anything inside the boundary has een included in the entity selection, what happens depends on he "style" of hatching. Three styles are available.

et us consider a circle with a square inside it which has a riangle inside it as illustrated at the left below. There is lso some text in the figure; the effect of Text entities on atching will be discussed later. For these examples, assume hat a Window selection has been performed, so all the entities re included in the selection.



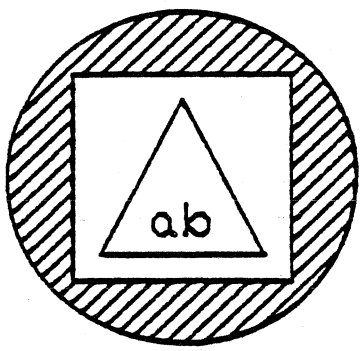
AREA TO BE HATCHED



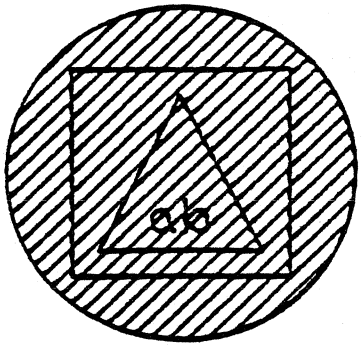
NORMAL STYLE

he **Normal** (default) style of hatching is illustrated in the igure on the right. This style hatches inward starting at the rea boundary, at each end of each hatch line. If it encounters n internal intersection, it turns off hatching until another ntersection is encountered. Thus, areas separated from the ıter boundary by an odd number of intersections are hatched, ıile areas separated by an even number of intersections are not.

ıe second style of hatching is **Outermost**. This style also ıtches inward from the area boundary, but it turns hatching off : an internal intersection is encountered, and does not turn it ıck on. Since this process starts from both ends of each hatch ine, the effect is that only the outermost level of the :ructure is hatched, and all internal structure is left blank. ıe drawing on the left, below, illustrates the effects of ıtermost hatching on our example figure.



OUTERMOST STYLE



IGNORE STYLE

The third style of hatching is Ignore. Here, all inner struct is hatched right through. In our example, each hatch line wo start and end on the circle (the outer boundary), and pass ri through the inscribed triangle and square as if they were there, as shown at the right, above. Normally the effect Ignore style hatching can be achieved simply by not selecting internal entities, but the style option is provided in case wish to do a Window selection for convenience, or in case blo are involved which would prevent your selecting the bound without the internal structure.

Hatching Text, Shapes, Traces, and Solids

Text, Shape, Trace, and Solid entities behave specially w regard to hatching. If a hatch line would pass through such entity, it will be automatically turned off, so that th entities won't be hatched through. In fact, Text and Sh entities will be surrounded by a magic invisible box wh encloses the entity with a margin for readability, which disab all hatch lines which would otherwise pass through it. T means, for example, that you can draw a pie slice, label it w text, and then hatch it in confidence that the text will rem readable. This does not apply, of course, if Ignore st hatching is selected--in that case the text will be hatched ri through.

Composite Entities and Hatching

INSERTed blocks work correctly with hatching. This means t what you see on the screen will be hatched the same regardless of whether none, some, or all of what you see i member of one or more inserted blocks. This is true regardl of the X scale, Y scale, and rotation with which the block were inserted. If scaling turns a circle or arc into an elli or elliptical arc, the hatch lines will stop at the elli correctly.

REPEATs are another matter. HATCH cannot detect that a selected entity is a member of a REPEAT, and it will hatch 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.

9.7.2 HATCH Command

Hatching is performed using the HATCH command:

```
Command: HATCH
  Pattern (name,style / U / ?):
```

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 will look 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 will be printed. This is handy when you forget the name of a pattern.

If you answer "U" to this prompt, you will be allowed to define a simple pattern on the fly. AutoCAD will ask the three questions:

```
Angle for crosshatch lines:
Spacing between lines:
Double hatch area (Y/N) <N>?
```

which should be answered with the angle, interline spacing, and whether you want a second set of lines at 90 degrees to the original lines to also be drawn. You can answer the first two prompts with numbers or by "showing" AutoCAD via two points.

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 will normally block together all hatch lines generated one HATCH command. If you do not wish this to happen, prece the reply to the "Pattern" prompt with an asterisk, "*". Thu if you want to hatch an area with the "BRASS" pattern, and ha the hatch lines entered individually, reply "*BRASS" to t prompt.

If you have selected a pattern from a file (e.g., not "U"), y will receive two additional prompts:

```
Scale for pattern <1>:
Angle for pattern <Ø>:
```

Each pattern is defined with an initial size, and with a rotati of zero degrees. You may expand or contract the pattern, rotate it with respect to the axes by supplying the desir values to these prompts. The defaults, as indicated, leave t pattern as defined. Most standard patterns have a characterist size of 1 drawing unit, so you may respond to the "Scale" prom with the desired size of the pattern, and have the pattern automatically scaled to that size. You may respond to the prompts with two points to "show" AutoCAD the size or angle.

After the pattern and style are chosen, you will be prompted:

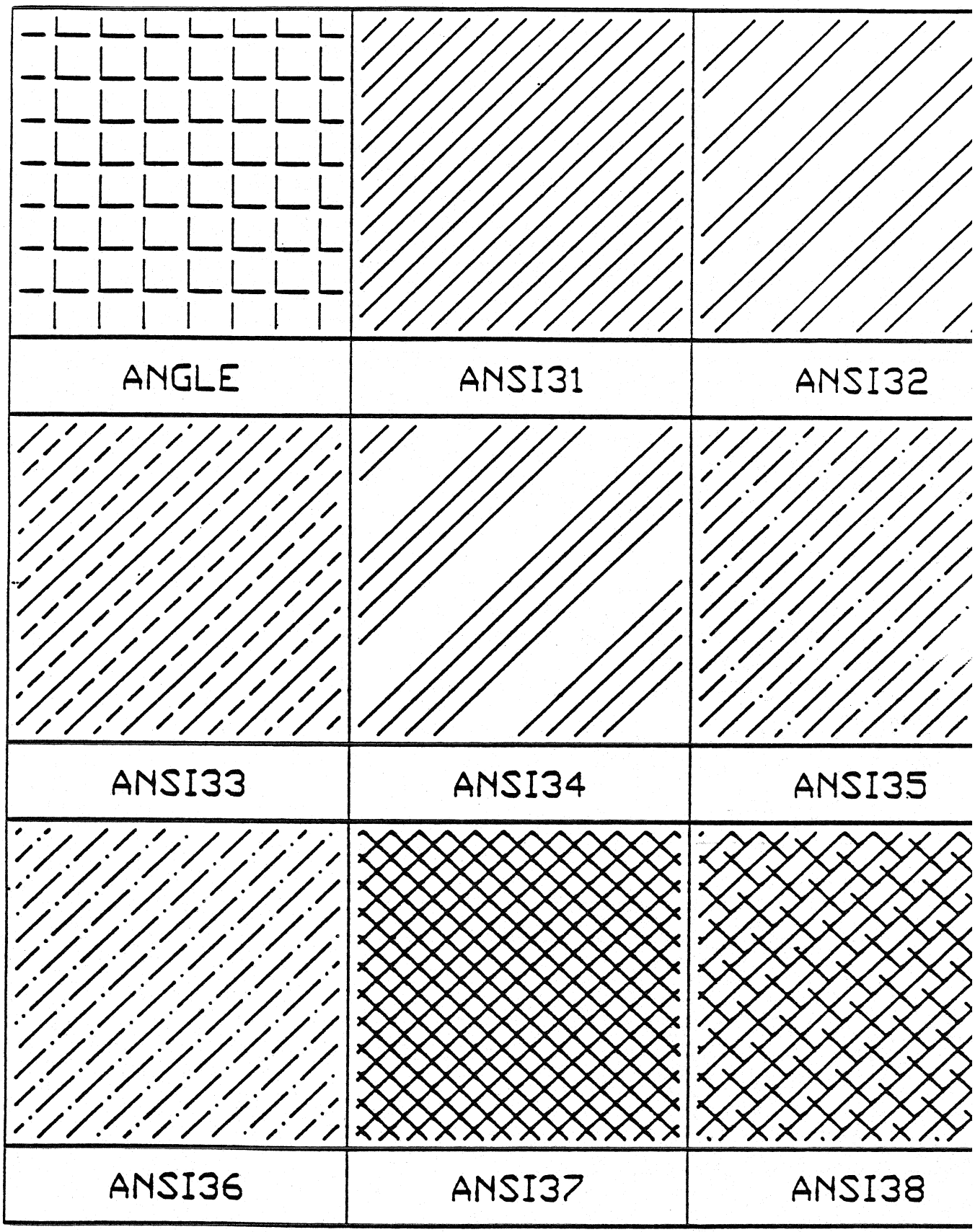
```
Select objects or Window or Last:
```

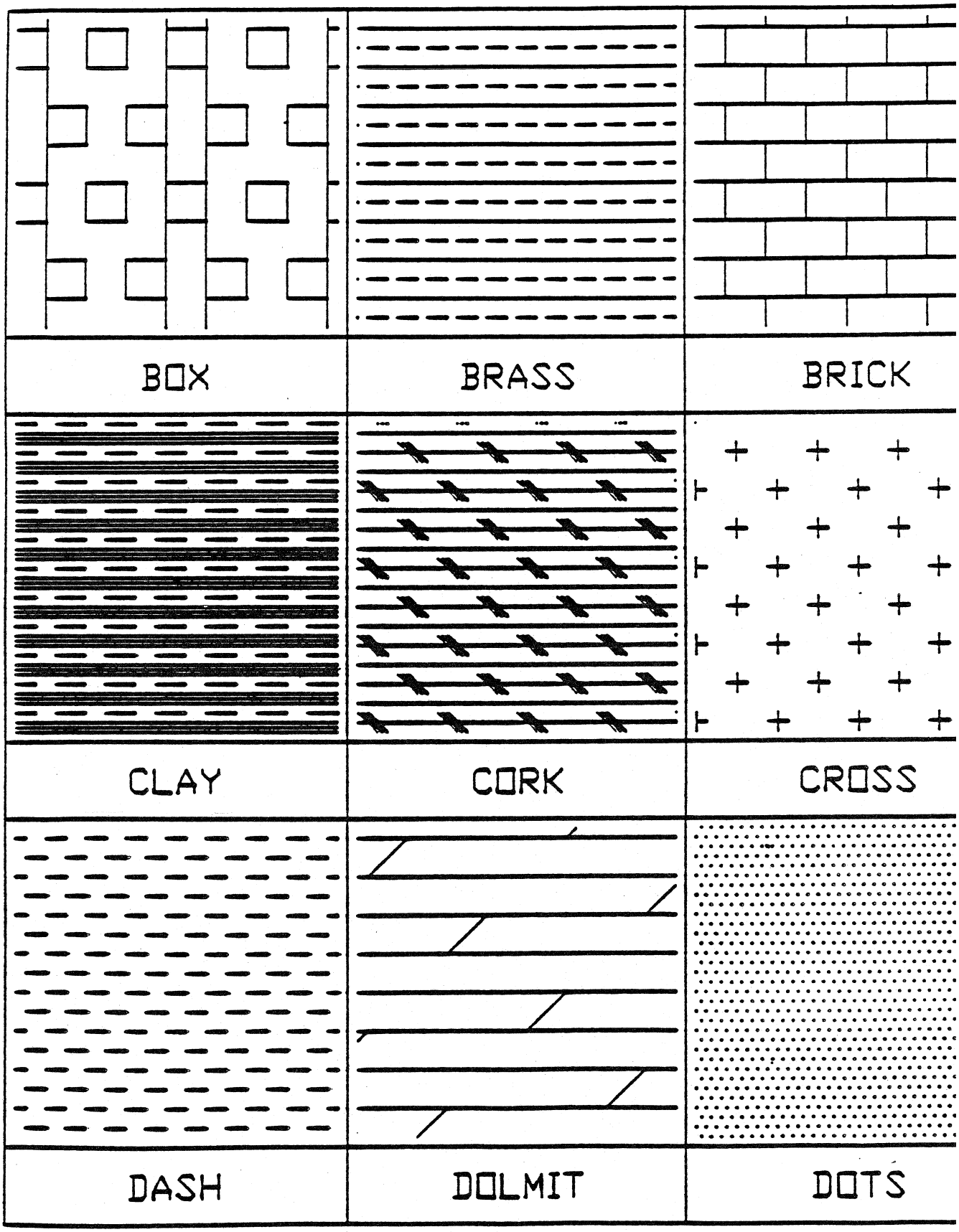
You should select, by any of the standard methods, the objec which define the boundary of the area to be hatched, and thc objects internal to the boundary which you wish hatching to aware of. Note that Window selection is particularly convenie in many hatching applications.

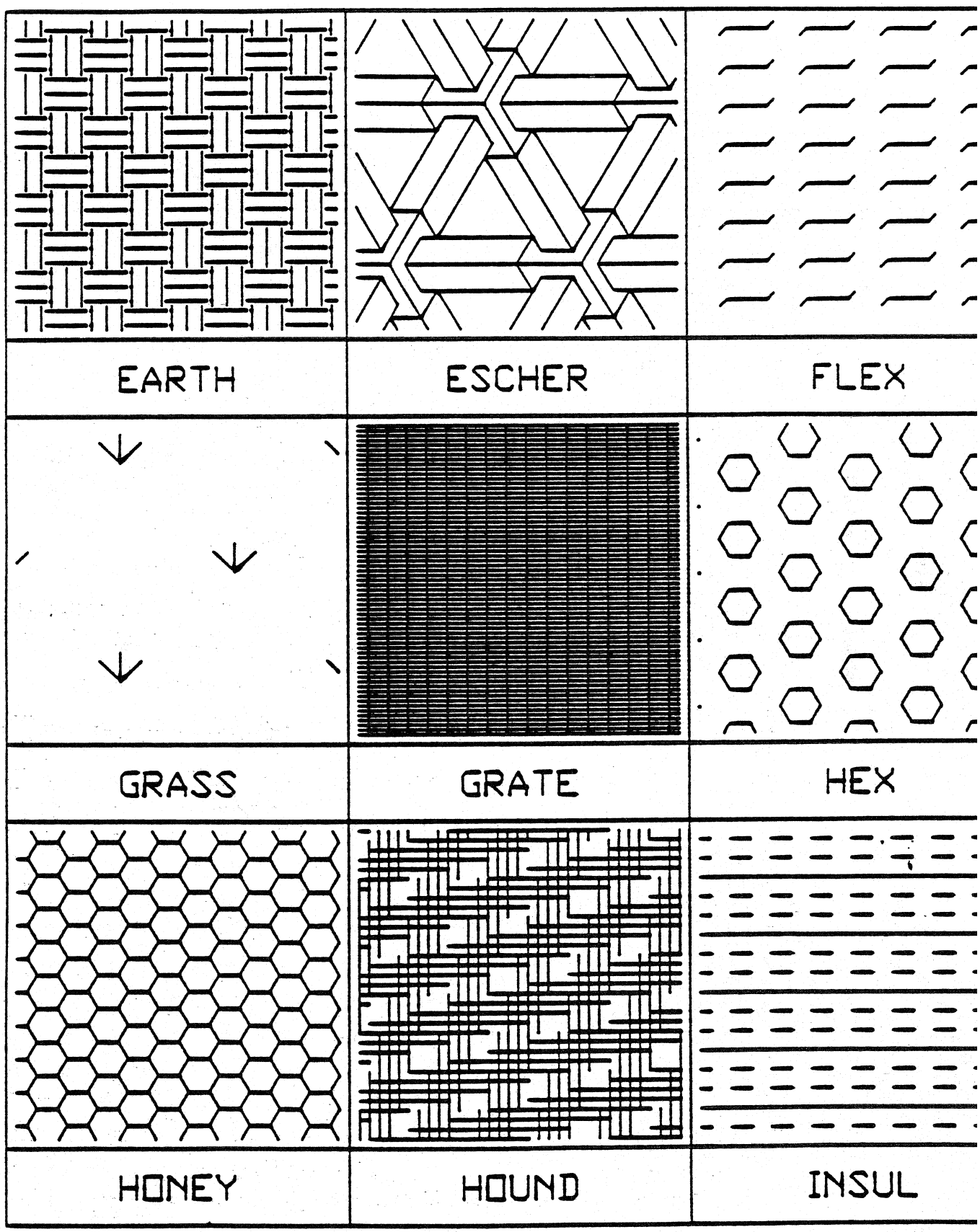
Once the boundary is selected, hatching will commence. Hatchi can take a while, particularly when the pattern, boundary, and/ internal structure are complex; you can watch the hatch lir appear as they are generated, or you can go out to lunch. Y may terminate the hatching process at any time with **CTRL C**. T pattern generated up to that point will be added to the drawir If the hatch lines were being grouped into a block (the defau case), ERASE Last can be used to delete them.

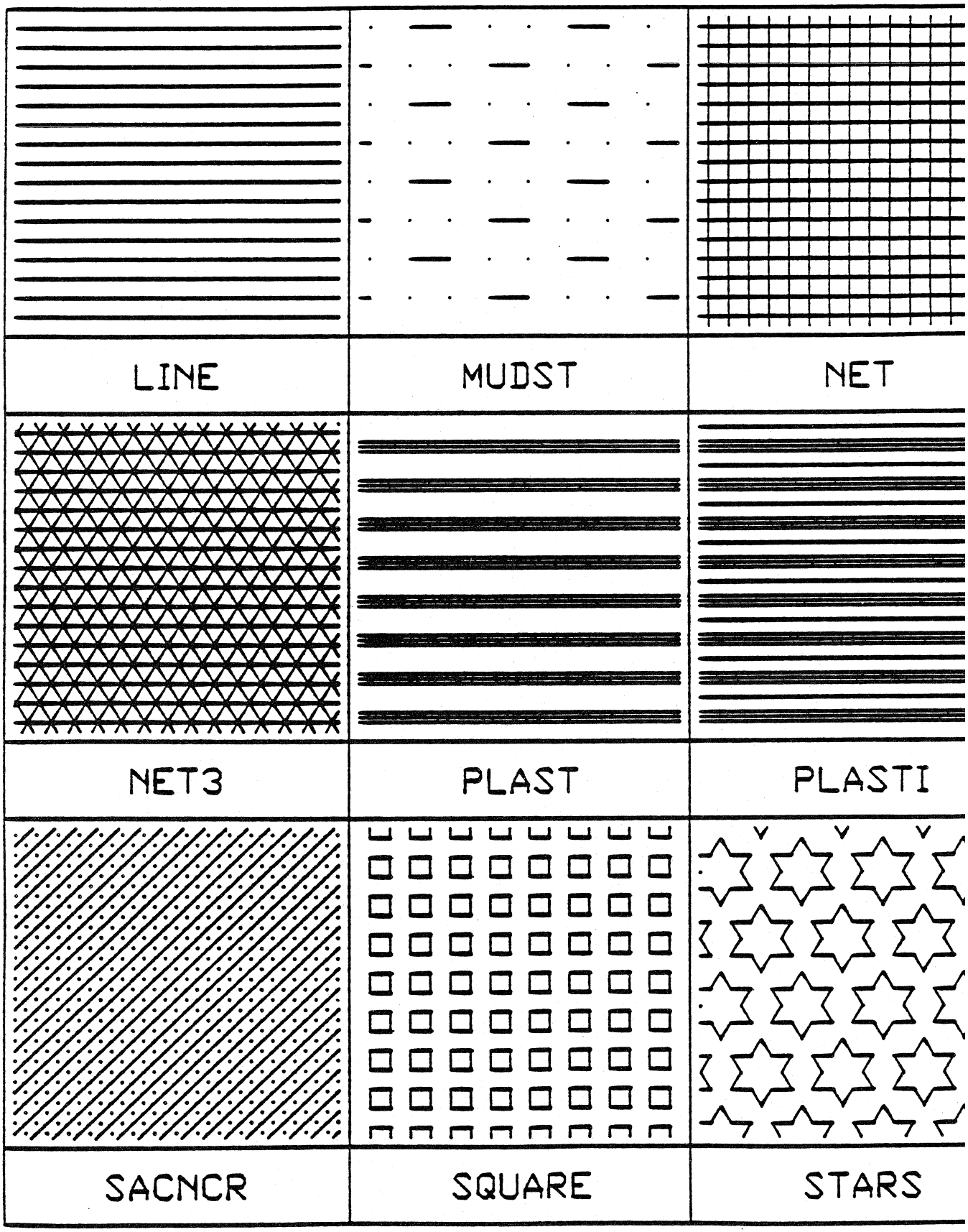
9.7.3 Standard Hatch Patterns

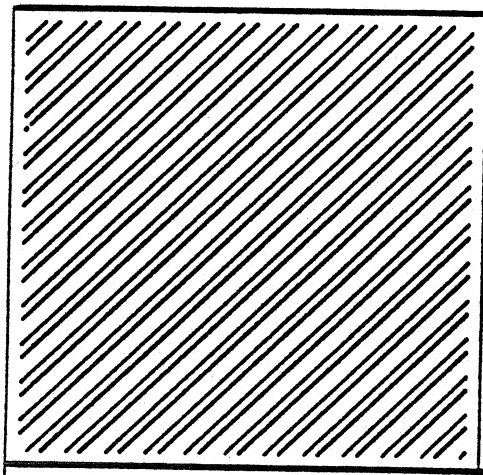
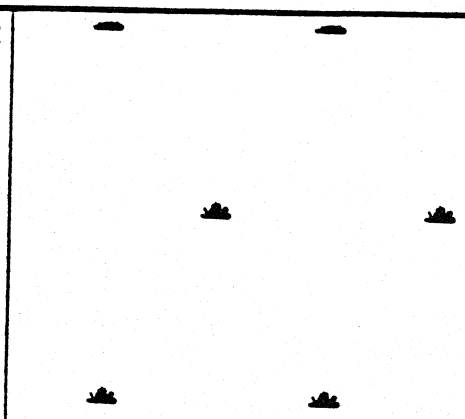
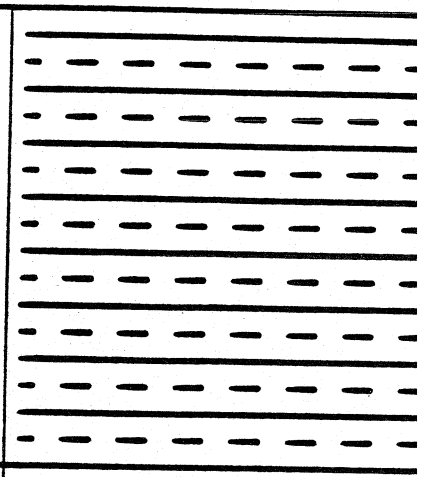
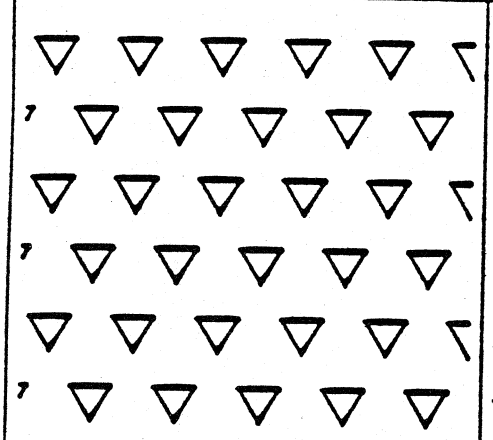
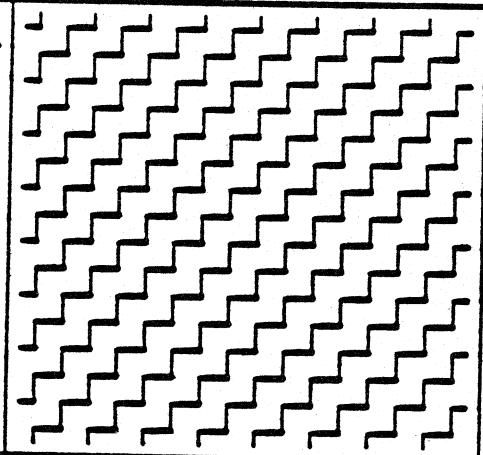
The following pages illustrate the standard hatch patter supplied in file ACAD.PAT.


