PC-CAPS™

User's Manual

Schematic Capture

September 1986

p-cad™

PERSONAL CAD SYSTEMS INC.

000-0060-05

COPYRIGHT

Personal CAD Systems, Inc. provides this manual "as is" without warranty of any kind, either expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. P-CAD may make improvements and/or changes in the product(s) and/or the program(s) described in this manual at any time and without notice.

Although P-CAD has gone to great effort to verify the integrity of the information herein, this publication could contain technical inaccuracies or typographical errors. Changes are periodically made to the information herein. These changes will be incorporated in new editions of this publication.

TRADEMARKS

P-CAD, PC-CAPS, PC-CARDS, PC-LOGS, PC-BACK, PC-DRC/NLC, PC-DRILL, PC-FORM, PC-LINK, PC-MODEL, PC-NODES, PC-PACK, PC-PHOTO, PC-PLACE, PC-PLOTS, PC-PRINT, PC-ROUTE, POSTSIM, PREPACK, and PRESIM are trademarks of Personal CAD Systems, Inc. (P-CAD).

PREFACE

This manual describes PC-CAPS, P-CAD's schematic capture program. This manual is designed to serve as a learning aid and as a reference manual.

Be sure you first read the *P-CAD Installation Guide* to find out what you need to do before using PC-CAPS, and then read the *System Overview* manual to find out how PC-CAPS interacts with other P-CAD programs.

Chapter 1, INTRODUCTION, provides an overview of PC-CAPS, a schematic capture program. This chapter also describes P-CAD conventions. Read this chapter first to become familiar with PC-CAPS.

Chapter 2, USING PC-CAPS, shows you how to use PC-CAPS to design a schematic. It is divided into two parts. Read Part A for an overview of the main commands and concepts. Then, for step-by-step hands-on practice, use the examples in Part B.

Chapter 3, COMMANDS, provides step-by-step instructions for each PC-CAPS command. When you start your own design activity, use this chapter to refer to commands you are not familiar with.

Chapter 4, DOS REFERENCE, describes what you need to know about the Disk Operating System (DOS) to use P-CAD software.

Appendix A, ERROR MESSAGES, provides a list of error messages that may appear on the screen, the cause of each error, and appropriate user action.

Appendix B, EXECUTABLE FILES, provides information that is not necessary to know before using the system, but may be helpful to advanced users. We recommend that you become familiar with your system before working with executable files.

Appendix C, USING PC-CAPS WITH A DIGITIZER, describes how to configure your system for digitizing, how to set up the digitizer, and how to use the digitizer to enter a drawing.

Appendix D, COLOR SELECTION, describes the default colors for PC-CAPS and how to change them. This appendix is intended for expert users.

Appendix E, THE ADD_DOTS PROGRAM, describes how to use the P-CAD ADD_DOTS program, which adds solder dots to a schematic that was created using an earlier than 1.3 version of PC-CAPS.

An index is also provided at the back of this manual.

NOTATION

This manual gives step-by-step procedures and examples. To make it easy for you to follow these procedures, we use the following notation.

<xxxx> Angle brackets around lowercase letters indicate a variable name that may be entered by the system or by you. For example:

<filename>.SCH

[ ] Square brackets indicate the name of a key. For example:

[Return]

[Return] [Return] indicates the key that is used to execute a command or accept an option. This key may be labeled differently, depending on your system. For example:

[RETURN], [←┘], [Enter], [Enter ←┘], [ENTER].

[ ]-[ ] Square brackets connected with a hyphen indicate keys that must be pressed simultaneously. For example:

Press [Ctrl]-[Alt]-[Del].

UPPER Uppercase letters indicate a command or an element that must be typed as shown. For example:

Type PCPLOTS and press [Return].

/ A forward slash separates main menu and submenu command combinations. For example:

DRAW/ARC

A forward slash also begins a keyboard command. For example:

/EXE

* An asterisk in a filename or in a filename extension indicates that any character(s) can occupy that position and all the remaining positions in the filename or extension. For example, the DOS command

DIR *.SYM

displays a list of all the filenames with the .SYM extension in the current directory.

Select Select means move the cursor with the mouse or the arrow keys to a command in the menu or to a point in the drawing area and press Button 1 or the space bar.

CONTENTS

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

FIGURES

FIGURES (Continued)

FIGURES (Continued)

FIGURES (Continued)

TABLES

CHAPTER 1. INTRODUCTION

This chapter introduces you to Personal CAD (P-CAD) Systems' PC-CAPS and describes P-CAD conventions.

OVERVIEW

PC-CAPS is a schematic capture program that runs on personal computers. PC-CAPS is an integral part of P-CAD's electronic design automation systems.

With PC-CAPS, you can draw and edit schematic diagrams and create a database which may be used to generate plots and connectivity netlists. For systems designs, netlists can be packaged into physical components and connections for input into PC-CARDS, P-CAD's printed circuit board (PCB) design program. For semi-custom IC designs, the gate level logic can be tested with PC-LOGS, P-CAD's logic simulation program.

Using P-CAD's integrated family of utility programs, you can link multiple schematic sheets into a single netlist and print wire, node, and component lists, and a bill of materials. Data added or changed during physical PCB layout, such as reference designators and gate swaps, can be automatically back-annotated to the schematic.

Netlist translators which interface to other simulators and CAD systems are also available, enabling you to integrate P-CAD into your existing CAE/CAD environment.

The key features of PC-CAPS include:

- **Hierarchical design.** Schematics are drawn in detail mode. A symbol can be drawn in symbol mode that represents the detail of the circuit. That symbol can then be used as a component in other higher level schematics. PC-CAPS supports up to 15 levels of nesting and special commands for looking inside symbols drawn with hierarchy.

- **Intelligent database.** PC-CAPS is built around P-CAD's intelligent integrated database that continually keeps track of components and connectivities. Because the database is integrated with P-CAD's PC-CARDS program, end-to-end design can be achieved in one system.

- **Editing and viewing commands.** The on-screen menu includes a complete set of commands to draw, edit, rotate, copy, delete, move, and enter components and wires. Many of these commands operate on single elements or groups of elements. Viewing commands include capabilities to pan, zoom in and out, save and restore views, and view selected windows.

- **Error minimization.** PC-CAPS performs real-time continuity checking against wiring mistakes, and queries you before merging or shorting two nets.

- **Complete design documentation.** Any component or net may be named; if you don't name a net, PC-CAPS will automatically generate its own names. Also, text may be placed anywhere at different angles and sizes, and with left, right and center, top or bottom justification.

P-CAD CONVENTIONS

We strongly recommend that you follow our filename, layer structure, and symbol library conventions in your design work for compatibility with P-CAD libraries, programs, and future software releases. These conventions are described below.

Filenames

P-CAD recommends that you use the following filename extensions.

.SCH for schematic database files

.SYM for symbol database files

.PLT for plot files

.SDT for solder dot files

.SSF for special symbol files (solder dot definition files)

Also, when you name your files, use alphanumeric characters only. Some special characters, such as percent (%) and underscore (_), are not recognized by some P-CAD programs.

Layer Structure

Layers are like clear plastic sheets that are stacked on top of each other to form a complete drawing. So that you can selectively edit and plot portions of your design, you can assign each type of data to a different layer. In a schematic, for example, you draw interconnections between components on the WIRES

layer; pin names on the PINNAM layer; and attributes on the ATTR layer. You may view or plot layers individually or in combination with other layers.

Layers can be set up in either of two ways:

1. Viewing the PC-CAPS default layer screen using the VLYR command and changing and/or adding layers as desired.

2. Loading an existing <filename>.SCH or <filename>.SYM file that contains the desired layer structure using the FILE/LOAD command, and then erasing the graphics using the FILE/ZAP command.

When you create a schematic, we strongly recommend that you use the P-CAD layer names; some P-CAD programs look for data by layer name. Also, when you create additional symbols, use the P-CAD layer structure for your library so that the symbols you create are compatible with P-CAD library symbols.

P-CAD also provides standard size sheet files called ASIZE.SCH, BSIZE.SCH, CSIZE.SCH, DSIZE.SCH and ESIZE.SCH on the symbol library diskettes. The standard size sheet files include a border and a layer structure.

Table 1-1 shows the PC-CAPS default layer structure.

Table 1-1. PC-CAPS Default Layer Structure

Layer Position	Name	Pen	Status	Use
1	WIRES	1	ABL A	Interconnecting wires
2	BUS	1	ABL	Interconnecting busses/wires
3	GATE	2	ON	Symbol geometries (ANSI)
4	IEEE	2	OFF	Symbol geometries (IEEE)
5	PINFUN	3	OFF	Pin functions (IEEE)
6	PINNUM	1	OFF	Pin numbers
7	PINNAM	6	ON	Pin names
8	PINCON	4	ON	Pin connections
9	REFDES	2	OFF	Reference designators
10	ATTR	6	OFF	Visible attributes
11	SDOT	1	OFF	Connect dots
12	DEVICE	5	ON	Component name or cell number

Table 1-1 Continued

Layer Position	Name	Pen	Status	Use
13	OUTLIN	5	ON	Component outlines (used in semicustom designs)
14	ATTR2	6	OFF	Invisible attributes
15	NOTES	6	OFF	Notes/text/ documentation
16	NETNAM	4	OFF	Net/signal names
17	CMPNAM	5	OFF	Component instance names
18	BORDER	5	OFF	Drawing/schematic border

Scale and Sizing

Schematics are drawn and edited within a rectangular, gridded coordinate system. The distance between the grid points is measured in database units (DBUs), which specify the physical measurement for each unit of length for schematic layouts. In PC-CAPS, 1 DBU is 0.01 inch (10 mils) if plotted at 1:1 scale. The default grid is 10 DBUs in both x and y axes. You can change the grid to any increment of 1 DBU (0.01 inch).

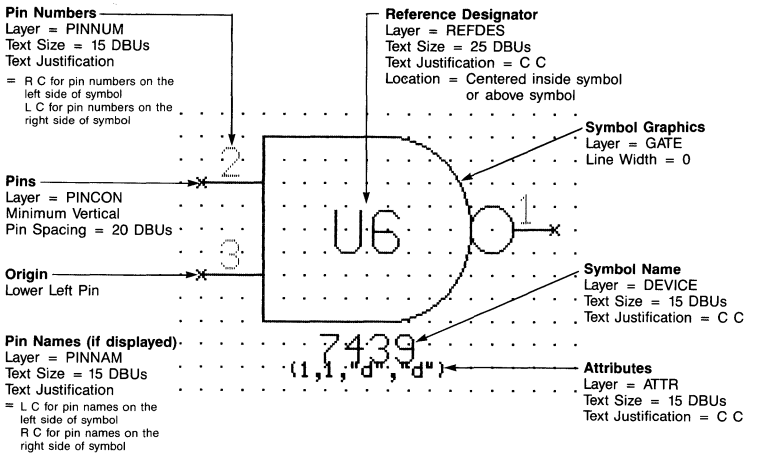
PC-CAPS provides virtual sheet capacity, which means that you may create a schematic at any size, or at standard sheet sizes. Due to this virtual sheet capability, you can enter components at 1/2 or 3/4 size as well as full size.

Libraries

P-CAD provides a growing selection of symbol libraries to increase the efficiency of your design activity.

P-CAD component libraries are based on ANSI standards for graphic symbol size, pin spacing, and text positioning. Where possible, IEEE symbol representations have also been included.

Figure 1-1 shows the basic specifications that P-CAD uses to design library symbols.



85037

Figure 1-1. P-CAD Symbol Specifications

For instructions on how to create P-CAD compatible symbols, see Chapter 2, "Using PC-CAPS."

CHAPTER SUMMARY

This chapter presented an overview of PC-CAPS and P-CAD conventions. The next chapter, "Using PC-CAPS," provides an overview of PC-CAPS commands and concepts and provides hands-on, step-by-step instruction and examples for using the program.

CHAPTER 2. USING PC-CAPS

This chapter explains how to use PC-CAPS to design schematics. It is divided into two parts.

Part A provides an overview to using PC-CAPS. As you read through this part, feel free to select menu commands and enter data into the drawing area. Then, if you want more practice, use the examples in Part B.

Part B provides the examples listed below with step-by-step instructions.

Example 1. Creating a Schematic
Example 2. Editing a Schematic
Example 3. Designing with Hierarchy
Example 4. Assigning Attributes
Example 5. Creating a P-CAD Compatible Symbol
Example 6. Creating a Bus
Example 7. Creating a Solder Dot and a Special Symbol File

PART A. OVERVIEW TO USING PC-CAPS

This part describes how to start the program, read the display, save and retrieve files, and use detail and symbol modes of operation.

STARTING THE PROGRAM

Before starting PC-CAPS, be sure that:

- The security device is correctly installed. (If it is not correctly installed, the system will display an error message.)
- The P-CAD directory structure is set up.
- The PC-CAPS program file (PCCAPS.EXE) is in the \PCAD\EXE directory.
- The AUTOEXEC.BAT, CONFIG.SYS, and PCADDRV.SYS files are in the root directory.
- The driver files (<filename>.DRV) are in the \PCAD\DRV directory.

  NOTE: For more information on the items listed above, see the *P-CAD Installation Guide*.

First, change the current directory to the project directory you will be working in. At the DOS prompt C:\>, type:

CD \PCAD\PROJ0 [Return]

Then, to start PC-CAPS at the **C:\PCAD\PROJ0>** prompt, type:

PCCAPS [Return]

The PC-CAPS Title Screen is displayed. Press any key to enter the PC-CAPS environment.

The PC-CAPS Opening Menu is displayed as shown in Figure 2-1.

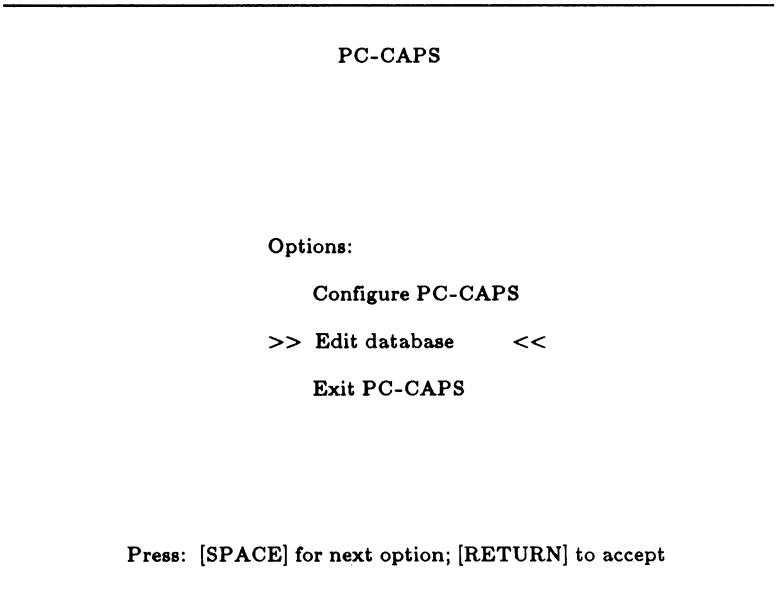


Figure 2-1. PC-CAPS Opening Menu

This screen provides three options.

Configure PC-CAPS - Allows you to call the Configuration Screen for adapting PC-CAPS to your specific system.

Edit database - Allows you to call the PC-CAPS program for creating and editing schematics.

Exit PC-CAPS - Allows you to return to DOS.

To move from one option to the next, press the space bar.

To accept and initiate the selected option, press [Return].

Select **Configure PC-CAPS** and the Configuration Screen is displayed as shown in Figure 2-2.

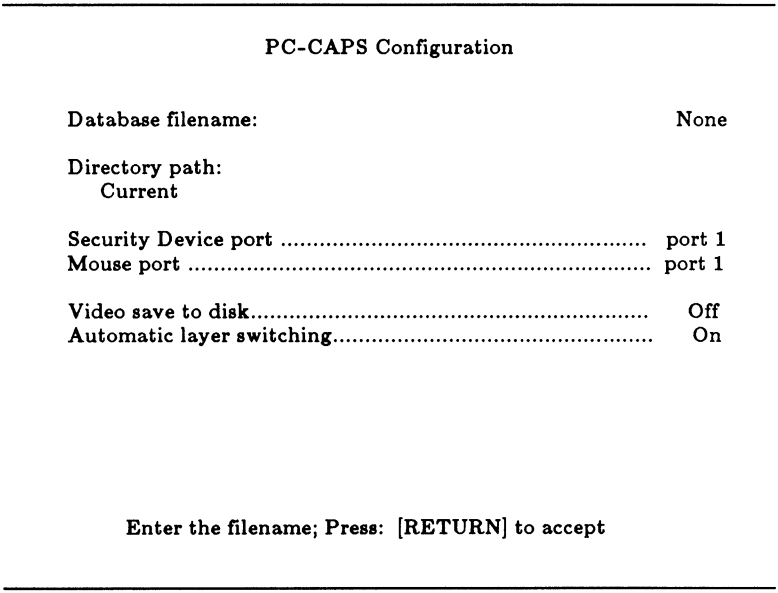


Figure 2-2. PC-CAPS Configuration Screen

The PC-CAPS Configuration Screen lets you select the following parameters.

Database filename - The file that is automatically loaded when the PC-CAPS program is started. For example, you can load an existing schematic here, or you can load a layer structure different from the default.

Note that you can enter a DOS path in front of the filename. The default is "None."

Directory path - The file directory that will be searched when you enter components or load schematic database files. For example:

\PCAD\SYM

NOTE: For P-CAD software releases 1.24 and up, you can enter more than one library directory. Use a semicolon to separate the libraries. For example:

C:\PCAD\SYM;C:\PCAD\SYM2

To continue an entry on the next line, end the line with a semicolon.

The default is "Current."

Security device port - The port the security device is connected to. The choices are "port 1" and "port 2." The default is "port 1."

Mouse port - The port the mouse is connected to. The choices are "port 1" and "port 2." The default is "port 1."

Video save to disk - Allows you to store a pixel image to disk when using certain commands such as VLYR, LPAN, MASK, and STO/MAP. The choices are "On" and "Off." The default is "Off." When you are working with a large database file, this feature saves time. For example, if the **Video save to disk** option is on, and you use the STO/MAP command to store a map view of your schematic, and then use the RCL command to recall the view, the MAP view comes up automatically; you don't need to wait for the system to redraw the screen. If the **Video save to disk** option is off, and you use RCL to recall the MAP view, you must wait for the system to redraw the screen before continuing.

Automatic layer switching - Allows you to turn off or on the automatic layer switching feature. The choices are "On" and "Off." The default is "On." If the **Automatic layer switching** option is on, any layer associated with a command is automatically set active when the command is selected. If the **Automatic layer switching** option is off, you set the layer you want active using the VLYR command or by toggling through the enabled layers from the status line.

To use PC-CAPS in this overview, press [Return] for **Database filename.** Then, press [Return] to accept the default for **Directory path.** Use the space bar to cycle through the choices for **Security Device port, Mouse port, Video save to disk, Automatic layer switching** and press [Return] to select a choice. After you set the last option, if you have changed any options, the system prompts:

Save this configuration? NO

To use this configuration for this program session only, press [Return] to select NO. To keep this configuration for all program sessions, type YES and then press [Return]. You can change these parameters later, if needed, by entering new data on the PC-CAPS Configuration Screen. The system returns you to the PC-CAPS Opening Menu.

NOTE: If you save this configuration, the system saves it to a file called PCCAPS.CFG in the current directory. This configuration is only active when the directory path is the current directory.

Select the **Edit database** option to enter the PC-CAPS environment.

THE DISPLAY

When you enter PC-CAPS, the screen display looks like Figure 2-3.

NOTE: If the security device is not correctly installed, the dotted grid is not displayed and the system displays the following message:

Security device not connected

The system then returns you to the PC-CAPS Opening Menu. See the *P-CAD Installation Guide* for instructions on installing the security device.

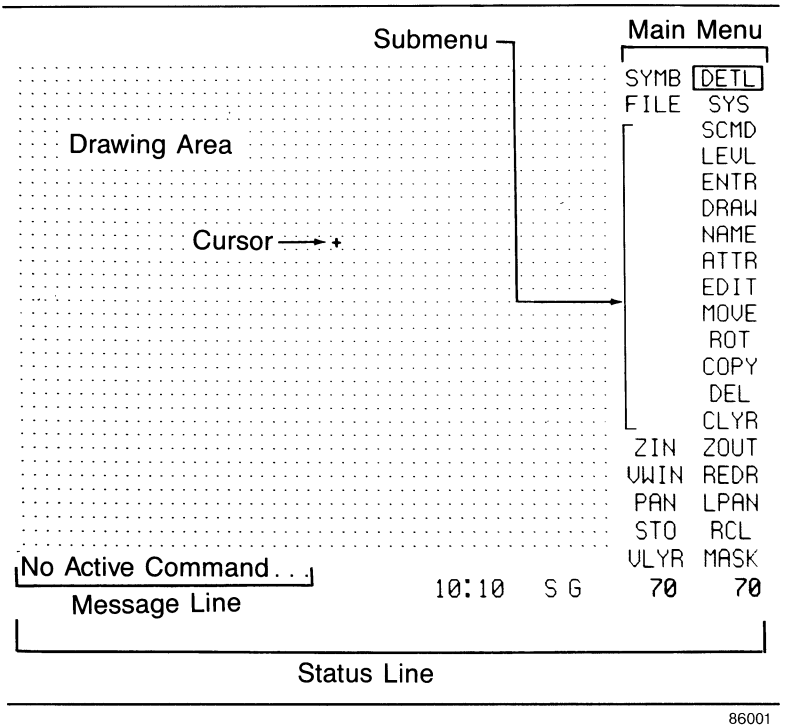


Figure 2-3. Initial Screen Display

Commands

Commands are displayed in the two columns on the right of the screen. There are two sets of commands that are invoked by the SYMB (Symbol) command and DETL (Detail) command at the top of the menu. The SYMB commands are used for drawing a symbol to be added to the symbol library and the DETL commands are used for drawing a schematic.

If a command has subcommands, the subcommands are displayed in yellow in the submenu area when the command is selected.

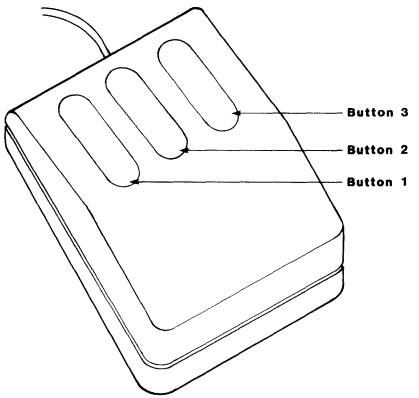
The viewing commands REDR (Redraw), VWIN (View Window), PAN, LPAN (Long Pan), ZIN (Zoom In), ZOUT (Zoom Out), STO (Store), and RCL (Recall) are nested commands and may be used while one of the operational commands is active.

Command nesting saves time. You can redraw the screen, specify a new view window, pan to another view of the screen, or zoom in or out from the screen without having to exit from a command function.

For example, while working in the detail (DETL) mode and placing components on the screen with the ENTR/COMP command, you can select PAN and center a new area on the screen. Press Button 2 to end the PAN command and you can then continue placing components; you don't have to call up the symbol again. You can also zoom in or zoom out, or specify a new view window and continue using the ENTR/COMP command without interruption.

Cursor

The small cross is the cursor. You can change the cursor display to full-screen crosshairs by typing an X on the keyboard and then change it back to the default cursor by typing another X. The cursor is controlled by the mouse shown in Figure 2-4 or by the arrow keys.



85038

Figure 2-4. Mouse

Mouse

Button 1 is the outer left button on the mouse. Use it to select a command or a point in the drawing area. Button 2 is the center button. Use it to end a command or to escape. You can also end a command by selecting another command from the menu. Button 3 is the outer right button. It performs the same function as Button 1.

Move the mouse, and the cursor on the screen moves with it. Notice that the cursor changes to a box as you move it over the command menu on the right or down to the status line on the bottom. You can also use the [F10] key to toggle the cursor location between the drawing area of the screen and the menu on the right. Note that the cursor returns to the same location where you left the drawing area even though you have moved the mouse.

To select a command from the menu, use the mouse to move the cursor over the command and press Button 1.

> NOTE: If you accidentally press [CTRL]-[S] on the keyboard, the mouse won't work until you press another key.

Arrow Keys

On the keyboard, the arrow keys move the cursor. The space bar functions as Button 1 and Button 3, and [Esc] functions as Button 2.

In addition, you can enter any of the menu commands from the keyboard by typing a slash (/). The system prompts:

Menu command:

Type the command and press [Return]. Then, if you want to select a submenu command, follow the same procedure: type a slash, type the command, and press [Return]. For example, you can type:

/SYS [Return]

and then for the submenu command, type:

/PLOT [Return]

to create a plot file.

Keyboard Commands

Some PC-CAPS commands must be entered from the keyboard. To enter one of these commands, type a slash (/). The system prompts:

Menu command:

Type the command and press [Return]. For more information on keyboard commands, refer to Chapter 3, "Commands."

Message Line

The message line is located at the bottom of the screen. The system displays prompts for input and displays error messages on the message line. All messages are displayed until you press any key or move the mouse.

Status Line

The status line is located at the bottom of the screen, even with the message line. The status line shows the current operating parameters, including active layer, current grid, cursor coordinates, and other information, depending on the command selected. You can change these parameters without having to exit from a command function.

You can use [F9] to toggle the cursor location between the drawing area and the status line. Note that the cursor returns to the same location where you left the drawing area even though you moved the mouse.

Drawing Area

The drawing area is 60,000 by 60,000 database units (DBUs). The current x,y coordinates of the cursor are always shown in DBUs in the right-most field on the status line. Although the scale of DBUs per inch is user-definable and may represent any length, the P-CAD libraries are built assuming 1 DBU = 0.01 inch.

If you use P-CAD library symbols and plot your schematic in 1:1 scale, the symbols will be approximately 0.7 inches high, and you can produce a schematic measuring 50 by 50 feet.

The grid is a design aid to ensure that lines are straight and symbols are lined up with each other. You can toggle the grid on or off at any time either by selecting the S on the status line or by pressing [F7]. The grid is on when the S is green and off when the S is red.

When you zoom out, the grid dots get closer together. When the dots are too close to be useful, they are suppressed, even though the grid display is on. The point at which they are suppressed depends on the resolution of your monitor and graphics card.

You can lock the grid (so that the cursor moves from grid point to grid point) by selecting the G on the status line or by pressing [F8] to toggle grid lock on or off. Grid lock is on when the G is green and off when the G is red. We recommend that you set grid lock on.

You can also change the spacing between the grid points by selecting the X:Y parameter on the status line. The default is 10:10 DBUs.

The viewing commands, PAN, LPAN (Long Pan), STO (Store), and RCL (Recall) allow you to move around the drawing area quickly.

For example, if you select LPAN, the entire drawing is shown with the current screen area indicated by the rectangle in Figure 2-5. This rectangle appears yellow on your display.

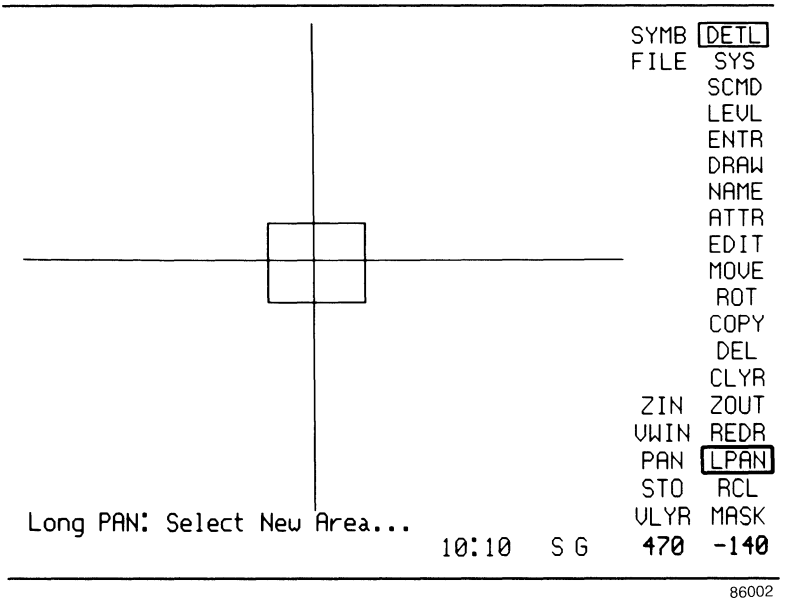
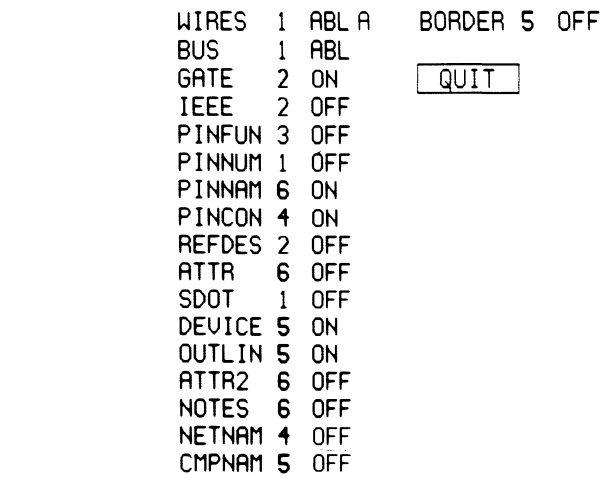


Figure 2-5. LPAN Screen Display

Layer Screen

Use the VLYR (View Layer) command to display the layer screen. The PC-CAPS default layer structure is displayed as shown in Figure 2-6.



86003

Figure 2-6. PC-CAPS Default Layer Screen

The left column shows the layer name.

The second column shows the layer color. The color column sets any layer to be one of up to 15 colors. If you have an IBM standard color card, there are only 3 colors available: red, green, and yellow. If you have a TI, there are 7 colors, and if you have a high-resolution graphics card in the IBM, all 15 colors are available.

You can use the numbers in this column to assign plotter pen colors.

The third column shows the layer status:

OFF - not visible
ON - visible and editable but not accessible
ABL - visible and may become the active (accessible) layer

To change the status of a layer, move the cursor to the status designation of the layer. Press Button 1 to cycle the status to OFF, ON, or ABL.

The fourth column shows the active (accessible) layer:

A - currently active layer

The active layer is the layer on which you place data. To make a layer active, move the cursor to the fourth column and press Button 1. An "A" appears in the fourth column making the layer active. Note that only layers with ABL status may be made active.

Fifty layers are available to permit showing or plotting pertinent information. To add layers, refer to the description of the VLYR command in Chapter 3, "Commands."

Use the QUIT command or Button 2 to exit the layer display and to return to the drawing view.

The active layer may be changed from the status line. If you set all the layers you will be using during a design session to ABL status and set the first layer you will be using to the active layer, then you can change the active layer as needed from the status line. Press [F1] or move the cursor to the active layer parameter on

the status line and press Button 1 until the desired layer is displayed.

When you select the ATTR/ACOM, ENTR/PIN, ENTR/WIRE, NAME/COMP, NAME/NET, or the SCMD/PNLC command, the associated default layer is set ABL and active automatically.

You can change the associated default layer of these commands for a design session, if desired. For example, if you select ATTR/ACOM, the ATTR layer is set active. You can use the VLYR command to change the active layer to ATTR2, and this layer will become the associated default layer for the rest of the design session.

SAVING AND RETRIEVING DATABASE FILES

Drawings created with PC-CAPS may be saved as schematic database files and as plot files.

The FILE/SAVE command stores the entire database file. PC-CAPS always "remembers" the parameters when a drawing is saved. When you load an existing drawing to continue work on it, the parameters will be set just as they were when you saved it.

> NOTE: If you do not save a file in which you were working (for example, if you accidentally use the FILE/ZAP command), you can recover the lost data by renaming and then replaying the command log file. Refer to Appendix B, "Executable Files," for more information.

The database file is left active on the screen after FILE/SAVE so that work can continue.

> CAUTION: If you don't have enough disk space for the schematic, you will not be able to save the file.

The SYS/STAT (System Statistics) command displays a list of internal storage usage and capacity for the database file currently displayed on the screen.

> CAUTION: A saved database file automatically overwrites the old file unless you enter a new filename.

The SYS/DOS command allows you to execute commonly used DOS commands from within the PC-CAPS environment.

The SYS/PLOT (System Plot) command stores a plot file of the drawing area. PC-PRINT and PC-PLOTS utility programs output the schematic diagram on a dot-matrix printer or a pen plotter, respectively.

The FILE/ZAP command can be used after a FILE/SAVE to erase the graphics data from the screen for a new project.

The SYS/QUIT command exits PC-CAPS.

The FILE/LOAD command loads a file from disk and displays it on the screen.

DETAIL MODE

Detail mode is used for creating and editing schematic diagrams. It is invoked by the DETL (Detail) command at the top of the menu. The main menu color is green when the system is in detail mode.

Creating a Schematic

The following section provides a general description of how to create a schematic diagram. For step-by-step instructions, see Example 1 in Part B of this chapter.

> NOTE: If you plan to interface to any other system, read the interface program manual to find out if there are any special attributes you need to assign or other special requirements in creating a schematic.

Setting Up the Environment

Make sure the system is set to DETL (detail) mode.

Placing Components

Use the ENTR/COMP (Enter Component) command to call symbols from the library.

The symbols may be scaled in size when called by entering a value for the optional "x/y scales" prompt. The "x/y scales" entry is expressed as a percentage of the original size. For example, if you enter <filename>.SYM 50/50, the symbol is entered at 50% of the original size.

Once called, a symbol may be placed in as many locations as desired. It may be placed in any of four rotations by toggling the F's on the status line until the appropriately positioned F is green, or it may be mirrored with the M parameter by toggling the M on the status line to green.

Components that have been placed may be duplicated with the COPY command, repositioned with the MOVE command, or rotated with the ROT (Rotate) command.

Wiring the Circuit

Use the ENTR/WIRE (Enter Wire) command to interconnect components with wires. (Components connected using the DRAW/LINE command may look the same on the screen, but the lines do not form nets that may be used for netlist extraction, simulation, or PCB design.) Wires are automatically placed on the WIRES layer.

Wires may be entered at 90 degree angles using the ORTH parameter, at any angle using the ANGL parameter, or at 45 degree angles using the 45D parameter on the status line. You can use [F2] to toggle among Orthogonal, Angle, and 45 degree angle modes of wire or line entry.

Wire-to-pin connection on the components is indicated by a small cross on each unconnected pin that disappears when a wire connection is made. The snap-to-pin feature snaps wires to the nearest pin location within the tolerance radius set with the /SGAT command. The snap-to-pin feature is enabled by toggling the L on the status line to green.

When you enter a wire that intersects another wire to form a "T," PC-CAPS automatically adds a default solder dot at the T-junction. Solder dots are automatically placed on the SDOT layer. When a T-junction is broken by moving or deleting a wire, the solder dot is automatically deleted. You can create your own solder dot and attach it to your schematic as described later in the section titled "Creating a Solder Dot."

You can also add solder dots to a schematic that was created using an earlier than 1.3 version of PC-CAPS by using the P-CAD utility program ADD_DOTS. For more information, refer to Appendix B, "ADD_DOTS."

Components and nets may be identified with names using the NAME/COMP (Name Component) and NAME/NET commands. The NAME/COMP command is used primarily in semicustom designs to identify components for simulation. Component names are automatically placed on the CMPNAM layer and net names are automatically placed on the NETNAM layer. You can use [F3] to assign the active net name while using the ENTR/WIRE command. If you don't name the components and nets, the system automatically names them for you. These names are used by other P-CAD programs such as PC-NODES, PC-LINK, and PC-FORM to generate a netlist of the components used in the schematic and the nets that are connected to those components.