ANGLE
ANSI31
ANSI32
ANSI33
ANSI34
ANSI35
ANSI36
ANSI37
ANSI38


BOX
BRASS
BRICK
CLAY
CORK
CROSS
DASH
DOLMIT
DOTS


EARTH
ESCHER
FLEX
GRASS
GRATE
HEX
HONEY
HOUND
INSUL


LINE
MUDST
NET
NET3
PLAST
PLASTI
SACNCR
SQUARE
STARS

		
STEEL	SWAMP	TRANS
		
TRIANG	ZIGZAG	

9.7.4 Defining Your Own Patterns

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 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 will be 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_ 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 (e.g., is measured perpendicular to the lines). Delta_x gives the displacement in the direction of the line between members of the family, and is meaningful only for dashed lines.

The process of hatching consists of taking each pattern line in the pattern definition (each line is of infinite length, but may have a dash pattern), and expanding it to its infinite family of parallel lines. All selected entities are then checked for

intersections with any of these lines, which cause the hatch lines to be turned on and off as governed by the chosen hatching style. If the hatch line is dashed, it will be drawn with the dash pattern in those areas in which it is on.

Since all hatch lines are generated from families of lines which have an absolute origin, and are generated by parallel transport from the initial line, 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 so 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 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
```

Note that 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:

```
- - - - - - - - - -
- - - - - - - - - -
- - - - - - - - - -
```

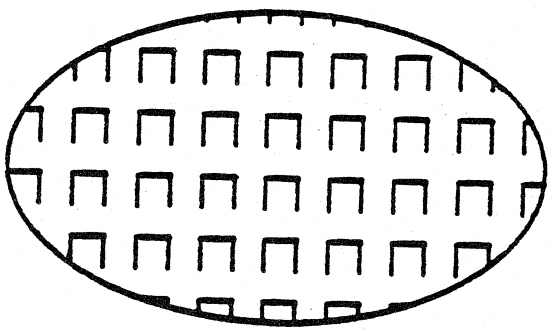
Now let's change the pattern to be:

```
*SKEWED
Ø, Ø,Ø, .5,.5, .5,-.5
```

This is the same, except we've set delta_x to .5. This w
offset each successive member of the family by .5 in
direction of the line (in this case, parallel to the X axis).
since the lines are, of course, infinite, this will have
effect of sliding the dash pattern down the selected amount.
this case, our hatched area would become:

```
 - - - - - - - - - -
- - - - - - - - - - -
 - - - - - - - - - -
```

So far, all the patterns we have examined use origin points
(Ø,Ø), and thus one member of the line family passes through
origin, with its dash pattern starting at that point.
composing more complex patterns, it is often required to st
the dash pattern and offset the line family so that it ali
with another line making up the pattern. Suppose we want to d
a pattern which is a squared off inverted "U". That is, one l
up, one over, and one down: a box with the bottom missing. Le
define a pattern of these that repeats every 1 unit. Each w
be .5 high and wide, as illustrated below:



This pattern would be defined as:

```
*IUS,Inverted U's
9Ø,   Ø,Ø,   Ø,1, .5,-.5
Ø,    Ø,.5   Ø,1, .5,-.5
27Ø,  .5,.5, Ø,1, .5,-.5
```

The first line (up bar) is a simple dashed line with (Ø,
origin. The second line (top bar) should begin at the end of t
up bar, so we have set its origin to (Ø,.5). This will offs
this line by .5 up so it aligns with the end of the up bar. T

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:

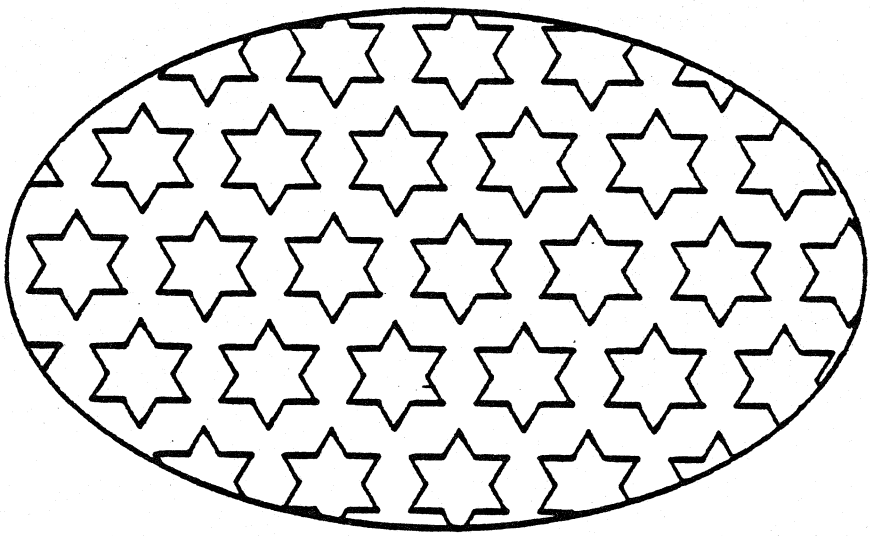
```
90, .5,0, 0,1, .5,-.5
```

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 patterns opposed 180 degrees are not alike. Two solid patterns are, obviously.

Finally, consider the following pattern of six-pointed stars.



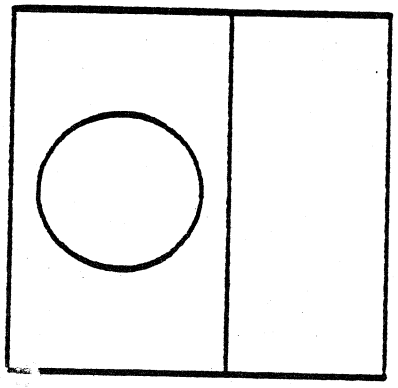
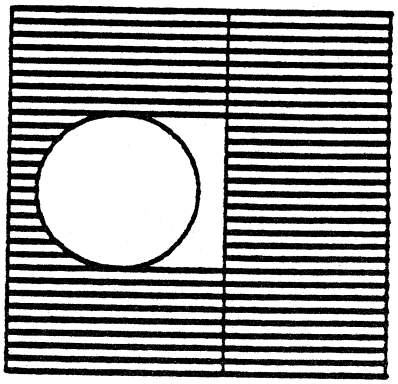
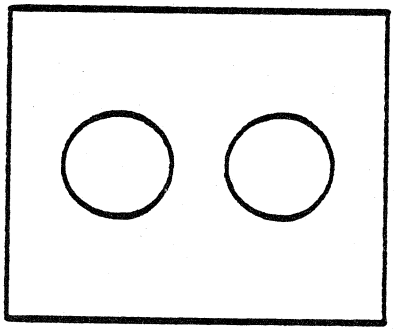
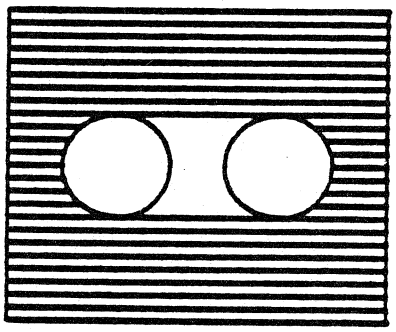
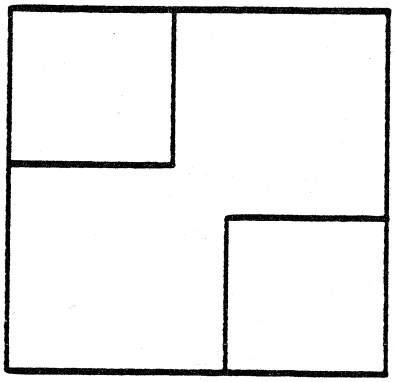
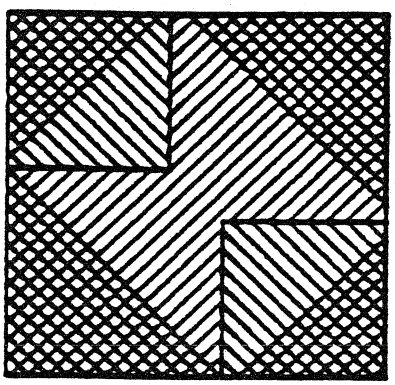
Here is AutoCAD's definition of this pattern. This is a good example to pick apart to hone your skills at pattern definition:

```
*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
```

Hint: .866 is the sine of 60 degrees.

9.7.5 Odd Cases

There are some odd cases that hatching may not handle as desir
Remember that hatching always proceeds inward from the ou
boundary, and consider the following examples. All entities
each figure have been selected for hatching.

Pattern, style	Before hatching	After hatching
Horizontal lines Normal style		
Horizontal lines Outermost style		
45 & 135-degree lines Normal style		

9.8 SKETCH Command - Freehand Drawing

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 may perform limited editing on these lines before recording them in the AutoCAD drawing database. Once recorded, all the normal facilities of AutoCAD may be used on the freehand material--you may move it, delete all or part of it, make it part of a block, and so forth.

Freehand drawings may be made only with a pointing device (digitizing tablet, mouse, pen, etc.). It is not practical to make freehand drawings with the console keys, thus this facility is not implemented.

You should be aware that freehand drawing, especially with very fine accuracy, generates a large number of lines. While 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 will justify generating a new line. This establishes the resolution or accuracy of the sketch in the following way: starting at the first point, movement of less than the record increment will not generate a new line. Once you move a distance equal to the record increment, a line will be generated to that point, then you will have to move another record increment (in any direction) to generate another line.

Assuming your drawing limits are from 0,0 to 10,10, and that t screen shows the entire drawing, a record increment of .1 wou generate a reasonably high resolution sketch (100 lines acro the drawing), while a record increment of 1 would generate terrible resolution sketch. (Only movement of 1/10 the drawi size would record a new line.)

Let's enter .1 and try some sketching. After you type .1 a RETURN, SKETCH will display the following message:

Sketch. Pen eXit Quit Record Erase Connect .

to remind you that you're sketching, and list some subcomman you can use while sketching (which we'll discuss later).

9.8.1 The Sketching Pen

Sketching uses the concept of a "pen", which should not confused with a physical pen you may be using as a pointer. Wh the pen is up, you are not drawing. When the pen is down, y are. If you are using a physical pen as the pointer, whether is on the screen or not has nothing to do with whether t sketching pen is down. Henceforth, "pen" will refer to t sketching pen and "pointer" to the pointing device, whatever may be.

The pen is initially up, and while up the crosshairs on t screen track movements of the pointer as usual in AutoCAD. make a sketch, move to the start point of your sketch line a press the "P" key on the keyboard. If your pointer has a swit (e.g., digitizer cursor or stylus, or mouse button), pressing t switch you use to select points normally in AutoCAD will beha the same as pressing the "P" key. The "P" key lowers the pen up, and raises it if down. Once you lower the pen, a line started at the point you lowered the pen, and as you move t pointer, new line segments are added. The rubber band cursor used to show you lines not yet long enough to add to the sketch line. When you raise the pen by pressing "P" again (or t pointer button), the sketch line will be extended to the poi where the pen was raised (even if closer to the last point th the record increment), and the normal crosshairs will return.

You may lower the pen again and sketch another line, and contin to do so without leaving the SKETCH command. Sketched lines a not immediately added to the drawing; they are kept as tempora lines until you exit the SKETCH command or use one of i facilities to process them in some way. If you have a colo display, the temporary lines you sketch will be displayed green, unless you are drawing in green, in which case red will used. When the lines are added permanently to the drawing, th will be redrawn in the current drawing colour.

9.8.2 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 may do so by using the BLOCK command to make it into a block.

9.8.3 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 which may be used while a sketch is in progress. These commands are all very simple compared to normal AutoCAD commands, and all can, in fact, be invoked by a single key. For convenience, if your pointer has more than one button, these subcommands may also be invoked by pressing the appropriate button on the pointer. Note that the normal invocation of menu items from the pointer buttons does not occur while a SKETCH command is in progress. However, SKETCH subcommands may be entered from either the screen or the tablet menu by the normal pointing and selecting process.

The following is a list of SKETCH subcommands, and the pointer buttons which correspond to them:

Command Character	Pointer Button	Function
P	Point select	Raise/Lower Pen
.	1	Line to point
R	2	Record lines
X, Space, RETURN	3	Record lines, exit
Q, CTRL C	4	Discard lines, exit
E	5	Erase
C	6	Connect

In addition, the special keys which toggle Snap, Ortho, and Grid modes may be used while the SKETCH command is active.

All of the SKETCH subcommands take effect immediately on entry--no RETURN is required after them.

9.8.3.1 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.

9.8.3.2 . (period) - Line to point

The "." command is used to draw a normal line from the endpoi of the last sketched line to the current location of the pointe It is meaningful only when the pen is up. You might use it, i example, while tracing the outline of a country upon encounteri a portion of the border which is a straight line. After addi the straight line, the pen returns to the up position.

9.8.3.3 R - Record

The "R" command records all the temporary lines sketched so f as permanent lines, without changing the up/down state or t position of the sketching pen. Remember that once recorded, t lines cannot be edited with SKETCH subcommands, but only with t regular AutoCAD facilties. The number of lines added to t drawing by this command will be displayed.

9.8.3.4 X - Record and Exit

The "X" command records all the temporary lines entered a returns to the AutoCAD command prompt. The space bar and RETU key have the same effect, and the "X" is provided mainly for mo convenient use in menus. The number of lines added to t drawing by this SKETCH command will be displayed as:

n lines recorded.

9.8.3.5 Q - Quit

The "Q" command discards all temporary lines sketched (since t start of the SKETCH command or the last "R" command) and retur to the AutoCAD command prompt. CTRL C has the same effec serving as the universal command terminator in AutoCAD.

9.8.3.6 E - Erase

The "E" command allows you to selectively erase from any porti of the line you have sketched to the end. Type "E" to activa the command, and you will receive the message:

Erase: Select end of delete.

If the pen was down, it will be raised by this command. Move t crosshairs to the point at which you want to chop the line. you do this, the line will be displayed with the portion from t point closest to the crosshairs to the end blanked out. When y have selected the portion you wish to erase, press the "P" k (or point select button on the pointer), and the indicat portion of the temporary line will be erased. Remember that y cannot erase lines by this method once you've recorded them.

you decide not to erase anything, just press "E" again (or any other command), and the message:

Erase aborted.

will appear, and you will be 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.

9.8.3.7 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 will receive the prompt:

Connect: Move to endpoint of line.

and the crosshairs will 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 will be automatically lowered, the SKETCH prompt will reappear, and you may 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.

9.8.4 Effects of Other Modes

Operation of the SKETCH command is affected in various ways by the current setting of Snap, Ortho, and Tablet modes.

9.8.4.1 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 frc it into the AutoCAD drawing. Coordinates will be preserved a with normal data entry from the digitizer in Tablet mode. T make more of the digitizer available in Tablet mode, the scree menu may not be pointed to from the digitizer while Tablet moc is in effect. This area may be used for your drawing. If yc are using a tablet menu, it will continue to work (althoug reducing the free area for the drawing, of course). During th SKETCH command, CTRL T cannot be used to turn Tablet mode on an 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 no mix input to the SKETCH command from these two sources. Once yo use one, you must continue to use it for the duration of th SKETCH command. You can, of course, end the SKETCH command an issue another using the other device. This restriction applie only when Tablet mode is on. Attempting to mix input will resul in the message:

Screen and tablet mode input may not be mixed.

and further input from the second device used will be ignored fo the rest of the SKETCH command.

While sketching in Tablet mode, the crosshairs on the screen wil appear at the screen coordinates corresponding to the coordinate generated from the tablet. Thus, it is possible for th crosshairs and the lines you are sketching to be totally of screen. If you wish to refer to the screen while sketching, yo should do a ZOOM Window to the desired work area before th SKETCH command.

Also when sketching in Tablet mode, AutoCAD must do far mor computation per point from the pointer than in screen mode, s when using very high resolution (low record increment values) you may have to trace more slowly to preserve accuracy. Se Section 9.8.3 for details on how the SKETCH command can tell yo when you should slow down.

9.8.4.2 Sketching and Snap Mode

When Snap mode is on, all coordinates received from the pointe will be 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 will be the Snap value anyway.

9.8.4.3 Sketching and Ortho Mode

When Ortho mode is on, the SKETCH command will draw only vertical and horizontal lines. If the movement between two adjacent

points from the pointer is diagonal, an orthogonal line will be drawn first in the longer direction between them, then over to the second point. This makes a "staircase" line which follows the pointer. While this mode would seem less than useful for tracing, it is actually 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.

9.8.5 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 which require 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, because when the computer next looks at the pointer, it will have moved farther than 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 will use the record increment as if it were positive, but will test every point received from the pointer against twice the record increment. If the point is more than two record increments away, the console bell will sound as a warning that you should slow down to avoid losing accuracy. Having the bell sound 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 will fit in memory and no delays will 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 while it was happening. To prevent this, the SKETCH command constantly

watches the available memory. When a disk access will l
required within the next hundred points or so, it displays tl
message:

Please raise the pen!

and sounds a continuous bell. You should stop tracing (you don
have to stop instantly, SKETCH gives you reasonable warni
time), and raise the pen with the "P" key or the pointer buttor
AutoCAD will then perform the required disk operations while tl
pen is up and not tracing. The message:

Thank you. Lower the pen and continue.

will then appear and you may lower the pen and continue tracin
Most users of the SKETCH command will never experience this, bu
the protection is there in case your machine or drawing needs i
On machines with just enough memory to barely run AutoCAD, y
may see the message:

Warning -- low memory -- accuracy may be low.

If this message appears, there is so little free memory that tl
SKETCH command cannot protect you against disk accesses, so
you hear the disk click, you'd better stop tracing right awa
This message will appear only if the physical memory available
marginal to run AutoCAD; it is <u>not</u> related to the drawi
complexity or the amount of information sketched.

Disk Space Considerations

Remember again that SKETCHing with a small record increment m
generate a phenomenal number of line entities, and this c
rapidly consume free space on the drawing disk. When tracing
complex drawing, it is wise to leave the SKETCH command from ti
to time and check STATUS to make sure you're not about to r
out.

Chapter 10

PLOTTING

You can plot a drawing in AutoCAD either by selecting the "Plot a drawing" function of the main menu, or by entering the PLOT command from within the Drawing Editor:

```
Command: PLOT
```

The PLOT program handles your request in either case. The only difference is that the menu function will ask you for the name of a drawing to plot, and the PLOT command will plot the drawing you are already working on.

When the plot is finished, AutoCAD will return to where you were when the plot was initiated (either the Main Menu or the "Command:" prompt). You can use CTRL C to abort the plot at any time. Some plotters have large data buffers, however, and will not stop drawing the moment you enter the CTRL C.

The PLOT program is capable of drawing plots of any desired size, to any scale, and with various line types, pen numbers, and speeds (if these features are available on your plotter). It stores these plot specifications in the master AutoCAD configuration file, ACAD.CFG, on the AutoCAD program disk. When entered, the PLOT program will display all the stored plot specifications, as in:

```
Drawing colour assignments:

Colour  Line type  Pen nr  Pen speed
  1         Ø        1       16
  2         Ø        2       16
  3         1        3       16
 ...       ...      ...     ...
 15         Ø        8       16

Sizes are in Inches
Plot origin is at (Ø.ØØ,Ø.ØØ)
Plotting area is 15.ØØ wide by 1Ø.ØØ high (MAX size)
Pen width is Ø.Ø1Ø
```

It will then ask:

```
Do you want to change anything? (Y/N)
```

If you answer "No", the current plot specifications will be used without change. If you answer "Yes", you will be given an opportunity to change them, as described below.

10.1 Changing Plot Specifications

If you answer "Yes" to the "change anything?" question, the PLO program will prompt you for new values for each of the plo specifications you can change. Each prompt message will show th current value inside corner brackets, "< >", and you can kee that value by just pressing RETURN or the space bar. If you wan to change something, type in the new value. You can us **Backspace** and **CTRL X** to correct typing errors just as in th Drawing Editor, or enter **CTRL C** at any time to cancel the entir plot. The specifications you can change are as follows:

Drawing Colour Assignments

The drawing colour assignments allow you to control how th different layer colours will be plotted. If you have single-pen plotter, you can't actually plot using differen colours, but some such plotters can draw using different lin types (dashed, dotted, etc.). Also, some plotters are capable o controlling the speed of pen motion. AutoCAD makes use of thi feature by allowing you to assign a different speed to eac drawing colour. This is useful if, for example, you have one pe that skips if you move it too fast.

In the following discussion of the colour assignments dialogue we will assume that the configured plotter has multiple pens line types, and variable pen speeds.

First, representations of the various line types will b displayed on the screen to remind you of the types available o the plotter. Possible pen speed values will be displayed, also For example:

```
Line types: 0 = continuous    Pen speed codes:
            1 = .........
            2 = . . . . .       In/sec: 1,2, 4, 8,16
            3 = ----------      Cm/sec: 3,5,10,20,40
            4 = - - - - -

Enter line types, pen numbers, pen speed codes
blank=go to next, Cn=go to Colour n,
S=Show current choices, X=Exit
```

Then, for each colour, you will be prompted for a line type, pe number, and pen speed. When entering pen numbers, do not specif a number for a pen that isn't present in its stall. Only colour 1 to 15 may be assigned pen numbers and line types; colour greater than 15 will plot the same as colour 15.

As indicated in the display, a few special responses are honored

Blank Retains the current value shown in corner brackets, "< >", and proceeds to the next prompt. RETURN does the same thing. When the last value for a particular colour has been reached, this proceeds to the next colour. If this was the last colour, AutoCAD will simply cycle back to the first colour.

Cn Retains the current value, skips any remaining prompts for this colour, and proceeds directly to colour number "n". If a "C" is entered without a number, this will proceed to the next colour. This allows you to change colour assignments selectively.

S Displays the current settings for all colours, then returns to the current prompt.

X Exits from the drawing colour assignments dialogue. Enter an "X" when you are satisfied with the line types, pen numbers, and pen speeds you have selected.

Here's a sample colour assignments dialogue:

```
Colour number 1  Line type <0>: blank
Colour number 1  Pen number <1>: 6
Colour number 1  Pen speed <16>: C3
Colour number 3  Line type <1>: 0
Colour number 3  Pen number <3>: X
```

Size Units

When you have finished the colour assignments dialogue (by entering "X"), the following prompt will appear:

```
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. Again, the current value is shown in corner brackets; you may retain this value by simply pressing RETURN or the space bar.

Plot Origin

The next prompt is:

```
Plot origin in (units) <current X,Y>:
```

where "(units)" will be "Inches" or "Millimeters," depending on which plot size units you specified. Normally, the plot will begin in the lower left corner of the paper (the plotter's "home" position, point 0,0). You may elect to place the plot origin at another location on the paper. To do this, simply enter the plotter coordinates of the desired origin point, using the "size

units" (inches or millimeters) that you selected above. Fc example, if you have selected inches as your plot size unit: entering "2,3" would set the plot origin to the point 2 inches t the right of and 3 inches above the home position.

Plot Size

The prompt for plot size is:

    Plot size (list) or width in (units) <current>:

Here again, "(units)" will be either "Inches" or "Millimeters" "List" is a list of size mnemonics. Depending on the selecte plot size units (Inches or Millimeters), this list will includ the ANSI or DIN standard sizes accomodated by the configure plotter model, minus a standard border width. The dimensions use are:

ANSI Sizes (Inches)		DIN Sizes (Millimeters)	
A	10.5 x 8.0	A4	285 x 198
B	16.0 x 10.0	A3	396 x 273
C	21.0 x 16.0	A2	570 x 396
D	33.0 x 21.0	A1	817 x 570
E	43.0 x 33.0	A0	1165 x 817

The size mnemonic list will always include two special items "MAX" and "?". If you enter "?", the dimensions of the standar sizes will be displayed, plus the maximum possible size for you plotter model; this size is selected by entering "MAX".

You can respond to the "Plot size" prompt by pressing space o RETURN to retain the current value, or you may enter one of th listed mnemonics, or you may enter a width in the specified plo size units. If you enter a width, you will be asked to supply height as well. The size determines the maximum distance th plotter will move horizontally and vertically away from the plo origin. You shouldn't specify a width or height larger than you plotter can accomodate.

Pen Width

The last plot specification you will be asked to supply is th width of the pen tip to be used. This governs how much wor AutoCAD must do to "fill" a Solid or Trace.

Saving Plot Specifications

If you have changed any plot specifications, AutoCAD will ask:

    Save changes for future plots? <N>:

Reply "Y" if you want the changes to remain on file, or "N" (or just RETURN) if the changes were for this plot only.

10.2 Readying the Plotter

Once you have set the plot specifications (or adopted the previous settings), you will be asked to ready the plotter. A prompt such as:

```
Position paper in plotter.
Press RETURN when ready.
```

will appear. Follow its directions, and the plotter will be initialized.

10.3 Setting the Scale

The PLOT program doesn't save the drawing scale, so it must be entered every time you make a plot. When the prompt:

```
Scale (N, 1:N, or RETURN):
```

appears, you may enter a number, "1:" or "1/" followed by a number, or just RETURN. RETURN will cause the portion of your drawing that was last displayed to be plotted. Since the plotting area isn't generally the same shape as the display screen, the plot won't extend out to the full limits of the plotting area in both the vertical and horizontal directions. A message will be printed telling how far it actually does extend (effective plotting area).

If you reply with a number, that is the number of inches (or millimeters) one drawing unit will occupy. Thus, if your size units are millimeters and you set the scale to "5", a line that is 2 drawing units long will span 10 millimeters when plotted.

If you enter "1:" or "1/" followed by a number, that is the number of drawing units that will be drawn per inch (or millimeter). For example, if you are working on a house plan and your drawing units represent feet, you can get a plot at a scale of 1/4 inch to the foot by entering "1:4" or "1/4".

10.4 Plot Alignment

Normally, the plot origin will contain the point that was at the lower left corner of the screen the last time you saved the drawing (via the END command). If you just enter a RETURN for the scale, the plot will contain exactly what was on the screen. If a numeric scale is used, the plot may include portions of the drawing that weren't on the screen, or parts may be truncated.

AutoCAD also may be configured to allow a point other than t drawing's lower left corner to be placed at the plot origin. you have configured it to permit this, AutoCAD will prompt:

```
Drawing coordinates of point to
place at plot origin <lower left corner of last view>:
```

Respond with the point in your drawing that you want at the pl origin, or RETURN to use the default.

10.5 OPLOT - Quick Plots

On some computer systems, AutoCAD can "dump" the contents of the graphics screen to a dot matrix printer if such a printer is configured as the operating system's list device. This is often much quicker than using the plotter, although it lacks the accuracy of the plotter and will not make a plot to scale.

Consult your AutoCAD-86 Installation Guide / User Guide Supplement to determine whether the quick plot capability is provided for your computer and printer. If it is, you can request a quick plot from within the Drawing Editor by entering:

```
Command: qplot
```

You will be asked:

```
Horizontal or vertical? (H/V):
```

to which you should respond "H" or "V". When the plot is done, control will be returned to the Drawing Editor. You may cancel the plot at any time by typing **CTRL C**. (If the printer has a large data buffer, it may not stop plotting right away.)

Appendix A

DEFINING SHAPES AND FONTS

This appendix explains how to define Shape files for drawing parts and text fonts. See Section 7.3 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.

A.1 Shape Definitions

Shape definitions are loaded from a disk file with the LOAD command. This file has a file type of ".SHP" and contains descriptions of one or more shapes. To create or modify the file, use a text editor or word processor in "programmer" mode. This section explains what goes into the file.

Every shape must have a unique number between 1 and 255. The values 1-127 are used by the text fonts, so we suggest that your non-text shape numbers be limited to 128-255. (A text font file is simply a file of shape definitions for each character, where the shape number is the ASCII value of the character).

Each shape definition has a header line of the form:

*<shapenumber>,<defbytes>,<shapename>

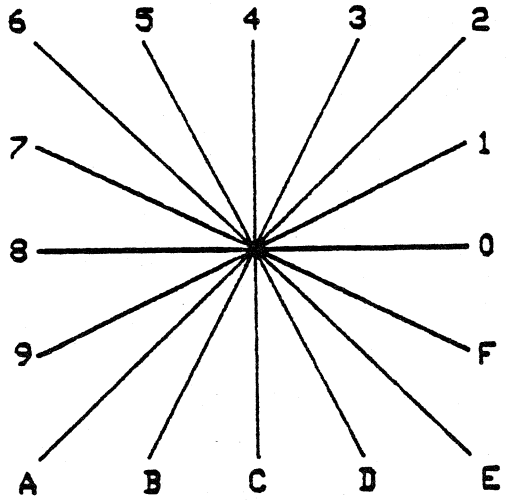
followed by one or more lines containing specification bytes separated by commas and terminated by a 0. All the numbers in the shape definition file are scanned with the convention that a leading zero signifies hexadecimal. Hence, 10 is decimal 10, but 010 is hexadecimal with decimal value 16.

The shape name is ignored if it contains 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 directi encoded into one byte. The high-order hex nybble (4 bits) is t number of vector lengths (relative to the specified "height" the shape) and the low-order nybble is the vector direction cod (See diagram, below.)

Vector Length and Direction Encoding



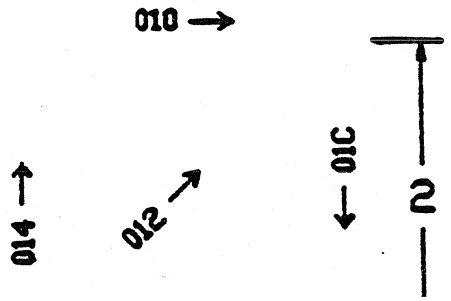
All the vectors in this figure were drawn with the same lenc specification. Diagonal vectors are "stretched" to match the or Y displacement of the closest orthogonal vector. As a worki example, let's construct a shape named DBOX with an arbitrari assigned shape number of 230.

```
*230,6,DBOX
014,010,01C,018,012,0
```

The above sequence of bytes defines a box one unit high by (unit wide, with a diagonal line running from the lower le corner to the upper right corner. If we LOADed the Shape fi containing this definition and then entered:

```
Command: SHAPE  Name: DBOX
  Starting point: 1,1
  Height: 2
  Angle: 0
```

the following 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 on, vectors will cause lines to be drawn. When 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 could 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 will result 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. <u>You must pop everything you push!</u> This stack is only 4 locations deep. An example would be to allow subscripts and superscripts in text.

Code 7: Subshape

Code 7 is a subshape reference. The byte following the code 7 a shape number from 1 to 255. That shape will be drawn at t time. (Note that DRAW mode is not reset for the new shape). W the subshape is completed, the rest of the current shape will processed. By combining codes 3 and 4 with 7, subshapes t draw high-resolution curve segments can be defined and used other shapes.

Codes 8 & 9: X-Y Displacements

The normal vector bytes are only capable of drawing in the predefined directions, and the longest length is 15. Th restrictions help make Shape definitions efficient, but sometimes very limiting. Therefore, codes 8 and 9 are availa to permit "nonstandard" vectors to be drawn using X-Y displa ments. 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. leading "+" is optional, and parentheses may be used to impr readability. For example:

8,(-1Ø,3)

would result in a vector which draws (or moves) 1Ø units to left and three units up. Following the two displacement byt the shape returns to normal vector mode.

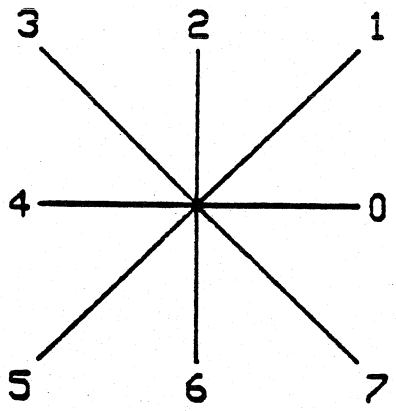
Code 9 may be used if a sequence of "nonstandard" vectors is be drawn. This code may be followed by any number of displacement pairs, and is terminated by a (Ø,Ø) pair. instance:

9,(3,1),(3,2),(2,-3),(Ø,Ø)

would draw three nonstandard vectors and then return to nor vector mode. Note that you must terminate the sequence of displacement pairs with a (Ø,Ø) pair before any normal vectors special codes will again be recognized.

Code ØØA: Octant arc

Special code ØØA (or 1Ø) uses the next two bytes to define arc. This type of arc is called an "octant" arc because it sp one or more 45-degree octants, starting and ending on an oct boundary. The octants are numbered counterclockwise from th o'clock position, as shown below:



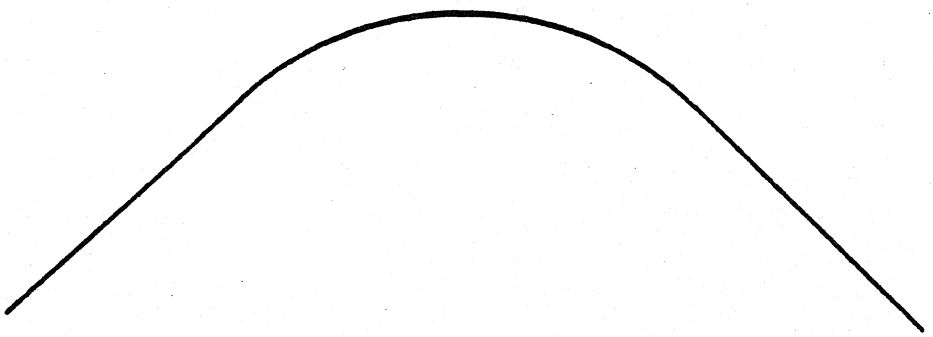
he arc specification is:

```
10,<radius>,<(-)0SC>
```

he radius may be any value from 1 to 255. The second byte ndicates the direction of the arc (counterclockwise if positive, lockwise if negative), its starting octant ("S", a value from 0 o 7), and the number of octants it spans ("C", a value from 0 to with zero meaning eight octants or a full circle). Parentheses ay be used to improve readability. For example, consider the ollowing fragment of a shape definition:

```
...012,10,(1,-032),01E,...
```

his would draw a 1-unit vector up and to the right, a clockwise rc from octant 3 with a radius of 1 unit for 2 octants, and then 1-unit vector down and to the right, as illustrated below.



ode 00B: Fractional arc

pecial code 00B (11) is used to draw an arc which does not ecessarily start and end on an octant boundary. The definition ses five bytes:

```
11,<start offset>,<end offset>,<high radius>,
  <low radius>,<(-)0SC>
```

The start and end offsets represent how far from an octa boundary the arc begins or ends. The <high radius> is the mo significant 8 bits of the radius; it will be zero unless t radius is greater than 255 units. Aside from that, the radi and control byte are the same as for the octant arc specificati (code ØØA, above). The octant count ("C") is the number octants containing any portion of the arc. Again, Ø means 8.

The start offset is found by calculating the difference degrees between the starting octant's boundary (a multiple of degrees) and the start of the arc. Then multiply this differer by 256, and divide by 45. If the arc starts on an octa boundary, its start offset will be zero.

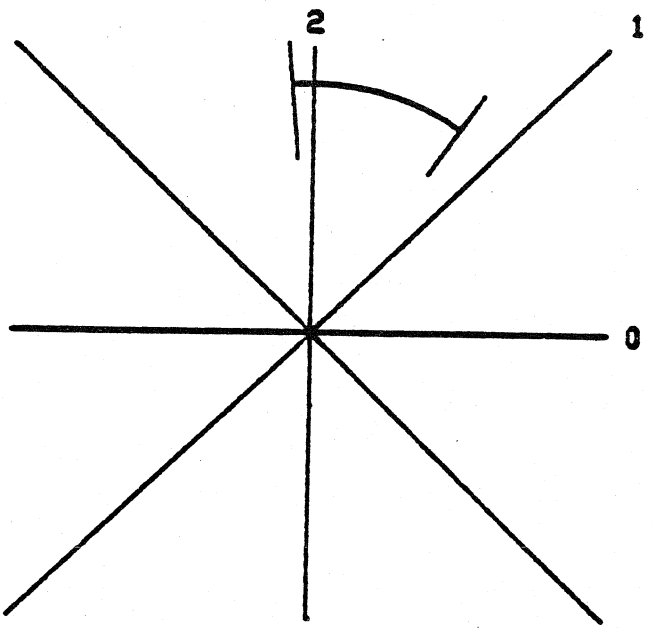
The end offset is calculated in a fashion similar to that of t start offset, but using the number of degrees from the la octant boundary crossed to the end of the arc. If the arc er on an octant boundary, its end offset will be zero.

For example, a fractional arc from 55 degrees to 95 degrees wi a 3-unit radius would be coded:

```
11,(56,28,3,Ø12)
```

Explanation:

```
starting octant =  1 = 45 degrees
ending octant   =  2 = 9Ø degrees

start offset    = 56 = ((55 - 45) * 256 / 45)
end offset      = 28 = ((95 - 9Ø) * 256 / 45)
```



A.2 Text Fonts

Text fonts are files of shape definitions. The shape number corresponds to the ASCII value of each character. The shape name should be specified in lower-case letters to save memory. (Lower-case names are not saved, and text characters are not referenced by name.)

All text fonts require one special shape definition, for shape number 0.

```
*0,4,<Font name>
<height>,<depth>,<flag bits>,0
```

This defines the name of the text font; although this is not yet used by AutoCAD, it is useful for documentation. The <height> specified should be the number of vector lengths that the upper case letters extend above the base line, and <depth> should indicate how far the lower case letters descend below the base line. These values will be used to scale the height specified by the user during the TEXT command, like an automatic "3,scale" special code. The <flag bits> byte is currently unused, and must be zero.

All text fonts also require definitions for three special control characters:

SOT - 1 The Start of Text (SOT) character must push the current location to the stack.

CR - 13 The Carriage Return (CR) must restore ("pop") the starting position from the stack.

LF - 10 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. The spacing of the lines may be modified by adjusting the amount of downward movement.

Samples of these are in the file TXT.SHP, which contains the standard text font provided with AutoCAD. Using a text editor, you may modify it or create your own font file.

Appendix B

DRAWING INTERCHANGE FILES

AutoCAD can be used by itself as a complete drawing editor. In some applications other programs must examine drawings created by AutoCAD, or must 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 could 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 application would be to use AutoCAD to describe structures which are then submitted to a large computer for finite element structural analysis. You could 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 very hard for user programs to read directly. In addition, every implementation of AutoCAD uses a different internal format 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 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.

B.1 Drawing Interchange Format

A Drawing Interchange File is simply an ASCII text file with a file type of ".DXF" . Each entity in the file occupies at least one line, with additional lines used for most entities.

The format of lines in the file is outlined on the following pages. The numbers used for the values in the items are floating-point for all coordinates and values, and may contain an exponent specification, for example, "-3.233E-5". Layer numbers, shape numbers, and repeat counts for ENDREP items are integers, which may not contain decimal points or exponents. Text items, such as the contents of a text string, a block name, or a file name for a LOAD item, are on a line by themselves, terminated by a carriage return (end of line). All lines start in column 1.

Drawing Entities

LINE

```
LINE,<layer>
<x1>,<y1>,<x2>,<y2>
```

where <x1>,<y1> are the coordinates of the "from point" a
<x2>,<y2> are the coordinates of the "to point". Note that lin
extended on data entry will actually be stored as separate line

POINT

```
POINT,<layer>
<x>,<y>
```

<x>,<y> are the drawing coordinates of the point.

CIRCLE

```
CIRCLE,<layer>
<centrex>,<centrey>,<radius>
```

The circle is specified by its centre point and radius. Note th
the circle is stored in the database in this form even if it w
entered using the "three point" method.

ARC

```
ARC,<layer>
<x>,<y>,<radius>,<start angle>,<end angle>
```

An arc is represented in the database as a segment of a circ
rather than by the three points which originally specified i
The centre of the circle is given by <x> and <y>, and its radi
by <radius>. The part of the circle drawn is the part betwe
<start angle> and <end angle>, which are in degrees.

TEXT

```
TEXT,<layer>
<x>,<y>,<size>,<angle>
<text>
```

The text start location is given by <x>,<y>, the size (in drawi
units) by <size>, and the angle in degrees by <angle>. T
following line is the actual text, terminated by end of line.

TRACE

```
TRACE,<layer>
<x1>,<y1>,<x2>,<y2>
<x3>,<y3>,<x4>,<y4>
```

A trace is stored in the database as a quadrilateral which defines its borders. (The width, start, and end points used in data entry are used to calculate these numbers.) This representation is required to make traces which abut connect without overlap or gaps. The original parameters may be calculated from the four enclosing points. If a trace has been continued by entering successive "To point"s in data entry, it will be stored as separate entities for each segment.

SOLID

```
SOLID,<layer>
<x1>,<y1>,<x2>,<y2>
<x3>,<y3>,<x4>,<y4>
```

A solid is simply stored as the four points that bound each part of it. If the solid has only three points, then the <x3>,<y3> and <x4>,<y4> coordinates will be the same. If a complex solid is entered with the continuation feature of the SOLID command, it will be stored in the database as separate quadrilateral or triangular "pieces".

SHAPE

```
SHAPE,<layer>
<x>,<y>,<size>,<angle>,<shapenumber>
```

<x> and <y> give the coordinates of the origin point for the shape (normally its lower left corner). <size> is the size entered when the shape was inserted, and <angle> is the orientation, in degrees. <shapenumber> is the code for the shape (the shape name is used only to determine this number on input--it could be found by searching the shape specifications file).

REPEAT start

```
REPEAT,<layer>
```

This item is just one line (since the information is stored at the REPEAT end). The <layer> is irrelevant for this item, but is included to make scanning programs easier to write.