Assigning Attributes

An attribute assigned in detail mode is text or numeric information about a component in a schematic. It is used as data for other P-CAD programs and report generation. A component may have many attributes.

You can add, delete, or change attributes of a component in a schematic using the ATTR (Attribute) commands. Attributes are automatically placed on the ATTR layer.

When you create symbols, we recommend that you add attributes to your symbols in the symbol mode and then, if desired, change the attributes of the symbols in your schematic in detail mode.

For a step-by-step example on assigning attributes using the ATTR commands, see Example 4 in Part B of this chapter.

Adding Packaging Information

You can pre-assign several logic symbols to the same physical package and/or you can assign a symbol to a specific gate in the package for use by PC-PACK using the SCMD/PNUM (System Command/Pin Number) command and the SCMD/PNLC (System Command/Pin Number Location) command. The SCMD/PNUM command is used in detail mode to pre-assign a reference designator and/or pin numbers to a schematic symbol. This command relies on the proper definition of the symbol using the SCMD/PNLC command. The SCMD/PNLC command is used in symbol mode and is described in Example 5 in Part B of this chapter.

Creating a Bus

A bus is a group of wires in a schematic that collectively transfer digital data, such as an Address Bus.

In PC-CAPS, a bus may be represented by a thick line drawn with the DRAW/LINE command. Place the bus line on the BUS layer. The line is for graphical representation only. Electrical connections are made with the ENTR/WIRE command and by naming each individual bus signal. Connectivity is established between net segments having the same signal name.

For a step-by-step example on how to create a bus, see Example 6 in Part B of this chapter.

Assigning Solder Dots

After you create a schematic, you can link a special symbol file that assigns your own solder dot graphics at T-junctions. These dots replace the default solder dots. Solder dots are described later in this chapter in the section titled "Creating a Solder Dot" and in Example 7 in Part B of this chapter. Special symbol files are described in Example 7 in Part B of this chapter.

Use the SCMD/GSSF (System Command/Get Special Symbol File) command to link the special symbol file (<filename>.SSF) to your layout. The special symbol file, in turn, links the solder dot file (<filename>.SDT) described in the special symbol file to your layout at T-junctions. Use the REDR (Redraw) command to change the display of default solder dots to that of the solder dot described in the special symbol file.

To unlink the special symbol file, use the SCMD/GSSF command and press Button 2 or [Esc] instead of entering a filename.

Creating a Plot File

After you have created the schematic, and attached the user-created solder dots (if desired), you can create a plot that is used by PC-PRINT or PC-PLOTS to output plots on a dot-matrix printer or a pen plotter, respectively. Use the SYS/PLOT command to create the plot file. For a step-by-step example on creating a plot file, see Example 1 in Part B of this chapter.

Saving the Schematic

Use the FILE/SAVE command to save the schematic.

Creating Multisheet Schematics

Multisheet schematics are created by producing a number of single sheet schematics and then instructing PC-LINK to link all the sheets together. See the *PC-LINK User's Manual* regarding this process.

In order to indicate that a given schematic is a single sheet, assign the SHEET attribute to the schematic. To do this, in symbol mode, use the ATTR/ACOM command and enter, anywhere in the design, the following attribute:

SHEET=<sheet id>

where <sheet id> is typically a number, with the first being 01 and so on. For example, you could assign the following attributes to three sheets that make up one design:

SHEET=01
SHEET=02
SHEET=03

The sheet id must be no more than three characters for each sheet.

NOTE: The NX-HILO interface program requires the identifier to be alphanumeric and to be no more than two characters long. If you use a two-character identifier, do not use 0 (zero) as the second character.

The sheet id in a multiple sheet schematic must be unique. The program attaches this sheet id to all unnamed nets and components within each sheet.

CAUTION: If you assign SHEET=1 to FILE1.SCH and SHEET=01 to FILE2.SCH, some unnamed nodes may be merged after linking both schematics by PC-LINK. PC-LINK interprets both sheet ids as 001.

Connections between sheets are indicated by giving connected wires the same net name on each sheet. Wires with the same net name on different sheets are considered connected.

Editing a Schematic

You can edit wires in a schematic using the following EDIT and DELETE commands:

EDIT/ADDV adds a vertex.
EDIT/DELV deletes a vertex.
EDIT/MOVV moves a vertex.
EDIT/LAYS changes the layer a segment is on.
EDIT/DELS deletes a segment.
EDIT/MOVS moves a segment.
EDIT/MOVA moves a wire segment or a vertex and all attached wire segments and solder dots on all layers.
DEL deletes a selected object. (Note that entire nets are deleted, not just segments.)
DEL/IDEN deletes a collection of individually identified objects.
DEL/UNDO restores the last deleted object when you are doing single deletions.
DEL/WIN deletes a collection of objects enclosed in or intersected by a window.

> CAUTION: When using the DELETE commands, some data can be deleted even if the layer that data is on is turned off. For more information, see the command descriptions for the DELETE commands in Chapter 3, "Commands."

In addition, you can uncommit a pin using the ENTR/UCOM (Enter/Uncommit a Pin) command. This command removes the connectivity of a pin and the pin is no longer connected to a net.

For a step-by-step example on editing a schematic, see Example 2 in Part B of this chapter.

SYMBOL MODE

Symbol mode is used for creating symbols and solder dots. It is invoked by the SYMB (Symbol) command at the top of the menu. The main menu color is red when the system is in symbol mode.

Creating a P-CAD Compatible Symbol

The following section provides a general description of how to create a P-CAD compatible symbol. For step-by-step instructions, see Example 5 in Part B of this chapter.

> NOTE: If you plan to interface to any other system, read the interface program manual to find out if there are any special attributes you need to assign or other special requirements for creating a symbol.
>
> If you plan to use PC-LOGS, refer to the *PC-LOGS Primitives Library* manual when creating primitives.

When you create new symbols, P-CAD recommends that you use its layer structure and relative sizes to be compatible with P-CAD library symbols. See the section titled "Libraries" in Chapter 1, "Introduction," for P-CAD symbol specifications.

Setting Up the Environment

Select the SYMB command to set the system to symbol mode.

Drawing the Graphics

The DRAW commands are used to create the graphics. For example, use the DRAW/LINE command to draw lines representing inputs, outputs, and other connections. Place graphics on the GATE layer.

Entering Input and Output Pins

Use the ENTR/PIN (Enter Pin) command to specify the location, name, and function of each pin. Change the Input-Output-I/O parameter on the status line before entering the pin to correctly identify the pin type of each pin. Pins are automatically placed on the PINCON layer.

After you enter a pin, the system prompts you for a pin name. You can enter the pin name and make the name visible on the screen, or you can press Button 2 or [Esc] to make the name invisible.

Assigning the Origin

Use the ENTR/ORG command to assign the origin to the lower left pin.

Adding the Device Name

Use the DRAW/TEXT command to add the symbol name. Place the symbol name on the DEVICE layer.

Assigning Attributes

An attribute assigned in symbol mode is text or numeric information about a symbol. You can add, delete, or change attributes of a symbol using the ATTR (Attribute) commands. For example, you can assign an attribute to a symbol, such as TECH=TTL, as described in Example 4 in Part B of this chapter. Attributes are placed automatically on the ATTR layer.

Adding Packaging Information

You can pre-assign packaging for use by PC-PACK or for listing pin number references in the PC-FORM reports using the SCMD/PNLC (System Command/Pin Number Location) command. This command lets you assign the location of the reference designator and the pin numbers used during pre-assignment. It can also be used to specify the number of gates to be assigned to each physical package and the pin numbers that correspond to each gate location. In detail mode, when you select the SCMD/PNUM (System Command/Pin Number) command, the system relies on the proper definition of the symbol as described above. Pin

numbers are placed automatically on the PINNUM layer and reference designators are placed automatically on the REFDES layer. For a step-by-step example using the SCMD/PNLC command, see Example 5 in Part B of this chapter.

Every symbol you create except nonhierarchical symbols that will not be simulated by PC-LOGS must be assigned a Component Type ID for use by other P-CAD programs. (Nonhierarchical symbols that will not be simulated are automatically assigned a Component Type ID of 255.) Use the SCMD/SCAT (System Command/Set Component Atrribute) command to assign the Component Type ID.

For symbols you create that will be simulated by PC-LOGS, refer to the *PC-LOGS Primitives Library* manual for the corresponding Component Type ID.

For hierarchical symbols you create, assign a Component Type ID of 256. A Component Type ID of 256 enables PC-LINK to "flatten" the hierarchical database. For more information, see the *PC-LINK User's Manual.*

For nonelectrical symbols, such as title blocks, logos, and solder dots that you don't want listed in the netlist, assign a value of 0. A Component Type ID of 0 prevents PC-NODES from extracting the symbol into the netlist.

For symbols you create that cannot be packaged, such as power and ground, but that you want listed in the netlist, assign a value of -1. A Component Type ID of -1 prevents PC-PACK from packaging the symbol.

Saving the Symbol

Use the FILE/SAVE command to save the symbol.

Creating a Solder Dot

A solder dot is a database file that defines the solder dot characteristics for pen plotting.

If you don't want to use the system default solder dot, you can create one of your own and attach it to your layout at T-junctions. This section provides a general description of how to create a solder dot. For a step-by-step example, see Example 7 in Part B of this chapter.

Setting Up the Environment

Select the SYMB command to set the system to symbol mode.

Because the solder dot is smaller than a symbol or a circuit, it must be drawn on a smaller scale than that of symbols or circuits. Adjust the current viewing scale so that the grid spacing is more convenient for this smaller-scale creation. Use the ZIN command to zoom in.

Turn the visibility status of the SDOT (solder dot) layer to ABL and A (active) using the VLYR command.

Drawing the Graphics

Use the DRAW commands, such as DRAW/CIRC or DRAW/RECT, to make the graphic representation on the SDOT layer.

Assigning the Origin

Use the ENTR/ORG command to set the origin to the center of the solder dot.

Setting the Component Type ID

Use the SCMD/SCAT command to assign a Component Type ID of 0 to the solder dot to prevent PC-NODES from extracting the symbol into the netlist.

Saving the Solder Dot

Use the FILE/SAVE command to save the solder dot.

After the solder dot is created, the special symbol file that contains the solder dot description can be created or edited. See Example 7 in Part B of this chapter for a step-by-step example.

DESIGNING WITH HIERARCHY

Detail and symbol modes allow designing with hierarchy.

You can draw a schematic diagram in detail mode and then draw a symbol in symbol mode that represents the detail of the circuit. You can then save the entire database file, call the representative symbol, and place it as a component when you are creating other higher level schematics.

The LEVL/PUSH (Level Push) command lets you look inside symbols drawn with hierarchy and the LEVL/POP (Level Pop) command lets you return to the overall schematic.

The system allows up to 15 levels of hierarchy.

For a step-by-step example on designing with hierarchy, see Example 3 in Part B of this chapter.

ENTERING TEXT

Text may be added to drawings using the DRAW/TEXT command. You may specify parameters on the status line at the bottom of the screen when you draw text. These parameters are:

- The layer the text is on
- The current size of text
- The horizontal and vertical justification of text
- The current orientation of text
- The mirroring of text

The layer that text is entered on may be changed by selecting the layer parameter on the status line.

The system text sizes are given in DBUs, the same as the coordinates. The size of the text that you enter may vary from the size of the text in your actual drawing, depending on the text dimensions specified for PC-PRINT or PC-PLOTS. You can change text size by selecting the SIZ:n (Text Size) parameter on the status line. Or, you can use [F5] to enter a new text size when using the DRAW/TEXT command.

The horizontal position of text relative to the cursor may be changed by selecting L, C, or R on the status line for left, center, or right, respectively.

The vertical position of text relative to the cursor may be changed by selecting T, C, or B on the status line for top, center, or bottom, respectively.

Text orientation may be changed by selecting one of the four F's on the status line. You can also use [F6] to cycle the orientation of text.

Mirror image orientation of text may be changed by selecting the M on the status line to toggle the setting to on (green) or off (red).

Negation may be specified by typing the text string desired, followed with an apostrophe (') and [Return]. A negation bar appears over the text string.

PART B. EXAMPLES

This part provides step-by-step instructions for using PC-CAPS. The examples are organized as follows:

These examples assume the example data files are loaded in the PROJ0 subdirectory.

To make sure the files are loaded, at the DOS prompt, C:\>, type:

```
DIR \PCAD\PROJ0
```

Your directory listing should include the following files:

NANDT.SYM	TUTORD.SCH
INV.SYM	74LS373.SYM
TUTORC.SCH	

NOTE: All the figures in the following examples were produced on a high-resolution monitor; a low-resolution monitor may produce differences in the appearance of the grid and the graphics.

When you load your example data files, you may need to zoom out using the ZOUT command or zoom in using the ZIN command, depending on the resolution of your monitor, to match the figures shown in the examples.

You may also need to change the current viewing window using the PAN command.

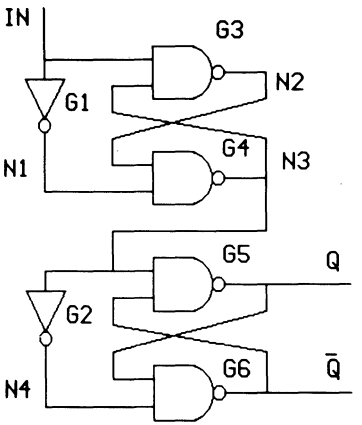
EXAMPLE 1. CREATING A SCHEMATIC

Schematics are constructed from symbols. Symbols are also referred to as components in PC-CAPS. In this example, you will create a simple flip-flop circuit using two symbols: an inverter and a NAND gate. The steps in creating a schematic are:

1. Set up the environment.
2. Place the components.
3. Wire the circuit.
4. Name the components and nets.
5. Save the file.

You will also create a plot file containing graphic information that can be used to generate a plot of the schematic.

The schematic you will create is the simple master-slave flip-flop shown in Figure 2-7.



86004

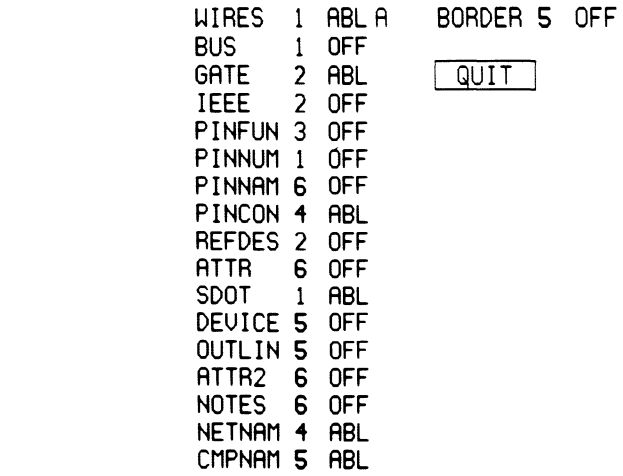
Figure 2-7. Master-Slave Flip-Flop

Step 1. Setting Up the Environment

1. Make sure the system is set to DETL (detail) mode. If it isn't, move the cursor to the DETL command and press Button 1.

2. Move the cursor to the VLYR command and press Button 1. The View Layer Screen is displayed. Move the cursor to the ABL status column (third column) and press Button 1 to set WIRES to ABL status. Set GATE, PINCON, SDOT, NETNAM and CMPNAM to ABL status also. Then move the cursor to the active column (fourth column) and press Button 1 to make WIRES the active layer. Turn the rest of the layers off.

Figure 2-8 shows what your screen should look like.



86005

Figure 2-8. Layer Assignments Set for Creating a Circuit

3. Move the cursor to the QUIT command and press Button 1 to exit the layer display.

   NOTE: In the rest of the examples in Part B, the sequence in step 3 is presented as:

   Select QUIT to exit the layer display.

Step 2. Placing the Components

1. Examine the symbols available.

   a. Select SYS/DOS. The system prompts:

      Enter DOS command (BUTTON 2 to quit)>

   b. Type DIR *.SYM and press [Return].

      The system responds by displaying all the files in the PROJ0 subdirectory with the .SYM extension. Notice in your listing the files:

      NANDT.SYM
      74LS373.SYM
      INV.SYM

      You will use these symbols in this example.

      The system continues prompting:

      Enter DOS command (BUTTON 2 to quit)>

   c. Press Button 2 when you have examined the directory listing. The view returns to the grid and menu.

2. Select ENTR/COMP. The system prompts:

   :
   Comp-file-name x/y-scales

NOTE: The scaling factor is not used in this example. See the ENTR/COMP command in the command reference chapter for information about using scaling.

3. Type NANDT and press [Return].

   NOTE: The system assumes the filename extension is .SYM.

   The system prompts:

   Select loc to place comp. (Orientation OK?)

4. Move the cursor around in the drawing area. Notice the box that moves with the cursor. The size of this box shows the boundaries of the component you are placing.

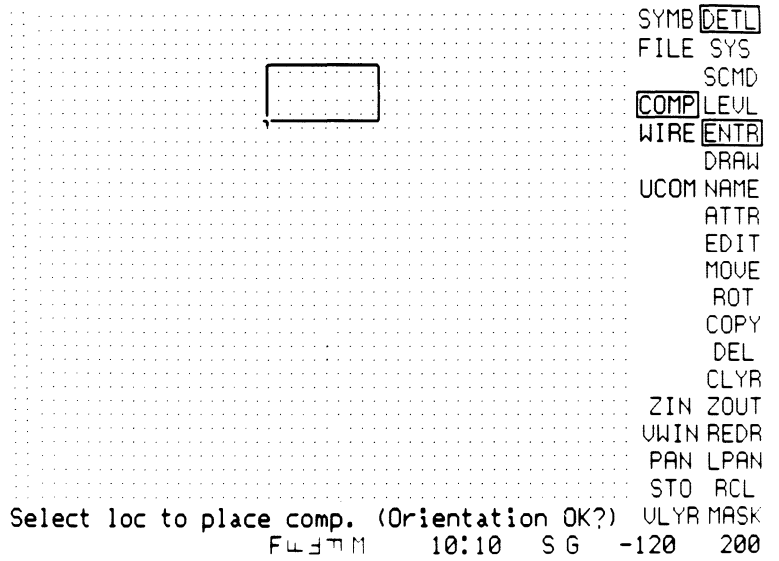
   NOTE: If the size of your component does not match the size of the component in Figure 2-9, you may need to zoom out with the ZOUT command or zoom in with the ZIN command before placing the component.

   Note the part of the prompt:

   (Orientation OK?)

   You can change the orientation of the component as it is placed. This feature is shown later in this example when you place the INV gates.

5. Place the first NAND gate by moving the box to the location shown in Figure 2-9 and press Button 1. The component appears inside the box.

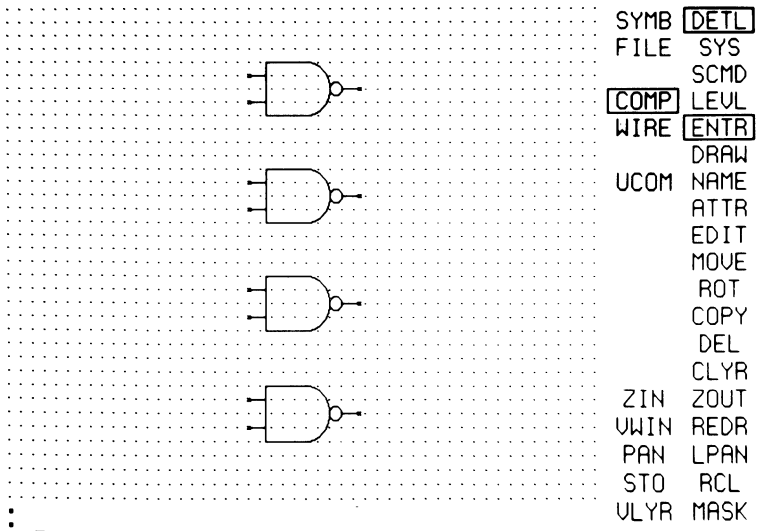


86006

Figure 2-9. Location of First NAND Gate

6. Place the remaining three NAND gates as shown in Figure 2-10. Move the cursor to the desired position and press Button 1. Each time you press Button 1, a copy of NANDT.SYM is placed.

7. After you have placed the fourth NAND gate, press Button 2.



86007

Figure 2-10. Four NAND Gates in Place

8. Type INV and press [Return] to enter the name of the second component.

9. Move the cursor to the set of four F's on the status line. Press Button 1 three times to highlight the right-most F.

   This part of the status line controls the orientation with which components are placed. By selecting the F pointing down, you have chosen to enter the INV gates pointing down.

10. Move the cursor to the desired location, and press Button 1 to place the INV gates as shown in Figure 2-11.

    NOTE: To simplify wiring in the next step, place the gates as shown, relative to the NAND gates. Your coordinates may not match those in the figures.

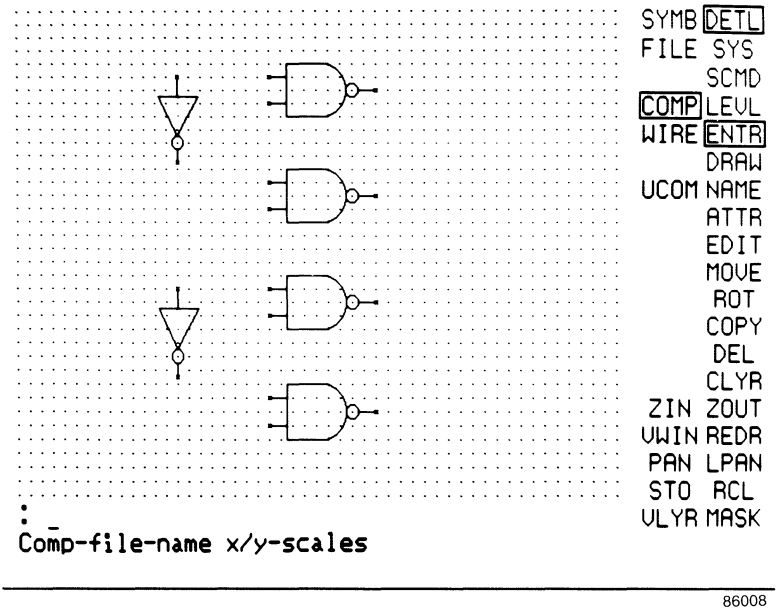


Figure 2-11. Two Copies of INV.SYM in Place

11. Press Button 2 twice to end the ENTR/COMP command.

Step 3. Wiring the Circuit

When the components are in place, the next step is to enter the wires between them.

1. Select ENTR/WIRE to enter a wire. The system prompts:

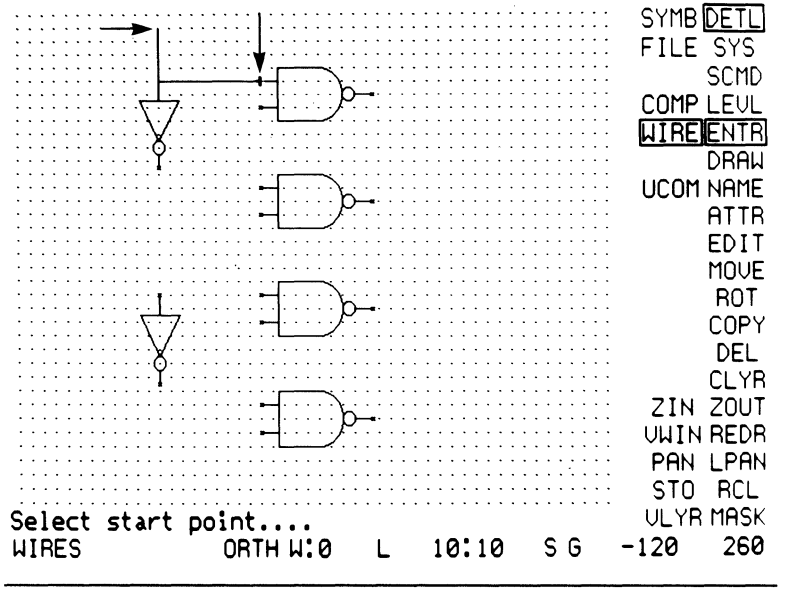
   Select start point....

2. Check the status line parameters for entering wires. The active layer should be WIRES, the line mode should be ORTH (Orthogonal), the line width should be 0, and the lock snap-to-pin feature (L) should be on (green).

3. Select starting and ending points with Button 1 to create the wire shown in Figure 2-12. Then select Button 2 to end the wire.

   NOTE: Be sure that the wire ends on the input pin of the NAND gate to ensure a connection to the pin.

   Unconnected pins are displayed as X's. The X disappears when a wire is connected to the pin.

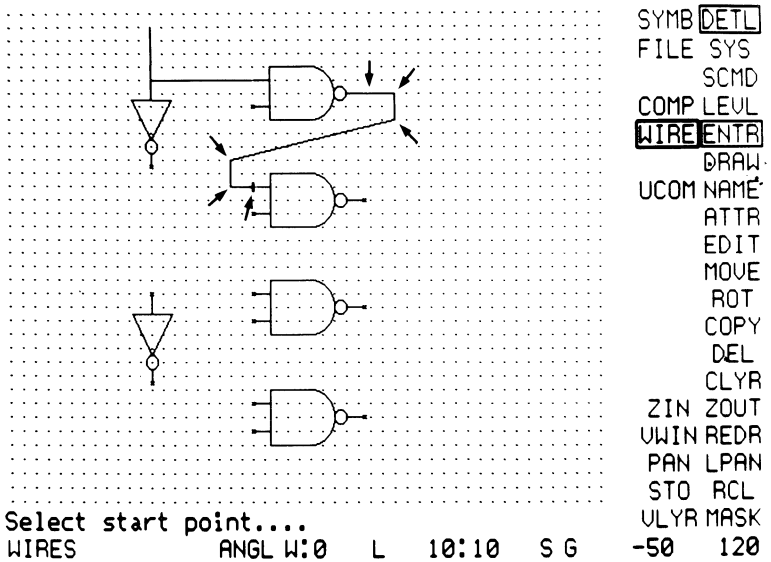


86009

Figure 2-12. First Wire in Place

4. Change the wiring mode. Select ORTH on the status line to toggle the wiring mode to ANGL, so that any angle can be entered.

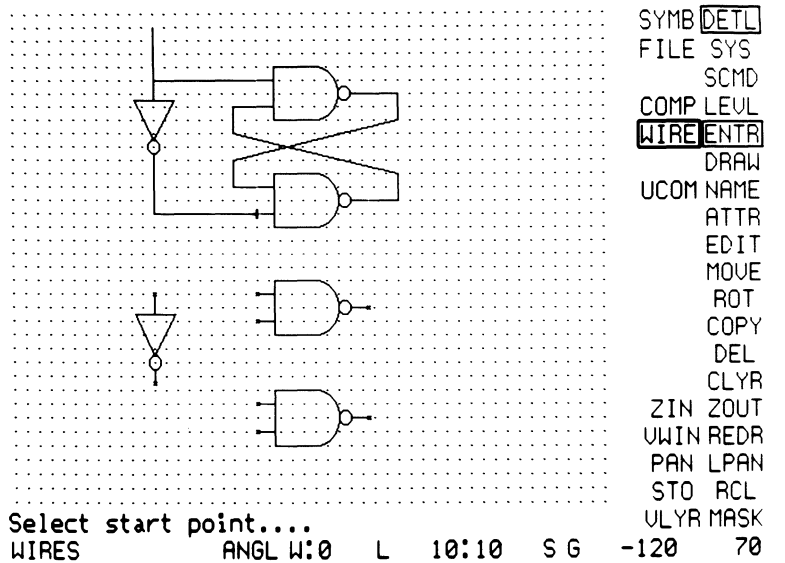
5. Add the angled wire shown in Figure 2-13. Press Button 1 at each location indicated by the arrows.



86010

Figure 2-13. Angled Wire Added

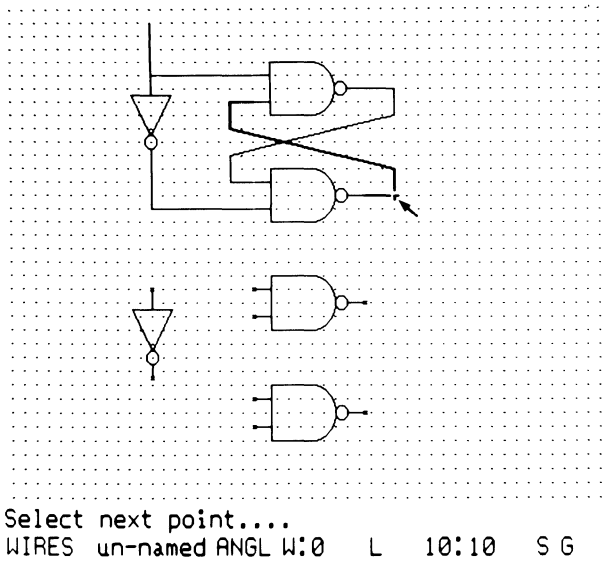
6. Complete the wiring of the upper part of the circuit as shown in Figure 2-14. Change the wiring mode from ANGL to ORTH and from ORTH to ANGL, as needed.



86011

Figure 2-14. Wires to Complete Upper Part of Circuit

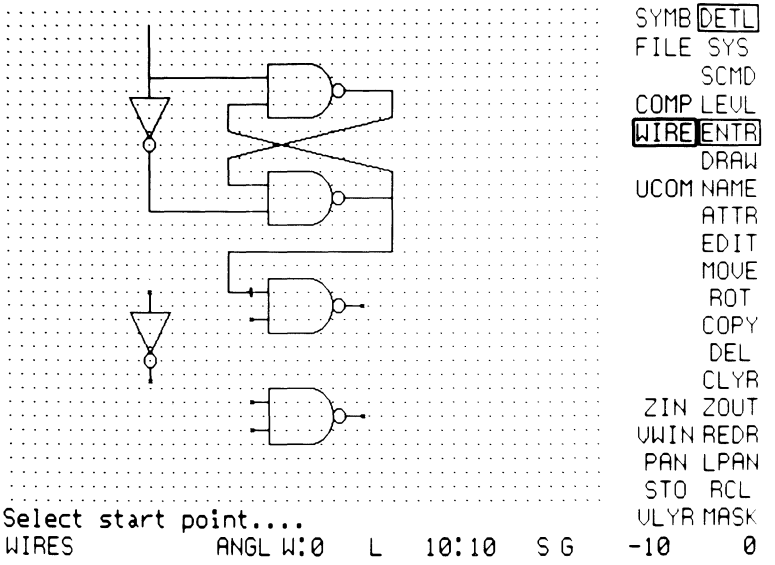
7. Begin a wire at the point shown in Figure 2-15. Notice that the existing wire is highlighted, which indicates that the wire you are starting is considered part of the same electrical connection, or net.



86012

Figure 2-15. Start of Wire Showing Existing Wire Highlighted

8. Complete the wire as shown in Figure 2-16.



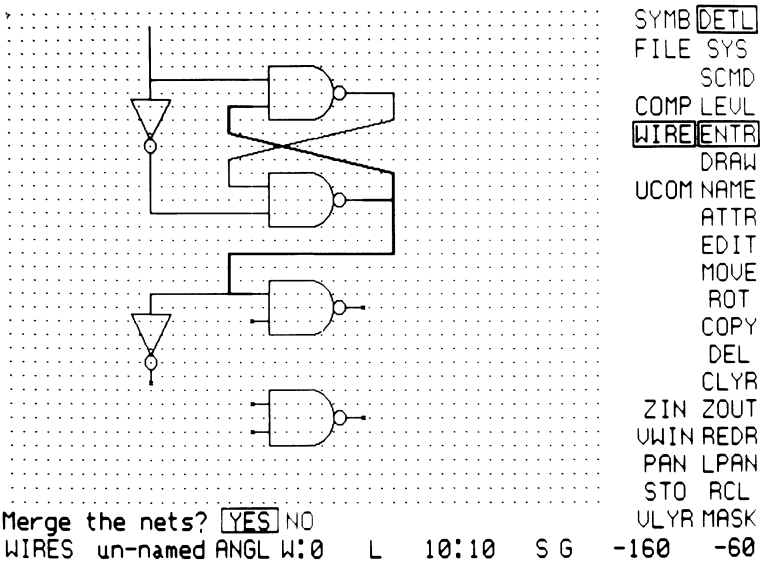
86013

Figure 2-16. Completion of Wire

9. Add the short wire from the INV input to the existing wire shown in Figure 2-17. The system prompts:

   Merge the nets? YES NO

10. Select YES. This joins the wire to the existing net.



86014

Figure 2-17. New Wire Showing Net-Merging Prompt

11. Complete the wiring of the circuit as shown in Figure 2-18.

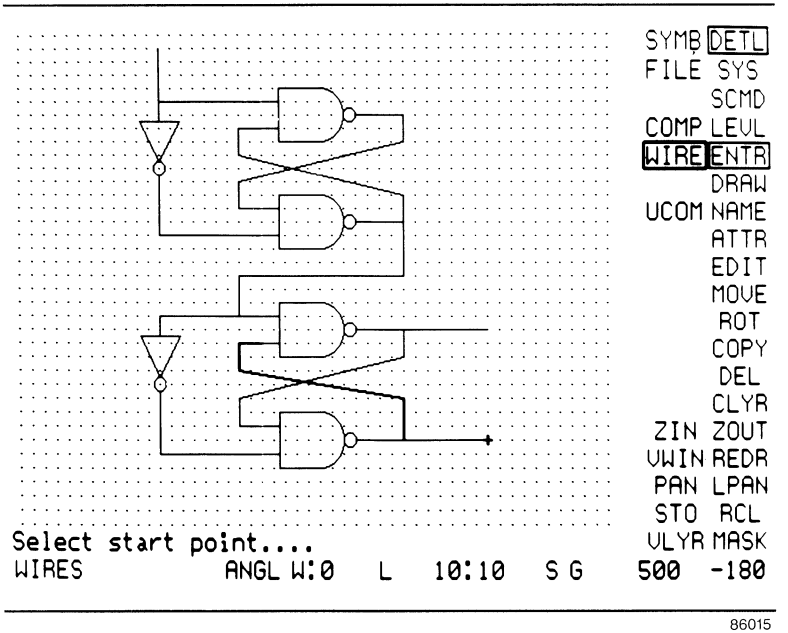


Figure 2-18. Completed Wiring of Flip-Flop

Step 4. Naming the Components and Nets

To make the schematic more readable as well as to uniquely identify data for simulation, you can add names for the components and nets.

Name the Components

1. Select NAME/COMP to name the components. The system sets the CMPNAM layer active and prompts:

   Select a component....

2. Change the text size to 30.

   a. Select SIZ: on the status line. The system prompts:

      Enter text size:

   b. Type 30 and press [Return].

3. Move the cursor to the upper INV gate and press Button 1. The system prompts:

   Enter component name:

4. Type G1 and press [Return]. The system prompts:

   Name = G1. Select location....

5. Move the cursor to the location to the right of the gate as shown in Figure 2-19 and press Button 1.

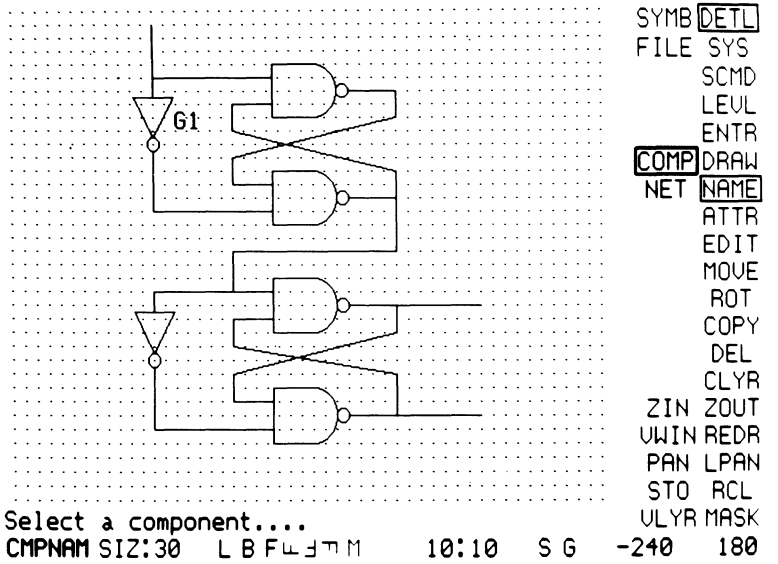


Figure 2-19. Location for G1 Text

6. Name the remaining components as shown in Figure 2-20.

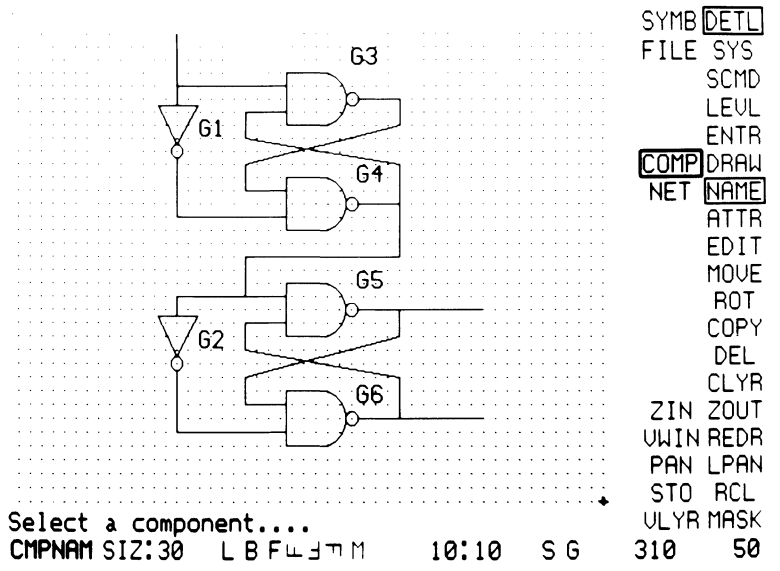


Figure 2-20. Names for All Components

Name the Nets

1. Select NAME/NET to name the nets. The system sets the NETNAM layer active and prompts:

   Select a net....

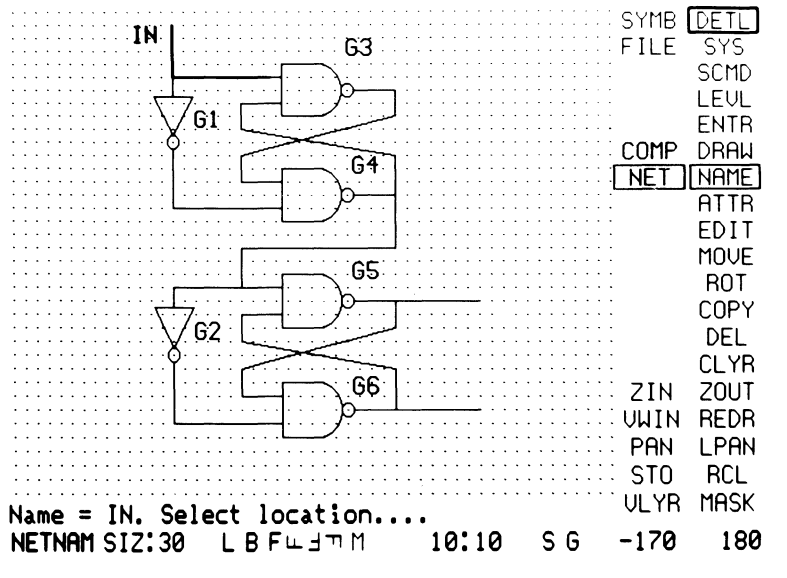
2. Move the cursor to any point on the input net (the upper left net connecting the inputs of G1 and G3) and press Button 1. The net is highlighted and the system prompts:

   Enter net name:

3. Type IN and press [Return]. The system prompts:

   Name = IN. Select location....

4. Move the cursor to the location shown in Figure 2-21 for IN and press Button 1.



86018

Figure 2-21. Location for Net Name IN

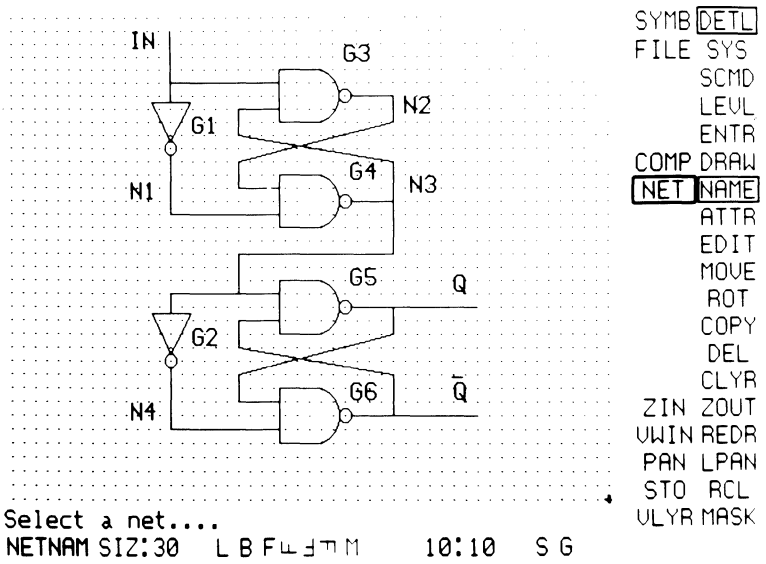
The prompt remains the same. If you move to another point and press Button 1, another copy of net name IN is placed.

5. Press Button 2. The command NAME/NET stays active, and the system prompts:

   Select a net....

6. Name the remaining nets as shown in Figure 2-22.

   NOTE: To specify negation, name the bottom output net Q'. The system will display this as a Q with a bar over it as shown in Figure 2-22.



86019

Figure 2-22. All Nets Named

Step 5. Saving the File

You have completed a simple master-slave flip-flop circuit. Now save the schematic.

1. Select FILE/SAVE. The system prompts:

   Enter file name:

2. Type TUTORA and press [Return].

   NOTE: You don't need to type a filename extension. The system automatically adds a .SCH extension to the filename.

Step 6. Creating a Plot File

Before you clear the screen, create a plot file. Schematic plot files are used by P-CAD's PC-PRINT and PC-PLOTS utilities programs to output plots on a dot-matrix printer and a pen plotter, respectively.

If you are using PC-PLOTS to plot your schematic, note that the numbers displayed on the View Layer Screen correspond to the pen numbers of your plotter. The pen number assignments are saved with the plot file. To change pen number assignments, use the VLYR command.

To create a plot file:

1. Select SYS/PLOT. The system prompts:

   PLOT: Select Page Corner 1...

2. Position the cursor as shown in Figure 2-23 and press Button 1. The system prompts:

PLOT: Select Page Corner 2...

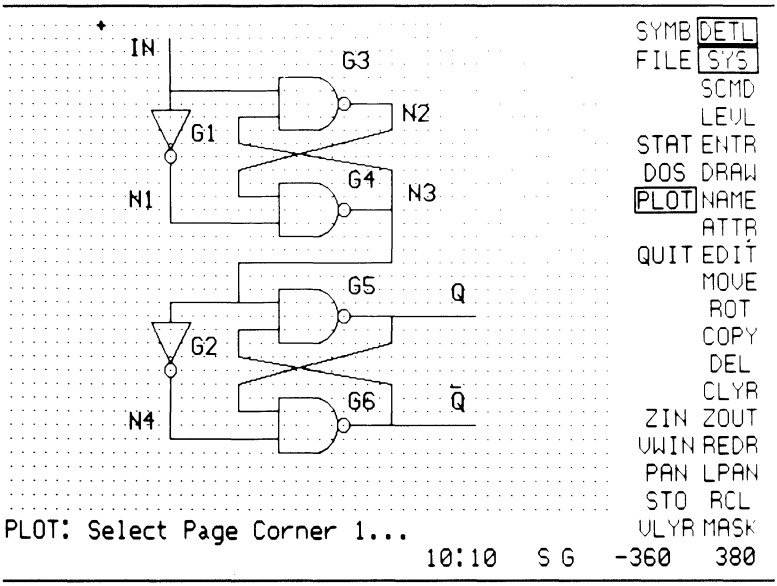
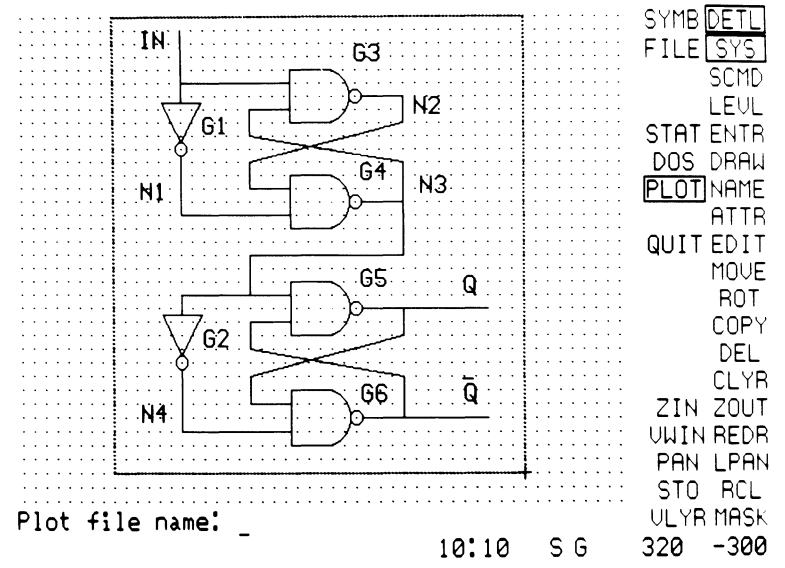


Figure 2-23. Cursor Location for Page Corner 1

3. Position the cursor as shown in Figure 2-24 and press Button 1.



86021

Figure 2-24. Cursor Location for Page Corner 2

Notice the dashed box that moves with the cursor. The box shows the boundaries of the plot. The system prompts:

Plot file name:

4. Type TUTORA and press [Return] or Button 1.

   NOTE: You don't need to type a filename extension. The system automatically adds a .PLT extension to the filename.

   The system displays the following message:

   Writing to plot file "tutora.PLT"

   and then

   Plot file complete.

5. Now you can clear the memory and screen. Select FILE/ZAP.

Summary

By completing this example, you have learned how to create a schematic. Review the steps below to check your understanding of the process.

Table 2-1. Creating a Schematic

Step	Commands Used
Setting up the environment	DETL VLYR
Placing the components	SYS/DOS ENTR/COMP
Wiring the circuit	ENTR/WIRE
Naming the components and nets	NAME/COMP NAME/NET
Saving the file	FILE/SAVE
Creating a plot file	SYS/PLOT
Erasing the screen	FILE/ZAP

EXAMPLE 2. EDITING A SCHEMATIC

In your design work, you may need to edit an existing schematic.

The EDIT commands operate on segments and vertices. Each point you enter when creating a wire or line is a vertex. The part of the wire or line between two vertices is a segment.

The EDIT commands are shown in Table 2-2.

Table 2-2. EDIT Commands

Command	Function
EDIT/ADDV	Adds a vertex
EDIT/DELV	Deletes a vertex
EDIT/MOVV	Moves a vertex
EDIT/LAYS	Changes the layer of a segment
EDIT/DELS	Deletes a segment
EDIT/MOVS	Moves a segment
EDIT/MOVA	Moves a wire segment or a vertex and all attached wire segments and solder dots on all layers

In this example, you will work with an existing schematic and make several modifications to it. Figure 2-25 shows the schematic to be used and indicates the desired modifications.

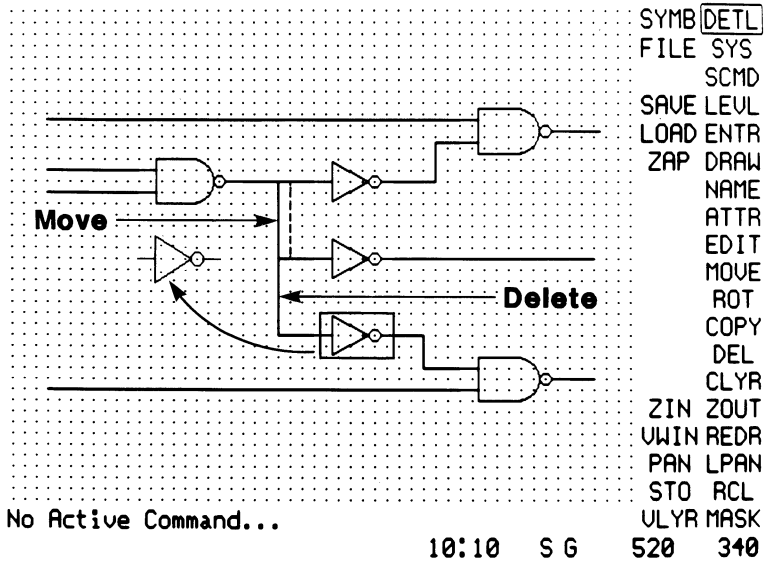


Figure 2-25. TUTORC Circuit Showing Modifications

The modifications you will make are:

- Delete the segment shown.
- Move the lower INV gate to the new location shown.
- Make all wire angles 90 degrees.

First, load the schematic file. Follow the steps below.

1. Select FILE/LOAD. The system prompts:

   Enter file name:

2. Type TUTORC and press [Return]. The system assumes the filename extension is .SCH.

Deleting a Segment

1. Select EDIT/DELS. The system prompts:

   Select a segment....

2. Move the cursor to any point on the vertical segment shown in Figure 2-25 and press Button 1. The segment is deleted as shown in Figure 2-26.

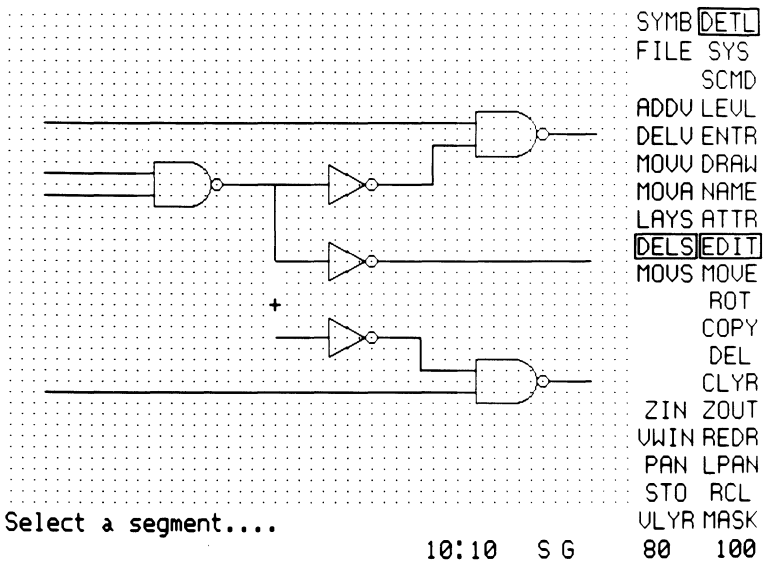


Figure 2-26. Segment Deleted

Moving a Component

1. Select MOVE to move the lower INV gate. The system prompts:

   Select the object(s)....

2. Move the cursor to any point inside the lower INV gate and press Button 1. The gate appears as a solid box as shown in Figure 2-27. The system prompts:

   Move object to. (Select point)...

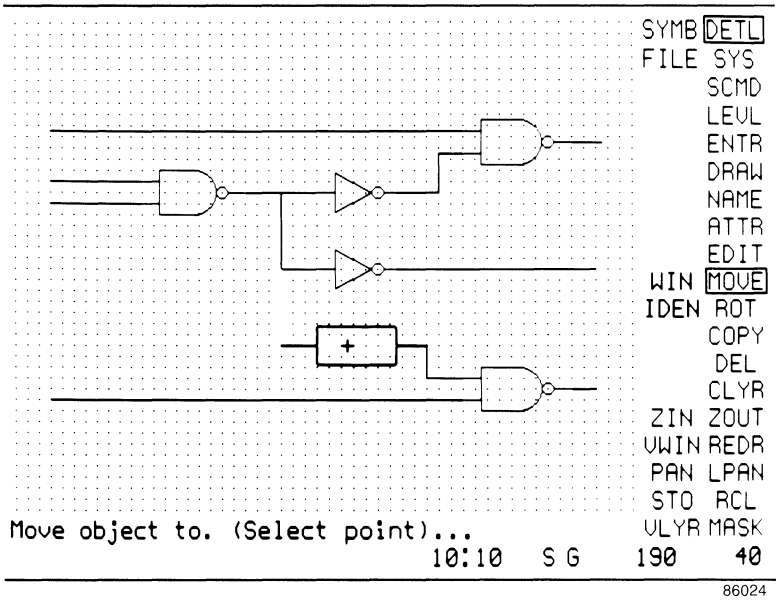


Figure 2-27. INV Gate Selected for Moving

3. Move the gate to the new location shown in Figure 2-28. Notice that the wires stretch as you move the gate. This PC-CAPS feature is called "rubberbanding."

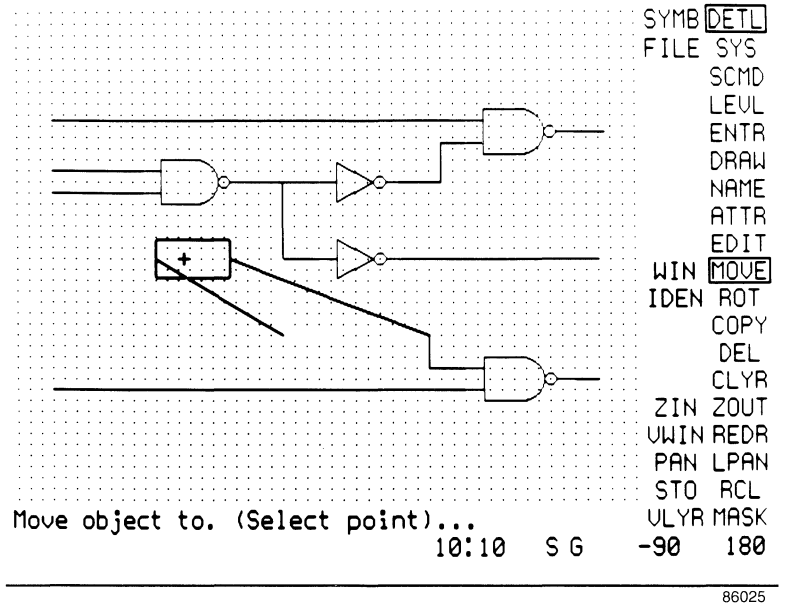


Figure 2-28. INV Gate During Move

4. Press Button 1. The gate is placed in the new location as shown in Figure 2-29.

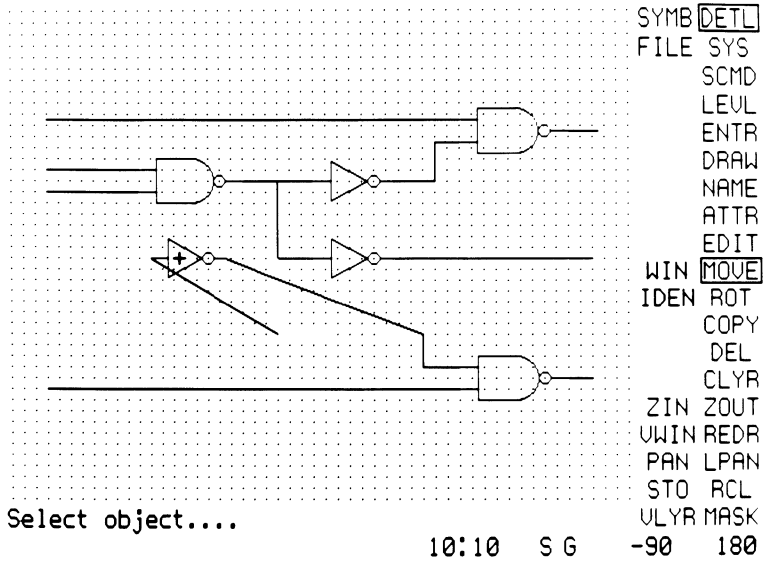


Figure 2-29. INV Gate in New Location

Moving a Vertex

1. Select EDIT/MOVV to move the first vertex. The system prompts:

   Select vertex....

2. Move the cursor to the vertex shown by the arrow in Figure 2-30 and press Button 1. The system prompts:

   Move the vertex to....

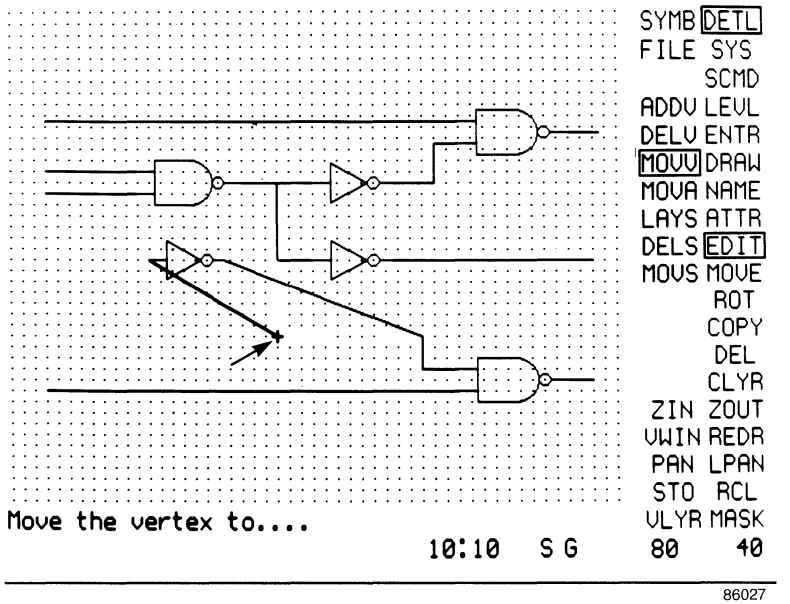


Figure 2-30. Vertex to Be Selected

3. Move the vertex to the new location shown in Figure 2-31 and press Button 1.

4. Select REDR to redraw the screen.

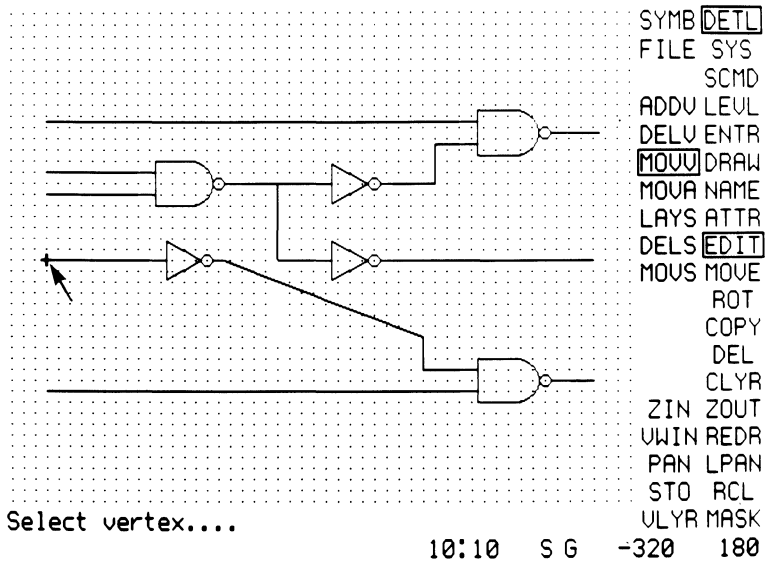


Figure 2-31. Input Wire of INV Gate Moved to New Location

Adding a Vertex

1. Select EDIT/ADDV to add a vertex. The system prompts:

   Select a point....

2. Move the cursor to a point on the end of the segment shown by the arrow in Figure 2-32 and press Button 1. The segment is highlighted and the system prompts:

   Select a new vertex....

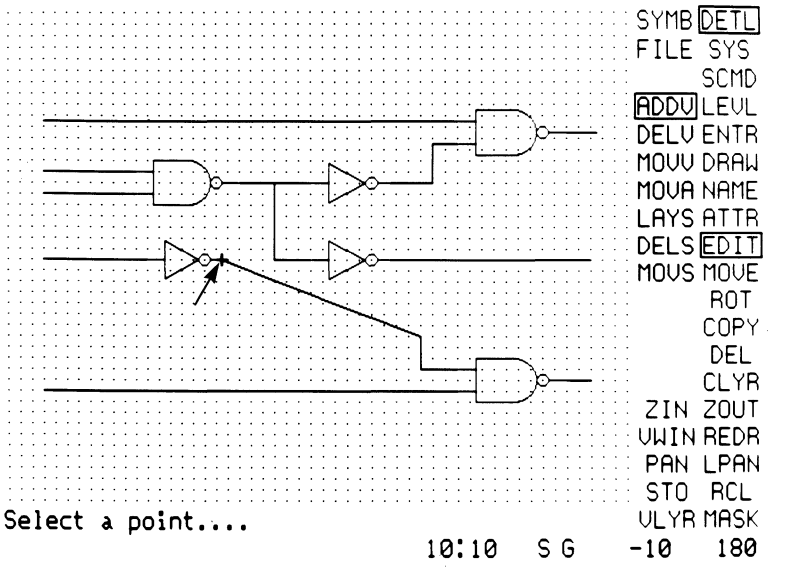
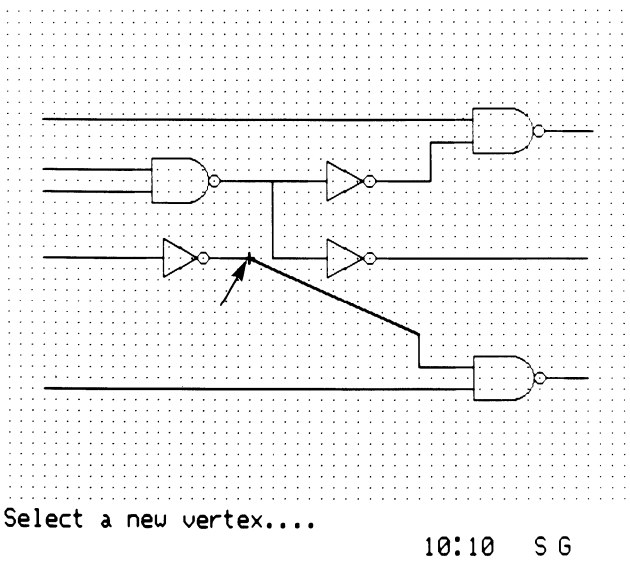


Figure 2-32. Point on End of Segment Selected

3. Move the cursor horizontally to the new location shown in Figure 2-33 and press Button 1.



86030

Figure 2-33. New Vertex Added

Moving Another Vertex

1. Select EDIT/MOVV to move a vertex. The system prompts:

   Select vertex....

2. Move the cursor to a vertex on the segment shown by the arrow in Figure 2-34 and press Button 1. The line segment is highlighted and the system prompts:

   Move the vertex to....

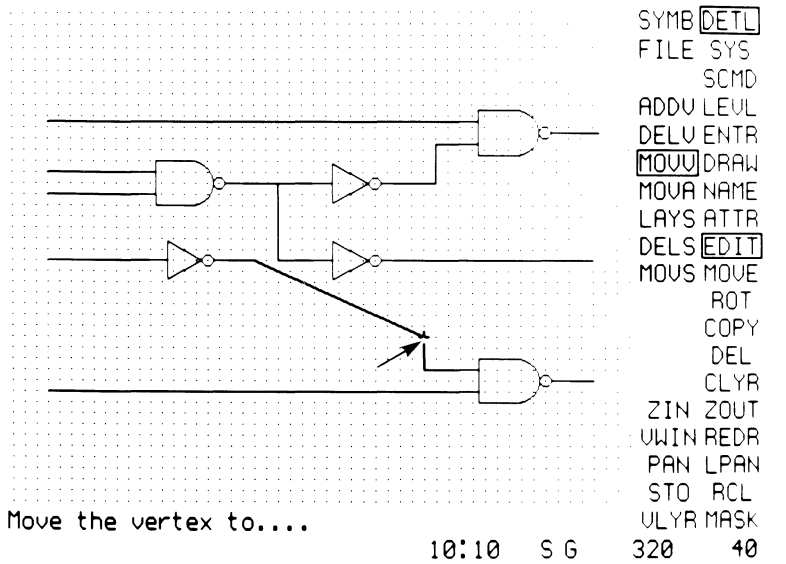
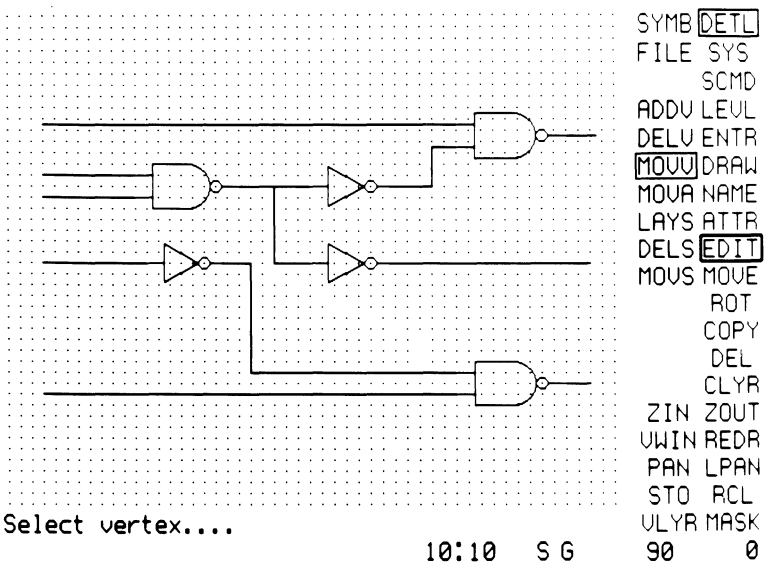


Figure 2-34. Vertex to Be Moved

3. Move the vertex to the new location shown in Figure 2-35 and press Button 1.

4. Select REDR to redraw the screen.



86032

Figure 2-35. Vertex Moved

Moving a Segment and All Attached Solder Dots and Segments

1. Select EDIT/MOVA. The system prompts:

   Select a point....

2. Select a point on the wire shown in Figure 2-36. The system rubberbands the segment and the connected segments and solder dot. The system prompts:

   Select new position....

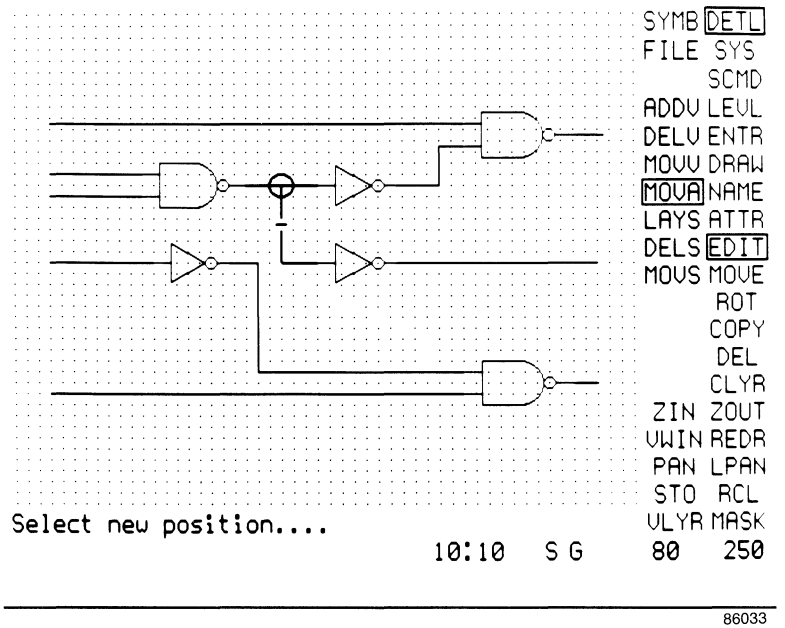
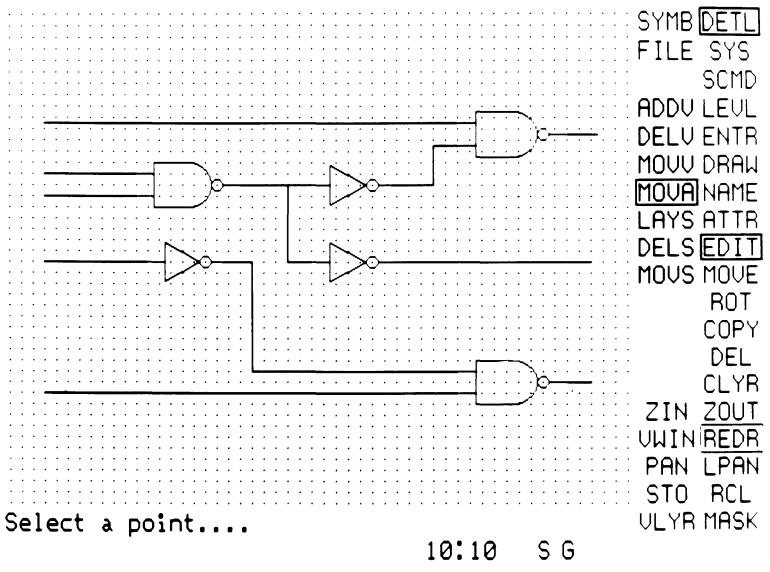


Figure 2-36. Segment to Be Moved

3. Move the cursor to the new location shown in Figure 2-37 and press Button 1.

4. Select REDR to redraw the screen.



86034

Figure 2-37. Segment in New Location

You have now completed the wiring corrections.

5. Select FILE/ZAP to erase the screen and clear the memory. The system prompts:

   Workfile modified. Sure to zap? YES NO

6. Select YES.

Summary

Now that you have completed this example, you are familiar with the EDIT commands listed below and know how to use them to make minor changes in a layout.

Table 2-3. Editing a Schematic

Step	Commands Used
Deleting a segment	EDIT/DELS
Moving a component	MOVE
Moving a vertex	EDIT/MOVV
Adding a vertex	EDIT/ADDV
Moving a segment and all attached solder dots and segments	EDIT/MOVA

EXAMPLE 3. DESIGNING WITH HIERARCHY

This example illustrates hierarchical design methods. Designing with hierarchy is a useful way to design complex circuits consisting of many elements by defining modules for portions of the circuit. The modules can then be used just like symbols.

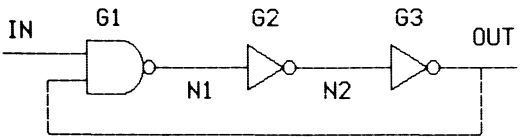
In this example, you will create the details of an oscillator circuit, create a symbol to represent the oscillator circuit, and save the file. Then you will use the symbol in a schematic. The steps in this example are:

1. Create the circuit detail.
2. Create the symbol for the circuit.
3. Save the file.
4. Use the symbol.
5. Examine the hierarchy.

Step 1. Creating the Circuit Detail

The oscillator circuit you will create is shown in Figure 2-38. The procedure is:

- Set up the environment.
- Place the components.
- Wire the circuit.
- Name the components.
- Name the nets.



86035

Figure 2-38. Oscillator Circuit Detail

Setting Up the Environment

1. Make sure the system is set to detail (DETL) mode.

2. Select VLYR to view the layer screen. Set the WIRES, GATE, PINNAM, PINCON, DEVICE, NETNAM, and CMPNAM layers to ABL status and make WIRES the active layer. Turn the rest of the layers off.