REPEAT end

```
ENDREP,<layer>
<columns>,<rows>,<column offset>,<row offset>
```

<Columns> and <rows> are integers which specify the number of columns and rows to be generated. <column offset> and <row offset> specify the distance between columns (in X) and rows (in Y), respectively. The <layer> is included on the first line, but is irrelevant.

LOAD request

```
LOAD,<layer>
<file name>
```

The LOAD item records a request for a shape definition file to loaded. The <file name> specifies the name of the sha definition file. The ".SHP" file type is not given as part the file name. Since AutoCAD loads the standard text sha definitions when it first encounters a text entity, an explic load for this file will not be in the database unless the us uses the LOAD command to load the "TXT" shapes manually. T <layer> is present on the first line of this item, but irrelevant in this entity.

BLOCK definition

```
BLOCK,<layer>
<basex>,<basey>
<block name>
```

A block definition entity is inserted in the drawing databa before the first entity transcribed from an INSERTed file. T insertion base (set when the file being INSERTed was edited the BASE command), is given by <basex>,<basey>. This specifi the point to which coordinates within the block are relativ Coordinates from the file are not changed when the file inserted; all transformations are done when the block is drawn the screen. The .<block name> is simply the name by which t file is referenced. It is an ASCII item, terminated by end line. The <layer> item is included on this entity, but irrelevant.

END BLOCK definition

```
ENDBLK,<layer>
```

The end block item delimits the end of an INSERT. The <layer> present but irrelevant. Remember that since INSERTed files m themselves contain other INSERTs, BLOCK/ENDBLK pairs may nested to any depth.

BLOCK INSERT

```
INSERT,<layer>
<x>,<y>,<xscale>,<yscale>,<angle>
<block name>
```

The block insert entity causes a defined block to be drawn. T insertion point, over which the insertion base of the block overlaid, is given by <x>,<y>. The scale for the block is giv by <xscale> and <yscale>. Since the X and Y scales are separat the aspect ratio of the part may be changed as well as its tot size. If the X and Y scales are both 1, the part will be insert

Example Header Information

```
EXTENTS,1
0.000000,1.000000,0.000000,1.000000
LIMITS,1
0.000000,1.000000,0.000000,1.000000
BASE,1
0.000000,0.000000
DWGVIEW,1
0.753802,0.500000,1.000000
DIMARROW,1
0.100000
MODERES,1
0,1.000000
MODEGRID,1
0,1.000000
MODEORTHO,1
0
MODEFILL,1
1
TXTSIZE,1
0.200000
TRACEWID,1
0.050000
LAYER,1
127
LAYERC,1
0,15,-4,255,255,255,255,255,255,255,255,255,255,255,255,255
255,255,255,3,255,255,255,255,255,255,15,255,255,255,255,255
255,255,255,255,255,255,255,1,1,255,255,255,255,255,255,255
255,255,255,255,255,255,255,255,255,255,255,255,7,255,255,255
255,255,255,4,255,255,255,255,255,255,255,255,255,255,255,255
-2,-4,255,255,255,255,255,255,255,255,255,255,255,255,255,255
255,255,255,255,6,255,255,255,255,255,255,255,255,255,255,255
255,255,255,7,7,255,255,255,255,255,255,255,255,255,255,15
```

B.2 Making a DXF File

A drawing interchange file is generated from an existing drawi file simply by selecting the "Make drawing interchange fil function from the main AutoCAD menu. This function will ask f a drawing name (you may use the "current name" by default), a generate a drawing interchange file in the format described abo with the same name, but a type of ".DXF" instead of the ".DW for the drawing. If an older drawing interchange file with t same name exists, it will be deleted.

B.3 Loading a DXF File

A drawing interchange file can be converted into an AutoC drawing by selecting the "Load drawing interchange file" functi from the main menu. This function asks for a file name a converts the ".DXF" drawing interchange file into a ".DWG" fi with the same name. If a ".DWG" file with that name alrea exists, AutoCAD will append the interchange file to the existi drawing; and the entities from the interchange file will added to the existing drawing. If you wish to replace t existing drawing with the interchange file, you must first dele the existing drawing file.

at the same size it was originally drawn. The <angle> specifies the angle, in degrees, the part should be rotated around its insertion point, after scaling but before drawing. If <angle> is Ø, no rotation will be done and the part will retain the orientation with which it was drawn. <block name> specifies the name of the INSERT block, and corresponds to the <block name> in a "BLOCK" item earlier in the drawing file.

Drawing Header Information

AutoCAD's drawing database includes a header record defining the current state of various drawing modes, extents, layers, and so forth. This information is also included in the Drawing Interchange File, using the following data formats (all values are floating point unless otherwise indicated):

```
EXTENTS,1
<low x>,<high x>,<low y>,<high y>

LIMITS,1
<low x>,<high x>,<low y>,<high y>
```

The drawing extents and limits are specified in terms of drawing units.

```
BASE,1
<base x>,<base y>
```

This item defines the drawing coordinates of the insertion base point.

```
DWGVIEW,1
<centre x>,<centre y>,<height>
```

This item defines the current view of the drawing, using the drawing coordinates of the centre of the view, and its height in drawing units.

```
MODERES,1
<Snap mode flag>,<snap resolution>

MODEGRID,1
<Grid mode flag>,<grid interval>

MODEORTHO,1
<Ortho mode flag>

MODEFILL,1
<Fill mode flag>
```

These items define the current setting of the SNAP, GRID, ORTHO, and FILL flags for the drawing (integer Ø if off, 1 if on). The snap resolution and grid interval are given in terms of drawing units.

```
TXTSIZE,1
<default text height>

TRACEWID,1
<default trace width>

DIMARROW,1
<arrow size>
```

These items define the default text height, trace width, a dimension arrow size for the drawing, in terms of drawing unit The DIMARROW item will be present only if the Dimensioni feature has been used in this drawing.

```
LAYER,1
<current insertion layer>
```

This item defines the current insertion layer number (integer).

```
LAYERC,1
<Layer-colour table: 8 lines of 16 integers each>
```

The current colour number for each layer (Ø through 127) contained in this item, 16 integers per line. A negative val indicates a layer which is currently turned off, and a value 255 indicates a layer which has never been turned on. The ent for layer Ø is meaningless.

Appendix C

CONSTRUCTING MENU FILES

A menu file is simply a text file with the type ".MNU". 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 commands, indentation, etc.

A menu may contain many items. Each item should be on a separate text line, and each may be up to 80 characters long.

Simple Menus

The simplest menu, suitable for use with the digitizing tablet, is just a series of commands or other pieces of text:

```
line
ZOOM A
ZOOM W
GRID
ON
GRID ON
GRID .1
SNAP 0.001
```

When you select any of these from the menu, AutoCAD treats it as if it had been typed from the keyboard. For instance, selecting GRID, then ON, is the same as typing GRID ON or selecting the line that says GRID ON.

Command Names

Only the first eight characters of a menu line appear on the screen menu; therefore, the "SNAP 0.001" command above would show as "SNAP 0.0". To give a short name to a line, you can enclose the name in square brackets [] at the start of the line. For instance,

```
[FINE]snap 0.001
[COARSE]snap 0.1
[SAVE]end
[GIVE UP]quit
[PC]menu pc
```

SAVE and GIVE UP are simply alternate names for the two termination commands. GIVE UP will still ask whether you really want to discard all changes to the drawing; you could circumvent this safety measure by saying "quit y" in the menu. (We do not

recommend this sort of thing, but as OSHA knows too well, peop will always shortcircuit the devices one installs to keep th from cutting off their fingers.)

PC could be used to load a special menu used for printed circu work. Thus a general menu could lead to a series of specializ menus.

Commands which Require Input

Sometimes it's useful to take input from the keyboard digitizer in the middle of a menu command. To do this, put backslash character (\) at the point where you want input. F instance,

```
[CIRCLE1]circle \1
[ERASE 1]erase \;
```

CIRCLE1 will ask the user for the centre point, then read radius of 1 from the menu. Note that there is no space befo the 1. ERASE 1 will let the operator select one object and wi immediately erase that object; the normal ERASE command requir you to enter a blank or RETURN after the last object digitized.

Menu Miscellany

If you have examined the menu files on the release disk, you wi have noticed some odd things that haven't been explained ye These unclassifiable features have been lumped together in th section.

The screen menu may be limited by the capacity of the graphi monitor -- often it is impossible to display all the menu ite at once. In this case, AutoCAD will display as many of the ite as will fit, followed by a "NEXT" item. Selection of the "NEX item will cause AutoCAD to display another group of menu items the screen. (Note, by the way, that the NEXT line is not in t menu file; AutoCAD automatically places this item at the end the screen menu.) In many cases there will be a natural bre between groups of menu items that should be displayed at the sa time, but the break is not likely to fall on just the right lin You can force a shorter display by using an asterisk, "*", at t start of the first line of a new group. For example, if a me contains the lines:

```
[ANDGATE]insert andgate \1 1
*[FREEHAND]snap off
```

ANDGATE will be the last thing on a screen menu (apart from t NEXT line), and FREEHAND will be the first thing on the ne section of menu.

When a menu item is selected, AutoCAD automatically puts a blank after it before processing it: if the line in the menu file is

```
LINE
```

AutoCAD will see it as if the user typed "L I N E blank". There are times when this is wrong; for instance, the text string supplied to a TEXT or DIM command is terminated by a RETURN, not a space. Also, it is sometimes necessary to supply more than one space (or RETURN) to complete a command, but many text editors strip trailing blanks from a line before filing it. There are two special conventions that get around these problems.

- If a menu line contains a semicolon (;), AutoCAD will substitute a RETURN for it.

- If a line ends with a control character (value less than 32 in ASCII), a backslash (\), or a semicolon, AutoCAD will not add a blank after it.

For instance, look again at the "ERASE 1" menu item on the preceding page. It uses a semicolon to force a RETURN after the user input. Further examples are:

```
HELP
[HELP]HELP ;
[ADDRESS]TEXT \.4 Ø DRAFT Inc;;53 1st St.;;City, State;
```

The first line acts as "HELP <space>" and will result in a prompt for a command name. The second will act as "HELP <space> <RETURN>", and will display 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 line is displayed on the screen, and the result would look funny on the screen.

The third line above would prompt the user for a starting point, and then draw the address on three lines. Again, the semicolons will be interpreted by AutoCAD as RETURNs. 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. As with command scripts, you must be familiar with the prompts issued by these commands, in order to supply the proper data at the proper times.

Lastly, consider the menu line:

```
[*Cancel*]^C
```

Here, the ^C represents the ASCII control character ETX, or CTRL C. Selecting this line will abort an operation, just like entering CTRL C from the keyboard. If your text editor is able to handle control characters as text, it is useful to have this

in menus. A blank after the ^C would cause trouble, and AutoC does not insert one. (NOTE: The notation "^C", while used he to represent CTRL C, is not interpreted as such in the actu menu file. There, the actual ASCII control character must present.)

Appendix D

CONFIGURING AutoCAD

Before AutoCAD can be used, it must be tailored to the graphics devices present on your computer system. Displays, digitizers, and plotters made by different manufacturers (and sometimes even by the same manufacturer) must often be handled differently; for each device, the appropriate handler (or "driver") must be chosen, and various information must be provided 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 will automatically invoke 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 change some aspect of a device driver, or to change to a new device driver (if you buy a new piece of hardware). 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 is given control and will display its menu:

```
Configuration menu

  Ø.  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 system console
  7.  Configure operating parameters

Enter selection: _
```

To select a configuration task, simply enter the task number, followed by RETURN. When the task is complete, AutoCAD will return to the Configuration Menu and ask you to select another task. Continue selecting tasks until you are satisfied with the configuration as it stands; then select task Ø to exit from the The tasks are described below.

In each task, reasonable defaults are provided for all option The defaults are displayed within corner brackets, "< >"; you m choose the default simply by pressing space or RETURN. (Spa and RETURN are equivalent throughout the configurator.)

Task Ø. Exit to Main Menu

This task exits from the Configurator and returns to AutoCAD Main Menu. Before doing so, however, it asks whether you real 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 I port to which your digitizer, plotter, and (perhaps) display a connected. Reasonable defaults are provided, however, and t ports are often of no concern to you. Therefore, t Configurator will not ask about I/O ports, and will not inclu them in the "Show current configuration" display, unless y first select Task 2 and indicate that you are prepared to de with I/O port addresses and other nasty hardware stuff.

If your computer has standard names for its serial and paral I/O ports, you may use them during port configuration.

The digitizer and plotter (or even the video display and plotte may share the same I/O port, since AutoCAD does not use the devices at the same time. A switch or some other means swapping cables must be provided if this type of connection necessary. However, the video display and digitizer may never connected to the same I/O port. If such a condition is detecte the Configurator will display the message:

```
**I/O conflict:  video display
  and digitizer are on the same port**
```

and will not permit this configuration to be used. The "al I/O port configuration" switch will be turned on automatical so that you may see and reconfigure the I/O ports in question.

Task 3. Configure video display

On some computers, AutoCAD can operate with a variety of graph displays. This task lists the available displays, and asks to indicate which one you will be using. You may also prompted to set various parameters associated with the displ

For instance, if a display has a light pen or Touchpen (tm) capability, you may 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-86 Installation Guide / User Guide Supplement.

The Advanced Drafting Extensions package includes two features which may be used with most video displays; the AXIS command's ruler line, and the Mode/Coordinate display or "Status Line". These features are described in Chapter 9. If you have purchased the Advanced Drafting Extensions package, the Configurator will allow you to customize these features to your liking.

Status Line You may disable the status line entirely, or you may enable it with or without the continuously updating coordinate display. The default for most displays is "enabled" without continuously updating coordinates. See Section 9.3 for more detailed information.

Axis Display On some displays, you may adjust the length of the longest tick marks in terms of "pixels" (picture elements or screen dots). On some displays, you may also choose whether the axis will be drawn inside or outside the screen's drawing display area.

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 will use. A list of the available digitizers will be displayed. When you choose one, additional information may be requested. (If you have no digitizer, you should so indicate by selecting "None".)

Your AutoCAD-86 Installation Guide / User Guide Supplement has the latest list of supported digitizers, along with their configuration and installation requirements.

Task 5. Configure plotter

AutoCAD supports several 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 will use. A list of the available plotters will be displayed. When you choose one, additional information may be requested. (If you have no plotter, you should so indicate by selecting "None".)

Your AutoCAD-86 Installation Guide / User Guide Supplement h the latest list of supported plotters, along with the configuration and installation requirements.

One option with which you will always be presented is useful f making large plots in several pieces. It allows you to specif each time you plot a drawing, the point in the drawing whi should be positioned at the plot origin. If you do not sele this option, every plot will automatically be positioned so th the point that appears at the plot origin is the lower le corner of the last view of the drawing that was on your displ (unless you exited the Drawing Editor via the QUIT command).

Task 6. Configure system console

On some computers, AutoCAD must be told about certa characteristics of the system console. If this is necessary your computer, appropriate self-explanatory questions will asked.

Task 7. Configure operating parameters

This task will display a submenu:

```
Operating parameters menu

  Ø.  Exit to configuration menu
  1.  Alarm on error
  2.  Initial drawing conditions
  3.  Drawing Editor menu
  4.  Dimension text orientation

Enter selection: _
```

The operating parameter subtasks are:

Subtask Ø This subtask will exit from Task 7, and return you the Configuration Menu.

Subtask 1 If you wish to hear an audible alarm whenever AutoC detects an invalid entry, you may enable it usi this subtask.

Subtask 2 This subtask allows you to specify the initi drawing limits, snap resolution, and coordina display format for all new drawings. These valu may be changed while drawing, using the LIMITS, SNA and UNITS commands.

Subtask 3 A default menu file for the Drawing Editor may selected using this subtask. The standard menu fi supplied with AutoCAD will be used if you dor specify one. If you prefer no default menu file

all, respond to the prompt with a period, ".", followed by RETURN.

If it is possible to use a tablet menu with your currently configured digitizer, this subtask will also ask if you wish to use the menu. If you are currently using a tablet menu, this subtask will ask if you wish to re-align it. An affirmative response to either of these questions will cause AutoCAD to ask you where the tablet menu is located the next time you enter the Drawing Editor. The prompts look like:

```
Align tablet menu

Digitize lower left corner of menu: <digitize>
Digitize upper right corner of menu: <digitize>
```

You should fasten the menu form to the top of the tablet, and digitize the two requested corner points.

Subtask 4 This subtask is provided only if your copy of AutoCAD has the optional Dimensioning feature. Using this subtask, you may select either horizontal or vertical alignment of dimension text when the dimension line runs vertically. You may select different orientations for the cases when the text will fit within the extension lines and when it will not. The default is horizontal for both these cases.

D.2 Error Recovery

Configuration is a complex process. While AutoCAD attempts t validate the choices you make, and provides reasonable default for everything, mistakes can occur. Therefore, four levels o error recovery are provided.

- Entering a **CTRL C** during any configuration task will disca all changes made since you selected that task, and will retu you to the most recent menu.

- Upon exiting from the Configurator, you are asked whether y really want to save the new configuration.

- Configuration data is kept in a disk file named ACAD.CFG. you are updating an existing configuration, the old file wi be retained, with its name changed to ACAD.BAK. If you produ a bad configuration, you may delete its ACAD.CFG file, rena ACAD.BAK as ACAD.CFG, and start over. Note that if you ha changed device drivers, you will have to go through t configuration process again, but you should be able to use t defaults from your old configuration file simply by enteri RETURN for most of the prompts.

- Once you have a working version of AutoCAD, make a backup co of the disk, and keep it in a safe place. Be sure to inclu your AutoCAD-86 serial number and the Autodesk, In copyright notice on the label. This is your "last ditc disaster recovery mechanism if all good copies of ACAD.CFG a lost.

Appendix E

REPORTING PROBLEMS

If you encounter problems when using AutoCAD, contact your dealer. You can assist in solving the problem if you supply the following important information:

- The release level and serial number from your AutoCAD-86 release disks. (This information is required whenever you need assistance.)
- A complete description of the hardware environment you are using (memory size, peripheral devices, etc.).
- The operating system and level being used.
- The circumstances, as best you can determine them, under which the problem occurs.
- The text of any messages accompanying the error.
- A diskette containing a copy of the ".DWG" drawing file (if the problem occurs when working with a particular drawing).
- A diskette containing a copy of your ACAD.CFG configuration file (if the problem appears to be configuration-related).
- Any supporting documentation (plotter output, listings, file dumps, etc.) which might help in diagnosis of the problem.

Appendix F

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.

F.1 Version 1.3Ø (August, 1983)

- Configuration facility added
- Layer-to-layer move via CHANGE command
- "Rubber band" and "box" crosshairs
- Right-justified text ability
- Large menus (no more 4Ø item limit)
- Erasure of screen menu
- Plot origin and alignment may be adjusted
- Grid spacing may be tied to the snap resolution
- CHANGE command for Text entities
- Large plotter support added
- Audible error alarm option
- Enhanced HELP command
- Header information included in interchange file
- CTRL C during menu dialogue returns to menu
- "New drawing" dialogue moved to Configurator, along with selection of plotter model and default menu file
- Changed "REDRAW ON/OFF" to "FILL ON/OFF"
- Changed "RESOLUTION" to "SNAP"
- Eliminated abbreviations in many prompts
- Standardized color numbers
- Support for several additional computer systems, plotters, digitizing tablets, and mice

F.2 Version 1.4Ø (October, 1983)

- Screen crosshairs now operate while in Tablet mode
- LINE command enhancements (automatic polygon close, start line at end of last line or arc)
- Alternate arc and circle specifications
- ZOOM enhancements (ZOOM Previous, ZOOM Extents, magnification relative to current magnification)
- New ARRAY facility, including circular arrays
- Block enhancements (WBLOCK command writes block to file, blocks may be redefined)
- Text size may be specified by width of area
- Font definition now includes height above base line, to make text height specification more accurate
- New text fonts are included
- Text for vertical dimensions may be aligned vertically
- Control keys to toggle Snap, Grid, Ortho, and Tablet modes
- INSERT rotation angle governed by Ortho mode
- Display snapped crosshair if Snap mode is on
- Enhanced Shape definitions (X-Y displacments, arcs)
- Specification of REPEAT or ARRAY column/row distance using two corners of a rectangle
- FILES command and new main menu task for access to disk file directories
- The grid is no longer automatically rescaled
- Space and RETURN are now equivalent in all cases except text strings (affects HELP and LAYER commands)
- The old REDRAW ON/OFF command was removed
- Documented the command script facility
- The Dimensioning option was renamed "Advanced Drafting Extensions". This package now includes Dimensioning and:

  - Mode / coordinate display line
  - HATCH command: Crosshatch / pattern-fill
  - SKETCH command for free-hand drawing
  - AXIS command to display ruler lines
  - UNITS command for feet-and-inches input and display
  - BREAK command to delete part of a line, trace, circle, or arc
  - FILLET command to connect two lines with a smooth arc

Appendix G

AutoCAD COMMAND REFERENCE

Commands

ARC	DELAY	HELP	ORTHO	SKETCH *
AREA	DIM *	ID	PAN	SNAP
ARRAY	DIST	INSERT	PLOT	SOLID
AXIS *	END	LAYER	POINT	STATUS
BASE	ENDREP	LIMITS	(QPLOT)	TABLET
BLOCK	ERASE	LINE	QUIT	TEXT
BREAK *	FILES	LIST	REDRAW	TRACE
CHANGE	FILL	LOAD	REGEN	UNITS *
CIRCLE	FILLET *	MENU	REPEAT	WBLOCK
COPY	GRID	MOVE	RESUME	ZOOM
DBLIST	HATCH *	OOPS	SHAPE	?

() = machine dependent command
* = Advanced Drafting Extensions feature

Entity Types

ARC	ENDREP	POINT	SOLID
BLOCK	LINE	REPEAT	TEXT
CIRCLE	LOAD	SHAPE	TRACE

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.