Placing the Components

1. Select ENTR/COMP to place the components. The system prompts:

   :
   Comp-file-name x/y-scales

2. Type NANDT and press [Return]. The system prompts:

   Select loc to place comp. (Orientation OK?)

3. Move the cursor to the location shown in Figure 2-38 to place the NAND gate and press Button 1.

NOTE: In the rest of this example and the examples in Part B, the sequence above is presented as:

Select the location

or

Select a point

"Select" means move the cursor to a point in the drawing area or to a command in the menu and press Button 1.

4. Press Button 2 and the system prompts:

:
Comp-file-name x/y-scales

5. Type INV and press [Return]. The system prompts:

Select loc to place comp. (Orientation OK?)

6. Select the location to place one of the two inverted gates as shown in Figure 2-38. The system continues prompting:

Select loc to place comp. (Orientation OK?)

7. Select the location to place the other inverted gate as shown in Figure 2-38. Press Button 2 twice to end the ENTR/COMP command.

Wiring the Circuit

1. Select ENTR/WIRE to wire the circuit. The system prompts:

   Select start point....

2. Enter the wires as shown in Figure 2-38. Make sure the wiring mode is set to ORTH.

   NOTE: When ending a wire on an existing net, the system prompts:

   Merge the nets? YES NO

3. Select YES to merge the nets. This joins the wire to the existing net. The system responds:

   Nets merged into worknet.

4. Press Button 2 to end the ENTR/WIRE command.

Naming the Components

1. Select NAME/COMP to name the components. The system sets the CMPNAM layer active and prompts:

   Select a component....

2. Change the text size to 30.

3. Select the NAND gate. The system prompts:

   Enter component name:

4. Type G1 and press [Return]. The system prompts:

   Name = G1. Select location....

5. Select the location of the name G1 as shown in Figure 2-38.

6. Press Button 2 and the system prompts:

   Select a component....

7. Select the INV symbol located in the center of the circuit. The system prompts:

   Enter component name:

8. Type G2 and press [Return]. The system prompts:

   Name = G2. Select location....

9. Select the location of the name G2 as shown in Figure 2-38. The system prompts:

   Select a component....

10. Select the INV symbol located on the right side of the circuit as shown in Figure 2-38. The system prompts:

    Enter component name:

11. Type G3 and press [Return]. The system prompts:

    Name = G3. Select location....

12. Select the location of the name G3 as shown in Figure 2-38 and press Button 1.

Naming the Nets

1. Select NAME/NET to name the nets. Notice that you don't have to select NAME again, just NET. The system sets the NETNAM layer active and prompts:

   Select a net....

2. Select any point on the input net (the net connected to the input of G1). The net is highlighted and the system prompts:

   Enter net name:

3. Type IN and press [Return]. The system prompts:

   Name = IN. Select location....

4. Select location of IN net name as shown in Figure 2-38.

5. Press Button 2 and the system prompts:

   Select a net....

6. Select any point on the output net (the net connected from the output of G3 to the input of G1). The net is highlighted and the system prompts:

   Enter net name:

7. Type OUT and press [Return]. The system prompts:

   Name = OUT. Select location....

8. Select the location of OUT net name as shown in Figure 2-38.

9. Then press Button 2 and the system prompts:

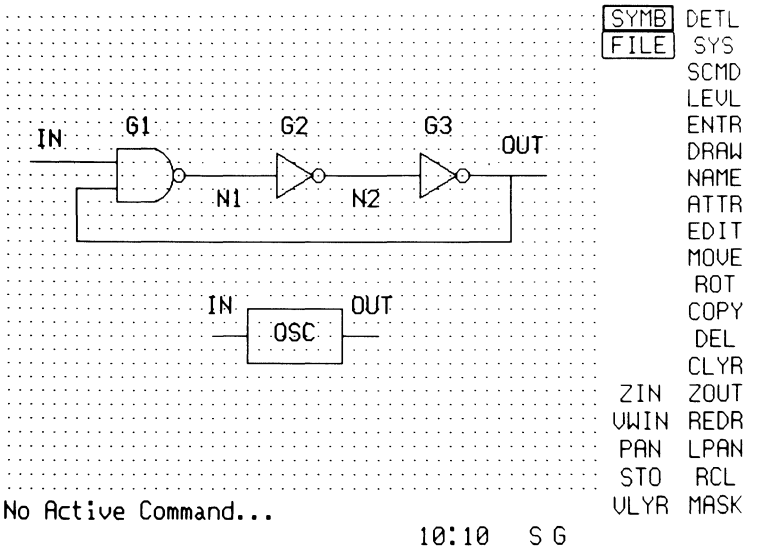
   Select a net....

10. Name the rest of the nets as shown in Figure 2-38. Name one net N1 and the other net N2.

Step 2. Creating the Symbol for the Circuit

Create the symbol for the oscillator circuit as shown in Figure 2-39. The symbol graphics is the part of the oscillator that will be placed in a higher-level schematic. Symbol and detail are alternate representations of the same circuit. The procedure is:

- Create the graphics.
- Enter input and output pins.
- Add the symbol name.
- Assign the origin.



86036

Figure 2-39. Oscillator Circuit Symbol

Create the Graphics

Before you create the graphics, select SYMB to set the system to symbol mode.

1. Select DRAW/RECT to create the rectangle part of the graphics as shown in Figure 2-39.
2. Change the active layer to GATE.

3. Select DRAW/LINE to draw the input and output lines as shown in Figure 2-39.

Enter Input and Output Pins

1. Select ENTR/PIN. The system sets the PINCON layer active and prompts:

   Select pin location: .....

2. Set the pin type parameter on the status line to INPUT.

3. Select location of input pin. The system sets the PINNAM layer active and prompts:

   Select pin name location. (Attrb OK?)...

4. Change the justification parameters as needed and select location of pin name. The system prompts:

   Enter pin name.

5. Type IN and press [Return]. The system sets the PINCON layer active and prompts:

   Select pin location: ....

6. Change the pin type parameter on the status line to OUTPUT.

7. Select location of output pin. The system sets the PINNAM layer active and prompts:

   Select pin name location. (Attrb OK?)

8. Select location of pin name. The system prompts:

   Enter pin name.

9. Type OUT and press [Return].

Add the Symbol Name

1. Select DRAW/TEXT to assign device name. The system prompts:

   Select location. (Attributes OK?)

2. Change the active layer to DEVICE.

3. Select location of text as shown in Figure 2-39. The system prompts:

   Type in text.

4. Type OSC and press [Return].

Assign the Origin

1. Select ENTR/ORG. The system prompts:

   Select the origin....

2. Select the input pin.

Save the File

1. Select FILE/SAVE to save the file. The system prompts:

   Enter file name:

2. Type OSC. A .SYM extension is automatically added to the filename.

3. Select FILE/ZAP to erase the screen.

Step 3. Using the Symbol

Now you will add the oscillator symbol to a schematic using the ENTR/COMP command. The procedure is to:

- Set up the environment.
- Place the component.
- Enter wire connections.
- Name the component.
- Name the nets.

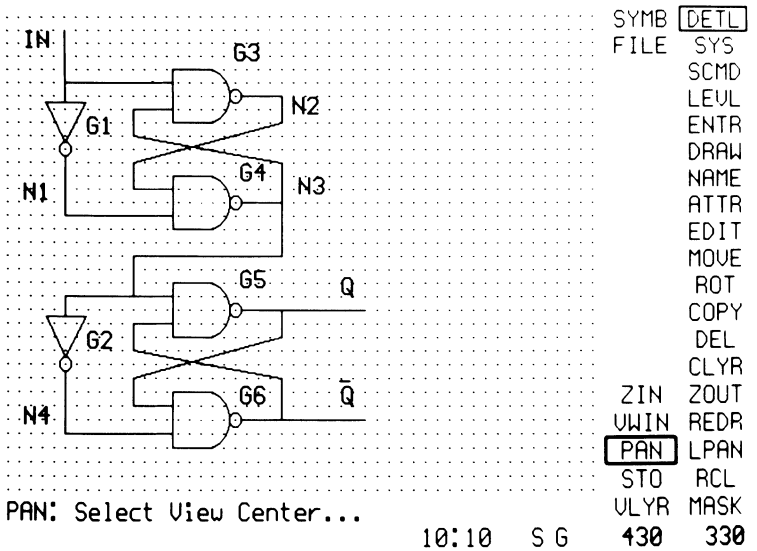
Set Up the Environment

1. Select FILE/LOAD. The system prompts:

   Enter file name:

2. Type TUTORA and press [Return].

3. Select PAN to change the center of the view as shown in Figure 2-40.



86037

Figure 2-40. TUTORA.SCH Loaded and View Center Changed

Place the Component

1. Select ENTR/COMP. The system prompts:

   :
   Comp-file-name x/y-scales

2. Type OSC and press [Return]. The system prompts:

   Select loc to place comp. (Orientation OK?)

3. Change the placement orientation on the status line. Select the four F's on the status line until the left-most F is highlighted. Also, make sure the DEVICE layer is on.

4. Select the component location as shown in Figure 2-41. Place the component so that the input pin does not touch the existing net.

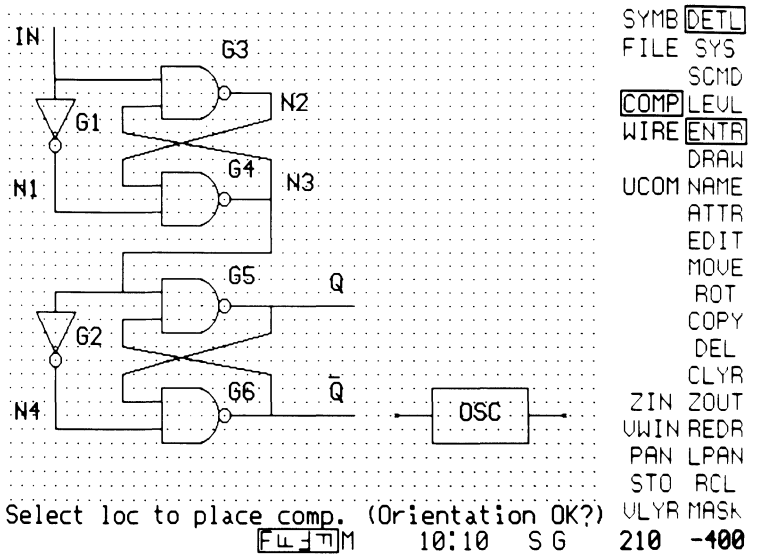


Figure 2-41. Placement of OSC.SYM

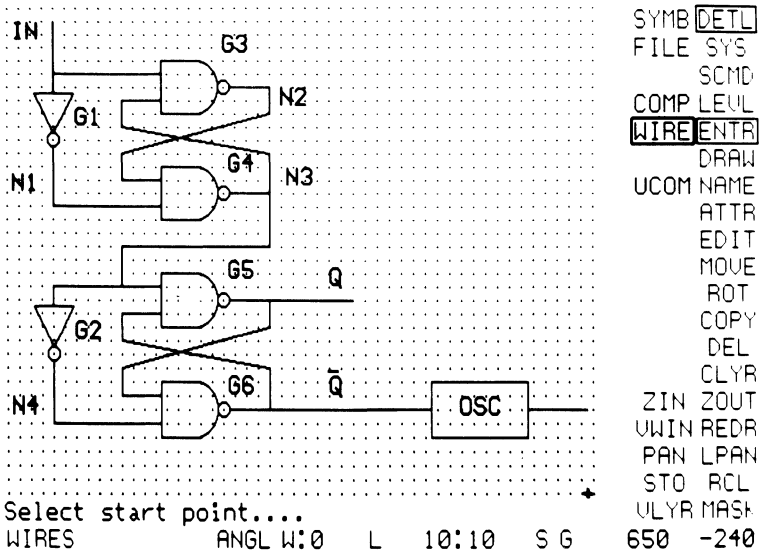
Even though OSC.SYM contains both symbol and detail versions, only the symbol version shows when you place the component with ENTR/COMP. Only the symbol graphics you create in SYMB mode are displayed when you place the component. "Step 4. Examining the Hierarchy" shows you how to see the detail of a component.

Enter Wire Connections

1. Select ENTR/WIRE to wire the OSC component. The system sets the WIRES layer active and prompts:

   Select start point....

2. Enter wires as shown in Figure 2-42.



86039

Figure 2-42. Wiring of OSC.SYM

Name the Component

1. Select NAME/COMP to name the component. The system sets the CMPNAM layer active and prompts:

   Select a component....

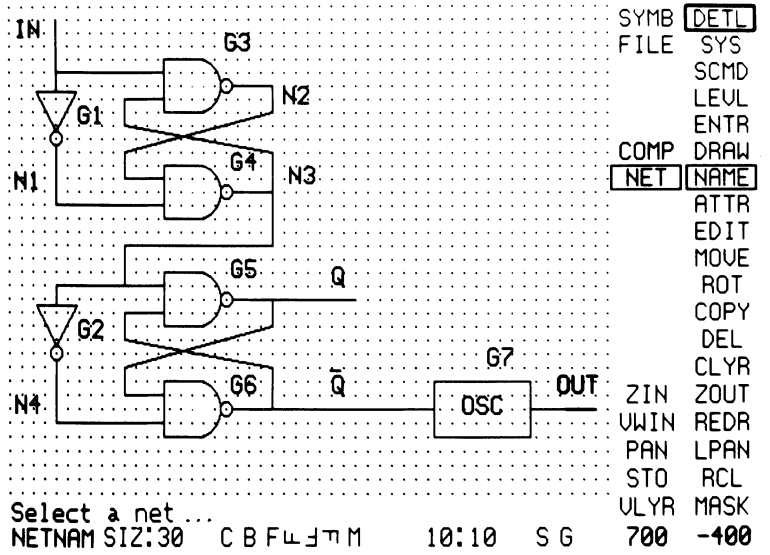
2. Select the component. The system prompts:

   Enter component name:

3. Type G7 and press [Return]. The system prompts:

   Name = G7. Select location....

4. Select component name location as shown in Figure 2-43.



86040

Figure 2-43. Naming of OSC.SYM

Name the Net

1. Select NAME/NET. The system sets the NETNAM layer active and prompts:

   Select a net....

2. Select a point on the line connected to the output of the OSC symbol. The net is highlighted and the system prompts:

   Enter net name:

3. Type OUT and press [Return]. The system prompts:

   Name = OUT. Select location....

4. Select net name location as shown in Figure 2-43.

Save the File

1. Select FILE/SAVE to save the file. The system prompts:

   Enter file name:

2. Type TUTORB and press [Return].

   Do not clear the working memory and screen until after the next step.

Step 4. Examining the Hierarchy

The OSC.SYM symbol represents the detail of the oscillator circuit. You can view this without exiting the current drawing. Viewing the detail of a component is called "pushing into the hierarchy."

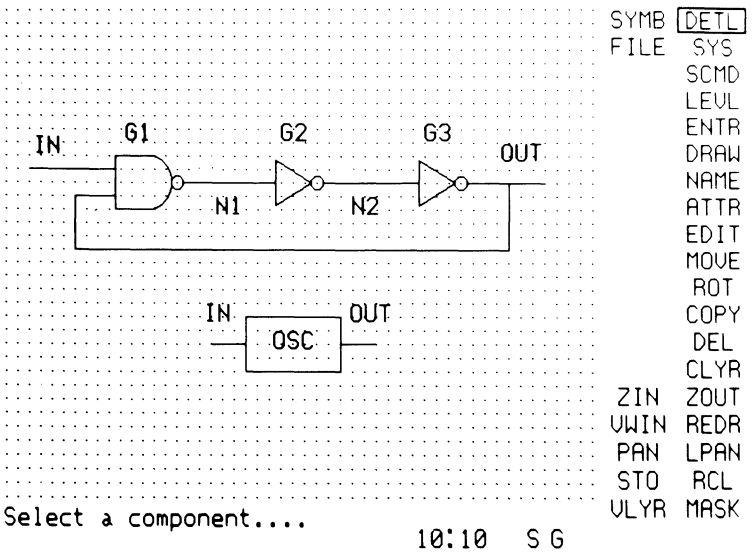
1. Select LEVL/PUSH to examine the hierarchy. The system prompts:

   Select a component....

2. Select the OSC.SYM component. The system prompts:

   WAIT! Pushing into OSC.SYM

   After a few seconds, the display changes to the view shown in Figure 2-44. This view is the pushed view of OSC.SYM.



86041

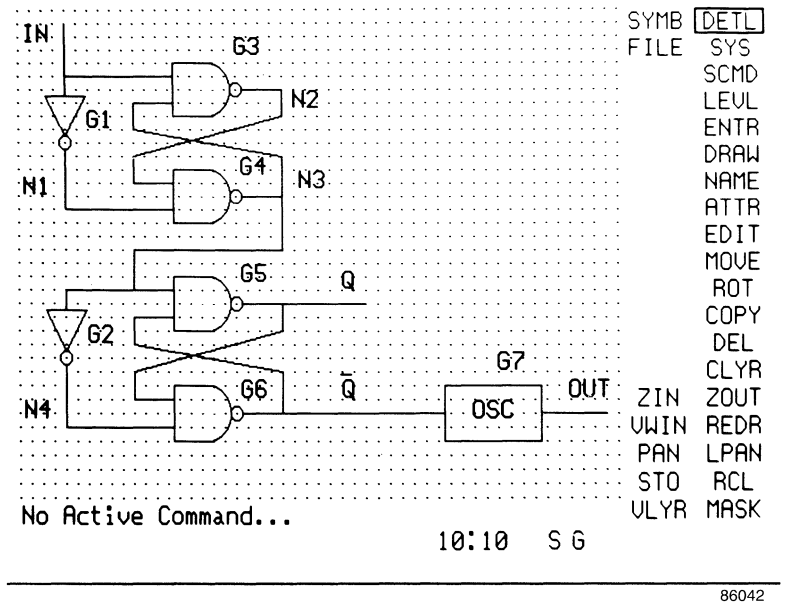
Figure 2-44. Pushed View of OSC.SYM

Returning to the view that existed before the LEVL/PUSH command was executed is called "popping."

3. Select LEVL/POP. The system prompts:

   WAIT! Popping back to the parent level.

   After a few seconds, the view returns to the original view as shown in Figure 2-45.



86042

Figure 2-45. Original View of Circuit

4. Select FILE/ZAP to clear the screen.

Summary

By completing this example, you have learned how to design with hierarchy. Review the steps below to check your understanding of the process.

Table 2-4. Designing With Hierarchy

Step	Commands Used
Building the detail of the circuit	DETL See Table 2-1
Building the symbol for the circuit	SYMB See Table 2-8
Using the symbol	ENTR/COMP
Examining the hierarchy	LEVL/PUSH LEVL/POP

EXAMPLE 4. ASSIGNING ATTRIBUTES

Attributes are data assigned by you and recorded in the database.

Attributes may be specified for components, pins, or wires, and used later for report generation, simulation, or layout.

With the ATTR (Attribute) commands, you assign the name of the attribute (Keyword) and the value of the attribute.

You can add, change, or delete attributes of a component in detail mode or of a symbol in symbol mode.

Table 2-5 lists the ATTR (Attribute) commands.

Table 2-5. ATTR (Attribute) Commands

Command	Function
ATTR/ACOM	In detail mode, adds an attribute to a component in a schematic. In symbol mode, adds an attribute to a symbol.
ATTR/DATR	In detail mode, deletes the selected attribute of a component in a schematic. In symbol mode, deletes the selected attribute of a symbol.
ATTR/SCHG	In detail mode, changes the value of a single attribute of a component in a schematic. In symbol mode, changes the value of a single attribute of a symbol.

In the following examples, you will learn how to add and change attributes using the ATTR commands.

Adding an Attribute

In this example, you will add an attribute called TECH=TTL to a 2-input NAND gate.

1. Load the NANDT.SYM file.

   a. Select FILE/LOAD. The system prompts:

      Enter file name:

   b. Type NANDT.SYM and press [Return].

      Figure 2-46 shows what your screen should look like.

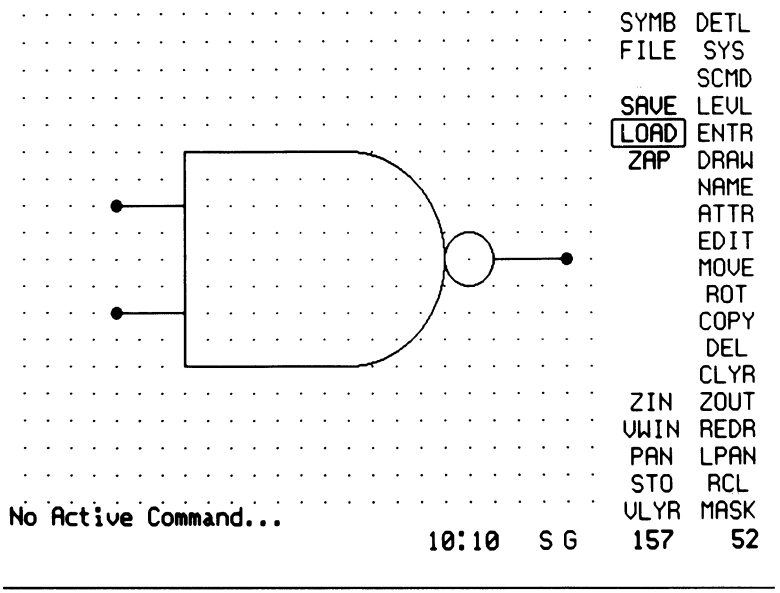


Figure 2-46. NANDT.SYM

2. Select ATTR/ACOM to add the attribute to the symbol. The system sets the ATTR layer active and prompts:

   Select location. (Text attributes OK?)...

3. Before selecting the location for the text attributes, set the text size to 15 on the status line.

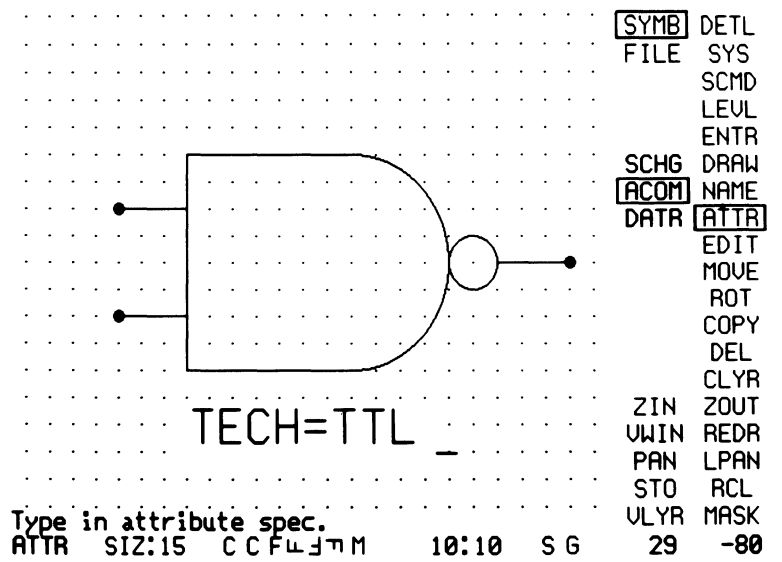
4. Make sure the vertical and horizontal positions of text are set to C C on the status line.

5. Select a point under the symbol. The system prompts:

   Type in attribute spec.

6. Type TECH=TTL as shown in Figure 2-47 and press [Return].

   NOTE: After you enter the attribute, the keyword (TECH) is suppressed, leaving only the attribute value (TTL) displayed. If you change the attribute using ATTR/SCHG, the entire attribute (keyword and value) will be shown while editing.



86044

Figure 2-47. TECH Attribute Added

Saving the File

1. Select FILE/SAVE. The system prompts:

   Enter file name:

2. Type NAND2 and press [Return].

3. Select FILE/ZAP to erase the screen.

Changing an Attribute

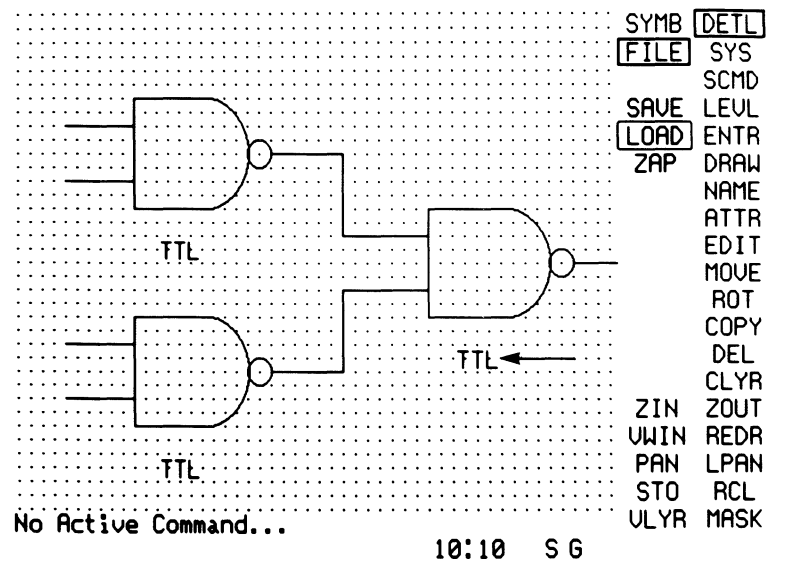
In this example, you will change the attribute of a component in a schematic using the ATTR/SCHG command.

1. Select FILE/LOAD. The system prompts:

   Enter file name:

2. Type TUTORD and press [Return].

   Figure 2-48 shows what your screen should look like.



86045

Figure 2-48. TUTORD.SCH Circuit

The TECH attributes appear on the screen because the ATTR layer is on. You can verify the ATTR layer is on by using the VLYR command.

3. Select ATTR/SCHG to change an attribute. This command changes a single attribute of a component in a schematic. The system prompts:

   Select one attribute....

4. Select the TECH attribute shown by the arrow in Figure 2-48 above. The attribute is highlighted and the system prompts:

   Enter new value.

5. Type CMOS and press [Return].

6. Select REDR to redraw the screen.

   Figure 2-49 shows what your screen should look like.

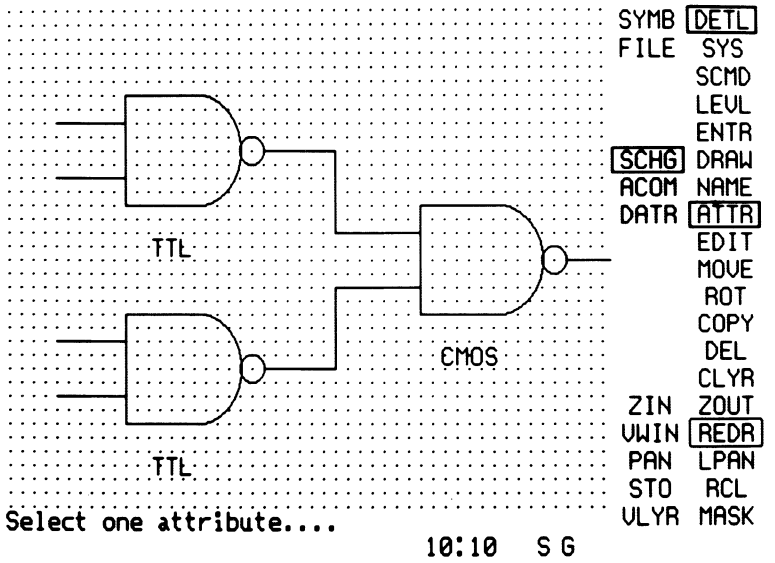


Figure 2-49. TECH Attribute Changed

7. Select FILE/ZAP to erase the screen.

Summary

Now that you have completed this example, you are familiar with the ATTR commands listed below.

Table 2-6. Assigning Attributes

Step	Commands Used
Adding an attribute	ATTR/ACOM
Changing an attribute	ATTR/SCHG

EXAMPLE 5. CREATING A P-CAD COMPATIBLE SYMBOL

Symbols are the building blocks of schematics designed with PC-CAPS. All symbols provided by P-CAD are created using certain layers and standards, such as pin spacing and text positioning. This example is provided so that if you need to create a P-CAD compatible symbol, you will know what layers, standards, and commands to use.

In this example, you will build a simple 2-input NAND gate. The steps in the process are:

1. Set up the environment.
2. Create the symbol graphics.
3. Enter input and output pins.
4. Add text for symbol name.
5. Assign the origin.
6. Add packaging information.
7. Set the Component Type ID.
8. Save the file.

Step 1. Setting Up the Environment

1. Select ZIN to zoom in. The system prompts:

   Select view center...

2. Select a point in the center of the screen.

3. Select SYMB to set the system to symbol mode.

   Figure 2-50 shows your screen display with the screen zoomed in.

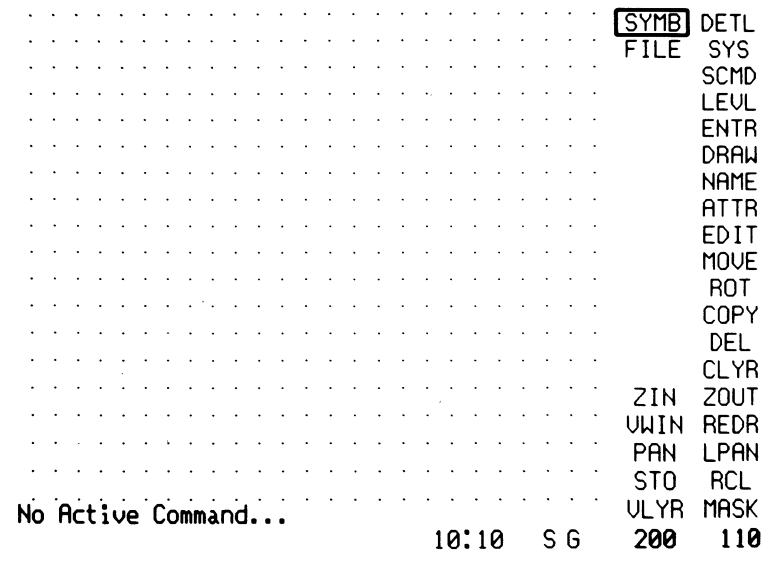
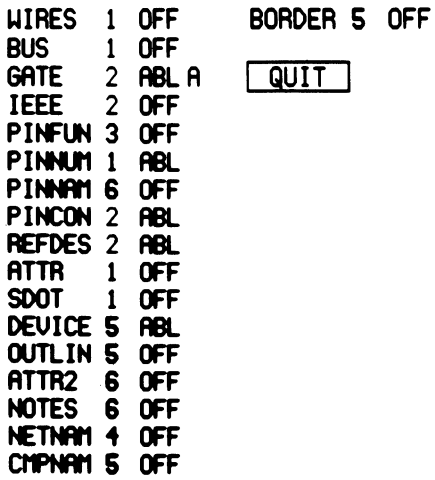


Figure 2-50. Screen Display

4. Select VLYR to display the layer screen and to set the layer assignments. Set the layers GATE, PINNUM, PINCON, REFDES, and DEVICE to ABL status and make GATE the active layer. Turn the rest of the layers OFF.

   Figure 2-51 shows what your screen should look like.



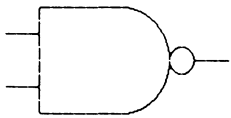
86048

Figure 2-51. Layer Assignments Set for Creating a Symbol

5. Select QUIT to exit the layer display and to return to the drawing view.

Step 2. Creating the Symbol Graphics

To build a 2-input NAND gate, you will create a symbol that looks like Figure 2-52.



86049

Figure 2-52. Graphics for 2-Input NAND Gate

The graphics that you create for a symbol will be displayed whenever you use the symbol in a schematic.

To create the symbol graphics, you will draw:

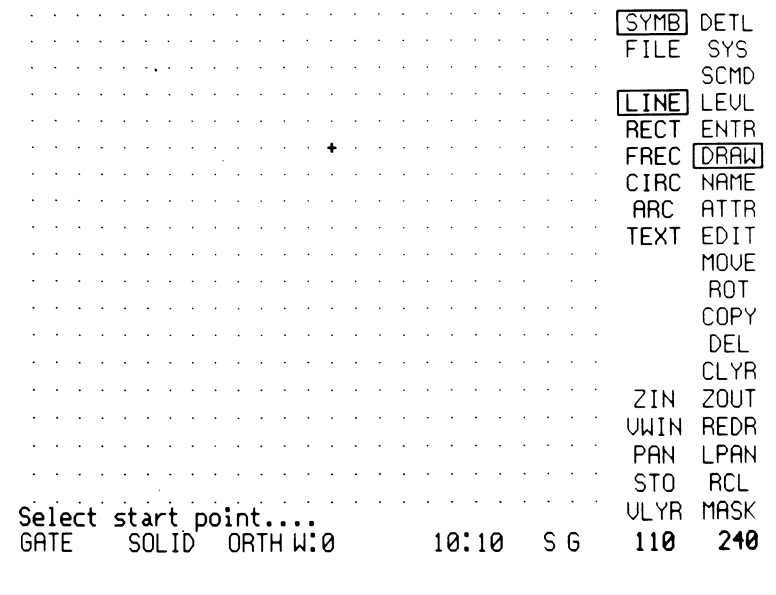
- A 3-sided box
- An arc

- A circle
- Input and output lines

Draw the Box

1. Select DRAW/LINE.

   Figure 2-53 shows the screen display.



86050

Figure 2-53. Screen Display After Selecting DRAW/LINE

Check the status line to review the default parameters for the DRAW/LINE command. These parameters are explained below.

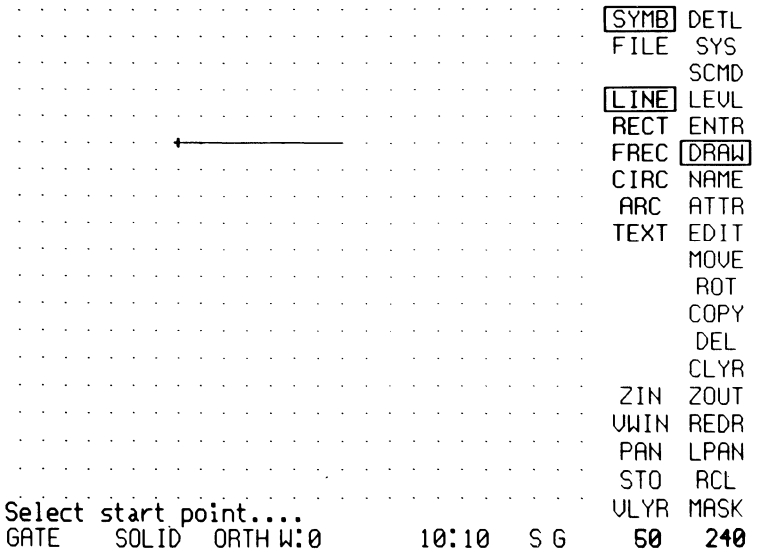
GATE	The active layer is the gate layer.
SOLID	Solid lines will be drawn. (Other choices are dotted and dashed.)
ORTH	Lines will be orthogonal (angles are 90 degrees only).
W:0	Lines will have zero width. This means that lines are drawn as narrow as possible on the screen.
10:10	Grid spacing is set to 10 database units in both x and y directions.
S	Screen grid is on (green is on, red is off).
G	Grid lock is set to on (green in on, red is off).
x,y	The cursor is positioned at the coordinates x,y.

All of these parameters can be changed. Use the defaults unless otherwise instructed.

2. Select the point shown in Figure 2-53. The system prompts:

   Select next point....

3. Select the next point as shown in Figure 2-54. Notice that the line is drawn as you move the cursor.



86051

Figure 2-54. First Line Segment of NAND Gate

NOTE: If you make a mistake while entering the line:

a. Select DEL. The system prompts:

Select the object(s)....