Command Options

Object selection:	L	= Last object drawn
	W	= Objects within window
ARC:	A	= Included angle
	C	= Centre point
	D	= Starting direction
	E	= End point
	L	= Length of chord
	R	= Radius
	blank	= (as reply to Start point) sets start poin and direction as end of last line or arc
ARRAY:	C	= Circular array
	R	= Rectangular array
AXIS:	ON	= Turn axis (ruler line) on
	OFF	= Turn axis off
	number	= Set tick spacing (Ø = use snap resolutior
	numberX	= Set spacing to multiple of snap resolutic
CHANGE:	L	= (as reply to Intersection point) change layers
CIRCLE:	2P	= Specify by 2 endpoints of diameter
	3P	= Specify by 3 points on circumference
	D	= To enter diameter instead of radius
DIM:	A	= Set arrow size
	B	= Continue from 1st extension line
	C	= Continue from 2nd extension line
	T	= Specify text orientation
FILL:	ON	= Solids and traces filled
	OFF	= Solids and traces outlined
FILLET:	R	= Set fillet radius
GRID:	ON	= Turn grid on
	OFF	= Turn grid off
	number	= Set grid spacing (Ø = use snap resolutior
	numberX	= Set spacing to multiple of snap resolutic
HATCH:	I	= Ignore internal structure
	N	= Normal style: turn hatch lines off and c 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 known block names

LAYER:	number	= Set current layer number
	ON x,y	= Turn on layers
	OFF x,y	= Turn off layers
	COLOUR x	= Set current colour & colour of layer
	COLOUR *	= Set each layer's colour to layer number
	?	= List layers and colours
LINE:	blank	= (as reply to From point) start at end of previous line or arc
	C	= (as reply to To point) close polygon
ORTHO:	ON	= Force lines to horizontal or vertical
	OFF	= Do not constrain lines
SKETCH	C	= Connect: restart sketch at end point
	E	= Erase (back up over) temporary lines
	P	= Raise/lower sketching pen
	Q	= Discard temporary lines, exit Sketch
	R	= Record temporary lines, remain in Sketch
	X	= Record temporary lines, exit Sketch
	.	= Draw line to current point
SNAP:	number	= Set snap alignment resolution
	ON	= Align designated points
	OFF	= Do not align designated points
TABLET:	ON	= Turn tablet mode on
	OFF	= Turn tablet mode off
	CAL	= Calibrate tablet
TEXT:	A	= Fit text within area, choose appropriate height
	C	= Centre text
	R	= Right justify text
WBLOCK:	name	= Write specified block definition
	=	= Block name same as file name
	*	= Write entire drawing
	blank	= Write selected objects
ZOOM:	number	= Multiplier from original scale
	numberX	= Multiplier from current scale
	A	= All
	C	= Centre
	E	= Extents
	L	= Lower left corner
	P	= Previous
	W	= Window

Index

Autodesk, Inc.

The AutoCAD-86 (tm) Drafting Package

Installation Guide / User Guide Supplement

Edition for the IBM Personal Computer
running the PC-DOS operating system

(also for Eagle, Columbia, and other IBM PC
"work-alikes" running the MS-DOS operating system)

This manual was prepared and published in February 24, 1984, and is based on Release 1.40 of the AutoCAD-86 (tm) drafting package.

AutoCAD-86 / IBM PC

Table of Contents

Chapter 1

INTRODUCTION

This guide is designed as a supplement to the main AutoCAD-86 User Guide. It contains some minor exceptions and clarifications to the information presented therein, and explains the process of installing the AutoCAD (tm) drafting package and graphic input/output devices on the IBM Personal Computer.

The software installation procedure assumes that you have mastered the general operation of your computer and "booting" of its PC-DOS operating system. In particular, the process of copying files and entire disks should be understood. This information should be available in the manufacturer's manuals, and will not be repeated here. The sections of this guide that concern peripheral devices assume a fair amount of technical skill and an understanding of some computer terminology. If you run into difficulty, ask your AutoCAD dealer or a technically-inclined friend for assistance.

If all else fails, technical support is available directly from Autodesk, Inc., but please note that it is often difficult to solve installation problems over the telephone. Consult Appendix E of the main AutoCAD-86 User Guide for the type of information you will be asked to supply.

1.1 Notational Conventions

As in the main User Guide, sample dialogues in this guide will have the user input underlined.

1.2 System Requirements

Depending on the equipment attached to your IBM Personal Computer, AutoCAD-86 can operate in either single-screen or dual-screen mode. In single-screen mode, one display monitor is used both for graphics and as the system console. In dual-screen mode, the graphics are drawn on one monitor, while the other becomes the system console.

Standard Configuration:

- IBM Personal Computer (or PC XT) with at least 256 K memory
- Two double-sided floppy disk drives, or one double-sided floppy and a hard disk drive
- IBM Color/Graphics Display Adapter with color or black-and-white monitor
- PC-DOS Version 1.1 or 2.0 Operating System
- Digitizing tablet or mouse (optional)
- Plotter (optional)
- Intel 8087 Numeric Processor Chip (optional)
- Asynchronous Communication Adapter (required for plotter and digitizer)

AutoCAD "pages" drawings to disk when they exceed the available memory. If your system contains extra memory, more of the drawing will fit before disk access becomes necessary; thus, performance will improve.

If your system includes an 8087 math co-processor, AutoCAD will sense this automatically and use it to speed up the drawing process. Speed gains of 300% or more can be expected, especially when drawing circles, arcs, and text.

Several alternate video display configurations are supported. These are described in Chapter 4.

If your computer is an IBM PC "work-alike", consult Chapter 7 of this guide for special notes regarding your machine.

Chapter 2

EXCEPTIONS AND CLARIFICATIONS

2.1 Commands Not Available

The QPLOT command (quick plot to dot matrix printer) is not available on the IBM PC.

2.2 Keyboard Functions

In the main User Guide, it was necessary to use generic function names for various keyboard control keys. The keys assigned to these functions on the IBM PC are listed below.

AutoCAD function		IBM PC key
CTRL	=	Ctrl
FLIP SCREEN	=	F1
MENU CURSOR	=	Ins
SCREEN CURSOR	=	Home
ABORT CURSOR	=	End
FAST CURSOR	=	Pg Up
SLOW CURSOR	=	Pg Dn
UP CURSOR	=	Up arrow
DOWN CURSOR	=	Down arrow
LEFT CURSOR	=	Left arrow
RIGHT CURSOR	=	Right arrow
TOGGLE COORD	=	F6 or Ctrl D
TOGGLE GRID	=	F7 or Ctrl G
TOGGLE ORTHO	=	F8 or Ctrl O
TOGGLE SNAP	=	F9 or Ctrl B
TOGGLE TABLET	=	F10 or Ctrl T

In dual-screen AutoCAD configurations, the FLIP SCREEN key has no effect. The TOGGLE COORD key is functional only if the status line feature (ADEX package) is present and enabled.

Chapter 3

SOFTWARE INSTALLATION

3.1 Release Format

AutoCAD-86 for the IBM Personal Computer is released on three disks. Two of the disks contain the necessary files to configure and run the program; these disks are covered by the AutoCAD-86 Software License Agreement and copyrighted by Autodesk, Inc. The remaining disk contains sample drawings for you to experiment with, and is not copyrighted.

The files on the disks include:

Disk 1: AutoCAD Program

ACAD.EXE	AutoCAD execution file
ACAD.OVL	overlay file
ACAD.HLP	standard help text
ACAD.MNU	standard menu
ACAD.PAT	standard hatch patterns (present only if the Advanced Drafting Extensions package was purchased)
TXT.SHP	standard text font
<name>.SHP	alternate text fonts

Disk 2: AutoCAD Device drivers

<name>.DRV	device drivers

Disk 3: AutoCAD Sample Drawings

<name>.DWG	sample drawings
<name>.MNU	menus for sample drawings
<name>.SHP	sample shape files and alternate fonts

In addition, one disk may include a file named "README.DOC". If such a file is present, list it out. It will contain information on late developments which may not have made it into the printed manual.

3.2 Making a Backup

When you receive the software, the first thing to do is to make backups of the three release disks using DISKCOPY, then store the original disks in a safe place. Be sure to write the serial number from the "Program" disk, and the Autodesk, Inc. copyright notice, on the label of the new copy. Do <u>not</u> write-protect the copy.

3.3 Running the Configurator

Place your copy (not the original!) of the "Program" disk in Drive A. Enter:

ACAD

and select the "Configure AutoCAD" main menu task. (This will be done automatically if AutoCAD senses that it has not yet been configured.) The Configurator will ask you to:

Insert device driver disk in free drive and
enter drive letter when ready:

Place your copy of the "Device drivers" disk in Drive B, and respond to the message with "B" followed by a RETURN. The "Device drivers" disk is never altered by the configuration process. In fact, it may be write protected. However, this disk must always be placed in Drive B as described above, whenever you want to re-configure AutoCAD to use a different device.

See Appendix D of the main AutoCAD-86 User Guide for general information on the Configurator. See later sections of this guide for details on the display, digitizer, and plotter options available.

When the configuration process is complete, remove the "Device drivers" disk from Drive B and store it.

3.4 Testing the Configured Version

Once the Configuration process has been completed, you may test AutoCAD using the sample drawings provided. To list the names of the sample drawings, place the "Sample Drawings" disk in drive B. Exit from AutoCAD (if you are still in it) and enter:

DIR B:*.DWG

To verify that AutoCAD-86 works properly on your equipment, try loading one of the sample drawings. Enter:

ACAD

After a short pause, AutoCAD-86 will display its main menu on your monitor. Try selecting menu task 2, and when it asks for a drawing name, enter:

B:OFFICE (one of the sample drawings)

The Drawing Editor should start drawing an office furniture layout on your monitor. At this point, you know that AutoCAD is properly configured for your video display.

When the drawing is complete (the "Command:" prompt appears), try your digitizer or mouse if you have configured one. Proper operation will be indicated if crosshairs appear on the screen and follow the movements of the digitizer. Using the commands and techniques described in Chapters 2 and 3 of the main AutoCAD-86 User Guide, you should be able to designate points and select commands from the screen menu.

Proper operation of the plotter should be tested next, if you have configured one. Enter the command "PLOT" to send the current drawing to the plotter. See Chapter 10 of the main User Guide for more information on plotting.

When the plot is complete, AutoCAD will return to the Drawing Editor. To exit to the main menu, enter:

quit

You can get back to the operating system prompt by selecting menu task 0. More detailed information on the main menu and use of the Drawing Editor may be found in Chapter 2 of the main User Guide.

If any problems arise during these tests, double check your configuration. Select the "Configure AutoCAD" task from the main menu, and use its "Show Configuration" subtask to list the current configuration. If you cannot determine what is wrong, contact your dealer for assistance. (See Appendix E of the main User Guide for the information you'll need to provide.)

3.5 Making Another Backup

Once you are satisfied that your AutoCAD is properly configured, make another copy of the "Program" disk and save the original copy in a safe place. Again, be sure to duplicate the serial number and copyright notice on the label of the new copy.

The copy you have just made will be your AutoCAD production disk.

3.6 Hard Disk Usage

AutoCAD's performance will be enhanced if it resides on a hard disk. If your system includes a hard disk, you can install AutoCAD on it simply by copying the contents of your configured "Program" disk onto the hard disk. Then make the hard disk your "logged" disk drive before calling ACAD.

Chapter 4

VIDEO DISPLAY OPTIONS

This chapter describes various video display components which may be used with AutoCAD, and discusses the hardware interconnections needed to utilize them. The AutoCAD software must also be tailored to your equipment; this process is described in Appendix D of the main AutoCAD-86 User Guide. The questions asked by the Configurator about each device being configured are designed to be self-explanatory. Little will be said about them in this document.

AutoCAD can operate with any of several video display configurations on the IBM PC, from which you must choose one.

4.1 Single-screen IBM Color

In this configuration, the IBM Color/Graphics Display Adapter is used with a color or black-and-white monitor for graphics drawing and as the system console. If you select this configuration, AutoCAD will ask whether you wish to use the color capability.

4.2 Dual-screen IBM Color

This configuration utilizes two monitors:

- The IBM Color/Graphics Display Adapter (with color or monochrome monitor) displays the graphics drawing.
- The IBM Monochrome Display/Printer Adapter (with another monitor) is used as the system console. (Under PC-DOS 2.0, an ASCII terminal on a serial I/O port may be used instead.)

If you select this configuration, AutoCAD will ask whether you wish to use the color capability.

IBM's documentation is a bit vague when discussing the setting of the monitor-selection switches for dual-screen configurations. The switches should be set to the "monochrome" setting if you are using the IBM Monochrome Display/Printer Adapter, whether or not you have another monitor as well.

4.3 Single-screen Hercules

In this configuration, the (monochrome) Hercules Graphics Card is used instead of the IBM Display/Printer Adapter. (The Hercules card will not operate if either the IBM Monochome Display Adapter or the IBM Color/Graphics Display Adapter is present in the machine.)

If you wish to use AutoCAD in this configuration, you must first make sure the Hercules Graphics Card is installed, and that any IBM Display Adapters which may be present have been removed. Clear directions are given in the Hercules Operation Manual. Note that the system switch settings required for use of the Hercules are the same as for the IBM Monochrome Display/Printer Adapter.

Once the Hercules Graphics Card is installed and is functioning properly, you are ready to configure AutoCAD to use it.

4.4 Dual-screen Vectrix Color

AutoCAD can operate in a dual-screen mode, using the Vectrix VX384 color graphics controller and monitor for the drawing, and a separate system console (for instance, a monochrome monitor with either the IBM or Hercules adapter). Set IBM's monitor-selection switches to match the system console. .

The Vectrix has a high resolution (672 X 480) flicker-free display. AutoCAD currently supports 64 simultaneous colors on it. (See sample drawing "COLORS".)

The Vectrix controller may be connected to either a serial or parallel (printer) I/O port, using cables provided with it. The parallel port is the default; if you wish to use a serial port, perform I/O port configuration using AutoCAD's Configurator (see Appendix D of the main AutoCAD-86 User Guide.) You may choose to run the Vectrix at either 9600 or 19200 baud.

4.5 Tecmar Graphics Master

The Tecmar Graphics Master color graphics board can be used to create a single-screen AutoCAD system with 640 x 400 resolution and up to 16 colors. An IBM-compatible RGB monitor is required.

For proper operation with AutoCAD, the Tecmar board must be installed as the PC's primary display device, with its switches set to act like an IBM Color/Graphics Display Adapter. AutoCAD will reprogram the board to achieve the 640 x 400 resolution. The monitor should have a medium-persistence phosphor to avoid flicker at this resolution; if you find the flicker annoying,

obtain a better monitor or use the Tecmar's 320 x 200 mode (by telling AutoCAD that this is an IBM single-screen display).

The switch settings outlined below were taken from the 10/30/83 version of the Tecmar manual, but many revisions of the board exist. If yours differs from the description given here, configure it to act like an IBM Color/Graphics Display Adapter, and it should work with AutoCAD.

On the Tecmar board:

Set the External Monitor Select switch to the down position.

Connect:	JPR1	Position A
	JPR1	Position C
	JPR4	
	JPR5	Position B
	JPR6	
Disconnect:	JPR1	Position B
	JPR1	Position D
	JPR1	Position E
	JPR1	Position F
	JPR7	
	JPR8	

On the IBM System unit board, set:

SW1: 5 on, 6 off

If you have told AutoCAD to use the Tecmar Graphics Master for the video display, the Configurator will permit you to select any of the available colors for the background, border, and menu/prompt text. This is to accomodate monitors with limited bandwidth, or personal color pattern preferences. The following prompts will appear.

Select border color (0 to 127) <5>:

This allows changing the border surrounding the graphics display to any AutoCAD logical color. The default is blue (5).

Clear screen color (0 to 127) <5>:

This specifies the background color used for the graphics display. The default is blue (5), and you can select a color number corresponding to any color except white.

Text color (0 to 127) <2>:

This specifies the color used for text in the menu, status line, and command prompt areas. The default is yellow (2).

Note thtt zero is a legal color number for these options; it means black. If you are using a monitor with low resolution, the defaults for these options should provide the optimal display.

Drawing Color Assignments

Sixteen colors are available on the Tecmar board. The first 7 color numbers adhere to the standard AutoCAD color sequence:

1 - Red
2 - Yellow
3 - Green
4 - Cyan
5 - Blue
6 - Magenta
7 - White

However, since the background color may be user specified, any color which would map into the background color (and thus be invisible) will be translated into white.

Color numbers 16 and above are not mapped, except to prevent their mapping into the background color; they may be used to access the native Tecmar colors, including black. Thus, if the background color is not black, black may be used as a drawing color. Color numbers 8 through 15 are not defined.

Chapter 5

DIGITIZER OPTIONS

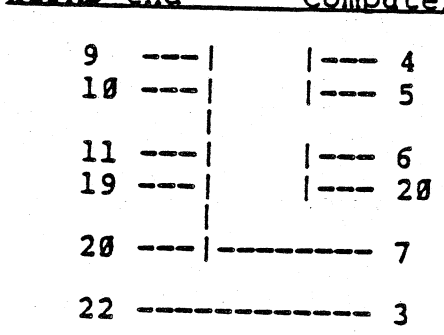
If you wish, you may select any one of several digitizing tablets and mice for use with AutoCAD. Each connects to a serial I/O port (the default is COM1, the first Asynchronous Communications Adapter port). Since the digitizer and plotter are not used at the same time, you can configure them on the same serial port, but you'll need a switch or some other means of swapping cables to connect the proper device when required.

You may have to make your own cable to connect the digitizer to the serial port. Some special wiring may be required at each end of the cable, and it is possible for both ends to require the same kind of connector. Therefore, be sure to clearly label the ends to distinguish them from one another. The cabling requirements and switch settings for individual digitizers will be noted in the following sections.

5.1 Houston Instrument HIPAD DT11AA Tablet

The Houston Instrument HIPAD digitizing tablet may be connected to an RS-232C serial I/O port. The HIPAD should be set for 4800 baud serial data output, with absolute coordinates on, and should be in STREAM mode when used with AutoCAD. A standard tablet menu area at the top of the tablet is supported.

A very strange interface cable is required to connect the HIPAD to the computer's serial port. It should have the following connections:



Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

5.2 Houston Instrument COMPLOT Series 7000

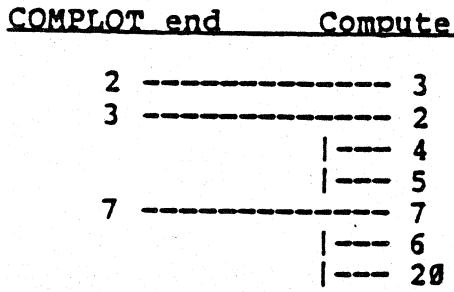
A Houston Instrument COMPLOT Series 7000 digitizing tablet (model 7012, 7024, 7048, or 7060) may be connected to an RS-232C serial I/O port. A standard tablet menu area at the top of the tablet is supported.

The tablet's controller should be set for 9600 baud serial data output, 7 data bits, even parity, and 2 stop bits. These parameters are selected using the following switch settings:

Switch	On	Off	Don't care
SW1	1,6,7	2,3,5	4,8
SW2	7	1,2,3,4,5,6,8	
SW3		1,2,3,4,5,6,7,8	
SW4	1,3,4,5,6	2,7,8	
SW5	1,3,4,5,6,7	8	2
SW6	1,3		2,4

In addition, the tablet must be in STREAM mode for operation with AutoCAD. The switches marked "don't care" control such things as whether or not the tablet beeps every time you press its button. These do not affect the tablet's operation with AutoCAD, and may be set as you wish.

Connect the MODEM port on the tablet controller to the desired RS-232C serial I/O port on the computer, using a cable with the following connections:



Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

A stylus, 1-button cursor, or 12-button cursor may be used. With the 12-button cursor, the "0" button is used to select points, and you can invoke the first 11 menu items using the remaining buttons (1-9, *=10, and #=11).

5.3 Hitachi HDG-1111 Tiger Tablet

Connect the Tiger tablet's "Port A" to an RS-232C serial I/O port on the computer. A standard 25-wire flat cable may be used, or you can construct a two-wire cable; it should run the following lines between the tablet and the computer:

Tablet end		Computer end
3	----------	3
4	---| |---	4
5	---| |---	5
7	----------	7
	|---	6
	|---	20

Three cursor types are available for the Tiger tablet; 4-button, 12-button, and stylus. The Configurator will ask which one you intend to use. You may also elect to use a tablet menu.

With the 4-button cursor, button 1 is used for point selection, and buttons 2, 3, and 4 can be used to select the first, second, and third menu items, regardless of the position of the cursor. With the 12-button cursor, button 0 is used for point selection, and buttons 1 through 9 select the first nine menu items. The buttons labeled "*" and "#" should not be used.

The Tiger tablet has numerous DIP switches on the back. For use with AutoCAD, they should be set for the following modes:

> Serial, Async, 9600 baud, Single port, Port A, 7 bit, Even parity, 1 stop bit, Test mode off, ASCII transmission, Leading zero suppression off, Function code output on, CR code output on, LF code output on, Buzzer on, Resolution 0.001 inch, English units, Run mode, Incremental mode off, Rate 1 (fastest), Menu disabled.

To achieve these modes, set ALL switches OFF except for the following:

DSW1	--	6	on
DSW2	--	2,5	on
DSW4	--	1	on

5.4 Hitachi HICOMSCAN HDG Series Tablets

A handler is provided for the Hitachi HICOMSCAN HDG series of tablets, models 1216, 2222, 3648, and 4460. The tablet must be equipped with the RS-232C serial interface, and the DIP switches and jumpers should be set as follows:

Switch	On	Off
MODE RATE	1,2,3,4	5,6,7,8
BAUD RATE	8	1,2,3,4,5,6,7

PC board jumpers (standard factory settings)

SYNC, OP, F0, F1	Removed
1S	Installed
RxC ASYNC	Installed
RxC SYNC	Removed
TxC ASYNC	Installed
TxC SYNC	Removed

The cable between the tablet and the computer's serial I/O port is shown below. Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

Tablet end	Computer end
2	3
3	2
4	4
5	5
8	
7	7
6	6
20	20

A stylus, 4-button cursor, or 12-button cursor may be used to designate points or to select the pointed-at item from a screen or tablet menu. If a multi-button cursor is used, its additional buttons will invoke specific menu items regardless of the position of the cursor.