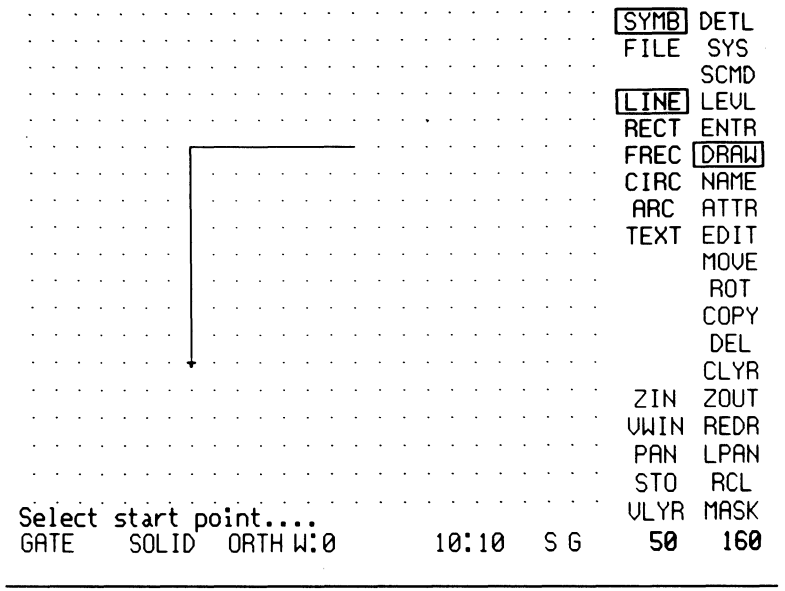
b. Select a point on the line. The line is deleted.

c. Select REDR to redraw the drawing.

Notice that the line reappears. If you select UNDO, the line is restored to the database. If you select any other command, the line disappears and is deleted from the database.

d. Select DRAW/LINE and draw the line correctly.

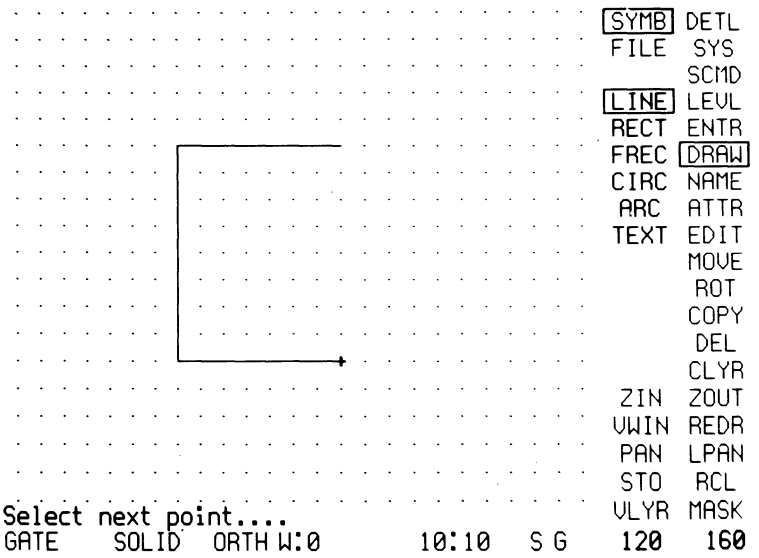
4. Select the next point as shown in Figure 2-55.



86052

Figure 2-55. Second Line Segment of NAND Gate

5. Select the next point as shown in Figure 2-56 to draw the third line segment. Then press Button 2 to end the line.



86053

Figure 2-56. Completed 3-Sided Box Part of NAND Gate

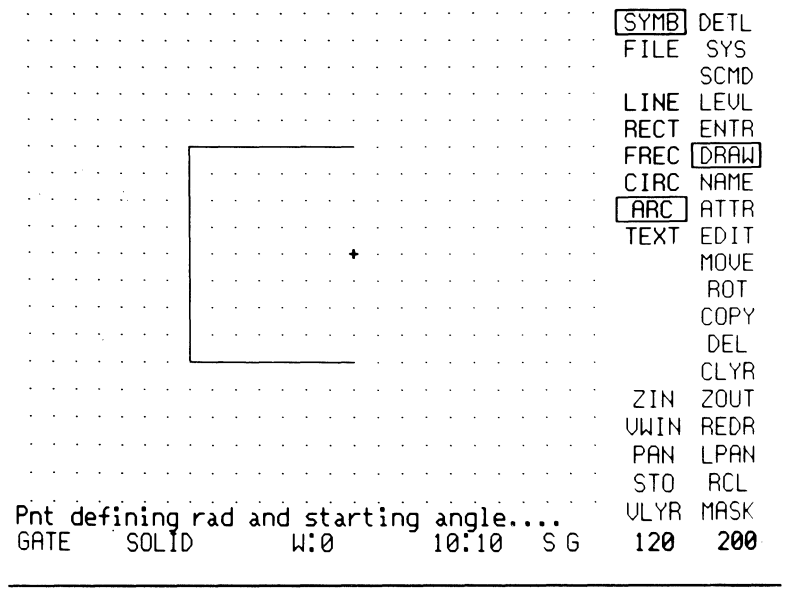
Draw the Arc

1. Select DRAW/ARC. The system prompts:

   Center point:...

2. Select the center point as shown in Figure 2-57. The system prompts:

   Pnt defining rad and starting angle....

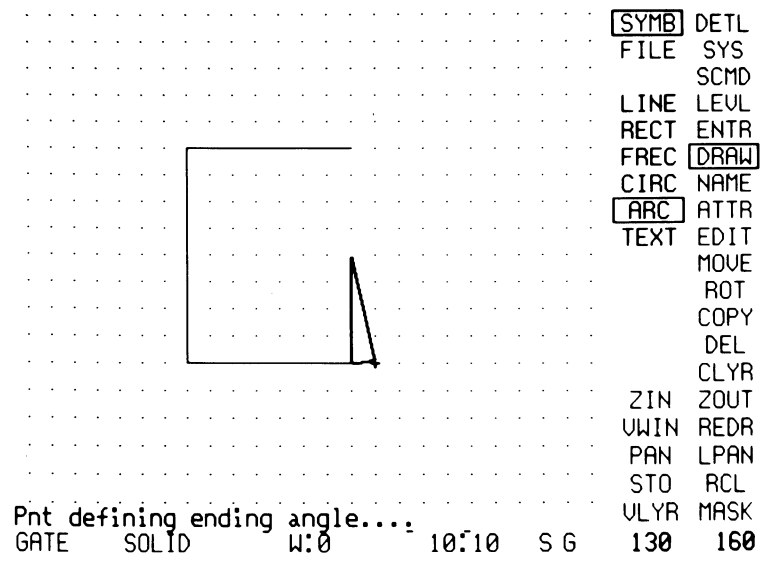


86054

Figure 2-57. Center Point for Arc

3. Select the next point as shown in Figure 2-58. Notice that the radius line moves with the cursor and the system draws the arc counterclockwise. The system prompts:

   Pnt defining ending angle....



86055

Figure 2-58. Radius for Arc

4. Select the next point as shown in Figure 2-59. Notice that the arc appears as you move the cursor.

5. Select REDR to redraw the screen.

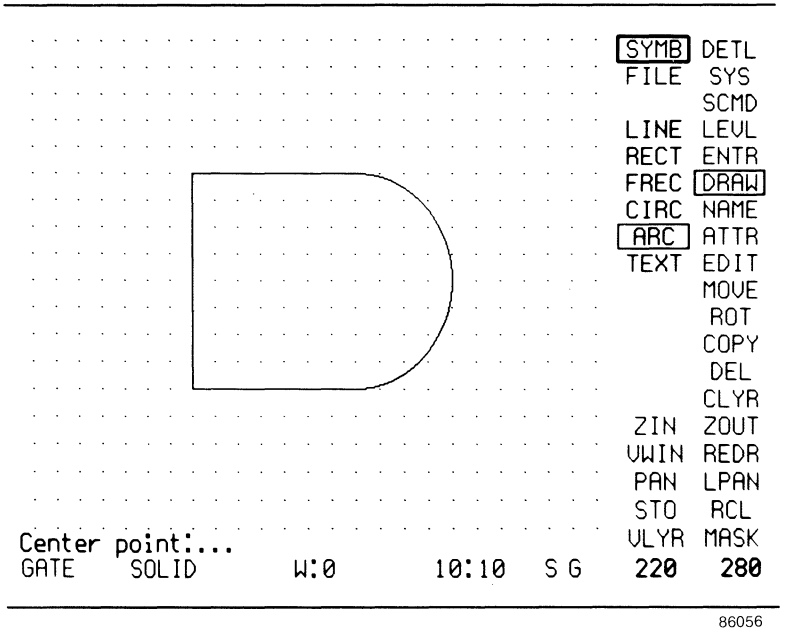


Figure 2-59. Completed Arc for NAND Gate

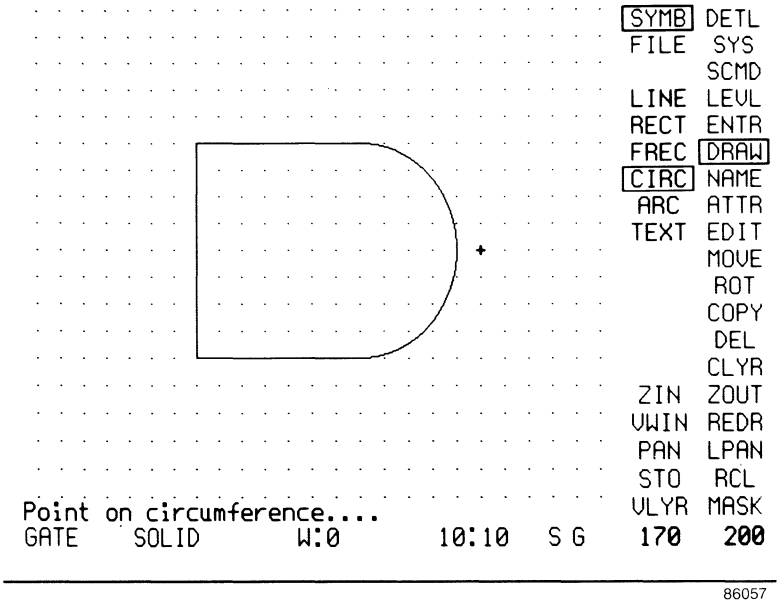
Draw the Circle

1. Select DRAW/CIRC. The system prompts:

   Center point.....

2. Select the point shown in Figure 2-60. The system prompts:

   Point on circumference....



86057

Figure 2-60. Center Point for Circle

3. Select a point one grid division away in any direction. A small circle appears as shown in Figure 2-61.

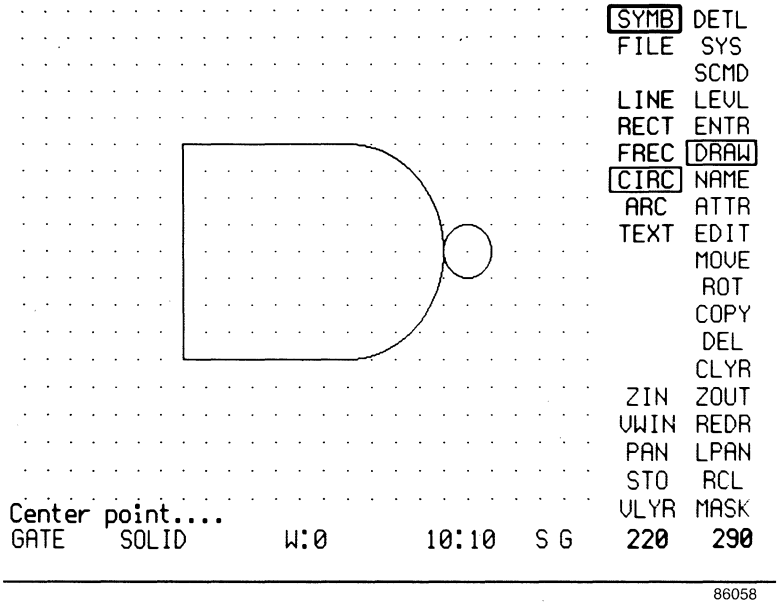


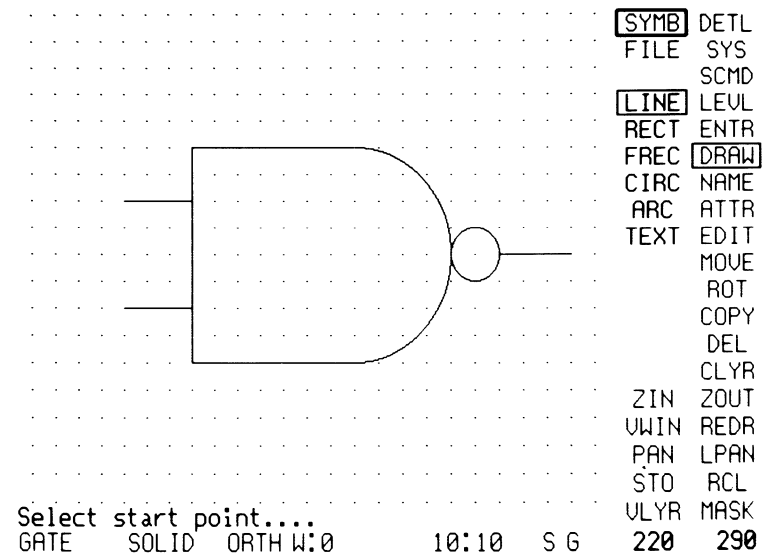
Figure 2-61. Completed Circle for NAND Gate

Draw the Input and Output Lines

1. Select DRAW/LINE to draw the input and output lines. The system prompts:

   Select start point....

2. Draw the input and output lines as shown in Figure 2-62. Press Button 2 to end each line.



86059

Figure 2-62. NAND Gate with All Input and Output Lines

You have completed the symbol graphics for the 2-input NAND gate.

Step 3. Entering Input and Output Pins

To use the NAND gate in a schematic drawing, you must enter the input, output, or I/O pins that are the electrical connection points.

1. Select ENTR/PIN to enter the pins. The system sets the PINCON layer active and prompts:

   Select pin location: ...

   Study the status line on the screen display. The layer that is currently active is displayed and the pin type is displayed.

2. Change the pin type to INPUT.

   Figure 2-63 shows the screen display.

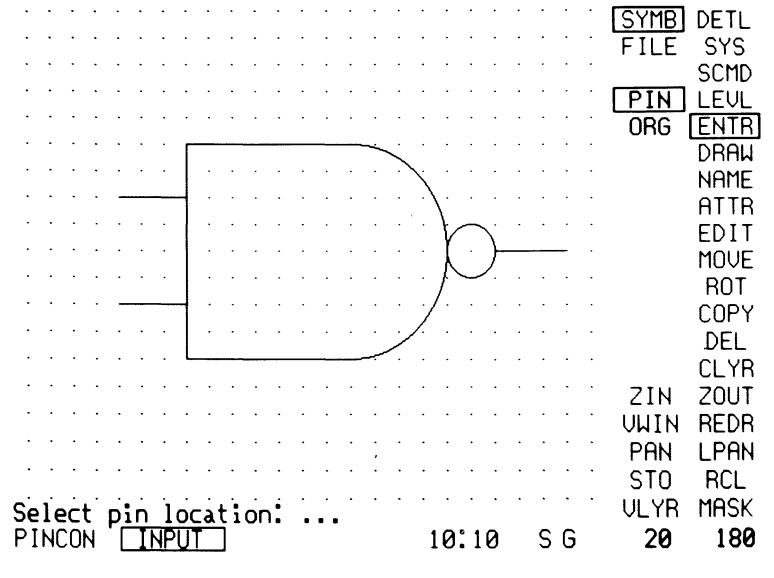
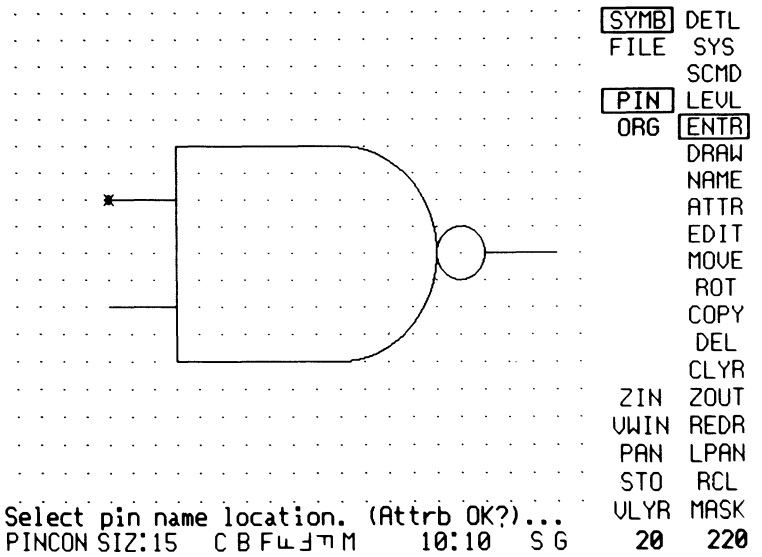


Figure 2-63. Screen Display Showing PINCON Layer Active

3. Select a point to enter input pin A as shown in Figure 2-64.



86061

Figure 2-64. Point to Enter Input Pin A

The pin connection, an X, is displayed and the system prompts:

Select pin name location. (Attrb OK?)...

Note the message

(Attrb OK?)...

This prompt refers to the text attributes of the pin names. For gate-level symbols, P-CAD names the pins for netlist purposes only, not for display on the screen. Therefore, you do not need to set text attributes.

4. Press Button 2. The system prompts:

   Enter pin name:

5. Type INA and press [Return]. Input pin A is displayed as shown in Figure 2-65.

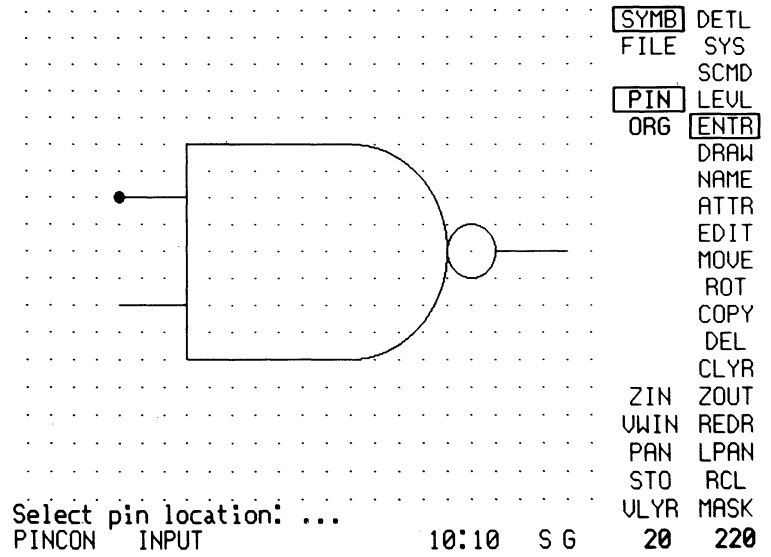


Figure 2-65. Input Pin A Entered

6. Select a point to enter input pin B. The system prompts:

   Select pin name location. (Attrb OK?)

7. Press Button 2. The system prompts:

   Enter pin name:

8. Type INB and press [Return]. Input pin B is displayed.

9. Change the pin type parameter to OUTPUT on the status line.

10. Select a point to enter output pin Y. The system prompts:

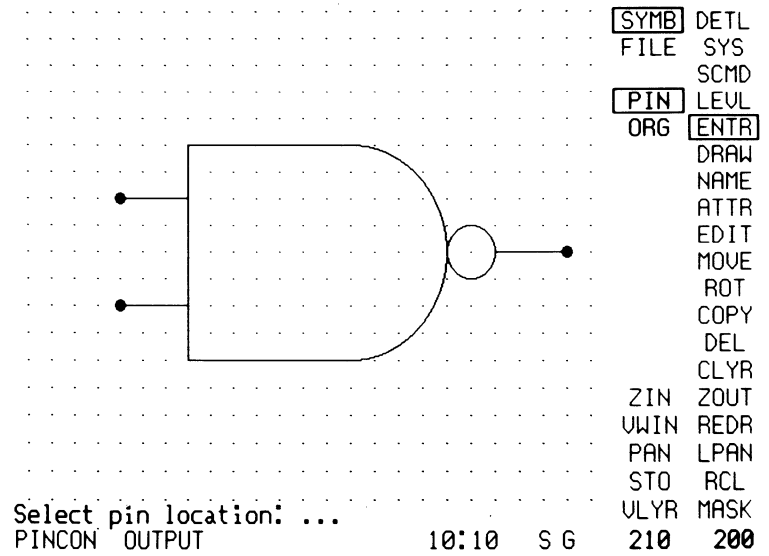
    Select pin name location. (Attrb OK?)

11. Press Button 2. The system prompts:

    Enter pin name:

12. Type OUTY and press [Return]. Output pin Y is displayed.

    Figure 2-66 shows all the pins entered.



86063

Figure 2-66. All Pins Entered

Step 4. Adding Text for the Symbol Name

1. Select DRAW/TEXT to add the symbol name. The system prompts:

   Select Location. (Attributes OK?)...

2. Change the active layer to DEVICE.

3. First set the status line parameters for entering text:

   a. Set the text size to 15.

   b. Make sure the text orientation is set to the first F on the status line.

   c. Set the horizontal text alignment parameter on the status line to C for center as shown in Figure 2-67. The text will be centered from left to right over the cursor position.

   d. Make sure the M for mirroring is set to off (red is off).

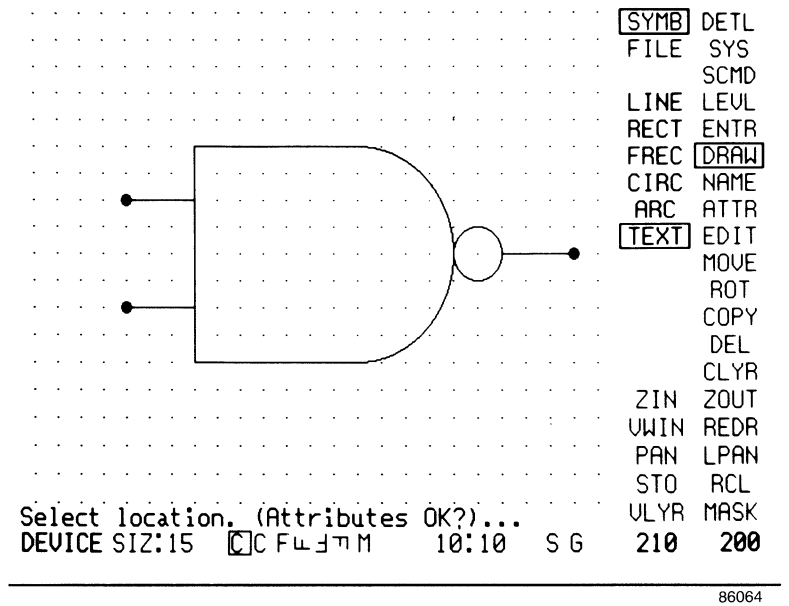
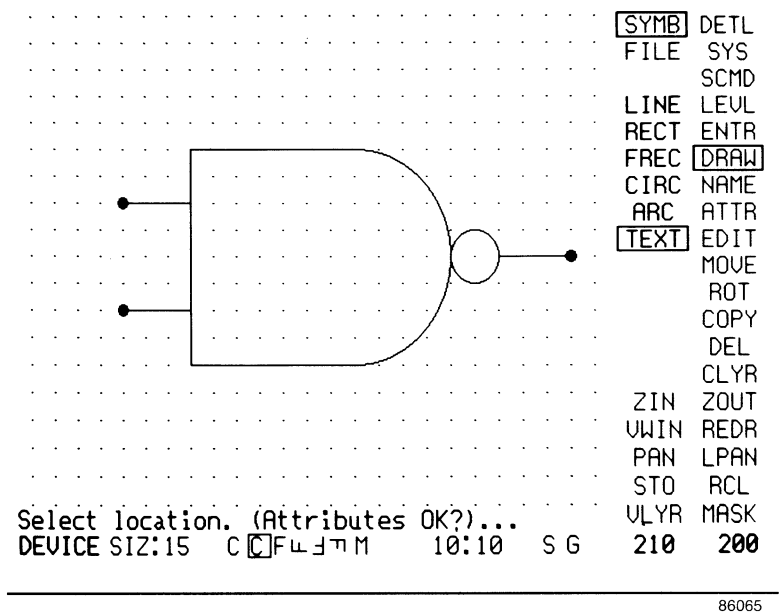


Figure 2-67. Horizontal Center Text Justification

e. Set the vertical text alignment parameter on the status line to C for center as shown in Figure 2-68. This sets the cursor position to the middle (vertically) of the text.



86065

Figure 2-68. Vertical Center Text Justification

4. Select a point to place the symbol name as shown in Figure 2-69. The system prompts:

   Type in text.

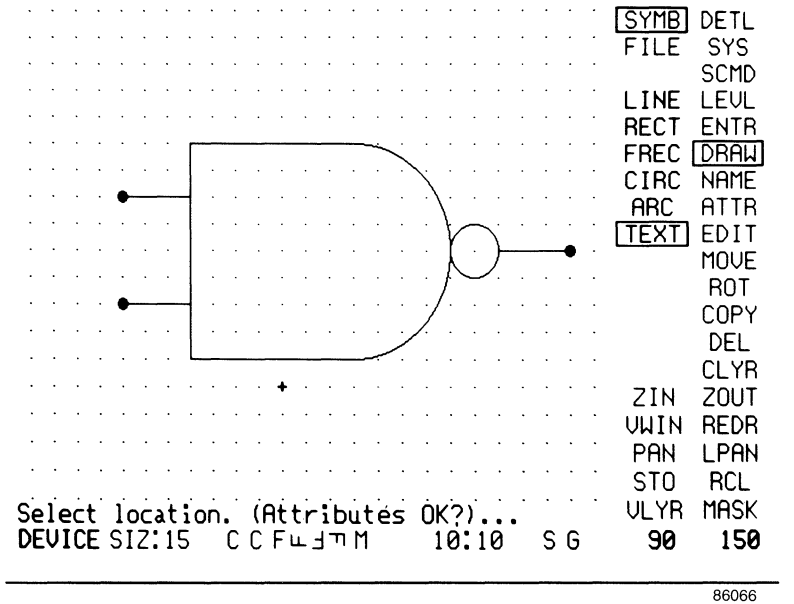
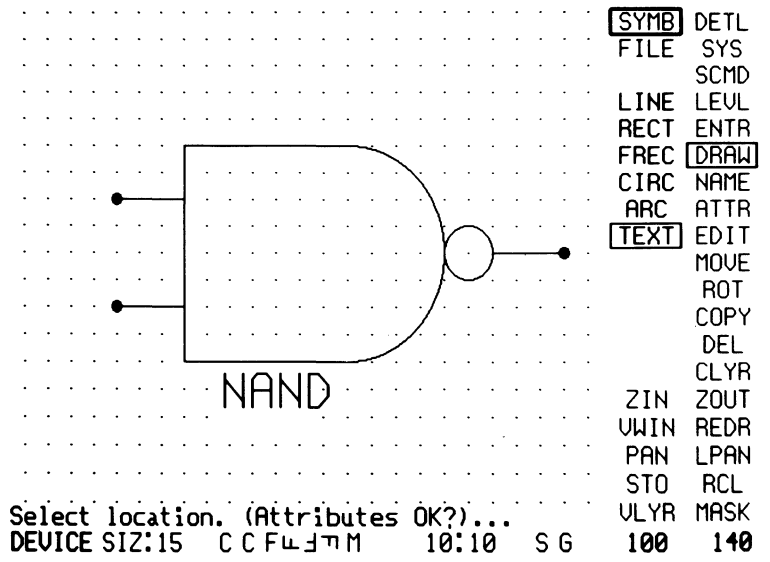


Figure 2-69. Point for Entering Text

5. Type NAND and press [Return].

   Notice that the text enters at the center point and is automatically centered as shown in Figure 2-70.



86067

Figure 2-70. NAND Gate with Text

Step 5. Assigning the Origin

The origin is the reference point used when you place the gate in a schematic drawing. Although you can choose any point in the drawing as an origin, P-CAD's convention is the lower left pin.

1. Select ENTR/ORG to assign the origin.
   The system prompts:

   Select the origin....

2. Select input pin B.

Step 6. Adding Packaging Information

NOTE: Adding packaging information is required only if you will be taking your design through physical layout.

1. Select SCMD/PNLC. The system prompts:

   Enter gates per package:

   as shown in Figure 2-71.

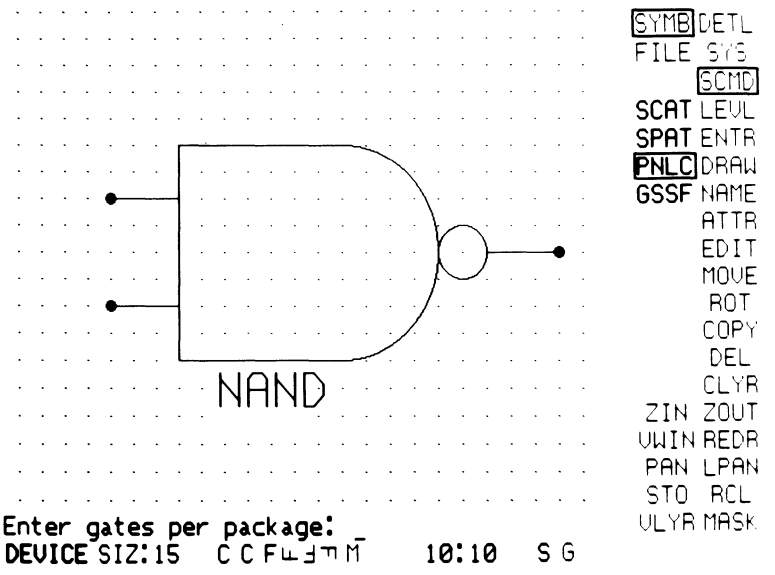


Figure 2-71. Prompt for SCMD/PNLC Command

2. Type 4 and press [Return]. The system prompts:

   Enter number of pins per gate:

3. Type 3 and press [Return]. The system sets the REFDES layer active and prompts:

   Select loc for ref designator....

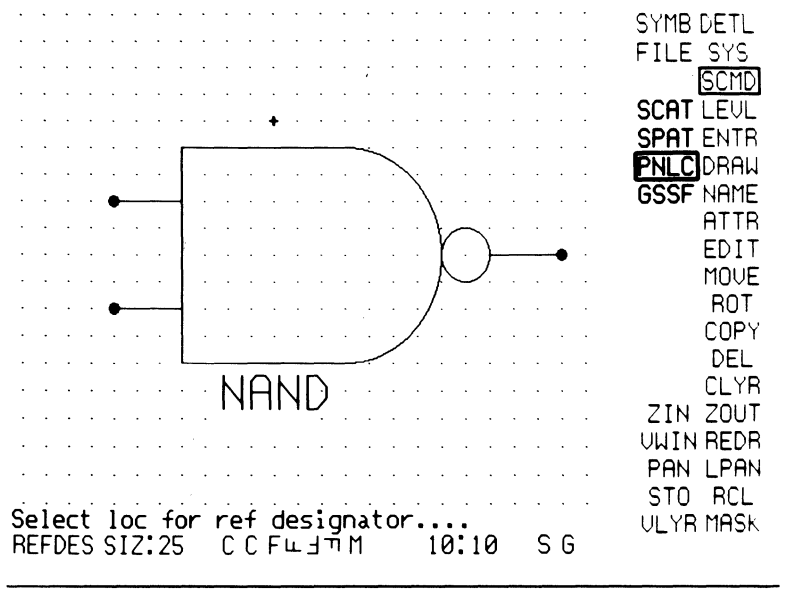
4. Before you enter the location for the reference designator, set the status line parameters:

   a. Set the text size to 25.

   b. Set the text justification to C C.

5. Select the point shown in Figure 2-72. The system sets the PINNUM layer active and prompts:

   Select loc for pin number....



86069

Figure 2-72. Cursor Location for Reference Designator

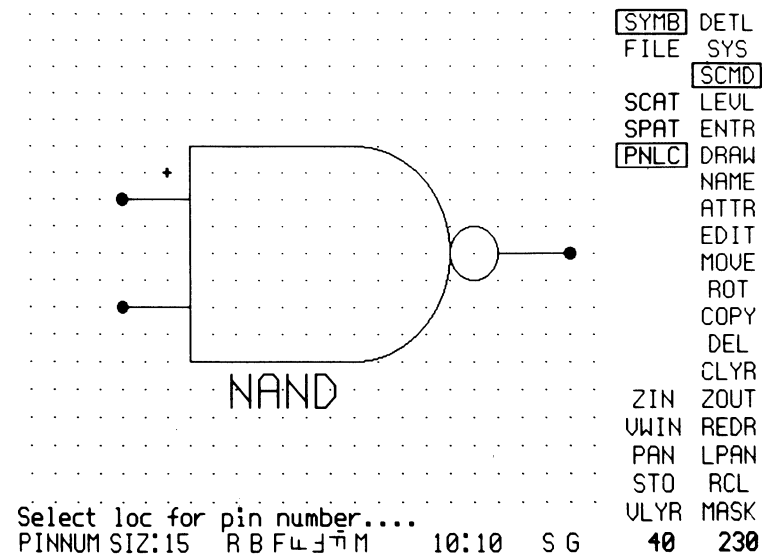
6. Before you enter the location for the pin numbers, set the status line parameters:

   a. Set the text size to 15.

   b. Set the text justification to R B.

7. Select the point shown in Figure 2-73 to indicate the location of the first pin number. The second input pin is highlighted and the system continues prompting:

   Select loc for pin number...



86070

Figure 2-73. Cursor Location for First Input Pin Number

8. Select the next point shown in Figure 2-74 to indicate the location of the second input pin number. The output pin is highlighted and the system continues prompting:

   Select loc for pin number...

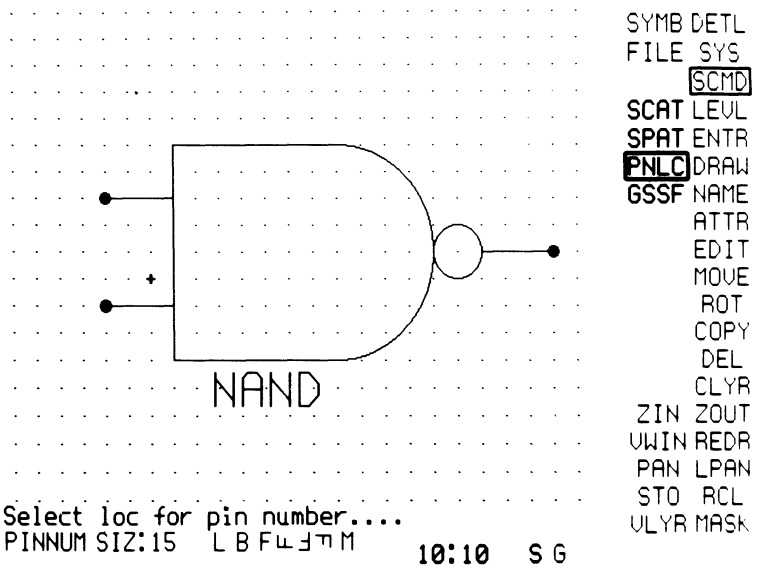


Figure 2-74. Cursor Location for Second Input Pin Number

9. Before you enter the location for the output pin number, change the text justification to L B.

10. Select the point shown in Figure 2-75 to indicate the location of the output pin number. Then the system prompts:

    Enter package pin number for INA:
    Gate assigned to section A.

    and input pin INA is highlighted.

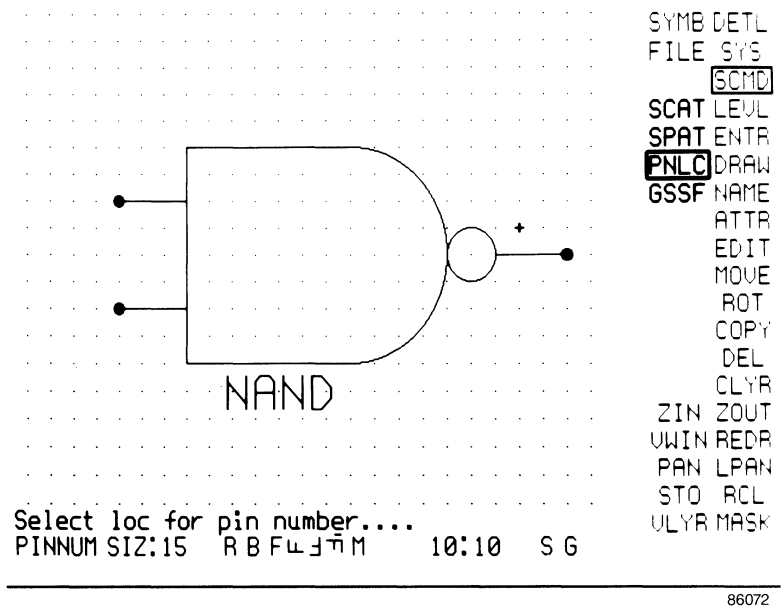


Figure 2-75. Cursor Location for Output Pin Number

11. Type 1 and press [Return].

12. Continue entering pin numbers in response to the prompts for pins INB and OUTY of section A and pins INA, INB, and OUTY in section B, C, and D as given in Table 2-7.

Table 2-7. Package Pin Numbers

Section	Pin Name	Component Pin
A	INA	1
A	INB	2
A	OUTY	3
B	INA	4
B	INB	5
B	OUTY	6
C	INA	10
C	INB	9
C	OUTY	8
D	INA	12
D	INB	13
D	OUTY	11

Step 7. Setting the Component Type ID

1. Select SCMD/SCAT to set the Component Type ID. The system prompts:

   Workcomp. Old type= 255 New type=

2. Type 255 and press [Return].

Step 8. Saving the File

1. Select FILE/SAVE to save the file. The system prompts:

   Enter file name:

2. Type NAND and press [Return].

3. Select FILE/ZAP to clear the screen.

Summary

By completing this example, you have learned how to create a P-CAD compatible symbol. Review the steps below to check your understanding of the process.

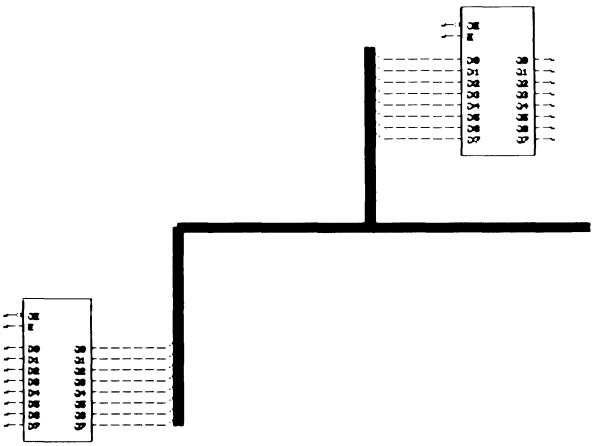
Table 2-8. Creating a Symbol

Step	Commands Used
Setting up the environment	ZIN, SYMB VLYR
Creating the symbol graphics	DRAW/LINE DRAW/ARC DRAW/CIRC
Entering input and output pins	ENTR/PIN
Adding text for symbol name	DRAW/TEXT
Assigning the origin	ENTR/ORG
Adding packaging information	SCMD/PNLC
Setting the Component Type ID	SCMD/SCAT
Saving the file	FILE/SAVE
Erasing the screen	FILE/ZAP

EXAMPLE 6. CREATING A BUS

In this example, you will create the address bus shown in Figure 2-76. The address bus contains eight individual signals named A0 through A7. These eight signals can be routed to various components on the schematic as a bus rather than eight individual wires. The steps in this example are:

1. Set up the environment.
2. Place the components.
3. Enter the wires.
4. Name the nets.
5. Draw the bus line.



86073

Figure 2-76. Address Bus

Step 1. Setting Up the Environment

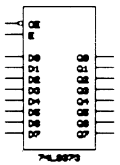
1. Make sure the system is set to detail (DETL) mode.

2. Using the VLYR command, set WIRES, BUS, GATE, and PINCON to ABL status and make WIRES the active layer. Set PINNAM to ON status. Turn the rest of the layers off.

Step 2. Placing the Components

1. Select ENTR/COMP to enter the component shown in Figure 2-77 twice. The system prompts:

   :
   Comp-file-name x/y-scales



86074

Figure 2-77. Component to Be Placed

NOTE: For this example, enter the component at half its size.

2. Type 74LS373 50 and press [Return]. The system prompts:

Select loc to place comp. (Orientation OK?)

3. Select two locations to place the component as shown in Figure 2-78.

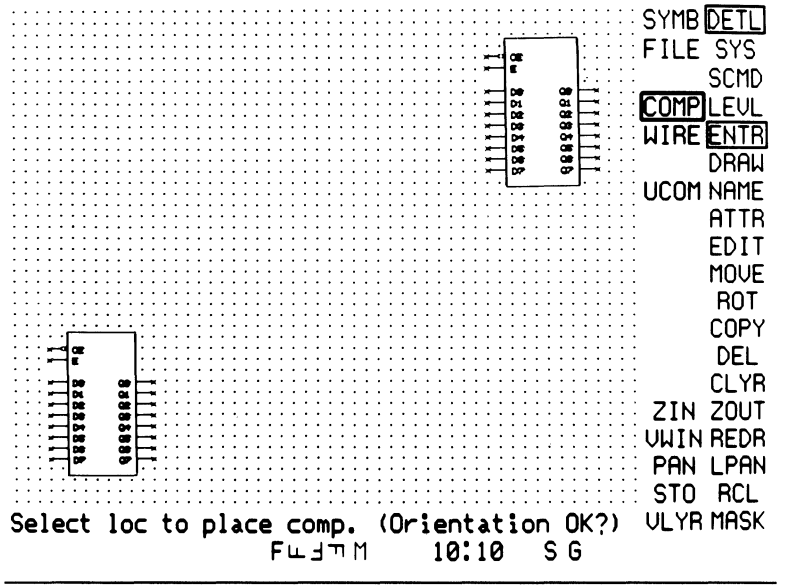


Figure 2-78. Component Entered Twice

Step 3. Entering the Wires

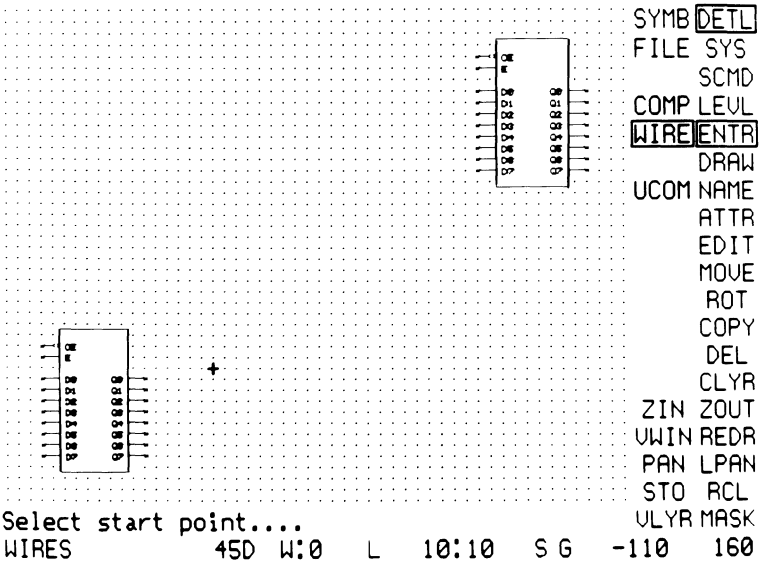
1. Select ENTR/WIRE to enter the wires. The system prompts:

   Select start point...

2. Change the line mode parameter on the status line to 45D.

3. Select the point shown in Figure 2-79 for the start point of the first wire. The system prompts:

   Select the next point...



86076

Figure 2-79. Cursor Location for Start of First Wire

4. Select the next point to enter the wire as shown in Figure 2-80.

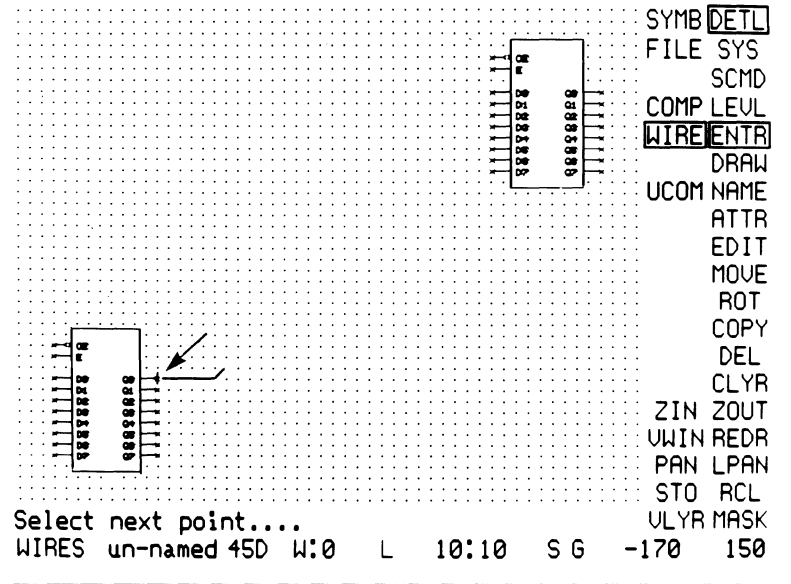


Figure 2-80. First Wire Entered on Lower Left Component

Now you may enter the rest of the wires using the ENTR/WIRE command, or you may place copies of the first wire you entered using the COPY command as described in the following steps.

5. Select COPY. The system prompts:

   Select the object(s)....

6. Select a point on the wire. The system prompts:

   Select new location....

7. Select the location for the second wire and press Button 1. The system continues prompting:

   Select new location....

8. Continue placing copies of the first wire until the output pins on the lower-left component have connecting wires.

Step 4. Naming the Nets

1. Select NAME/NET. The system sets the NETNAM layer active and prompts:

   Select a net....

2. Select a point on the top wire to identify the first net to be named. The net is highlighted and the system prompts:

   Enter net name.

3. Type A0 and press [Return]. The system prompts:

   Name = A0. Select location....

4. Press Button 2 to suppress the net name. The net name is not displayed.

   NOTE: If you want the net name displayed, move the cursor to the desired location and press Button 1.

5. Continue naming the nets on the lower left component as described in steps 2 through 4 above naming the nets sequentially A1 through A7.

6. Now enter wires on the upper right component as shown in Figure 2-81. Then use the NAME/NET command to name the nets sequentially starting with A0. When you enter the same net name for the wires as you did for the lower left component, the system highlights the nets and prompts:

   Net A0 exists. Merge? YES NO

7. Select YES. The system prompts:

   Name = A0. Select location .....

   Nets merged.

8. Press Button 2 to suppress the net name.

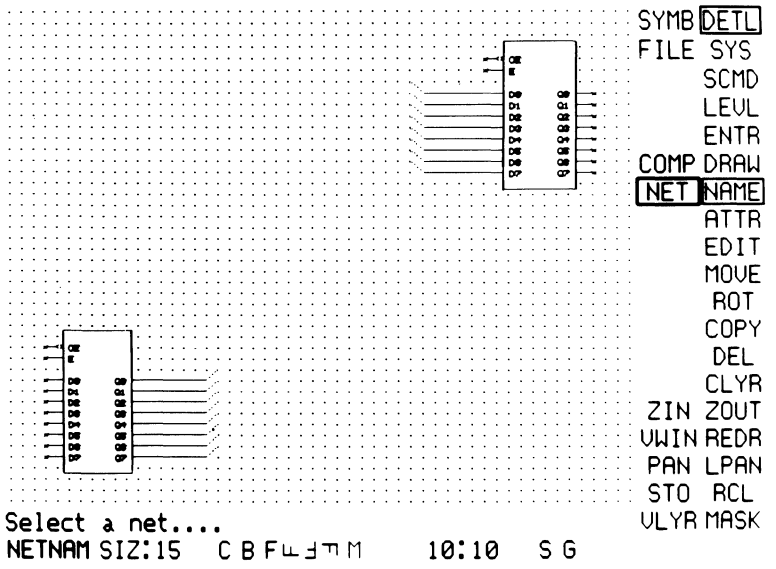


Figure 2-81. All Wires Entered

Step 5. Drawing the Bus Line

1. Select DRAW/LINE. The system prompts:

   Select start point....

2. Change the active layer to BUS.

3. Change the line mode on the status line to ORTH.

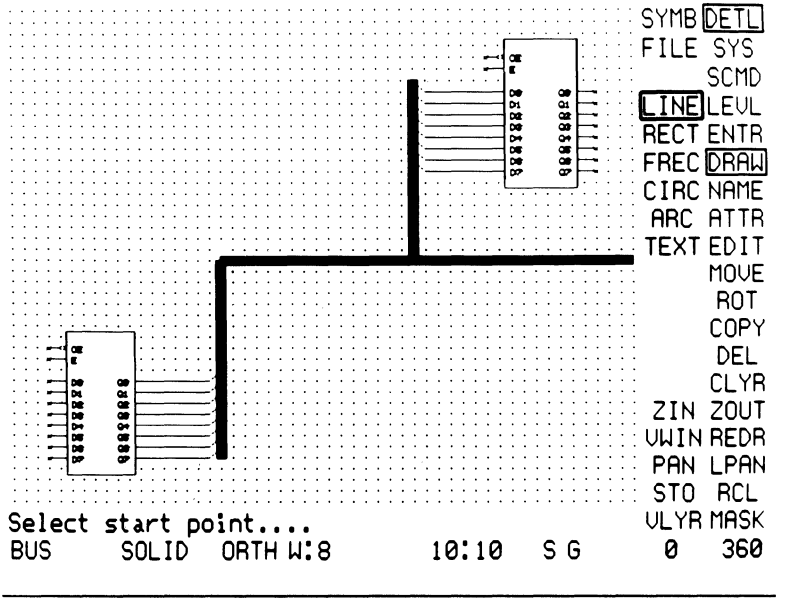
4. Select W:0 on the status line to change the line width. The system prompts:

   Enter line width:

5. Type 8 and press [Return].

6. Draw the line as shown in Figure 2-82.

   NOTE: The bus line is for graphical representation only. Electrical continuity was established when you entered the same net name for each of the wires between the bus signals.



86079

Figure 2-82. Address Bus

7. Select FILE/ZAP to clear the screen.

Summary

By completing this example, you have learned how to create a bus. Review the steps below to check your understanding of the process.

Table 2-9. Creating a Bus

Step	Commands Used
Setting up the environment	DETL VLYR
Placing the components	ENTR/COMP
Entering the wires	ENTR/WIRE
Naming the nets	NAME/NET
Drawing the bus line	DRAW/LINE

EXAMPLE 7. CREATING A SOLDER DOT AND A SPECIAL SYMBOL FILE

In "Example 1. Creating a Schematic," the system inserted a 0.1-inch round default solder dot at all T-junctions. In this example, you will create a 0.2-inch square solder dot and its associated special symbol file, and you will link the special symbol file to your schematic.

A user-created solder dot is a database file which contains the graphics for a solder dot. The solder dot may be any shape or size that can serve as a connection point at T-junctions; for example, circular, square, oval, or diamond. A user-created solder dot is linked to T-junctions in a schematic according to a special symbol file. If you don't link a special symbol file to your schematic, the system displays a 0.1-inch round solder dot at all T-junctions.

You create schematics and solder dots in independent databases. The PC-CAPS SCMD/GSSF command links the two databases.

The steps in the process are:

1. Create the solder dot.
2. Create the special symbol file.
3. Link the special symbol file.

Step 1. Creating the Solder Dot

In this step you will create a solder dot. The procedure you will follow in creating a solder dot is:

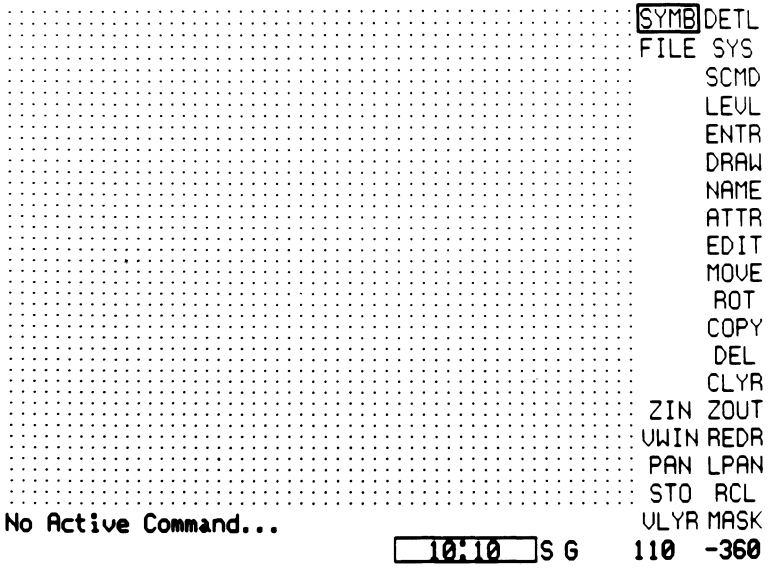
1. Set up the environment.
2. Draw the graphics.
3. Assign the origin.
4. Set the Component Type ID.
5. Save the solder dot file.

Setting Up the Environment

1. Select SYMB to set the system to symbol mode.
2. Select the X:Y grid spacing parameter on the status line as shown in Figure 2-83. The system prompts:

   Enter x grid size:

3. Type 5 5 and press [Return] to set the grid spacing to 5:5.



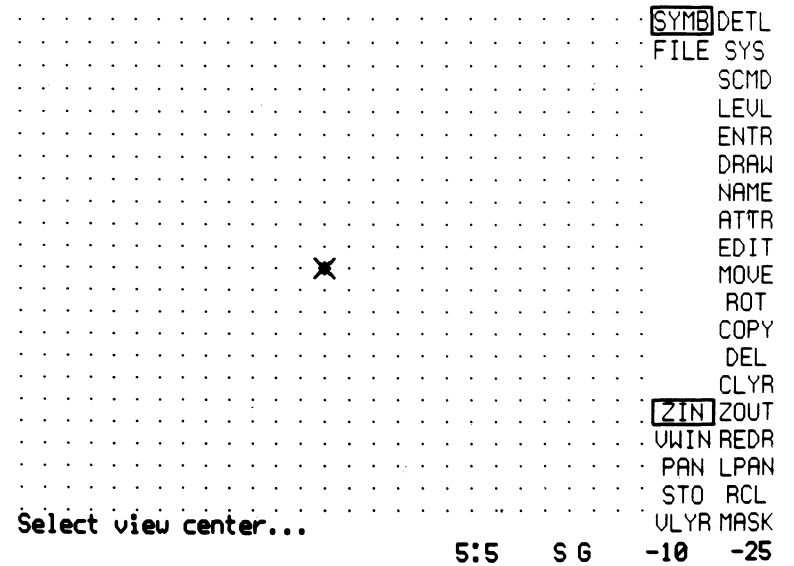
86080

Figure 2-83. Selecting X:Y Grid Spacing Status Line Parameter

4. Select ZIN to zoom in. The system prompts:

   Select view center...

5. Move the cursor to a point in the center of the screen and press Button 1 three times to zoom in three levels as shown in Figure 2-84. Press Button 2 to terminate the ZIN command.



86081

Figure 2-84. Screen Display After Zooming In

Drawing the Graphics

1. Change the active layer to SDOT.

2. Select DRAW/RECT to begin drawing the solder dot graphics. The system prompts:

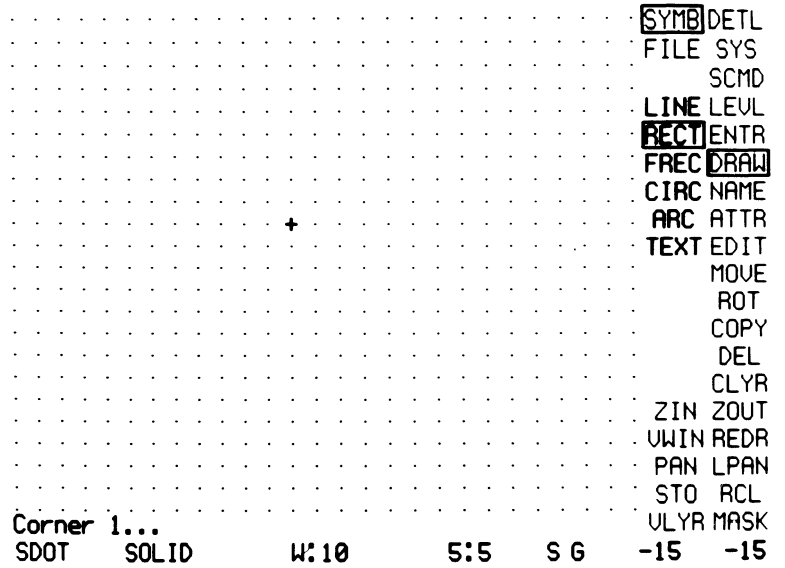
   Corner 1...

3. Before you select corner 1, make sure the line width (W: on the status line) is set to 10.

4. Draw a rectangle 20 mils square in the center of the screen as follows.

   a. Select the point shown in Figure 2-85. The system prompts:

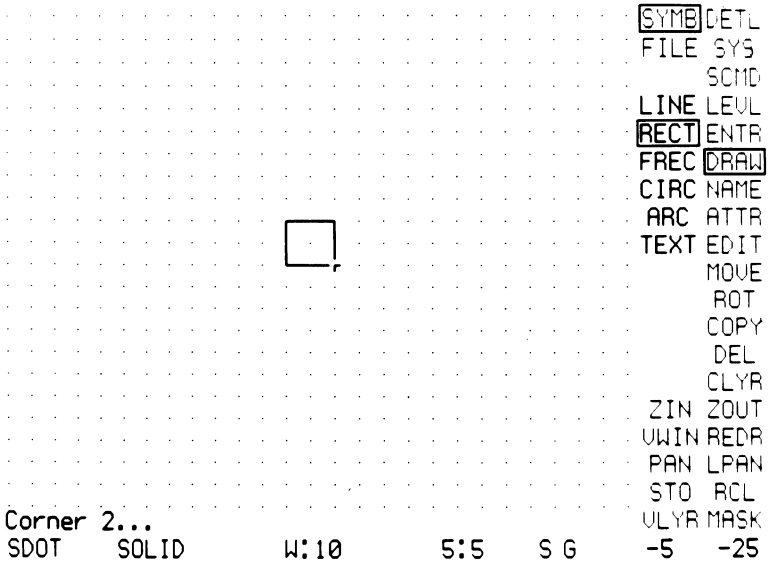
      Corner 2...



86082

Figure 2-85. Cursor Location for First Corner

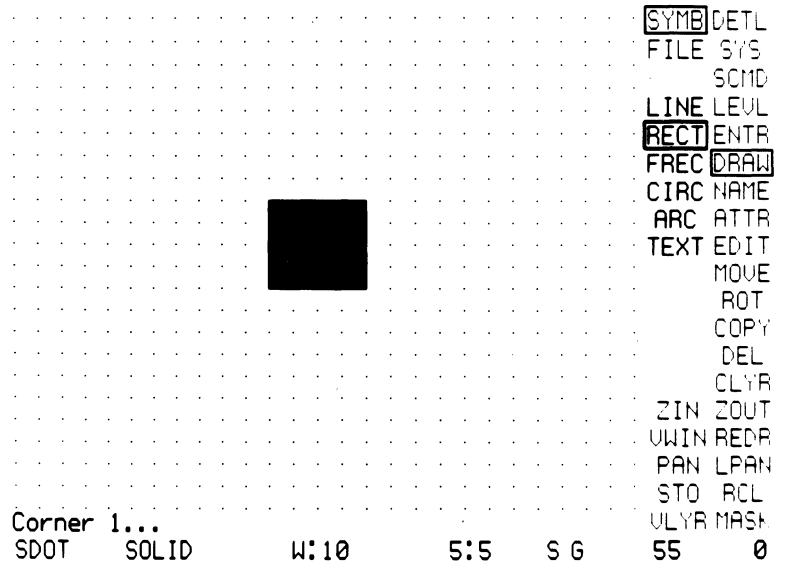
b. Select the next point shown in Figure 2-86.



86083

Figure 2-86. Cursor Location for Second Corner

The rectangle represents a 0.2-inch square solder dot. Your solder dot should look like the one in Figure 2-87.



86084

Figure 2-87. Solder Dot

Assigning the Origin

1. Select ENTR/ORG to set the origin of the solder dot. The system prompts:

   Select the origin...

2. Select a point in the center of the solder dot.

Setting the Component Type ID

1. Select SCMD/SCAT to set the Component Type ID. The system prompts:

   Workcomp. Old type= 255 New type=

2. Type "0" and press [Return].

Saving the Solder Dot File

1. Select FILE/SAVE to save your completed solder dot file. Name the file 2S.SDT to represent the 0.2-inch square solder dot you just created.

   The special symbol file you will create in "Step 2. Creating the Special Symbol File," will insert the solder dot graphics into the schematic when linked by the SCMD/GSSF command.

2. Select FILE/ZAP to clear the screen.

3. Select SYS/QUIT to exit from PC-CAPS.

Step 2. Creating the Special Symbol File

In this step you will create the special symbol file shown in Figure 2-88. You can do this with or without a text editor. In "Step 3. Linking the Special Symbol File," you will link the special symbol file to your schematic. The special symbol file will insert solder dot graphics at all T-junctions, overriding the default solder dot graphics.

```
% Special Symbol File
%
15 * 2S.SDT % 0.2-inch square
```

Figure 2-88. Special Symbol File

Figure 2-88 shows the format of the special symbol file.

The first column list the pin type. The pin type must be 15.

The second column shows an asterisk (*).

The third column lists the solder dot filename.

> NOTE: A percent sign (%) on a line indicates that the text to the right of the percent sign is a comment that is not read by PC-CAPS.

If your system has a text editor or word processing program, use it to create a file named <filename>.SSF, and then type the information shown in Figure 2-88 into it.

When creating the special symbol file, you must enter the space(s) shown in Figure 2-88 using the space bar. Only one space is required. Spaces entered with a tab key are not allowed.

If your text editor distinguishes between document mode and nondocument mode, use nondocument mode. Otherwise, use the procedure for creating a special symbol file without a text editor, which follows.

To create the special symbol file without a text editor:

1. From the DOS **C:\PCAD\PROJ0>** prompt, type:

   COPY CON: TUTORX.SSF [Return]

2. Type in the information exactly as shown in Figure 2-88.

3. Close the file TUTORX.SSF by pressing [Ctrl]-[Z] and then [Return].

   NOTE: If you make a mistake while creating TUTORX.SSF without a text editor, then repeat steps 1 through 3 above.

4. From the DOS prompt **C:\PCAD\PROJ0>**, re-enter PC-CAPS.

Step 3. Linking the Special Symbol File

1. Load the file TUTORA.SCH.

2. Make sure the SDOT layer is turned on.

3. Select SCMD/GSSF to link the special symbol file to the schematic. The system prompts:

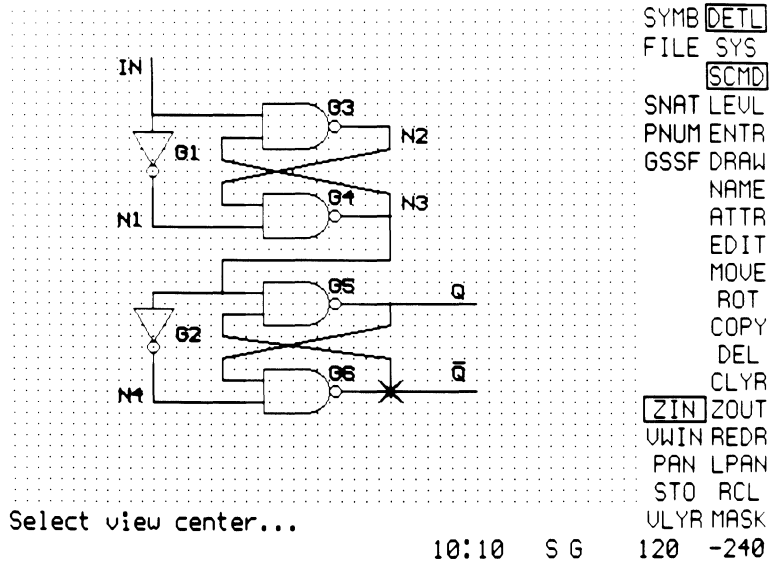
   Enter file name:

4. Type TUTORX.SSF and press [Return].

5. Select ZIN to zoom in. The system prompts:

   Select view center...

6. Move the cursor to the point shown by the arrow in Figure 2-89.



86085

Figure 2-89. Cursor Location to Zoom In

7. Press Button 1. Then press Button 2 to end the command. The screen display should look like Figure 2-90.

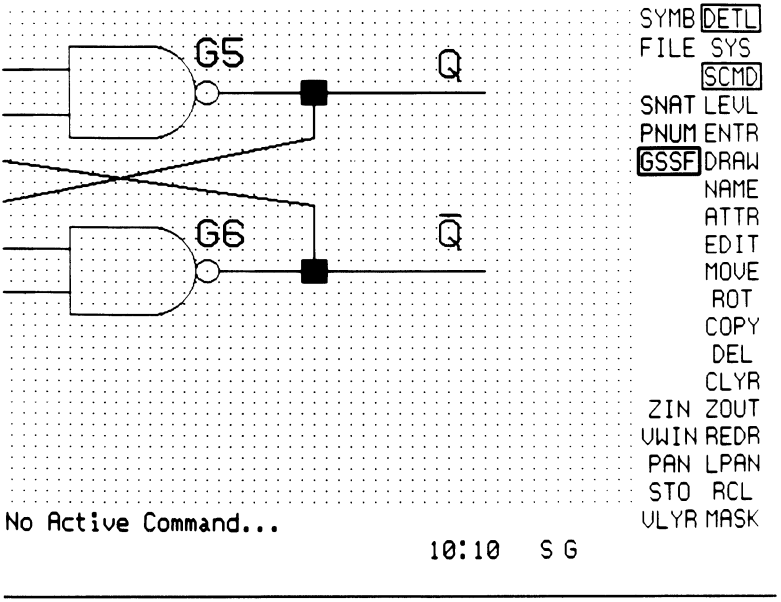


Figure 2-90. Linking the Special Symbol File

8. Select SCMD/GSSF again to unlink the special symbol file. Press Button 2 this time instead of entering a filename.

9. Select REDR to display the default solder dots.

10. Select FILE/ZAP to clear the screen.

Summary

You have now completed the essential steps in creating and using solder dots and special symbol files. Review the steps listed in Table 2-10 to check your understanding of the process.

Table 2-10. Creating a Solder Dot and a Special Symbol File

Step	Commands Used
Creating the solder dot	
a. Setting up the environment	SYMB ZIN
b. Drawing the graphics	DRAW/RECT
c. Assigning the origin	ENTR/ORG FILE/SAVE FILE/ZAP
Creating the special symbol file	
Linking the special symbol file	FILE/LOAD SCMD/GSSF

CHAPTER SUMMARY

After you have completed the examples in Part B, you are ready to start your own design work using PC-CAPS.

However, for additional assistance, the next chapter titled "Commands" contains detailed information about the PC-CAPS commands arranged in an easy-to-find format.

CHAPTER 3. COMMANDS

This chapter is a command reference for PC-CAPS. All menu and keyboard commands are listed in alphabetical order. Each command description is divided into four parts:

FUNCTION: A short description of the purpose of the command.

REMARKS: Some things you need to know about the command, including an example of the status line appropriate to the command.

OPERATION: Detailed procedures for using the command, including system prompts and user responses.

ERROR MESSAGES: Explanation and probable solutions for error messages where appropriate.

Status line parameters are listed in a section at the back of this chapter.

SYMBOL AND DETAIL MODES

Commands in PC-CAPS are executed in symbol (SYMB) or detail (DETL) mode.

In symbol mode, main menu commands are displayed in red. In detail mode, main menu commands are displayed in green.

The mode is chosen by selecting one of these two commands. For more information, see the command description for SYMB and DETL.

MAIN MENU AND SUBMENU COMMANDS

Many of the main menu commands contain a set of submenu commands that are displayed in a different color when you select the main command. This combination of main and submenu commands is always noted in this manual as capital letters separated by a slash, such as DRAW/LINE.

COMMAND SELECTION

To select a command from the menu, use the mouse to move the cursor over the command and press Button 1.

In addition, you can enter any of the menu commands from the keyboard by typing a slash (/). The system prompts:

Menu command:

Type the command and press [Return].

Then, to select a submenu command, follow the same procedure: type a slash, type the subcommand, and press [Return].

For example, type / and then SYS [Return] followed by / and PLOT [Return] to create a plot file.

To cancel a command, select another one from the menu with the mouse, press Button 2, or press [Esc].

KEYBOARD COMMANDS

Some PC-CAPS commands must be entered from the keyboard. Following is a list of the keyboard commands:

/CFIL (Command Log File)
/EXE (Execute)
/INTR (Interrupt Execution)
/LANG (Line Angle)
/LWID (Line Width)
/LSTY (Line Style)
/LYRN (Layer Name)
/MAC (Macro)
/MEND (Macro End)
/RESU (Resume Execution)
/SGAT (Set Global Attribute)
/WAIT

To enter a keyboard command, type a slash (/). The system prompts:

Menu command:

Type the command and press [Return]. For more information, refer to the specific command description in this chapter.

FUNCTION KEYS

The ten function keys on the keyboard provide quick access to the status line parameters and other functions.

Key	Function
[F1]	**ACTIVE LAYER** - Cycles through the ABLed layers to allow you to select the active layer. The current active layer is displayed on the status line.
[F2]	**LINE MODE** - Toggles between angle, 45 degree, and orthogonal modes of line or wire entry. The current mode is displayed on the status line as ANGL, 45D, or ORTH.
[F3]	**NET NAME** - Allows you to assign or change the active net name when using the ENTR/WIRE command.
[F4]	**COMPONENT NAME** - Allows you to name the component while you are placing it with the ENTR/COMP command.
[F5]	**TEXT SIZE** - Allows you to enter a new text size when using a text command.
[F6]	**TEXT ORIENTATION** - Cycles through the text orientation choices. The current orientation is shown by the green F on the status line.

Key	Function
[F7]	**GRID DISPLAY** - Toggles the grid display on and off. The S on the status line indicates the grid display status: green is on; red is off.
[F8]	**GRID LOCK** - Toggles the grid lock. The G on the status line indicates the grid lock status: green is on; red is off.
[F9]	**STATUS LINE TOGGLE** - Toggles the cursor location between the drawing area of the screen and the status line below.
[F10]	**MENU TOGGLE** - Toggles the cursor location between the drawing area of the screen and the menu on the right.

Mode: SYMB
DETL

ATTR

(Attribute)

FUNCTION:

Enables the submenu for changing, adding, or deleting attributes.

OPERATION:

Select ATTR and one of the following submenu choices:

SCHG (Single Change)
ACOM (Add Component Attribute)
DATR (Delete Attribute)

Mode: SYMB
DETL

ATTR/ACOM

(Attribute/Add Component Attribute)

FUNCTION:

Adds an attribute to a component.

REMARKS:

Attributes can be added to a symbol in the SYMB mode or to a component in a schematic in the DETL mode.