Cursor type	"Hit" button	Menu item buttons
Stylus	stylus tip	None
4-button	1	2,3,4 = items 1,2,3
12-button	0	1-9 = 1-9, *=10, $=11

5.5 Summagraphics Bit Pad One

The Summagraphics Bit Pad One digitizing tablet with RS-232C interface can be used with AutoCAD. AutoCAD expects the Bit Pad One to be set for 9600 baud, even parity, 2 stop bits, binary output, stream mode, fastest sample rate, and 0.005" resolution. These modes are implemented by the standard factory settings of the internal DIP switches, with one exception:

Position 7 of Switch 1 must be set to ON.

Access to the DIP switches can be gained by removing four Philips-head screws from the back of the Bit Pad One.

A two-wire cable can be used, with the following connections:

Bit Pad end		Computer end
2	----------	3
4	---| |---	4
5	---| |---	5
7	----------	7
	|---	6
	|---	20

Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

Either the stylus or the four-button cursor may be used. With the four-button cursor, button 0 is used for point selection and buttons 1, 2, and 3 select the first, second, and third menu items. A standard tablet menu area at the top of the tablet is supported.

5.6 Summagraphics MM Series Tablets

A handler is supplied for the Summagraphics MM series tablets, models 12Ø1 (12 x 12 inch) and 961 (9 x 6 inch). Model 961 may be used with its long direction oriented either vertically or horizontallly. Both tablets may be run in either a conventional digitizer mode, or in a relative-motion "mouse" mode, using either a 3-button cursor or a stylus. AutoCAD allows you to select any combination of these options, but assumes the standard factory settings for the internal jumpers.

These digittizers connect to an RS-232C serial I/O port. A three-wire cable should be sufficient, connecting the following pins between the tablet and the computer:

Tablet end	Computer end
2	3
3	2
	4
	5
7	7
	6
	2Ø

Note that some pins are strapped together in the connector at one end of the cable, without running them to the other end.

During digitizer configuration, AutoCAD will ask which model and cursor type you have, and which mode (digitizer or "mouse") you wish to use. If you have selected the model 961, the orientation of the tablet will also be requested. If you select "mouse" mode, AutoCAD's Tablet mode and tablet menu capabilities will be disabled. Also, the cursor/stylus must be lifted further from the surface when repositioning than is typical with real mice; this is an aspect of the hardware design, not of AutoCAD's tablet handler.

The tablet can be used to designate points on the screen and to select items from a menu. With the stylus, pressing the tip desginates a point or selects the pointed-to menu item, while the switch on the barrel of the stylus selects the first menu item, regardless of the position of the stylus.

When using the 3-button cursor, the button with one dot is the "hit" button, to designate a point or to select the item pointed to on the menu. The buttons with two and three dots immediately select the first and second items of the menu, respectively, regardless of the position of the tablet's cursor. Thus, these buttons can invoke frequently used commands.

A standard tablet menu area at the top of the tablet is supported, except if the tablet is operated in "mouse" mode.

5.7 Summagraphics SummaMouse

The SummaMouse connects to an RS-232C serial I/O port, using a cable with the following connections:

Mouse end		Computer end
2	----------	3
3	----------	2
	|---	4
	|---	5
7	----------	7
	|---	6
	|---	20

Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

AutoCAD uses the SummaMouse's standard "MM series delta" data format in remote request mode, and sets the mouse to 9600 baud. If the mouse has been set to a different data rate by another program, it must be powered off and back on before AutoCAD can use it.

Each time the mouse is powered up, you must calibrate it by moving it diagonally across the mouse pad (see the SummaMouse Technical Reference). However, calibration is not possible until the mouse's baud rate has been established. Therefore, when AutoCAD's Drawing Editor begins execution, it will check the mouse, prompt you to calibrate it if necessary, then wait for calibration to be completed. If calibration cannot be accomplished, you can tell AutoCAD to give up on the mouse by pressing CTRL C.

The SummaMouse can be used to designate points on the screen and to select items from the screen menu. The leftmost button is the "hit" button, to designate a point or to select the item pointed to on the screen menu. The middle and right buttons immediately select the first and second items of the menu, respectively, regardless of the position of the mouse. Thus, these buttons can invoke frequently used commands.

The tablet menu option is not supported for relative pointing devices such as the SummaMouse.

When you move the mouse quickly, the screen crosshair will move rapidly. For more precise pointing, simply move the mouse slowly. The fast and slow motion scale factors, and the threshold which distinguishes between "fast" and "slow" may be adjusted during configuration. Try the default values; you can experiment with the parameters if the defaults do not suit your style.

5.8 Calcomp 9000 Series Tablets

A handler is provided for the Calcomp 9000 series of tablets, models 9120, 9240, 9360, 9480, and 9600, using the RS-232C serial I/O interface. The DPU DIP switches should be set as follows:

Switch	Up	Down	Model-dependent
SW1	1,2,3,4,5,8	6,7	
SW2	4	1,2,3	5,6,7,8
SW3	2,3,6	1,4,5,7,8	

Switches 5,6,7,8 in SW2 depend on the tablet model:

Model	Up	Down
9120	5	6,7,8
9240	6	5,7,8
9360	5,6	7,8
9480	7	5,6,8
9600	5,7	6,8

The connection between the tablet and the computer is via Port A (Host, J7) on the tablet controller. The cable is shown below. Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

Tablet end	Computer end
2	2
3	3
	4
	5
7	7
	6
	20

A stylus or multi-button cursor may be used to designate points or to select the pointed-at item from a screen or tablet menu. If a cursor is used, its additional buttons will invoke specific menu items regardless of the position of the cursor.

Cursor type	"Hit" button	Menu item buttons
Stylus	stylus tip	None
4-button	Pick	1,2,3 = items 1,2,3
12-button	0	1-9 = 1-9, #=10, *=11
16-button	0	1-9 = 1-9, A-D = 10-13, #=14, *=15

NOTE: On the 12- and 16-button cursors, the "*" button may be used to select menu items only if the tablet's Cursor Function Disable sequence "**0" has been executed. See page 4-6 of the Calcomp 9000 Series Digitizer Operator's Manual, 50090-1.

5.9 USI OptoMouse

The USI OptoMouse pointing device connects to an RS-232C serial I/O port. AutoCAD sends output to the OptoMouse to set up its operating modes; the OptoMouse will not work if output cannot be sent to it.

The OptoMouse adapts to the data rate of the port to which it is connected; 9600 baud is used by AutoCAD. If the mouse has been set to a different baud rate by another program, it must be powered off and back on before using it with AutoCAD.

AutoCAD permits the OptoMouse to be used to designate points on the screen and to select items from the screen menu. Assuming the cable from the mouse is at the top, the rocker switch buttons are numbered as follows:

```
2 ---- 1
3 ---- 4
```

Button 1 is the "hit" button, to indicate a point selection, or to select an item pointed to on the screen menu. As an added convenience, buttons 2 through 4 immediately select the first three items of the menu regardless of the position of the mouse. Thus, these may be used for very frequently used commands.

The tablet menu option is not supported for the OptoMouse. The screen menu should be used instead. (Since the mouse is a relative positioning device, the tablet menu would not work.)

When you move the mouse quickly, the screen crosshair will move rapidly. For more precise pointing, simply move the mouse slowly. The fast motion scale factor, and the threshold which distinguishes between "fast" and "slow" may be adjusted during configuration. Try the default values; you can experiment with the parameters if the defaults do not suit your style.

5.10 Mouse Systems Mouse

The Mouse Systems Mouse comes with a mouse pad, a power supply, a connector box, and a cable with an RS-232C connector. The mouse tail, the RS-232C cable, and the wire from the power supply all plug into the connector box. The power supply plugs into the wall. The RS-232C connector plugs directly into an Asynchronous Communications Adapter port on the IBM PC. After connecting the mouse in this manner, calibrate it by moving it around on its pad in large circles at a comfortable speed for ten or 15 seconds; if you turn it over and see that one red light is on, it is calibrated and ready for operation.

AutoCAD allows you to configure this mouse for connection to any available serial port, but it sometimes doesn't work on non-IBM Asynchronous Adapters. So if you have difficulty, try using a different port if you have more than one.

When you are in the Drawing Editor, the crosshairs on the screen should move in correspondence with the mouse's motion on its pad. If the mouse runs off the pad before the crosshairs come to the edge of the screen, you can pick the mouse up and set it back down in the middle of the pad, then move it further in the same direction.

Of the three buttons on the mouse, the leftmost one is used for selecting points or menu items, depending on where the mouse is currently positioned. The middle and right buttons select the first and second items on the menu, respectively, no matter where the mouse is positioned.

5.11 Microsoft Mouse

(As of this writing, Microsoft only supports its mouse on the IBM Personal Computer; it may not operate on "work-alikes".)

The Microsoft Mouse comes with an interface card that plugs into an expansion slot inside the IBM Personal Computer, and a software driver which must be installed in the PC-DOS operating system. Consult the manuals supplied by Microsoft for instructions on installing the interface card and driver.

The mouse can be used to designate points on the screen and to select items from the screen menu. The leftmost button is the "hit" button, to designate a point or to select the item pointed to on the screen menu. The rightmost button immediately selects the first item of the menu, regardless of the position of the mouse. Thus, this button can invoke the most frequently used command.

The tablet menu option is not supported for relative pointing devices such as the Microsoft mouse.

When you move the mouse quickly, the screen crosshair will move rapidly. For more precise pointing, simply move the mouse slowly. The fast motion scale factor, and the threshold which distinguishes between "fast" and "slow" may be adjusted during configuration. Try the default values; you can experiment with the parameters if the defaults do not suit your style.

5.12 GTCO Digipad 5 Tablets

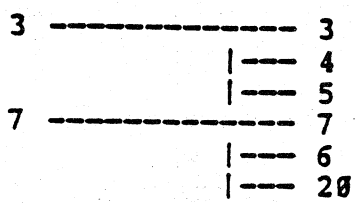
The GTCO Digipad 5 family of digitizing tablets is supported via an RS-232C serial I/O port. A standard tablet menu area at the top of the tablet is supported.

The tablet should be set for binary data transmission at 9600 baud, with 8 bits per character and one stop bit. On the IBM Personal Computer, we suggest using Port B (J6) of the tablet to connect to the computer. These parameters are selected using the following switch settings:

Switch	On	Off	Don't care
SW1	3,4,8	1,2,5,6,7	
SW2	5,7	6	1,2,3,4,8
SW3	3,4,7	1,2,6,8	Set 5 per cursor

The switches marked "don't care" do not affect the tablet's operation with AutoCAD, and may be set as you wish.

Connect Port B (J6) of the tablet to the desired RS-232C serial I/O port on the computer. A standard 25-wire cable may be used, or a simple 2-wire cable may be constructed as follows:



Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

A stylus, 5-button cursor, or 16-button cursor may be used. With the 16-button cursor, the "0" button is used to select points, and you can invoke the first 15 menu items using buttons "1" through "F". With the 5-button cursor, button "5" is used to select points; the action of the other four buttons is currently unknown. With the 1-button stylus, of course, the button is used to select points.

5.13 Logitech Logimouse R-5

The Logitech mouse connects to an RS-232C serial I/O port, using a cable with the following connections:

Mouse end		Computer end
2	----------	3
3	----------	2
	|---	4
	|---	5
7	----------	7
	|---	6
	|---	20

Note that some pins must be strapped together in the connector at one end of the cable, without running them to the other end.

AutoCAD uses the Logimouse's standard "Relative Bit Pad One Packed Binary" format in "remote request" mode, at a data rate of 9600 baud.

The mouse can be used to designate points on the screen and to select items from the screen menu. The leftmost button is the "hit" button, to designate a point or to select the item pointed to on the screen menu. The middle and right buttons immediately select the first and second items of the menu, respectively, regardless of the position of the mouse. Thus, these buttons can invoke frequently used commands.

The tablet menu option is not supported for relative pointing devices such as the Logitech mouse.

When you move the mouse quickly, the screen crosshair will move rapidly. For more precise pointing, simply move the mouse slowly. The fast motion scale factor, and the threshold which distinguishes between "fast" and "slow" may be adjusted during configuration. Try the default values; you can experiment with the parameters if the defaults do not suit your style.

Chapter 6

PLOTTER OPTIONS

To obtain hard copy output of your drawings, you may choose any one of several plotters for use with AutoCAD. Most connect to a serial I/O port (the default is COM1, the first Asynchronous Communications Adapter port). Since the digitizer and plotter are not used at the same time, you can configure them on the same serial port, but you'll need a switch or some other means of swapping cables to connect the proper device when required.

You may have to make your own cable to connect the plotter to the serial port. Some special wiring may be required at each end of the cable, and it is possible for both ends to require the same kind of connector. Therefore, be sure to clearly label the ends to distinguish them from one another. The cabling requirements and switch settings for individual plotters will be noted in the following sections.

Some plotters connect to the PC's parallel printer ("LPTR") port, rather than to a serial I/O port. This will be noted where applicable.

Hardware Buffer Devices

If a serial interface is used, a hardware buffer may be connected between the computer and plotter. With a sufficiently large buffer, most plots can be sent in a fraction of the normal plotting time, freeing the computer for other tasks while the remainder of the plot is sent from the buffer device to the plotter. For this to work, the buffer device must respond to XOFF/XON codes sent by the plotter (by suspending and resuming data flow, respectively), and it must in turn send XOFF/XON codes to the computer to avoid overflowing its own buffer. The only unit we are currently aware of that meets all the requirements is the MBIS Microbuffer from Practical Peripherals, Inc. It is available with up to 256K bytes of buffer memory.

6.1 Houston Instrument DMP-xx Plotters

Houston Instrument HI-PLOT plotters, models DMP-7, DMP-8, DMP-29, DMP-40, DMP-41, and DMP-42 are supported via an RS-232C serial I/O port.

A three-wire cable should be sufficient; it should connect the following pins between the plotter and the computer:

HI-PLOT end	Computer end
2	3
3	2
	4
	5
7	7
	6
	20

Additional connection for DMP-7 and DMP-8 only:

6	
14	

Note that some pins must be strapped together in the connector at one end of the cable, wirhout running them to the other end.

The plotter must be set to communicate at 9600 baud with 7 bits plus even parity, one stop bit, and XON/XOFF protocol. The method of accomplishing this depends on the model of plotter:

DMP-7 and DMP-8: In the cable connector that plugs into the plotter, strap pin 6 to pin 14.

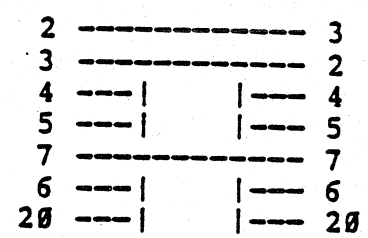
DMP-29: Connect the cable to the plotter's MODEM port. On the back of the plotter, set the switches as follows:

1	0 (closed)	9600 baud
2	1 (open)	"
3	1 (open)	"
4	1 (open)	parity enabled
5	1 (open)	even parity
6	any	
7	0 (closed)	inches
8	any	

DMP-40, DMP-41, DMP-42: Before plotting, set the plotter to use 9600 baud by depressing the ENTER button, then the up-arrow button.

6.2 Hewlett-Packard Plotters

Hewlett-Packard model 7220, 7470, 7475, 7580, and 7585 plotters are supported via an RS-232C serial I/O port, at 9600 baud with even parity (2400 baud for the model 7220). We recommend the following cable connection. Others are possible, but this one requires only a 3-wire cable and works with any H-P plotter. Where there is a choice, attach the cable to the plotter's "Computer" jack, rather than to the one marked "Terminal".



Note that some pins are strapped together in the connector at one end of the cable, without running them to the other end.

On the 7470 and 7475 plotters, establish the proper communication modes by setting the switches as follows:

B4	-	1	S1	-	1
B3	-	0	S2	-	0
B2	-	1	D-Y	-	D
B1	-	0			

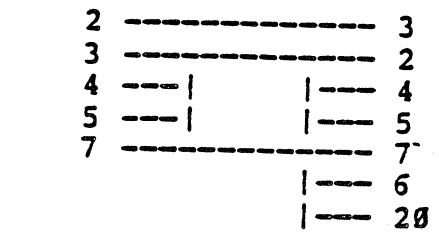
On the 7580 and 7585 models, set the RS-232C speed selector to 9600 and set Parity on, Even Parity, and Eavesdrop off. The Emulate and Expand switches should be set to Normal.

When you begin a plot, AutoCAD asks you to adjust the paper and hit RETURN. If the plotter does nothing in response to the RETURN, wires 2, 3, or 7 may be strung incorrectly or the plotter may be improperly set up (e.g., the Remote light isn't on, in the case of a 7580 or 7585). If the plotter's Error light turns on, then the speed or parity switches are probably wrong.

If the plotter starts a plot correctly but drops data and turns on the Error light, there is probably an error in wiring pin 2, 5, or 6 at the plotter end of the cable. In this case a 7580 or 7585 can also generate the message, "Invalid hard clip limits returned from plotter:".

6.3 Strobe Plotters

AutoCAD communicates with the Strobe model 100, 200, and 260 plotters via an RS-232C serial I/O port. (For the model 100, Strobe's optional IO5 serial interface is required.) A three-wire cable should be sufficient; it should connect the following pins between the plotter and the computer:

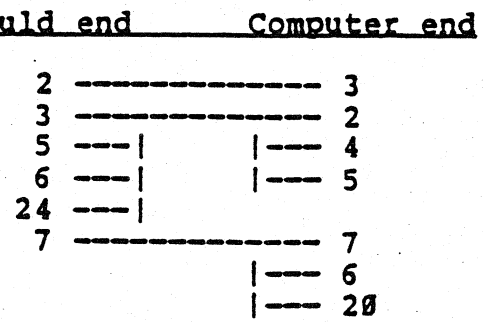


Note that some pins are strapped together in the connector at one end of the cable, without running them to the other end.

Set the plotter for 9600 baud communication (position "E" on the baud rate dial), and for NO parity.

6.4 Gould Colorwriter

The Gould Colorwriter plotters (models DS7 and DS1Ø) are connected, via the "P1" jack and optional RS-232C interface, to a serial I/O port. A three-wire cable should be sufficient; it should connect the following pins between the plotter and the computer:



Note that some pins are strapped together in the connector at one end of the cable, without running them to the other end.

Set the plotter for 96ØØ baud communication, using jumper position 1 on the RS-232C interface card. Switches SW3 on this card must also be set for 7 data bits, even parity, and one stop bit, as follows:

SW3: 1,3 Closed
2,4 Open

6.5 Watanabe / Western Graphtec MP1000

The Watanabe / Western Graphtec model MP1000 plotter connects to a serial I/O port via its RS-232C interface (model MP1000-01). A three-wire cable should be sufficient; it should connect the following pins between the plotter and the computer:

Watanabe end	Computer end
2	3
3	2
	4
	5
7	7
	6
	20

Note that some pins are strapped together in the connector at one end of the cable, without running them to the other end.

Set the plotter for 9600 baud communication, XON/XOFF protocol, Request to Send constantly on, 7 bits, even parity, and one stop bit. These parameters are established using the following settings of the ten switches on the back of the plotter:

Switches	1,4,7,9,10	OFF
Switches	2,3,5,6,8	ON

ON means that the switch is pushed down.

6.6 Sweet-P Model 100

The Sweet-P model 100 plotter by Enter Computer, Inc. connects to a Centronics-type parallel I/O port. We suggest that you use a cable specifically designed to match the Sweet-P to your IBM Personal Computer. Contact your Sweet-P dealer or Enter Computer, Inc. to obtain the proper cable.

Follow the instructions in the Sweet-P operator's manual to insert the pen and paper. AutoCAD expects the plotter to be at the "upper right" position when plotting begins.

6.7 Calcomp Models 81 and 84

The Calcomp model 81 and 84 plotters connect to an RS-232C serial I/O port. A three-wire cable should be sufficient; it should connect the following pins between the plotter and the computer:

Calcomp end		Computer end
2	----------	3
3	----------	2
	|---	4
	|---	5
7	----------	7
	|---	6
	|---	20

Note that some pins are strapped together in the connector at one end of the cable, without running them to the other end.

AutoCAD runs the model 81 plotter at a data rate of 2400 baud, and the model 84 at 9600 baud. Set the plotter for the proper baud rate, with XON/XOFF protocol, Request to Send constantly on, 7 bits, even parity, and one stop bit. On the model 84, these parameters are established using the following settings of the ten switches on the back of the plotter:

Switches	1,2,3,4,5,6,7,10	ON
Switches	8,9	OFF

ON means that the switch is pushed UP. The paper size switch should be set to "US" for ANSI "A" size drawings. Note that pen stall number one is the stall nearest the BACK of the plotter.

6.8 Roland DG Plotters

AutoCAD supports the Roland DG plotters, models DXY-800 and DXY-101, using either a Centronics-type parallel I/O port or an RS-232C serial I/O port. If a serial port is used, the plotter's DIP switches should be set for 9600 baud, even parity, and one stop bit, as follows:

Off	On
1,2,3,4,5,7	6,8,9,10

When using the Centronics parallel port, a standard parallel printer cable should be used to connect the plotter. To use the serial port, a cable with the following connections should be used.

Roland end		Computer end
2	----------	2
3	----------	3
4	----------	4
5	----------	5
6	----------	6
7	----------	7
20	----------	20

Operation of the Roland plotter is simple and straightforward; follow the instructions in the Roland manual. Remember to manually home the pen carriage to the bottom left before turning on power to the plotter. If you forget, the origin will be incorrect and the DXY-800 won't be able to change pens.

The DXY-800 multi-pen plotter has no command to put the pen back at the end of a plot. To compensate for this, AutoCAD will select pen number 8 when a plot is complete. If the stall for pen number 8 is left empty, this will have the effect of putting the last-used pen back in its stall and leaving the pen chuck empty. If you have a pen in stall number 8, it will be grabbed by this maneuver. You can cause the plotter to put this pen away by pressing the plotter's HOME button.