An attribute consists of two parts, a keyword and a value, separated by an equal sign (=) as in these examples:

TECH="LS"

PCL=(1,1,"D","D")

DIST="(AMPLE Electronics)"

A keyword must start with a letter, and can be up to seven characters long. A keyword must not have any spaces in it. A keyword is a text string that identifies the information supplied by the attribute value.

ATTR/ACOM

The value can be any text or numerical string desired. To include blank spaces in the text string of the value, add beginning and ending parentheses. Without parentheses, the value is truncated after a space.

To include more than one value with a keyword, separate the values with commas and enclose the values in parentheses.

In addition, P-CAD recommends you put literal strings in quote marks.

Some interface programs require quote marks. See the interface user's manuals for more information.

The keyword, equal sign (=), and value cannot exceed 39 characters. The system responds that input is full if you exceed 39 characters.

Use the PC-FORM utility program to list attributes by keyword and to include them in the component, packaging, and material listings.

The keyword PCL is reserved for PC-LOGS.

Use the ATTR/ACOM command in SYMB mode to add the sheet number to a multiple sheet schematic. The SHEET=<n> attribute can be placed anywhere in the design. The value may include alpha as well as numeric characters.

ATTR/ACOM

Status line for ATTR/ACOM:

Layer	Text Size	Text Justification	Text Orientation	Mirror
ATTR	SIZ:125	L B	F	M

OPERATION:

1. Select ATTR/ACOM.

   In DETL mode, the system prompts:

   Select a component....

   In SYMB mode, the system prompts as shown in step 2 below.

2. Move the cursor to the desired component and press Button 1. The component is highlighted. The system prompts:

   Select location. (Text attributes OK?)...

3. If needed, change the text attributes displayed on the status line. Then, move the cursor to the desired location and press Button 1. The system prompts:

   Type in attribute spec.

ATTR/ACOM

4. Type in the attribute data, <keyword>=<value>, and press [Return] or Button 1. The system displays the value entered at the chosen location.

5. Press Button 2. If you select REDR before you select another command, the attribute may appear to move.

   NOTE: After you enter the attribute, the keyword and the equal sign are hidden, leaving only the attribute value displayed.

   To display the keyword and value, select ATTR/SCHG (Attribute/Single Change). The system displays the entire attribute (keyword and value).

6. To include additional attributes for the same component, repeat steps 2, 3, and 4.

ATTR/ACOM

ERROR MESSAGES:

Message: Keyword must start with a letter.

Cause: The keyword does not start with a letter.

Action: Begin the keyword with a letter.

Message: Matching " or) expected.

Cause: The value begins with a " or a (, but does not end with a " or).

Action: End the value with a " or), if appropriate.

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: = expected

Cause: The keyword before the equal sign (=) has a space in it.

Action: Reenter the keyword without pressing the space bar.

Mode: SYMB
DETL

ATTR/DATR

(Attribute/Delete Attribute)

FUNCTION:

Deletes the selected attribute.

REMARKS:

The ATTR/DATR command deletes the entire attribute, including the keyword and the value.

This command only affects one attribute at a time in the component selected.

Attributes added in one mode must be deleted in the same mode. For example, an attribute added to a symbol in SYMB mode must be deleted in SYMB mode.

OPERATION:

1. Select ATTR/DATR. The system prompts:

   Select one attribute....

2. Move the cursor to the desired attribute and press Button 1. The attribute is deleted.

ATTR/DATR

ERROR MESSAGES:

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Mode: SYMB
DETL

ATTR/SCHG

(Attribute/Single Change)

FUNCTION:

Changes the value of a single attribute.

REMARKS:

This command only affects one attribute value at a time in the component you select. The keyword is unaffected.

See the ATTR/ACOM command description for details on entering values.

Attributes added in one mode can only be changed in the same mode. For example, an attribute added in DETL mode must be changed in DETL mode.

OPERATION:

1. Select ATTR/SCHG. The system prompts:

   Select one attribute....

ATTR/SCHG

2. Position the cursor over the desired attribute and press Button 1. The system displays the keyword and an equal sign (=) in front of the value for the selected attribute and prompts:

   Enter new value

3. Type the new value and press [Return] or Button 1. The keyword remains unchanged, and the new value is displayed.

   To delete the keyword as well as the value, use the ATTR/DATR command.

ERROR MESSAGES:

Message: **New command or BUT1 expected.**

Cause: An invalid input was detected.

Action: Select a new command.

Message: **No attribute found.**

Cause: The cursor was not in the right location or the wrong mode is active.

Action: Change modes (SYMB or DETL) or reposition the cursor and try again.

ATTR/SCHG

Message: No attribute spec present.

Cause: An empty attribute specification was detected.

Action: Re-enter the attribute.

Message: Value part expected.

Cause: The value part of an attribute was not found for the given entry.

Action: Re-enter the attribute. Refer to the ATTR/ACOM command description for an example.

Mode: SYMB
DETL

Keyboard Command: /CFIL

(Command Log File)

FUNCTION:

Enables a status line submenu allowing you to control the recording of the current command log file (PCCAPS.CMD) by turning it on or off, or by restarting it.

REMARKS:

PCCAPS.CMD is the command log filename used by the system to record your commands during the current session. The PCCAPS.CMD file is created automatically when you begin a session.

You may want to turn off, turn on, or restart the command log file. For example, you may want to turn off the command log recording to record a macro with the /MAC command; then turn the command log recording back on again when you finish a macro.

When you re-enter PC-CAPS, the command log file is renamed from PCCAPS.CMD to PCCAPS.CM$. Any existing PCCAPS.CM$ file is overwritten with the new file.

> NOTE: The /CFIL command works within macros; ignore the "Input full" message, press Button 1 or [Return] and select any submenu option (ON, OFF, or RESTART).

/CFIL

OPERATION:

1. Type /. The system prompts:

   Menu Command:

2. Type CFIL and press [Return].

   The system displays the following prompt on the status line:

   Command log file: RESTART OFF ON

   RESTART begins recording the command log file over again, erasing all commands recorded up to that point.

   OFF turns the recording off until you use the /CFIL command, followed by the ON option to resume recording.

3. Select the option desired or press Button 2 to ignore the /CFIL menu options and return to the main command menu.

Mode: SYMB
DETL

CLYR

(Change Layer)

FUNCTION:

Transfers objects from one layer to another.

REMARKS:

Objects are transferred from any enabled layer to the active layer and are redrawn with the color of the active layer.

The name of the active layer is shown in the lower left corner on the status line. You can select VLYR to enable other layers and to choose the active layer, or you can change the active layer by moving the cursor over the status line parameter and pressing Button 1 to cycle through the enabled layers. You can also use [F1] to select an active layer from among the enabled layers.

In DETL mode, components are not associated with any individual layer. Thus the CLYR command does not operate on them.

Wires cannot be selected with this command. Use the EDIT/LAYS command to transfer wires, or line segments, from one layer to another.

Status line for CLYR:

Layer
DEVICE

CLYR

OPERATION:

1. Select CLYR. The system prompts:

   Select the object(s)....

2. Make sure the desired layer is active.

3. Select the object to change. The object changes to the active layer and takes on the corresponding color. The system prompts:

   Select object....

4. Choose another object or a new command.

ERROR MESSAGES:

Message: **No object found.**

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: SYMB
DETL

CLYR/IDEN

(Change Layer/Identify)

FUNCTION:

Transfers a collection of individually identified objects from one layer to another.

REMARKS:

Objects are transferred from any enabled layer to the active layer and are redrawn with the color of the active layer.

The identified objects are treated as a set. Objects do not have to be adjacent to one another or lie within a certain area. They can be unconnected or separated by other objects.

You can select VLYR to enable other layers and to choose the active layer, or you can change the active layer by moving the cursor over the status line parameter and pressing Button 1 to cycle through the enabled layers. You can also use [F1] to select an active layer from among the enabled layers.

In DETL mode, components are not associated with any individual layer. Thus, the CLYR/IDEN command does not operate on them.

Wires cannot be selected with this command. Use the EDIT/LAYS command to move wires, or line segments, from one layer to another.

CLYR/IDEN

Status line for CLYR/IDEN:

Layer
DEVICE

OPERATION:

1. Select CLYR/IDEN. The system prompts:

   Select object. (BUT2 to terminate selection).

2. Make sure the desired layer is active.

3. Select an object to change. The object is highlighted.

4. Select other objects to change in the same manner.

5. Press Button 2 to terminate selection. The system prompts:

   OK to change? YES NO.

6. To transfer the object(s) to the active layer, select YES. To cancel the command, select NO or press Button 2.

CLYR/IDEN

ERROR MESSAGES:

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: SYMB
DETL

CLYR/WIN

(Change Layer/Window)

FUNCTION:

Transfers a collection of objects intersected by or enclosed by a window from one layer to another.

REMARKS:

Objects are transferred from any enabled layer to the active layer and are redrawn with the color of the active layer.

The number of objects within a window that can be transferred from one layer to another depends on how much memory is available on your system after PC-CAPS is loaded, up to the operating system limit. Also, the time it takes to do a CLYR/WIN operation depends on the number of objects within the window.

You can select VLYR to enable other layers and to choose the active layer, or you can change the active layer by moving the cursor over the status line parameter and pressing Button 1 to cycle through the enabled layers. You can also use [F1] to select an active layer from among the enabled layers.

In DETL mode, components are not associated with any individual layer. Thus, the CLYR/WIN command does not operate on them.

Wires cannot be selected with this command. Use the EDIT/LAYS command to move wires or line segments from one layer to another.

CLYR/WIN

Status line for CLYR/WIN:

Layer
DEVICE

OPERATION:

1. Select CLYR/WIN. The system prompts:

   Corner 1....

2. Make sure that the destination layer is active.

3. Select the first corner. The system prompts:

   Corner 2....

   As the cursor is moved, a window rubberbands to indicate the enclosed area.

4. Position the cursor to enclose the objects in this window and select the second corner. The system highlights the enclosed object(s) and prompts:

   OK to change? YES NO.

CLYR/WIN

5. To transfer the object(s) to the active layer, select YES. To cancel the command, select NO or press Button 2.

ERROR MESSAGES:

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL
SYMB

COPY

FUNCTION:

Copies an object one or more times.

REMARKS:

The COPY command functions as a single command to copy one object at a time, or works with the WIN and IDEN subcommands to copy several objects at a time.

In DETL mode, you can select wires as well as components.

All objects except for components are selectable only when they are on a currently visible layer. Components and/or wires cannot be copied if masked with the MASK command.

When copying a net, the net can be copied onto any group of pins only if the committed pins in the group all belong to the same net. If all pins in the group are uncommitted or if no pins are touched, a new net is created. If all committed pins in the group belong to the same net, then the copied wires and all uncommitted pins touched will be merged into the net.

OPERATION:

1. Select COPY. The system prompts:

   Select the object(s)....

COPY

2. Select an object to copy. The system highlights the object and prompts:

   Select new location....

3. Select a location for the copy.

4. To place more copies of the same object, repeat step 3.

5. To select another object to be copied, press Button 2 or [Esc], and then repeat steps 2 and 3.

ERROR MESSAGES:

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Message: Shorting more than one net.

Cause: An attempt was made to copy a wire onto pins that were owned by more than one named net.

Action: Re-enter the net so that it doesn't short any uncommitted pin or uncommit the pins that are hit before copying the wire.

Mode: **DETL**
SYMB

COPY/IDEN

(Copy/Identify)

FUNCTION:

Copies a collection of individually identified objects.

REMARKS:

You can make one or more copies of the selected objects.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of the layer visibility status.

Wires cannot be selected with this command.

OPERATION:

1. Select COPY/IDEN. The system prompts:

   Select object. (BUT2 to terminate selection).

2. Select an object. The system highlights the object and prompts you for more objects.

COPY/IDEN

3. Repeat this step until all desired objects are selected. Press Button 2 to quit selecting objects. The system prompts:

   Select reference point....

4. Select a reference point. The system prompts:

   Select new location....

5. Select a location for the copy.

6. To place more copies of the same object(s), repeat step 5.

7. To identify another set of objects for copying, press Button 2, then repeat steps 2 through 5.

ERROR MESSAGES:

Message: **Exceeded limit for number of components or pins.**

Cause: An attempt was made to exceed the maximum number of allowable components or pins. The component maximum is 499. The pin maximum is 2500.

Action: Redesign the schematic using hierarchy or multiple sheets.

COPY/IDEN

Message: Exceeded limit for number of nets.

Cause: An attempt was made to exceed the database limit for the maximum number of nets (600).

Action: Redesign your schematic using hierarchy or multiple sheets.

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Message: No source net.

Cause: Program error detected in copying net.

Action: Contact P-CAD Customer Support and save the database, the PCCAPS.CMD file, and the PCCAPS.DBG file for further analysis.

COPY/IDEN

Message: Shorting more than one net.

Cause: An attempt was made to copy a wire onto pins that were owned by more than one named net.

Action: Re-enter the net so that it doesn't short any committed pin or uncommit the pins that are hit before copying the wire.

Mode: DETL
SYMB

COPY/WIN

(Copy/Window)

FUNCTION:

Copies a number of objects intersected by or enclosed in a window.

REMARKS:

You can make one or more copies of the contents of the window.

The number of objects within a window that can be copied depends on how much memory is available on your system after PC-CAPS is loaded up to the operating system limit. Also, the time it takes to do a COPY/WIN operation depends on the number of objects within the window.

In DETL mode, the entire wire or line must be inside the window to be copied; partial wires cannot be copied.

> NOTE: The MASK command does not protect wires from the COPY/WIN command.

When copying a net, the net name is copied only if the net name and the entire net are enclosed by the window.

To name a net with a copied name location but no name, select NAME/NET, enter the net name, and press Button 2 in response to the location prompt. The old location will be used.

COPY/WIN

OPERATION:

1. Select COPY/WIN. The system prompts:

   Corner 1....

2. Select the first corner of the window. The system prompts:

   Corner 2....

3. Select the opposite corner of the window. The system highlights all of the objects intersected by and/or enclosed in the window and prompts:

   Select reference point....

4. Select a reference point. The system prompts:

   Select new location....

5. Select a location for the copy.

6. To place more copies of the same object(s), repeat step 5.

7. To select another window for copying, select Button 2, then repeat steps 2 through 5.

COPY/WIN

ERROR MESSAGES:

Message: Can not copy net.

Cause: An attempt to copy a net was unsuccessful. Either the maximum number of nets has been reached, or the new net would have shorted one or more named nets.

Action: Use the SYS/STAT command to determine if the maximum number of nets has been reached, or use ENTR/WIRE to add the new net.

Message: Exceeded limit for number of components or pins.

Cause: An attempt was made to exceed the maximum number of allowable components or pins. The component maximum is 499. The pin maximum is 2500.

Action: Redesign the schematic using hierarchy or multiple sheets.

Message: Exceeded limit for number of nets.

Cause: An attempt was made to exceed the database limit for the maximum number of nets (600).

Action: Redesign your schematic using hierarchy or multiple sheets.

COPY/WIN

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, then position the cursor, and try again.

Message: No source net.

Cause: Program error detected in copying net.

Action: Contact P-CAD Customer Support and save the database, the PCCAPS.CMD file, and the PCCAPS.DBG file for further analysis.

Message: No wire to copy.

Cause: When copying a window, a net was enclosed that contained no wires (only connectivity). The net was not copied and no new net was created for the parallel pin(s) in the copied window.

Action: None.

COPY/WIN

Message: Shorting more than one net.

Cause: An attempt was made to copy a wire onto pins that were owned by more than one named net.

Action: Re-enter the net so that it doesn't short any uncommitted pin or uncommit the pins that are hit before copying the wire.

Mode: DETL
SYMB

DEL

(Delete)

FUNCTION:

Deletes one or more objects that have been selected one at a time.

REMARKS:

The DEL command can be used as a single command to delete one object at a time, or can be used with the WIN and IDEN subcommands to delete several objects at a time.

The UNDO subcommand restores the last deleted item if selected immediately after the DEL command. DEL/UNDO cannot restore items deleted with DEL/IDEN or DEL/WIN.

In DETL mode, you can select wires as well as components.

In SYMB mode, you can select those objects that you have created in symbol mode.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of the layer visibility status.

> CAUTION: When using the DEL command, if a wire is selected, the entire net is deleted. Use the EDIT/DELS command to delete segments of a wire or line.

DEL

After an item is deleted, redrawing the screen highlights the deleted item until another command is selected. Selecting a different command makes the deleted item disappear.

OPERATION

1. Select DEL. The system prompts:

   Select the object(s)....

2. Select an object. The system deletes the object and prompts:

   Select object...

3. To delete another object, repeat step 2.

   NOTE: If the wrong item is deleted, use the DEL/UNDO command immediately. UNDO restores the last deleted item.

ERROR MESSAGES:

Message: **No object found.**

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL
SYMB

DEL/IDEN

(Delete/Identify)

FUNCTION:

Deletes a collection of individually identified objects.

REMARKS:

The DEL/UNDO command cannot restore items deleted with the DEL/IDEN command.

Use the MASK command to protect components, and/or wires from the DEL/IDEN command.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of layer visibility status.

OPERATION:

1. Select DEL/IDEN. The system prompts:

   Select object. (BUT2 to terminate selection).

2. Select an object. The system highlights the object and prompts for more items.

3. To finish selecting, press Button 2. The system prompts:

   O.K. to delete? YES NO.

DEL/IDEN

4. To delete the selected objects, select YES. To cancel the deletions select NO or press Button 2.

ERROR MESSAGES:

Message: **No object found.**

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Message: **New command or BUT1 expected.**

Cause: An invalid input was detected.

Action: Select a new command.

Mode: DETL
SYMB

DEL/UNDO

(Delete/Undo)

FUNCTION:

Restores the last deleted item when using the DEL command without a subcommand.

REMARKS:

The DEL/UNDO command can restore items deleted with the DEL command alone, provided that it is selected immediately after deleting an item. To restore a deleted item, you must use UNDO before any other operation.

The DEL/UNDO command cannot restore items deleted with the DEL/IDEN or DEL/WIN commands.

OPERATION:

Select the UNDO command to restore the last deleted item.

Mode: DETL
SYMB

DEL/WIN

(Delete/Window)

FUNCTION:

Deletes a collection of objects enclosed in or intersected by a window.

REMARKS:

To delete a net, enclose it completely in the window.

The number of objects within a window that can be deleted depends on how much memory is available on your system after PC-CAPS is loaded, up to the operating system limit. Also, the time it takes to do a DEL/WIN operation depends on the number of objects within the window.

The DEL/UNDO command cannot restore items deleted with the DEL/WIN command.

Use the MASK command to protect components from DEL/WIN. MASK does not protect wires from DEL/WIN.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of the layer visibility status.

OPERATION:

1. Select DEL/WIN. The system prompts:

   Corner 1....

DEL/WIN

2. Select the first corner of the window. The system prompts:

   Corner 2....

3. Select the opposite corner of the window. The system highlights all the enclosed items and prompts:

   O.K. to delete? YES NO.

4. To delete the windowed items, select YES. To cancel the deletion, select NO or press Button 2.

ERROR MESSAGES:

Message: Can not link pin to net.

Cause: Database error occurred when attempting to add a new pin to a net.

Action: Verify integrity of database and/or call P-CAD Customer Support.

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

DEL/WIN

Message: No object found.

Cause: The system cannot find an object.

Action: Check if the object is masked and make sure you are in the same mode in which the object was entered, position the cursor, and try again.

DETL

(Detail)

FUNCTION:

Sets the system to the DETL (detail) command mode.

REMARKS:

This mode is used for creating and editing schematic designs, graphics, or the detail (hierarchically internal) view of components or entities.

The main menu color is green when the system is in the DETL command mode.

Data created in the DETL command mode can only be edited in the DETL mode.

Symbols created in the SYMB command mode can be entered as components in the DETL command mode. Once a component is entered, commands affect it as a single entity.

OPERATION:

Select DETL to change to the DETL command mode. The main menu turns green.

Mode: SYMB
DETL

DRAW

FUNCTION:

Enables the submenu for drawing lines, rectangles, filled rectangles, circles, arcs, and text.

OPERATION:

Select DRAW to enable the following submenu choices:

LINE
RECT (Rectangle)
FREC (Filled Rectangle)
CIRC (Circle)
ARC
TEXT

Mode: SYMB
DETL

DRAW/ARC

FUNCTION:

Draws arcs.

REMARKS:

You can specify several parameters for drawing arcs. These parameters are shown on the status line when you select the DRAW/ARC command. They are:

- The layer the arc will be drawn on.
- The line type in effect. Choose SOLID, DASHED, or DOTTED.
- The line width in effect. The default is W=0. You can specify from 0 to 250 database units.

Status line for DRAW/ARC:

Layer	Line Type	Line Width
IEEE	SOLID	W:5

OPERATION:

1. Select DRAW/ARC. The system prompts:

   Center point:...

   This is the center point of an imaginary circle of which the arc is a part.

DRAW/ARC

2. Change the status line parameters as needed.

3. Select the center point. The system prompts:

   Pnt defining rad and starting angle....

4. Select a point defining the radius and the starting angle for the arc. The system prompts:

   Pnt defining ending angle....

   As you move the cursor, the system rubberbands the arc.

5. Select a point defining the ending angle. The system draws the arc and prompts again for a center point.

   NOTE: The arc is always drawn counterclockwise from the starting angle to the ending angle.

Mode: SYMB
DETL

DRAW/CIRC

(Draw Circle)

FUNCTION:

Draws circles.

REMARKS:

You can specify several parameters for drawing circles. These parameters are shown on the status line when you select the DRAW/CIRC command. They are:

- The layer the circle will be drawn on.
- The line type in effect. Choose SOLID, DASHED, or DOTTED.
- The line width in effect. The default is W=0. You can specify from 0 to 250 database units.

Status line for DRAW/CIRC:

Layer	Line Type	Line Width
IEEE	SOLID	W:5

OPERATION:

1. Select DRAW/CIRC. The system prompts:

   Center point....

DRAW/CIRC

2. Change the status line parameters as needed.

3. Select the center point for the circle. The system prompts:

   Point on circumference....

4. Select a point on the circumference of the circle. The system draws the circle and prompts again for a center point.

Mode: SYMB
DETL

DRAW/FREC

(Draw Filled Rectangle)

FUNCTION:

Draws filled rectangles.

REMARKS:

Filled rectangles are reproduced as hollow rectangles on photoplotters, pen plotters, and printers. They can be filled in manually. In the case of photoplotting, use opaque solution to paint the rectangle afterwards. Refer to the *PC-PHOTO User's Manual* for details.

You can specify the layer for the filled rectangle on the status line.

> NOTE: Line type and line width show on the status line for this command, but have no effect on the filled rectangle.

Status line for DRAW/FREC:

Layer	Line Type	Line Width
IEEE	SOLID	W:5

DRAW/FREC

OPERATION:

1. Select DRAW/FREC. The system prompts:

   Corner 1...

   Corner 1 is the first of two diagonally opposite corners.

2. Change the status line parameters as needed.

3. Select the first corner. The system prompts:

   Corner 2...

4. Select the second corner. The system draws the filled rectangle and prompts you again for Corner 1.

5. To draw another rectangle, repeat steps 3 and 4.

Mode: SYMB
DETL

DRAW/LINE

FUNCTION:

Draws lines.

REMARKS:

You can specify several parameters for drawing lines. These parameters are shown on the status line when you select the DRAW/LINE command. They are:

- The layer the line will be drawn on.
- The line type in effect. Choose SOLID, DASHED, or DOTTED.
- The line mode in effect: orthogonal, angle, or 45 degrees. Choose ORTH (for 90^{0} angles), ANGL (for any angles), or 45D (for 45^{0} angles).
- The line width in effect. The default is W=0. You can specify from 0 to 250 database units.

Status line for DRAW/LINE:

Layer	Line Type	Line Mode	Line Width
IEEE	SOLID	ORTH	W:0

DRAW/LINE

OPERATION:

1. Select DRAW/LINE. The system prompts:

   Select start point....

2. Change the status line parameters as needed.

3. Select the starting point. The system prompts:

   Select next point....

4. Move the cursor to the next point. As you move the cursor, a rubberband line appears between the starting point and the cursor.

5. Press Button 1 to select the second point. The rubberband line changes to the line type in effect, and the system prompts for the next point.

6. Continue to select points by pressing Button 1, or press Button 2 to terminate selection of points and to end the line.

   NOTE: To delete part of a line just drawn, trace the cursor backwards along the part to erase and press Button 1. The line segment that is traced over disappears.

DRAW/LINE

ERROR MESSAGES:

Message: Valid point or new command expected.

Cause: Invalid input was detected.

Action: Enter a point or select a new command.

Mode: SYMB
DETL

DRAW/RECT

(Draw Rectangle)

FUNCTION:

Draws rectangles.

REMARKS:

You can specify several parameters for drawing rectangles. These parameters are shown on the status line when you select the DRAW/RECT command. They are:

- The layer the rectangle will be drawn on.
- The line type in effect. Choose SOLID, DASHED, or DOTTED.
- The line width in effect. The default is W=0. You can specify from 0 to 250 database units.

Status line for DRAW/RECT:

Layer	Line Type	Line Width
IEEE	SOLID	W:0

DRAW/RECT

OPERATION:

1. Select DRAW/RECT. The system prompts:

   Corner 1...

   Corner 1 is the first of two diagonally opposite corners.

2. Change the status line parameters as needed.

3. Select the first corner. The system prompts:

   Corner 2...

4. Select the second corner. The system draws the rectangle and prompts again for corner 1.

Mode: SYMB
DETL

DRAW/TEXT

FUNCTION:

Draws text.

REMARKS:

You can specify several parameters for drawing text. These parameters are shown on the status line when you select the DRAW/TEXT command. They are:

- The layer the text will be on.
- The current text size in effect. [F5] allows you to change the text size.
- The horizontal justification in effect. L, R, and C stand for left, right, and center justification.
- The vertical justification in effect. B, T, and C stand for bottom, top, and center justification.
- The text orientation in effect. The current text orientation is indicated by the green F.
- Mirroring of text. The M specifies the mirrored text feature to be on (green) or off (red). Mirrored text appears mirrored on the screen.

The size of the text on the screen is approximately the same size of the text plotted using PC-PRINT and PC-PLOTS.

DRAW/TEXT

When the text is too small to be legible on the screen, a dotted line is drawn that is approximately the same length and direction of the text.

Status line for DRAW/TEXT:

Layer	Text Size	Text Justification	Text Orientation	Mirror
ATTR	SIZ:125	L B	F	M

OPERATION:

1. Select DRAW/TEXT. The system prompts:

   Select Location (Attributes OK?)...

2. Change the status line parameters as needed.

3. Select the starting location. The cursor blinks at the selected position, and the system prompts:

   Type in text.

4. Type the text string (up to 39 characters) and press [Return].

DRAW/TEXT

To enter a negation specifier, type the text string and add an apostrophe (') before pressing [Return]. For example, if you type:

CD'

and press [Return], a negation bar appears over the text string as shown below.

$\overline{CD}$

To add a negation bar after already entering the text, move the cursor back to the left end of the text.

Press the space bar until the cursor is at the end of the string to be affected. Type an apostrophe and press [Return].

The bar appears over the text. The text appears to be erased. Select the REDR command to view the text.

Mode: DETL
SYMB

EDIT

FUNCTION:

Enables the submenu for editing vertices and segments of lines and wires.

REMARKS:

Editing does not affect electrical connectivity. It only alters the graphics. A segment may appear graphically disconnected from the wire or line it was originally connected to, but the original connectivity is maintained.

Use the NAME/NET command to highlight the disconnected segments to verify which nets the segments are electrically connected to.

OPERATION:

Select EDIT to enable the following submenu choices:

ADDV (Add Vertex)
DELV (Delete Vertex)
MOVV (Move Vertex)
MOVA (Move All)
DELS (Delete Segment)
LAYS (Change Segment Layer)
MOVS (Move Segment)

Mode: **DETL**
SYMB

EDIT/ADDV

(Edit/Add Vertex)

FUNCTION:

Adds a vertex or a sequence of vertices to an existing wire or line.

REMARKS:

Use DETL mode to edit wires or lines originally created in DETL mode.

Use SYMB mode to edit lines created in SYMB mode.

Vertices are automatically added at T-junctions and automatically deleted when a T-junction is broken.

OPERATION:

1. Select EDIT/ADDV. The system prompts:

   Select a point....

2. Select a point on the desired segment. The segment is highlighted and the system prompts:

   Select a new vertex....

EDIT/ADDV

3. Move the cursor. The segment rubberbands to show the revised configuration of the wire or line.

4. Select a new vertex location. A new vertex is added and the line is redrawn.

5. Continue positioning the cursor and selecting new vertices until you are done.

ERROR MESSAGES:

Message: No segment/vertex found.

Cause: The system cannot find a segment or vertex.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL
SYMB

EDIT/DELS

(Edit/Delete Segment)

FUNCTION:

Deletes a segment of a wire or line.

REMARKS:

Use DETL mode to edit wires or lines originally created in DETL mode.

Use SYMB mode to edit lines created in SYMB mode.

OPERATION:

1. Select EDIT/DELS. The system prompts:

   Select a segment....

2. Select the desired segment of the wire or line. The selected segment of the wire or line is deleted. The system prompts again:

   Select a segment....

3. Select another segment, if appropriate.

EDIT/DELS

In the case of wires, deleting a segment may leave two separate portions of a net. These portions, although graphically unconnected, are electrically part of the same net.

If the selected segment is connected to a pin, then the pin becomes electrically uncommitted after this operation, provided that the netlist is open.

ERROR MESSAGES:

Message: Can not link pin to net.

Cause: Database error occurred when attempting to add a new pin to a net.

Action: Verify integrity of database and/or call P-CAD Customer Support.

Message: No segment found.

Cause: The system cannot find a segment.

Action: Make sure you are in the same mode in which the segment was entered, position the cursor, and try again.

Mode: DETL
SYMB

EDIT/DELV

(Edit/Delete Vertex)

FUNCTION:

Deletes a vertex of a wire or line.

REMARKS:

Use DETL mode to edit wires or lines originally created in DETL mode.

Use SYMB mode to edit lines created in SYMB mode.

Vertices are automatically added at T-junctions and automatically deleted when a T-junction is broken.

OPERATION:

1. Select EDIT/DELV. The system prompts:

   Select one vertex....

2. Select a vertex on the desired wire or line. The vertex is deleted.

   The system continues to prompt for the selection of a new vertex until another command is selected.

EDIT/DELV

If the vertex connects only to one other segment, then only the one segment is deleted. If it connects two other segments, then the selected vertex is deleted, but the overall connection of lines or wires is maintained.

ERROR MESSAGES:

Message: Can not link pin to net.

Cause: Database error occurred when attempting to add a new pin to a net.

Action: Verify integrity of database and/or call P-CAD Customer Support.

Message: No vertex found.

Cause: The system cannot find a vertex.

Action: Make sure you are in the same mode in which the wire was entered, position the cursor, and try again.

Mode: DETL
SYMB

EDIT/LAYS

(Edit/Change Segment Layer)

FUNCTION:

Changes the layer a wire or line segment is on.

REMARKS:

Use DETL mode to edit wires or lines originally created in DETL mode.

Use SYMB mode to edit lines created in SYMB mode.

Status line for EDIT/LAYS:

Layer
BUS

EDIT/LAYS

OPERATION:

1. Select EDIT/LAYS. The system prompts:

   Select a segment....

2. Set the active layer as needed.

3. Select the desired segment of the wire or line. The layer of the specified wire or line segment changes to that of the current active layer.

ERROR MESSAGES:

Message: No segment found.

Cause: The system cannot find a segment.

Action: Make sure you are in the same mode in which the segment was entered, position the cursor, and try again.

Mode: DETL

EDIT/MOVA

(Edit/Move All)

FUNCTION:

Moves a wire segment or a vertex and all attached wire segments and solder dots on all layers.

REMARKS:

This command works only with wires.

OPERATION:

1. Select EDIT/MOVA. The system prompts:

   Select a point....

2. Select a point on a wire or a vertex. The system rubberbands the segment and any connected segments and solder dots. The system prompts:

   Select new position....

3. Select the desired new location. The affected segments and solder dots will be redrawn at the new location.

EDIT/MOVA

ERROR MESSAGES:

Message: **No segment found.**

Cause: The system cannot find a segment.

Action: Make sure you are in the same mode in which the segment was entered, position the cursor, and try again.

Mode: DETL
SYMB

EDIT/MOVS

(Edit/Move Segment)

FUNCTION:

Moves a segment of a wire or line.

REMARKS:

Use DETL mode to edit wires or lines originally created in DETL mode.

Use SYMB mode to edit lines created in SYMB mode.

OPERATION:

1. Select EDIT/MOVS. The system prompts:

   Select a line segment....

2. Select the desired segment of the wire or line. The selected segment and any adjacent segment(s) are highlighted. The system prompts:

   Select new position for the line....

3. Move the cursor to the desired new location. The system rubberbands the segment.

EDIT/MOVS

4. Select the final location of the segment.

   If the selected segment is the part of a wire connected to a pin, then the pin can become graphically unconnected; however, the pin stays internally connected to the net.

   If you zoom in near the pin area, you can see the circle at the pin location. This means that the pin is electrically committed but is graphically unconnected.

ERROR MESSAGES:

Message: No segment found.

Cause: The system cannot find a segment.

Action: Make sure you are in the same mode in which the segment was entered, position the cursor, and try again.

Mode: DETL
SYMB

EDIT/MOVV

(Edit/Move Vertex)

FUNCTION:

Moves a vertex of a wire or line.

REMARKS:

Use DETL mode to edit wires or lines originally created in DETL mode.

Use SYMB mode to edit lines created in SYMB mode.

OPERATION:

1. Select EDIT/MOVV. The system prompts:

   Select vertex....

2. Select a vertex on the desired wire or line. The line segment(s) joining the vertex are highlighted. The system prompts:

   Move the vertex to....

3. Move the cursor to the new vertex position. The system rubberbands the joining segment(s).

4. Select the new vertex location. The system continues to prompt for the selection of a new vertex until another command is selected.

EDIT/MOVV

ERROR MESSAGES:

Message: **No vertex found.**

Cause: The system cannot find a vertex.

Action: Make sure you are in the same mode in which the wire was entered, position the cursor, and try again.

Mode: SYMB
DETL

ENTR

(Enter)

FUNCTION:

Enables the submenu for entering pins, origins, components, or wires.

OPERATION:

Select ENTR to enable the following submenu choices:

- In SYMB mode:

  PIN
  ORG (Origin)

- In DETL mode:

  COMP (Component)
  WIRE
  UCOM (Uncommit a Pin)

Mode: DETL

ENTR/COMP

(Enter/Component)

FUNCTION:

Enters components into a design.

REMARKS:

A component is a symbol that represents a functional entity. It is usually a schematic symbol that has been created and saved in a library.

A DOS path up to 39 characters long may be specified for the component filename. For example:

C:\PCAD\SYM\TTL\74LS151.SYM

Components may be entered into a design with a scale factor by including the x/y scaling factor after the filename when using the ENTR/COMP command. The scale factor is a percentage of the original size.

If a scale factor is omitted, the component is entered at the same scale as it was created, which is 100/100 or 100% scale.

The x and y dimensions may be scaled separately, although it is usually best to scale the x and y dimensions equally and at some multiple of 2 or 1/2 of the original symbol size to preserve the visual aspect ratio of the symbol.

The allowable range is from 1 to 10,000. A scale factor less than 100 means reduction, while a scale factor greater than 100 means enlargement.

ENTR/COMP

If a single number is entered, it applies to both the x and y scales. For example, entering 50 as a scale factor scales the component down to 50% of the original size on both axes.

Each instance of a component may be given a unique name (from 1 to 8 characters) with the [F4] key at the time it is entered. This name identifies the instance of the component to distinguish it from other instances, or placements, of the same graphic entity.