6.9 IBM Models XY/749 and XY/750

AutoCAD does not have a custom handler for the IBM XY/749 and XY/750 digital plotters. However, these plotters are equivalent to others for which handlers are provided. Refer to the guidelines for the equivalent plotter, elsewhere in this chapter, and use that plotter's name in all interactions with AutoCAD's Configurator.

IBM model	Equivalent plotter
XY/749	Calcomp model 84
XY/750	Calcomp model 81

Chapter 7

IBM PC "Work-alikes"

WARNING!

There are several computers which are "compatible" with the IBM PC to some degree. Unfortunately, the degree varies from model to model, and Autodesk, Inc. cannot guarantee that the IBM PC version of AutoCAD-86 will operate on every machine that claims to be a PC "work-alike". AutoCAD is a very large program, and addresses the graphics memory and maniuplates the I/O ports of the IBM PC at a very basic hardware level; if the "work-alike" computer does not behave exactly the same as an IBM PC in such areas as graphics, I/O, and memory, AutoCAD may work improperly, if at all.

Now that we've gotten that off our chests, we can get on with the topic at hand. We have successfully run AutoCAD on several of the PC "work-alikes", although some of them require special provisions. The following sections will document what is currently known about running AutoCAD on various PC-ish machines.

7.1 Columbia

AutoCAD runs without difficulty on the Columbia. Any of the video display configurations (including the Hercules Graphics Board and the Vectrix VX384) may be used.

7.2 Eagle PC-2

In order to use the Eagle PC high-resolution monochrome display (750 x 352), the special Eagle version of AutoCAD must be used. The other available display boards (Hercules, Tecmar, etc.) will work on the Eagle PC using either the Eagle or IBM versions of AutoCAD.

The Mouse Systems Mouse will not work on the Eagle PC.

On some Eagle PCs with early serial numbers, a transition from graphics display to text display takes a long time. This affects AutoCAD and other programs. New ROMs are available from Eagle to fix this.

The Eagle monochrome display causes a considerable amount of interference ("snow") as pictures are drawn on the screen. This is a function of the Eagle display hardware and may be improved in the near future.

A few of the special-function keys used by AutoCAD differ from those on the IBM PC.

AutoCAD function		Eagle PC key
FAST CURSOR	=	SHIFT-Up arrow
SLOW CURSOR	=	SHIFT-Down arrow
LEFT CURSOR	=	Alt-Backspace
ABORT CURSOR	=	F8 (END)
TOGGLE COORD	=	F6 or Ctrl D
TOGGLE GRID	=	F7 or Ctrl G
TOGGLE ORTHO	=	F14 (SLCT) or Ctrl O
TOGGLE SNAP	=	F9 or Ctrl B
TOGGLE TABLET	=	F10 or Ctrl T

7.3 Eagle 1600

The special Eagle version of AutoCAD must be used to run the Eagle 1600.

The Eagle high-resolution monochrome board and the Hercules board may be used on the 1600. All other display boards are currently unsupported. In order to use the Hercules display, the Eagle must have one of the newer 2-knob monitors (Model 16EM12-P39-4). The older, 3-knob monitor will not work with the Hercules display board.

The Mouse Systems Mouse and the Microsoft Mouse will not work on the Eagle 1600.

The Eagle monochrome display causes a considerable amount of interference ("snow") as pictures are drawn on the screen. This is a function of the Eagle display hardware and may be improved in the near future.

Also, the Eagle 1600's RS-232C serial I/O port appears like a modem to other equipment, whereas the IBM PC's (and Eagle PC's) serial port acts like a terminal. The digitizer and plotter cabling diagrams in this guide assume "terminal" mode for the computer's serial port; when connecting to the Eagle 1600's serial port, reverse the connections to pins 2 and 3 at the computer end of the cable.

A few of the special-function keys used by AutoCAD differ from those on the IBM PC.

AutoCAD-86 / IBM PC -- IBM PC "Work-alikes"

AutoCAD function		Eagle 1600 key
FAST CURSOR	=	SHIFT-Up arrow
SLOW CURSOR	=	SHIFT-Down arrow
LEFT CURSOR	=	Alt-Backspace
ABORT CURSOR	=	F8 (END)
TOGGLE COORD	=	F6 or Ctrl D
TOGGLE GRID	=	F7 or Ctrl G
TOGGLE ORTHO	=	F14 (SLCT) or Ctrl O
TOGGLE SNAP	=	F9 or Ctrl B
TOGGLE TABLET	=	F10 or Ctrl T

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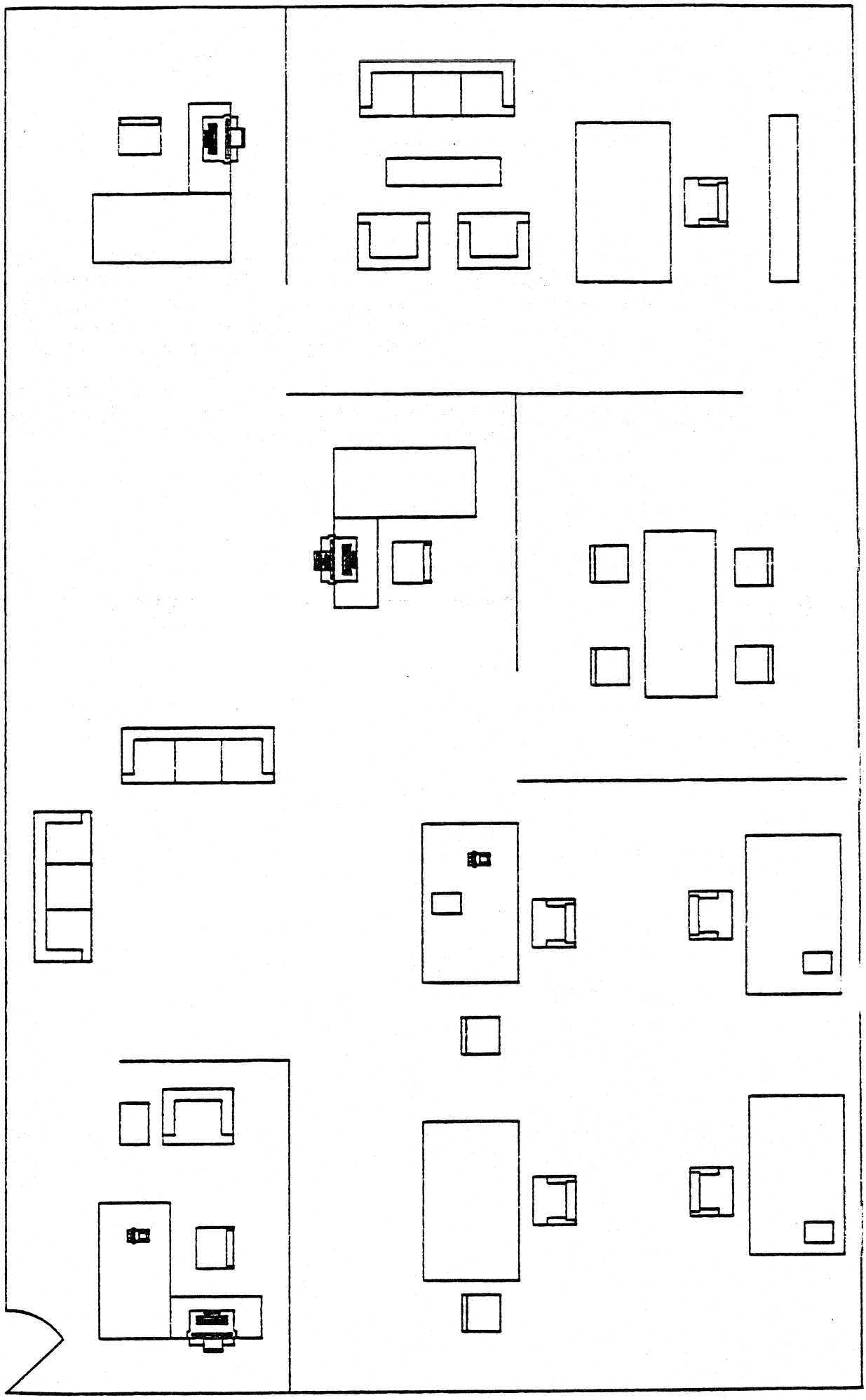
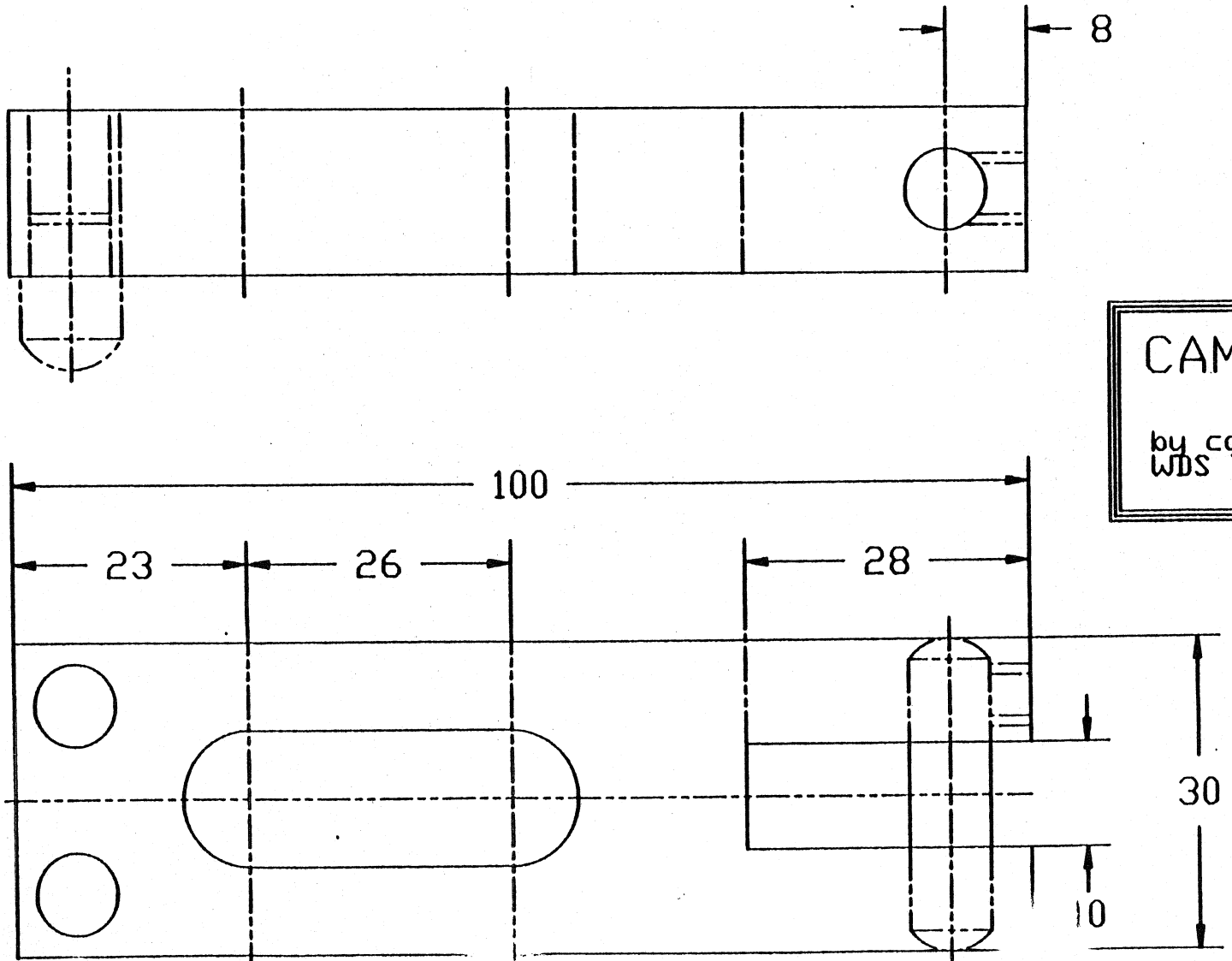


AutoCAD™ Menu

1	2	3	4	5	6	7	8	9	10
11	12	13	14	15	16	17	18	19	20
21	22	23	24	25	26	27	28	29	30
31	32	33	34	35	36	37	38	39	40

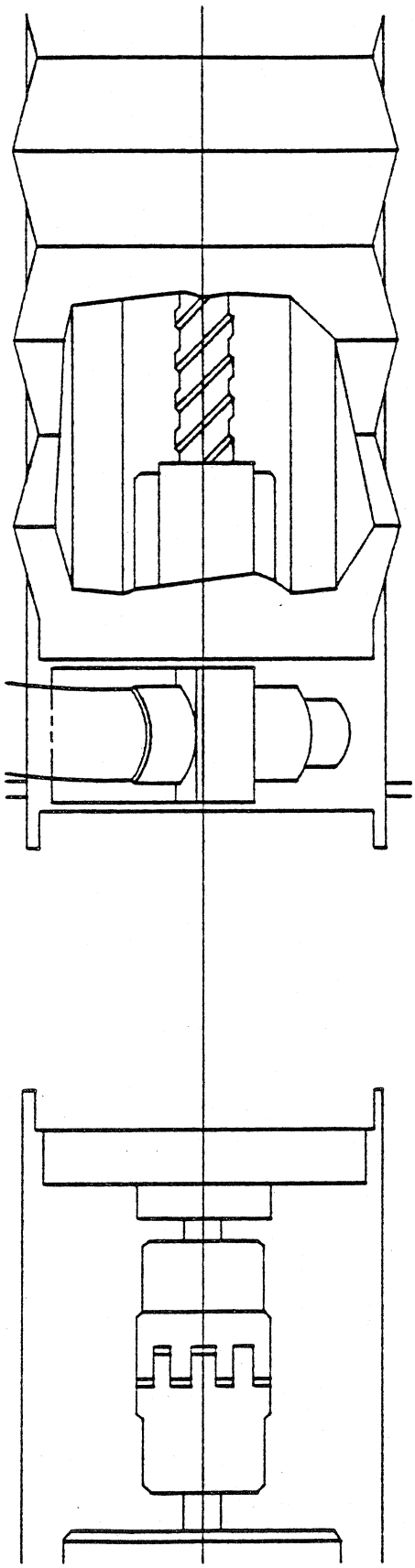
AutoCAD™ Menu

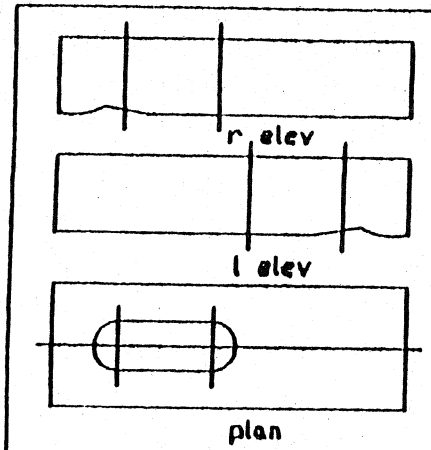
1	2	3	4	5	6	7	8	9	10
11	12	13	14	15	16	17	18	19	20
21	22	23	24	25	26	27	28	29	30
31	32	33	34	35	36	37	38	39	40



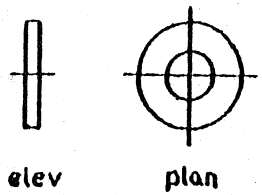
CAM CLAMP 212

by courtesy of
WDS TOOLING AIDS LTD

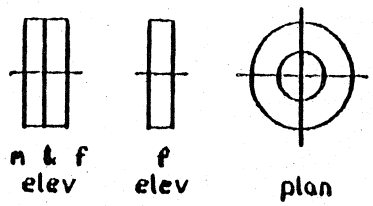




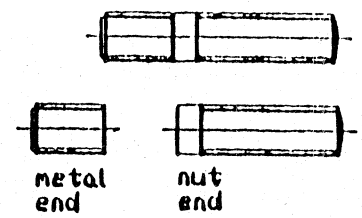
WDS202 CLAMP



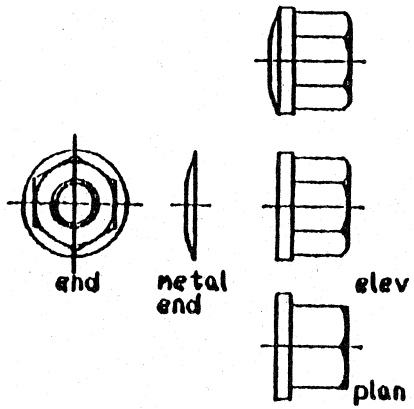
WDS400 WASHER



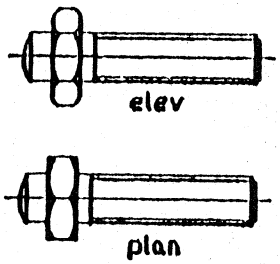
WDS401 SPH. SEATING WASHER



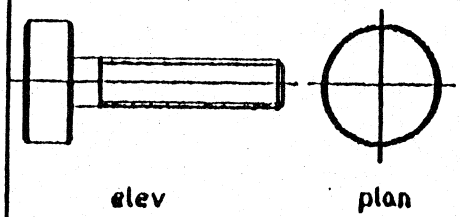
WDS405 STUD



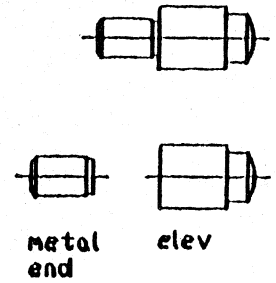
WDS407 SPH. SEATING NUT



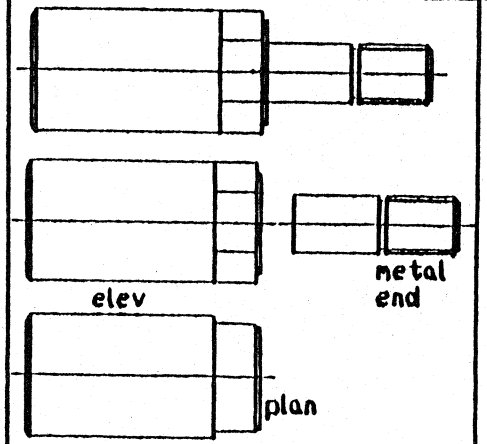
WDS601 LOCATING BUTTON



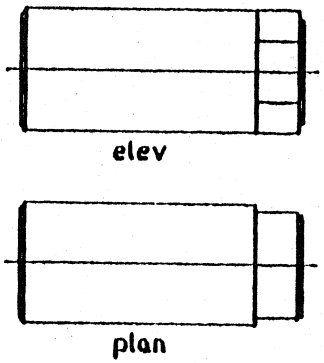
WDS603 ADJUSTING SCREW



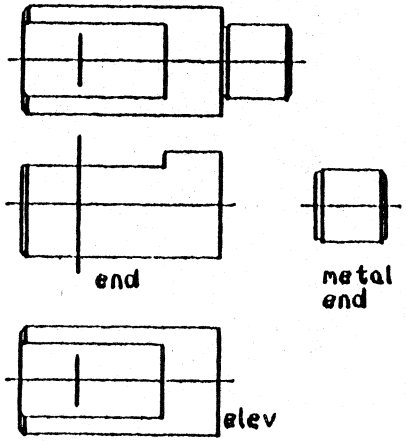
WDS607 HEEL PIN



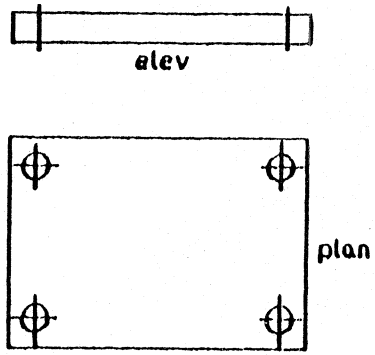
WDS626 FEET BOLT



WDS627 FEET NUT



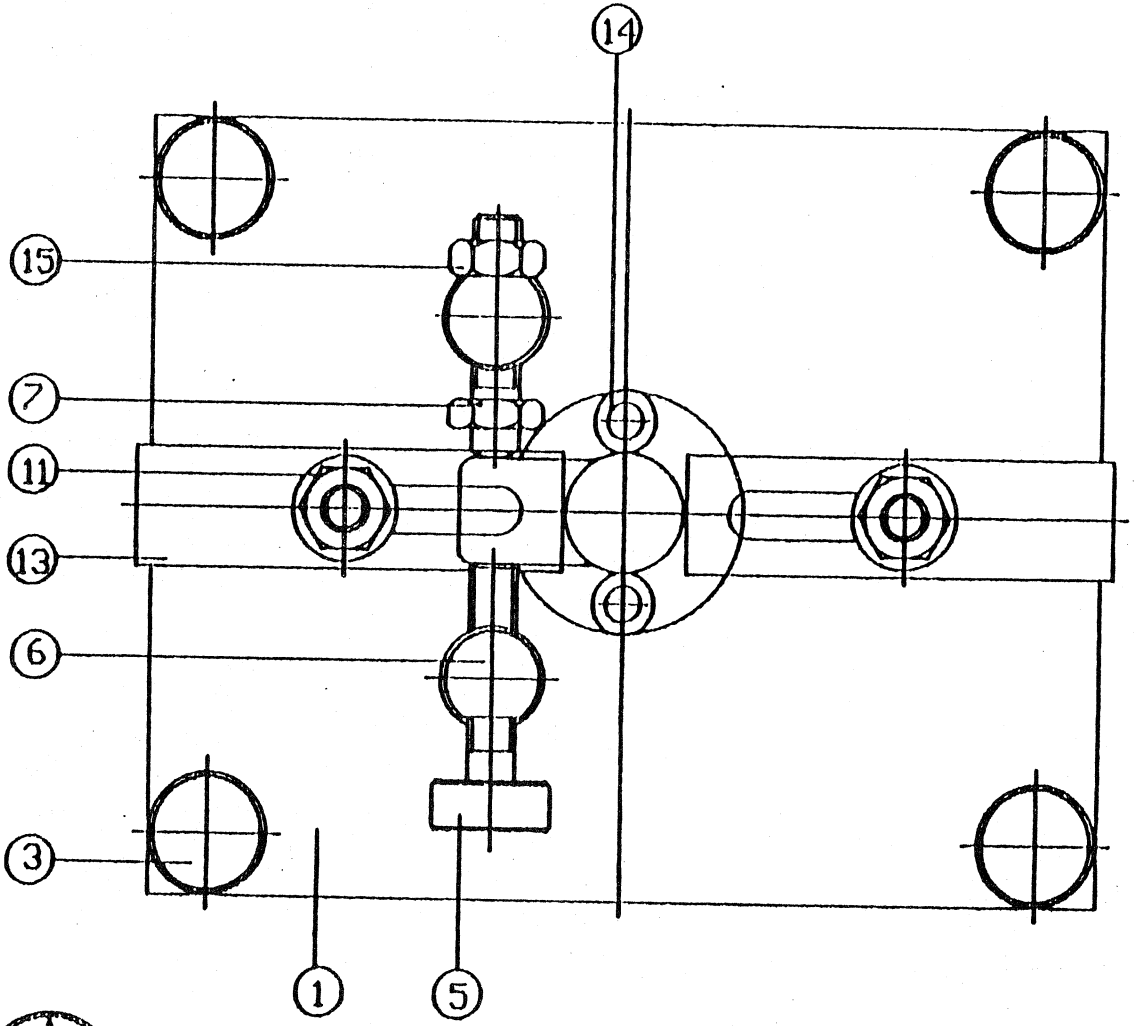
WDS632 PILLAR



WDS902 JIG PLATE

DO NOT SCALE

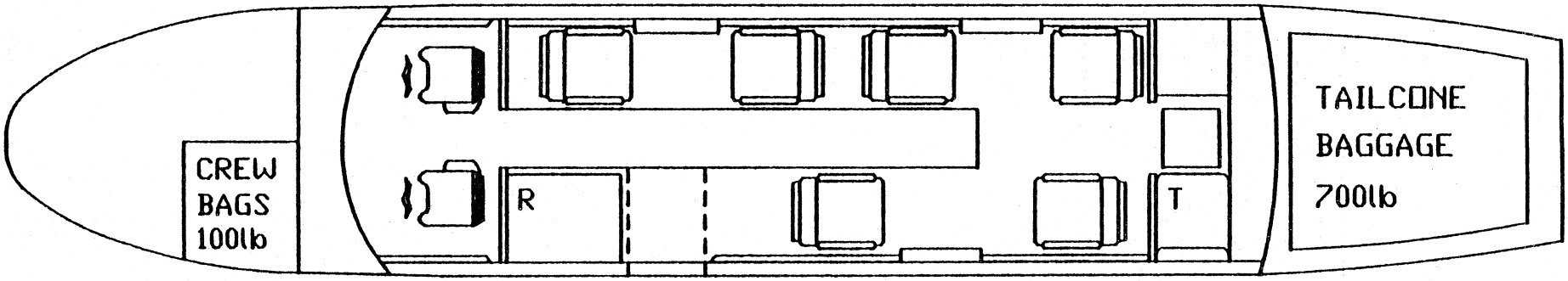
MODIFICATIONS		
No.	ALTERATION	DATE



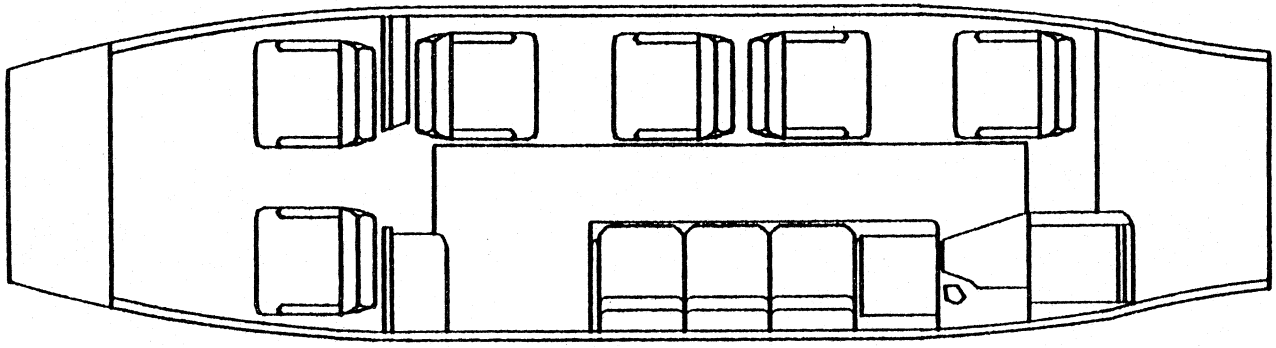
ITEM	QTY	MAT	DESCRIPTION	REMARKS
15	1	STK	HEXAGON NUT	M10
14	2	STK	DRILL BUSH	STD 7.5 DIA DRILL
13	2	STK	CLAMP	WDFS 202-204
12	2	STK	SPH SEATING WASHER	WDS 401-203(F)
11	2	STK	SPH SEATING NUT	WDS 407-033
10	2	STK	STUD	WDS 405-203 x 80
9	8	STK	WASHER	WDS 400-203
8	1	STK	CLAMP SPRING	WDS 640-203
7	1	STK	LOCATION BUTTON	WDS 601-204
6	2	STK	PILLAR	WDS 632-816
5	1	STK	ADJUSTING SCREW	WDS 605-203 x 60
4	8	STK	HEEL PIN	WDS 607-202 x 15
3	4	STK	JIG FEET BOLT	WDS 626-208 x 75
2	4	STK	JIG FEET NUT	WDS 627-208 x 30
1	1	STK	JIG PLATE	WDS 902-407

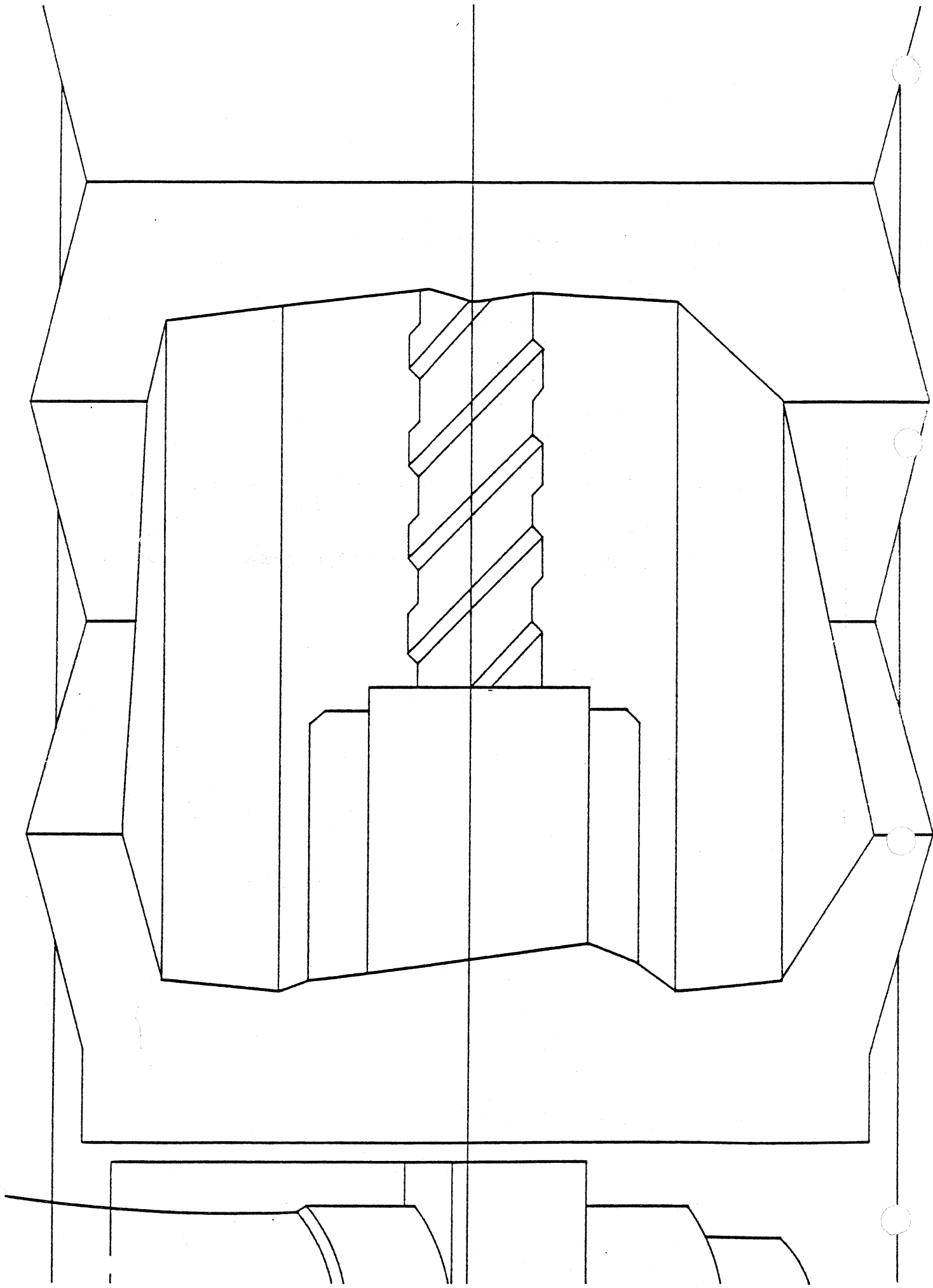
WDS TOOLING AIDS WOODSIDE WORKS LEEDS LS13 1EH	DRN R W BAKER CHK G EVANS APP P BLACK SHT No. 1 No. SHTS 3
DESCRIPTION	WDS NO.
DRILL JIG	WTC-3

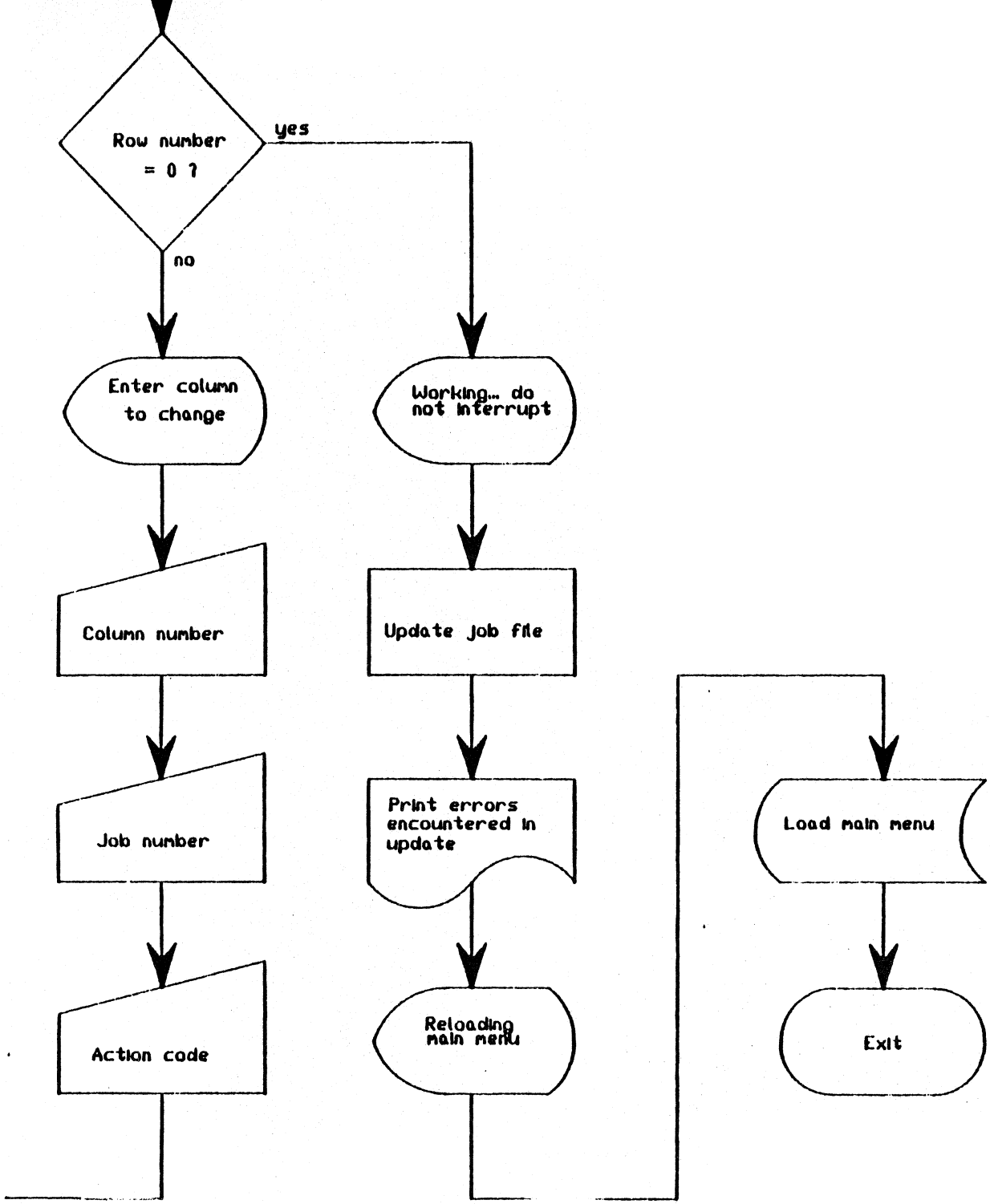
CITATION III

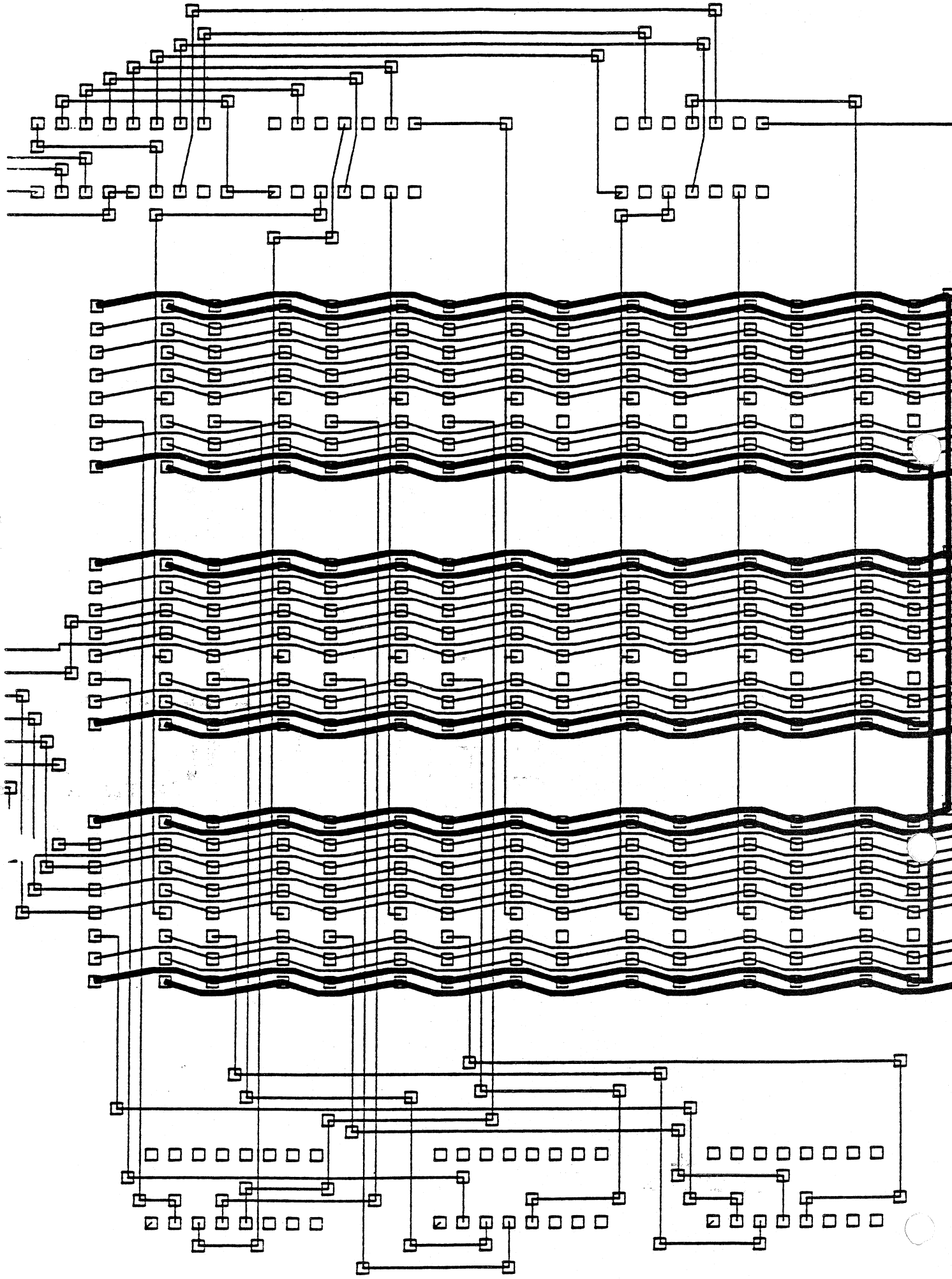


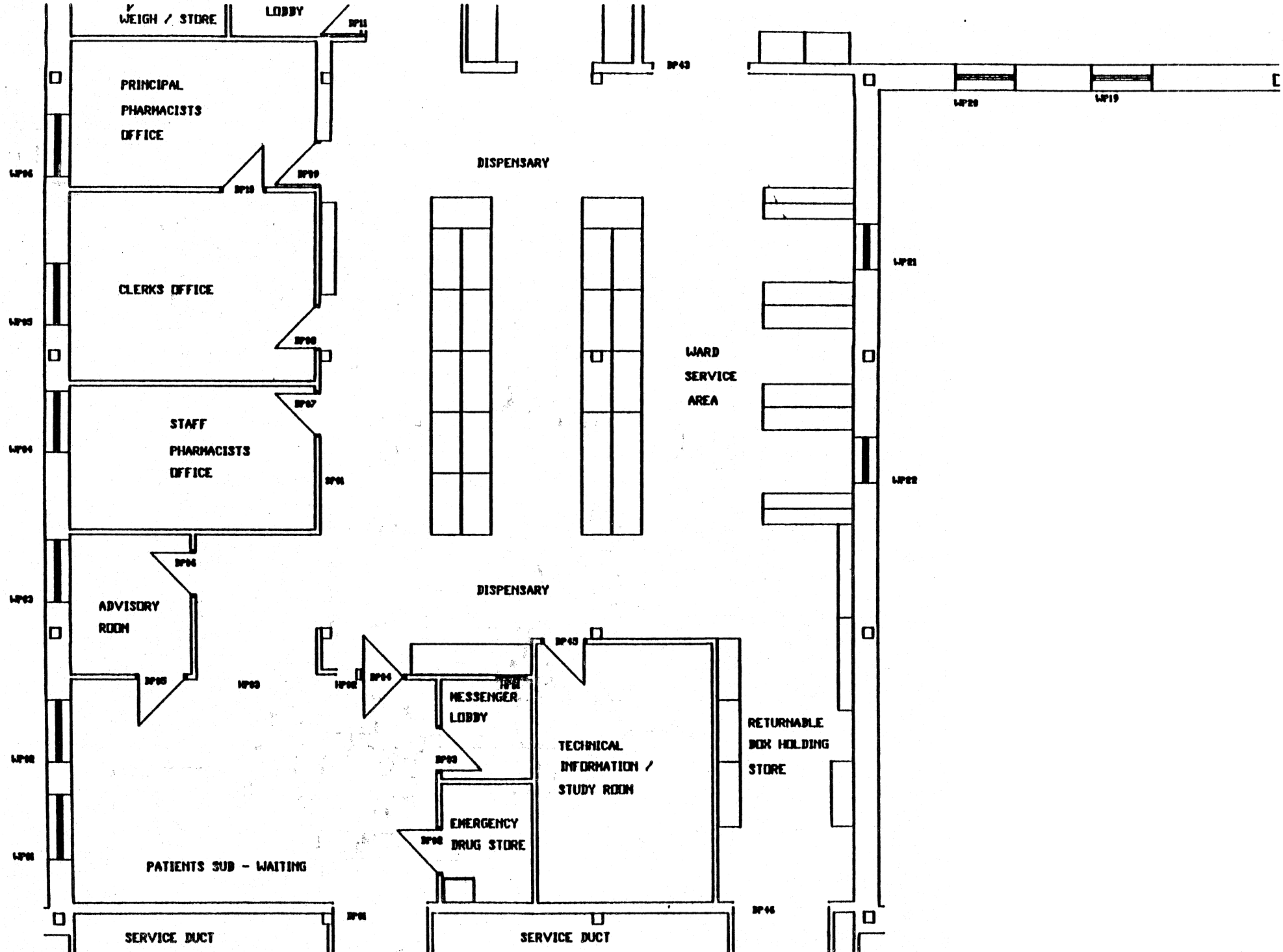
LEAR 55






Row number
= 0 ?
yes
no
Enter column
to change
Working... do
not interrupt
Column number
Update job file
Job number
Print errors
encountered in
update
Load main menu
Action code
Reloading
main menu
Exit




WEIGH / STORE
LOBBY
PRINCIPAL PHARMACISTS OFFICE
CLERKS OFFICE
STAFF PHARMACISTS OFFICE
ADVISORY ROOM
PATIENTS SUB - WAITING
SERVICE DUCT
DISPENSARY
DISPENSARY
WARD SERVICE AREA
MESSENGER LOBBY
TECHNICAL INFORMATION / STUDY ROOM
EMERGENCY DRUG STORE
RETURNABLE BOX HOLDING STORE
SERVICE DUCT