A component instance may also be named later with NAME/COMP.

Only one version of a component with the same filename can be used in a schematic.

When you enter a component with the M (Mirror) parameter on the status line, any text associated with that component will not be mirrored.

Status line for ENTR/COMP:

Comp Name	Orientation	Mirror
| _____ |	F	M

OPERATION:

1. Select ENTR/COMP. The system prompts:

   :
   Comp-file-name x/y scales

ENTR/COMP

2. Type the component filename.

   NOTE: You don't need to type a filename extension; the system automatically adds the .SYM filename extension. However, if you type a period (.) after the filename with no extension, the system assumes the filename has no extension.

   To scale the component, type a space after entering the filename and follow it with the scale factor percentage and press [Return] or Button 1.

   For example, if you type:

   <filename> 50/200 [Return]

   the component is entered at 50% of the original size on the X axis and 200% on the Y axis.

   If you type:

   <filename> 50 [Return]

   the component is entered at 50% of the original size on both axes.

   The system prompts:

   Select loc to place comp. (Orientation OK?).

   To change the orientation of the component, select the orientation or mirror feature on the status line.

ENTR/COMP

To name the component, press the [F4] key or move the cursor to the status line to the left of the orientation F until you see an empty rectangle. Press Button 1. The system prompts for a component name.

Type the name and press [Return] or Button 1. To add a negation bar over the name, add an apostrophe (') after the name before pressing [Return] or Button 1.

The name appears on the status line. It will not be visible in the drawing until you use NAME/COMP to name the component.

3. Move the cursor to the desired location. A "reference rectangle" the size of the component is dragged along with the cursor as an aid in placement.

4. Press Button 1. The component is entered at the cursor location.

5. To enter additional copies of the same component, repeat steps 3 and 4.

6. To enter a different component, press Button 2 and repeat steps 2 through 4.

ENTR/COMP

ERROR MESSAGES:

Message: Component has no symbol.

Cause: The specified component has no graphic data entered in SYMB mode.

Action: Check the spelling of the component filename. Edit the symbol file and/or check the symbol creation process against "Example 5. Creating a P-CAD Compatible Symbol" in "Using PC-CAPS."

Message: Database format not compatible.

Cause: An incompatible database format was detected upon opening the specified file name.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of the filename.

Message: Exceeded limit for number of components or pins.

Cause: An attempt was made to exceed the maximum number of allowable components or pins. The component maximum is 499. The pin maximum is 2500.

ENTR/COMP

Action: Redesign the schematic using hierarchy or multiple sheets.

Message: Exceeded limit for number of nets.

Cause: An attempt was made to exceed the database limit for the maximum number of nets (600).

Action: Redesign your schematic using hierarchy or multiple sheets.

Message: No component named <filename>.

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: Overlapping component. OK to enter? YES NO

Cause: The location you selected for the new component is already occupied.

Action: You have two options.

1. Select YES to enter the component in the occupied location anyway.

2. Select NO to select another location.

ENTR/COMP

Message: Path too long.

Cause: The path and filename for the specified database overflows the internal filename buffer.

Action: Shorten the path in the PC-CAPS Configuration Screen.

Mode: SYMB

ENTR/ORG

(Enter/Origin)

FUNCTION:

Enters or sets the origin point of a symbol.

REMARKS:

When a component symbol is entered in a design, the cursor shows at the origin. The origin serves as a reference point for the dragging and placement of the component.

For accuracy in placement, put the origin on a pin that is on a grid point to ensure that symbol pins line up on grid points when a symbol is placed in a design.

OPERATION:

1. Select ENTR/ORG. The system prompts:

   Select the origin....

2. Select the origin point. (There is no visible indication of the location of the origin until you enter the component with the ENTR/COMP command.) The origin is entered.

ENTR/ORG

If Button 2 is pressed instead of Button 1, no origin is placed. The system responds:

Select the origin....
Valid point or new cmnd expected.

This message indicates that the system expects you to select another origin point with Button 1 or to select another command from the menu.

ERROR MESSAGES:

Message: **Valid point or new command expected.**

Cause: Invalid input was detected.

Action: Enter a point or select a new command.

Mode: SYMB

ENTR/PIN

(Enter/Pin)

FUNCTION:

Enters pins on a component symbol.

REMARKS:

A pin name must be assigned in order to enter the pin into the symbol. However, you may make the pin name invisible.

> CAUTION: Leaving the pinname blank by pressing the space bar is not allowed by the system.

For pins, pin type indicates whether a pin is an input, output, or bidirectional (I/O) pin.

P-CAD recommends that grid spacing be compatible with that used for designing schematics (10:10 is P-CAD's standard). Use grid lock (the G on the status line) to ensure that pins are entered on grid points and that wires can be connected to them.

Certain analysis tools (such as logic simulators) may require that the pins be entered in a particular sequence.

The SCMD/SPAT command can be used to verify the pin sequence and pin type. This command can also be used to change the pin type. If the pin sequence is wrong, delete the pins and re-enter them in the correct sequence.

ENTR/PIN

The ENTR/PIN command allows you to enter up to 255 pins per gate.

To create a test pin, add graphics before saving the test pin as a symbol. For example, drawing a rectangle around a test pin guarantees that the pin can be used as a symbol for placement.

To enter a pin name with a negation bar over it, type an apostrophe (') at the end of the pin name and press [Return].

Status lines for ENTR/PIN:

Layer	Pin Type
PINCON	OUTPUT

Layer	Text Size	Text Justification	Text Orientation	Mirror
PINNAM	SIZ:50	L B	F	M

OPERATION:

1. Select ENTR/PIN. The system sets the PINCON layer active and prompts:

   Select pin location...

2. Before selecting the pin location, select the pin type on the status line.

ENTR/PIN

3. Position the cursor and select the pin location. The system places an X at the pin location, sets the PINNAM layer active, and prompts:

   Select pin name location. (Attrb OK?)...

   The status line changes to provide text features.

4. Select text parameters on the status line as needed.

5. To make the pin name visible, position the cursor and select the pin name location with Button 1.

   To make the pin name invisible, press Button 2 or [Esc]. The pin name is associated with that pin connection even though the name is not visible. The system prompts:

   Enter pin name.

6. Type the pin name and press [Return] or Button 1.

   Text entered in lowercase changes to uppercase after the screen is redrawn.

ERROR MESSAGES:

Message: Can not link pin to net.

Cause: Database error occurred when attempting to add a new pin to a net.

Action: Verify integrity of database and/or call P-CAD Customer Support.

ENTR/PIN

Message: **Invalid pin name.**

Cause: The specified pin name is illegal.

Action: Specify a nonblank pin name.

Message: **Overlapping pins. OK to enter? YES NO**

Cause: The location you selected for the new pin is already occupied.

Action: You have two options.

1. Select YES to enter the pin in the occupied location anyway.

2. Select NO to select another location.

Mode: DETL

ENTR/UCOM

(Enter/Uncommit a Pin)

FUNCTION:

Uncommits a pin. Connectivity of a pin is removed and the pin is no longer connected to a net.

REMARKS:

Committed pins appear as filled circles. Uncommitted pins appear as X's.

OPERATION:

1. Select ENTR/UCOM. The system prompts:

   Select a pin....

2. Position the cursor on the pin to uncommit and press Button 1. The pin changes to an X and becomes uncommitted. The system prompts again:

   Select a pin....

3. Continue selecting pins to uncommit, if appropriate.

ENTR/UCOM

ERROR MESSAGES:

Message: No pin found.

Cause: No pin was found at the specified location.

Action: Use the REDR and the ZIN commands to check the pin location and try again.

Message: Pin connected. Can't uncommit.

Cause: When attempting to uncommit a pin, the identified pin was connected and the pin was not allowed to be uncommitted.

Action: Unconnect the pin using the EDIT/DELS command and use the ENTR/UCOM command again.

Message: Pin is not committed.

Cause: When attempting to uncommit a pin, the identified pin was already uncommitted.

Action: None.

Mode: DETL

ENTR/WIRE

(Enter/Wire)

FUNCTION:

Enters wires into a schematic design.

REMARKS:

Wires provide the electrical connectivity for schematic design. Lines entered with the DRAW/LINE command, however, provide graphic information only.

Connectivity is established when a wire connects to a pin or when one wire forms a T-junction with another. Crossing wires does not establish connectivity.

Solder dots and vertices are automatically inserted at T-junctions and automatically deleted when a T-junction is broken.

The default solder dot is a 0.1-inch round dot. You can create a different solder dot, create a special symbol file that describes the solder dot, and then use the SCMD/GSSF (System Command/Get Special Symbol File) command to attach the solder dot graphics to the database file. (See the SCMD/GSSF command description for more information.)

Wires snap to pins or other wires if the end point is within the specified radius surrounding the pin or wire. This radius is set with the /SGAT command.

ENTR/WIRE

To enable the snap-to-pin feature, toggle the L on the status line to green (on). When entering a wire, it will snap to the nearest pin within the tolerance radius in effect.

When a wire is connected to a pin, the pin image changes from an "X" to a filled circle, then disappears when the wire segment is completed.

If you enter a wire from an uncommitted pin and then press Button 2 to abort the wire, the pin will be left uncommitted.

Nets may be given alphanumeric names or left unnamed. To make the net name visible, use the NAME/NET command.

Nets left unnamed are automatically assigned a numerical name by the system.

Status line for ENTR/WIRE:

Layer	Net Name	Line Mode	Line Width	Snap to Pin
WIRES	|_____|	ANGL	W:<n>	L

OPERATION:

1. To enter a new wire or create a net, select ENTR/WIRE. The system sets the WIRES layer active and prompts:

   Select start point....

ENTR/WIRE

2. Position the cursor and select. The system prompts:

   Select next point....

   The system displays the net name, if any, on the status line.

3. As you move the cursor, the wire rubberbands. When the cursor is at the desired end point, press Button 1. This establishes a wire segment. The system prompts again:

   Select next point....

4. To enter additional wire segments, repeat step 3. You can also backtrack over the previous segment. Backtracking erases previous wire segments.

5. To end a wire, press Button 2.

OPTIONS:

Naming a New Net - To name a wire or net while entering it, press [F3]. The system prompts:

Net name:

Type the net name and press [Return].

The active net name is displayed on the status line.

ENTR/WIRE

Renaming a Net - To rename a wire/net later, select ENTR/WIRE and then select the wire. When an existing wire is picked, the system highlights it and displays the active net name. Then, select the net name on the status line or press the [F3] key. The system prompts for the new name. Type the name and press [Return]. Press Button 2.

Unnaming a Net - To unname a net, select ENTR/WIRE. Then select the net to unname. The system highlights the net. Select the net name on the status line or press the [F3] key. When the system prompts for a net name, press Button 2. The status line displays "unnamed" as the net name. Select REDR to erase the deleted net name from the screen, if it is displayed.

Adding to a Net - If you are adding a wire to an existing net, the new wire segment merges with the existing wire/net and acquires the same net name.

Joining Two Nets - When you attempt to connect one existing wire/net with another existing wire and one of them is unnamed, the system highlights the first wire and then the second. The system prompts:

Merge the nets? YES NO.

ENTR/WIRE

To merge the wires, select YES. The system briefly responds:

Nets merged into worknet.

To prevent a merger, select NO or press Button 2. Continue with ENTR/WIRE, but avoid crossing the wires in a T-junction.

ERROR MESSAGES:

Message: Both nets named. Merge not allowed.

Cause: An attempt was made to merge two nets that have been assigned different names.

Action: Do not merge the nets, or unname one of the two nets and then merge the nets.

Message: Hit more than one net.

Cause: The starting point of a new wire is at the intersection of two or more wires.

Action: Choose a different start point.

Message: More than one named net. No merge.

Cause: An attempt to merge more than one named net was made.

ENTR/WIRE

Action: Reroute the net so that only one named net is merged at a time, or unname one of the nets.

Message: Net hit is named <netname>. Change? YES NO

Cause: The new wire that you are starting to enter is named, and you have hit a named net.

Action: You have three options.

1. Select YES to change the existing net name to the net name of the wire you are currently entering.

2. Select NO to merge the wires and have the new wire acquire the name of the existing wire.

3. Press Button 2 twice and enter the wire at different coordinates.

Message: Not allowed to short more than one net.

Cause: An attempt was made to merge more than one named net.

Action: Reroute the net so that only one named net is merged at a time.

Mode: SYMB
DETL

Keyboard Command: /EXE

(Execute)

FUNCTION:

Replays macro files and command log files.

REMARKS:

After a session when you have exited to DOS, the system assigns the filename PCCAPS.CMD to the session recording or command log file.

When you re-enter PC-CAPS, the command log file is renamed from PCCAPS.CMD to PCCAPS.CM$. Any existing PCCAPS.CM$ file is overwritten with the new file.

Use the PCCAPS.CM$ file with the /EXE command to replay a session.

The /CFIL command allows you to turn the automatic recording of the command log file on or off during a session and to restart the recording from scratch.

/EXE

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type EXE and press [Return]. The system prompts:

   Exec filename:

3. Type <filename.ext> and press [Return]. If you are replaying a command log file, the command log file begins execution. If you are replaying a macro, the system prompts:

   Macro location...

4. Select the location to begin execution of the macro or command log file. Use the mouse to position the cursor and press Button 1, or move the cursor to the status line and select the X Y coordinates in the lower right corner.

   The macro file specified begins execution.

   To pause a macro or a recorded command log file, press [Ctrl]-[S]. To resume execution, type two slashes (//).

/EXE

To include a pause as a built-in part of a command log file or a macro, use the /WAIT command during the recording. Type / and then type WAIT and press [Return]. The system prompts:

Wait time (0 = wait for a key):

You may add a specific time limit to the /WAIT command of <n> seconds. Enter a 0 in response to this message to cause the system to pause for a key to be pressed. Next, the system prompts:

Message:

Type a message that you want the command log file to display when the /WAIT command executes. This message serves as a prompt or reminder.

NOTE: You may edit the file after the recording is made and you exit to the DOS level. Use a word processor in the nondocument mode to make modifications.

You might want to add the /INTR command with a word processor. This command interrupts the execution of a command log file when the file is executed. Then you may use the software as you would normally to perform any desired tasks or modifications.

/EXE

To resume command file execution after an interrupt, type / and then RESU and press [Return]. The file resumes running at the point where it was interrupted.

To pause a command file that is running, press the space bar twice to interrupt it. Then type /RESU to continue.

Refer to Appendix B for more information on executable files.

ERROR MESSAGES:

Message: Can't open "<filename>"

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command to verify the existence or spelling of a filename.

Message: Another macro definition in progress.

Cause: The system is currently recording a macro definition and the /MAC command was entered. One macro cannot be nested within another.

Action: Finish the current macro in progress before starting another one. Type /MEND to end the current macro.

Mode: SYMB
DETL

FILE

FUNCTION:

Enables the submenu for FILE operations.

OPERATION:

Select FILE to enable the following submenu choices:

SAVE
LOAD
ZAP

Mode: SYMB
DETL

FILE/LOAD

FUNCTION:

Reads the disk and loads a previously saved file.

REMARKS:

The file is displayed exactly as it was when last saved, including the mode, display scale, grid setting, layer settings, and text size.

You may load files from a specific drive or directory as in these examples:

A:TESTFILE.SCH

\PCAD\PROJ3\TESTFILE.SCH

A DOS path up to 39 characters long may be specified for the component filename.

OPERATION:

1. Select FILE/LOAD. The system prompts:

   Enter file name: <filename>

FILE/LOAD

2. Type the filename and press [Return] or Button 1.

   NOTE: When loading a schematic file, you don't need to type the filename extension; the system automatically adds the .SCH filename extension. However, if you type a period (.) after the filename, with no extension, the system assumes the filename has no extension.

   The system responds:

   Loading database.

   If a file is already on screen when you select FILE/LOAD, and it has been changed since last saved, the following message appears:

   Workfile modified. O.K. to load? YES NO.

   Loading a file erases what is currently in memory.

   To save the changes, select NO or press Button 2 and select FILE/SAVE. If you do not want to save the changes, select YES.

FILE/LOAD

ERROR MESSAGES:

Message: Database format not compatible.

Cause: An incompatible database format was detected upon opening the specified file name.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of the filename.

Message: New database size too small.

Cause: The specified database has been built in the "large" (640K) database format and cannot be processed by the PC-CAPS (512K) program on a TI portable computer.

Action: Contact P-CAD Customer Support or use a version of PC-CAPS configured for 640K databases.

Message: No database named <filename>

Cause: The system cannot find the filename entered.

FILE/LOAD

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: **Path too long.**

Cause: The path and filename for the specified database overflows the internal filename buffer.

Action: Shorten the path in the PC-CAPS Configuration Screen.

Mode: SYMB
DETL

FILE/SAVE

FUNCTION:

Saves the current file and writes it to disk under the name already assigned or under a new name.

REMARKS:

After you save a file, it remains on the screen for further modifications. This feature allows you to back up a file before making changes to it that may need to be undone. After you save a file, you may continue to work on the file. Other options after saving a file include:

- FILE/ZAP to clear the memory and the display
- FILE/LOAD to retrieve a previously saved file
- SYS/QUIT to exit the system

NOTE: Although DOS accepts various characters in filenames (for example, #), avoid using them in symbol filenames (<filename>.SYM). Use only alphanumeric characters in symbol filenames.

FILE/SAVE

OPERATION:

1. Select FILE/SAVE. The system prompts:

   Enter file name: <filename>

   NOTE: You don't need to type a filename extension; the system automatically adds the .SCH filename extension in detail mode and the .SYM filename extension in symbol mode. However, if you type a period (.) after the filename, with no extension, the system doesn't add an extension to the filename.

   If a file has already been loaded, then that filename appears as the default in the above prompt.

2. Press [Return] to accept the default filename, or type a new name and press [Return] or Button 1. The system responds:

   Saving database.

   If the filename already exists, the system prompts:

   File <filename> exists. Overwrite? YES NO.

   To overwrite the existing file, select YES. To avoid overwriting the existing file, select NO and type a different name.

FILE/SAVE

ERROR MESSAGES:

Message: DISK FULL!!! FILE SAVED NO GOOD!!!

Cause: The destination disk for a FILE/SAVE operation is full.

Action: Use SYS/DOS command and then use the delete command to free up space on the destination disk or insert a new diskette with available space.

Mode: SYMB
DETL

FILE/ZAP

FUNCTION:

Clears the display and erases memory, except for the layer structure.

REMARKS:

Use this command with caution. Once FILE/ZAP has been used, the data cannot be recovered except through the use of the command log file. (See the /EXE command description.)

This command does not erase the layer structure. The layer structure remains unchanged unless it is edited with the VLYR command or overwritten by a new file with the FILE/LOAD command.

OPERATION:

1. Select FILE/ZAP. The system displays:

   No Active Command.
   Wait! Erasing database.

   The file is erased from memory.

   If the file has been changed since it was last saved, the following message appears:

   Workfile modified. Sure to zap? YES NO.

FILE/ZAP

2. To save the changes, select NO and then select FILE/SAVE.

   To abandon the file and any changes, select YES.

   If you select YES, the system responds:

   WAIT! Erasing database.

   The screen display is cleared.

Mode: SYMB
DETL

Keyboard Command: /INTR

(Interrupt Execution)

Active during: /EXE

FUNCTION:

Interrupts the execution of a command log file or macro when the file is executed.

REMARKS:

This command is used with the /EXE (Execute) command. See the /EXE command description for more information.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type INTR.

Mode: SYMB
DETL

Keyboard Command: /LANG

(Line Angle)

Active during: DRAW/LINE
and ENTR/WIRE

FUNCTION:

Toggles among angle, orthogonal, and 45 degree modes of line or wire entry, which are displayed on the status line.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type LANG and press [Return]. The line mode parameter will change to ANGL, ORTH, or 45D.

Mode: **SYMB**
DETL

LEVL

(Level)

FUNCTION:

Enables the submenu for pushing down or popping up a level in a schematic design.

REMARKS:

Level refers to the component level of the design currently on screen in the overall design hierarchy.

The PUSH command allows you to move down to view and edit a lower component level.

The POP command allows you to view and edit the next level up.

OPERATION:

Select LEVL to enable the following submenu choices:

PUSH
POP

Mode: SYMB
DETL

LEVL/POP

(Level/Pop)

FUNCTION:

Displays the next level up in a hierarchical design.

REMARKS:

Refer to the LEVEL/PUSH command notes for comments related to editing component symbols.

OPERATION:

Select LEVL/POP. The system responds:

WAIT! Popping back to the parent level

and completes the pop.

If any changes have been made, the system prompts:

Workfile modified. POP still? YES NO.

To forfeit any changes made to the displayed level in the hierarchy and to return to the parent level, select YES.

To remain at the displayed level and continue modifying, select NO. To save the modifications, select FILE/SAVE.

LEVL/POP

If the changes are saved under a new name, the system pops and then prompts:

Change the link to <filename>? YES NO.

If you want the revised version of the component detail incorporated in the design hierarchy, select YES; if not, select NO.

ERROR MESSAGES:

Message: **Already at the root. Can not POP.**

Cause: The top level in the hierarchy is already displayed.

Action: Select another command.

Message: **Database format not compatible.**

Cause: An incompatible database format was detected upon opening the specified file name.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of the filename.

LEVL/POP

Message: New database size too small.

Cause: The specified database has been built in the "large" (640K) database format and cannot be processed by the PC-CAPS (512K) program on a TI portable computer.

Action: Contact P-CAD Customer Support or use a version of PC-CAPS configured for 640K databases.

Mode: **SYMB**
DETL

LEVL/PUSH

(Level/Push)

FUNCTION:

Displays the next level down in a hierarchical design.

REMARKS:

Pushing into a component automatically saves the current design in memory (RAM). The system loads and displays the component database.

If the current design has not been assigned a filename, the system assigns one of the forms "levl{number}.lvl," where **number** represents the level of the current design in the hierarchy.

Both the symbol and detail views are visible and can be edited. Changes in the detail view of the component are automatically reflected in the design hierarchy.

Editing a symbolic view and the associated detail does not affect other databases in which the symbol has been used.

LEVL/PUSH

OPERATION:

1. Select LEVL/PUSH. The system prompts:

   Select a component....

2. Select a component with the cursor and press Button 1. The system responds:

   WAIT! Pushing into <component name>

   and completes the push.

ERROR MESSAGES:

Message: Database format not compatible.

Cause: An incompatible database format was detected upon opening the specified file name.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of the filename.

LEVL/PUSH

Message: Depth limit exceeded. Can not PUSH.

Cause: The number of supported hierarchy levels in a schematic design has been exceeded. The LEVL/PUSH command allows only 16 levels of hierarchy.

Action: Redesign the schematic to have fewer hierarchical levels.

Message: New database size too small.

Cause: The specified database has been built in the "large" (640K) database format and cannot be processed by the PC-CAPS (512K) program on a TI portable computer.

Action: Contact P-CAD Customer Support or use a version of PC-CAPS configured for 640K databases.

Message: No component found.

Cause: The system cannot find a component where the cursor was positioned, or the object selected is not a component.

Action: Make sure you are in the same mode in which the object was entered, reposition the cursor, and try again.

LEVL/PUSH

Message: No component named <filename>.

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: No DB exists for the comp selected.

Cause: No database could be found when attempting to push into the specified component using the LEVL/PUSH command.

Action: Check the path specified on the PC-CAPS Configuration Screen. Verify that the component database file is present in the path or the current directory using the SYS/DOS command.

Message: No DB name for current design.

Cause: No name has been assigned to the current database in which a LEVL/PUSH command was attempted.

Action: Save the file using the FILE/SAVE command.

LEVL/PUSH

Message: Path too long.

Cause: The path and filename for the specified database overflows the internal filename buffer.

Action: Shorten the path in the PC-CAPS Configuration Screen.

Mode: SYMB
DETL

LPAN

(Long Pan)

FUNCTION:

Displays the current viewing window, and any stored views, relative to the entire working database of 60K by 60K database units.

REMARKS:

The LPAN command replaces the screen display with a set of axes indicating the 0,0 point in the database. Shown with these axes is a rectangle indicating the location of the current viewing window, as well as rectangles indicating the location of stored views, if any.

The area shown by the LPAN command is approximately six times the current viewing window. The exact long pan window to current viewing window ratio depends on the zoom level at the current viewing window.

The LPAN command is a nested command and can be used while one of the main operational commands is active.

The LPAN command permits quick panning to any area within the world coordinate system.

LPAN

OPERATION:

1. Select LPAN. The system displays a reference rectangle representing the view of the database relative to the X and Y crosshairs. It also displays the stored views and shows the size and position relative to the crosshairs.

2. Move the cursor to relocate the reference rectangle.

3. To select the new location and view the new graphics display, press Button 1, or to restore the original viewing window unchanged, press Button 2.

Mode: SYMB
DETL

Keyboard Command: /LSTY

(Line Style)

Active during: DRAW/LINE, RECT, CIRC, and ARC

FUNCTION:

Specifies the line style, which is displayed on the status line. Cycles among SOLID, DOTTED, and DASHED.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type LSTY and press [Return]. The line style parameter will change to SOLID, DOTTED, or DASHED.

Mode: SYMB
DETL

Keyboard Command: /LWID

(Line Width)

Active during: ENTR/WIRE, DRAW/LINE, RECT, CIRC, and ARC

FUNCTION:

Specifies the current width assignment for line segments comprising lines, wires, rectangles, circles, and arcs.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type LWID and press [Return]. The system prompts:

   Enter line width:

3. Type the new width and press [Return].

Mode: SYMB
DETL

Keyboard Command: /LYRN

(Layer Name)

Active during: Most Commands

FUNCTION:

Specifies the active layer, which is displayed on the status line. Selects another enabled layer.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type LYRN and press [Return]. The active layer parameter will change to the next enabled layer.

Mode: SYMB
DETL

Keyboard Command: /MAC

(Macro)

FUNCTION:

Begins the interactive recording of a macro file.

REMARKS:

A macro is a file that contains a series of keystrokes and commands that may be replayed once it is recorded.

Macros may be used for the recording and subsequent playback of repetitive design tasks. Typical examples include:

- The creation of memory arrays
- The wiring of multiple bus structures
- The creation of symbols

Macros may be recorded any time during a session.

/MAC

Any macro recorded is also recorded within the current command log file (PCCAPS.CMD). To suppress recording of the command log file, refer to the /CFIL command description.

The /CFIL command can be selected during a macro recording.

> NOTE: Since a macro is stored and replayed with a reference origin, it may be executed at various coordinate locations.
>
> When you store macros, the sequence of toggles and changes to the status line settings (for instance, active layer, or grid spacing) are saved, but the settings themselves are not.
>
> Thus, when you execute a macro, make sure the current status line settings do not differ from those when you made the macro or objects may be drawn on different layers and/or with different grid spacing than originally specified.

/MAC

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type MAC and press [Return]. The system prompts:

   Macro Origin...

3. Select a starting reference coordinate. If no previous macro has been recorded, the system prompts:

   Macro filename:

4. Type <filename> and press [Return].

/MAC

The macro recording begins and continues until you enter the keyboard command /MEND (Macro end).

To run a macro, use the /EXE command.

NOTE: If a macro has already been recorded during the current session, the system suggests the name of the last recorded macro filename (step 3 above):

Macro filename: <filename>

where <filename> is the previous macro filename entered.

To accept this filename, press [Return] or Button 1. The original macro is overwritten by the new file.

/MAC

ERROR MESSAGES:

Message: Another macro definition in progress.

Cause: The system is currently recording a macro definition and the /MAC command was entered. One macro cannot be nested within another.

Action: Finish the current macro in progress before starting another one. Type /MEND to end the current macro.

Message: Macros nested too deep.

Cause: Nested log files exceeded the system limit of five.

Action: Do not nest the macros deeper than five.

Mode: SYMB
DETL

MASK

FUNCTION:

Enables a menu for protecting (masking) or unprotecting components or wires from editing commands.

REMARK:

The MASK command prevents editing of components and wires; that is, MASK protects components and wires from commands like ROT, EDIT, MOVE, and DEL by making the masked data unselectable by editing commands.

> NOTE: MASK does not protect wires from the /WIN subcommands.

The MASK command is a nested command and can be used while one of the main operational commands is active.

OPERATION:

1. Select MASK. The screen clears and displays the following submenu:

   SEARCH VALUES

   COMPONENT ON
   WIRE ON
   QUIT

MASK

"ON" indicates that component or wire data can be edited.

"OFF" indicates that component or wire data cannot be edited.

2. Select "ON" or "OFF." This status is a toggle.

3. Press Button 2 or select "QUIT" to exit the MASK command and to return to the graphics display.

Mode: SYMB
DETL

Keyboard Command: /MEND

(Macro End)

FUNCTION:

Ends the interactive recording of a macro file.

REMARKS:

Macros continue recording until this command is entered.

OPERATION:

1. Type /. The system prompts:

   Menu Command:

2. Type MEND and press [Return]. The system terminates the macro recording currently in progress and saves the macro file under the filename specified in the /MAC command.

Mode: **DETL**
SYMB

MOVE

FUNCTION:

Moves data or graphics from one location to another.

REMARKS:

MOVE can be used as a single command to move one object at a time or used with the WIN and IDEN subcommands to move several objects at a time.

If the object selected is a component, then all of its wire connections are rubberbanded.

You cannot select wires with this command. Use the EDIT/MOVS command to move wires.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of layer visibility status. Use the MASK command to prevent moving components.

Use grid lock for precision placement.

OPERATION:

1. Select MOVE. The system prompts:

   Select object(s).

MOVE

2. Select an object. The system highlights the object and prompts:

   Move object to. (Select point)...

3. Select the point where you want to move the object.

ERROR MESSAGES:

Message: **No object found.**

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL
SYMB

MOVE/IDEN

(Move/Identify)

FUNCTION:

Moves a number of selected objects.

REMARKS:

If a component is selected, then all of its wire connections are rubberbanded.

You cannot select wires with this command. Use the EDIT/MOVS command to move wires.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of layer visibility status. Use the MASK command to prevent moving components.

OPERATION:

1. Select MOVE/IDEN. The system prompts:

   Select object. (BUT2 to terminate selection).

2. Select an object. The system highlights the object and prompts again for the next object.

3. Repeat step 2 until you are finished with your selection.

MOVE/IDEN

4. Press Button 2 to terminate selection. The system prompts:

   Select from point....

5. Select a reference point for the move. The system prompts:

   Select to point....

6. Move the cursor to a new location and press Button 1 to relocate the reference point.

   The identified objects relocate in the design around the new reference point, in the same relative positions previous to the move.

   The system prompts again to select objects.

7. Continue selecting and moving objects or select another command to terminate.

ERROR MESSAGES:

Message: **New command or BUT1 expected.**

Cause: An invalid input was detected.

Action: Select a new command.

MOVE/IDEN

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL
SYMB

MOVE/WIN

(Move/Window)

FUNCTION:

Moves a number of objects enclosed in a window.

REMARKS:

The number of objects within a window that can be moved depends on how much memory is available on your system after PC-CAPS is loaded, up to the operating system limit. Also, the time it takes to do a MOVE/WIN operation depends on the number of objects within the window.

This command moves net names that are partially enclosed in the window.

For correct rubberbanding of wires, include part of the connecting wire in the window.

MASK does not protect wires from the MOVE/WIN command.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of layer visibility status. Use the MASK command to prevent moving components.

OPERATION:

1. Select MOVE/WIN. The system prompts:

   Corner 1....

MOVE/WIN

2. Select the first corner of the window. The system prompts:

   Corner 2....

3. Select the opposite corner of the window, enclosing the objects to be moved. The system highlights the enclosed items and prompts:

   Select from point.

4. Select a reference point for the move. The selected items disappear, leaving the window outline. The system prompts:

   Select to point.

5. Move your cursor to a new location and press Button 1 to relocate the reference point. The windowed items are relocated. The system prompts again for another window selection.

6. Continue windowing and moving objects or select another command.

MOVE/WIN

ERROR MESSAGES:

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL

NAME

FUNCTION:

Enables the submenu for naming components or nets.

OPERATION:

Select NAME to enable the following submenu choices:

COMP (Component)
NET

Mode: DETL

NAME/COMP

(Name/Component)

FUNCTION:

Names or unnames components. Also, makes visible the name of a component that has been previously entered or moves a component name.

REMARKS:

Each component may be given a unique name (from 1 to 8 characters) at the time it is entered with the ENTR/COMP command. The NAME/COMP command allows a name to be added to a component after the component has been entered.

Component names uniquely identify the component to distinguish it from other components, or placements, of the same graphic entity.

Status line for NAME/COMP:

Layer	Text Size	Text Justification	Text Orientation	Mirror
CMPNAM	SIZ:125	L B	F	M

NAME/COMP

OPERATION:

1. Select NAME/COMP. The system sets the CMPNAM layer active and prompts:

   Select a component....

2. Move the cursor to the desired component and press Button 1. The system highlights the component and prompts:

   Enter component name:

3. Type the name and press [Return] or Button 1. The system prompts:

   Name = <component name>. Select location....

4. Select status line parameters as needed.

5. Move the cursor to the desired location. A reference rectangle the size of the name is dragged along with the cursor as an aid in placement.

6. Press Button 1. The name displays at that location. If you are zoomed out, you may see only a dotted line instead of the name. Zoom in to verify the name.

NAME/COMP

NOTE: To enter a negation specifier, type the text string desired and follow it with an apostrophe ('), then press [Return].

OPTIONS:

Renaming - To rename a component with the NAME/COMP command as the active command, select the named component and press Button 2. Select the component again and rename it. Select the location for the new name.

Making a Name Visible - To make visible the name of a component that has already been named with the ENTR/COMP command, select the component with the NAME/COMP command. As you move the cursor, a rectangle the size of the name is dragged along with the cursor. Select the location with Button 1.

Changing Text Attributes - To change any of the attributes of the component name. Select the component. The text reverts back to the reference rectangle. Make any changes on the status line. Position the name and press Button 1.

Changing the Layer of a Name - To move a name to another layer, select the active layer parameter on the status line or with the VLYR command. Then select the NAME/COMP command. Select the corresponding component. Position the cursor and select. The name changes to the new layer.

NAME/COMP

ERROR MESSAGES:

Message: Component <name> exists. Another name?

Cause: An existing component name has been entered. The system highlights the named component and prompts for a different name.

Action: Since component names must be unique, enter a different name or press Button 2 to terminate the command.

Mode: DETL

NAME/NET

FUNCTION:

Names a net or makes visible the name of a net that has been previously entered.

REMARKS:

Nets may be given a name or identity (from 1 to 8 characters) with the NAME/NET command. Multiple instances of the net name may be placed at various places along the net.

They may also be named at the time they are entered with the ENTR/WIRE command, in which case, the NAME/NET command makes all line segments of the previously assigned name visible.

Text is changed to upper case when the screen is redrawn. The name can also be shown with a negation bar over it by including an apostrophe (') at the end of the text string.

To rename a net, see the ENTR/WIRE command description.

Status line for NAME/NET:

Layer	Text Size	Text Justification	Text Orientation	Mirror
NETNAM	SIZ:125	L B	F	M

NAME/NET

OPERATION:

1. Select NAME/NET. The system sets the NETNAM layer active and prompts:

   Select a net....

2. Select the desired net. The system highlights the net and prompts:

   Enter net name:

   If the net has already been named, skip step 3.

3. Type the name and press [Return] or press Button 1. The system prompts:

   Name = <net name>. Select location....

   If you do not want the name to be visible, skip the remaining steps and press Button 2.

4. Move the cursor to the desired location. A reference rectangle the size of the name is dragged along with the cursor as an aid in placement.

5. Press Button 1. The name displays at that location.

NAME/NET

6. To display multiple copies of the name, repeat step 5.

   If you enter an existing name, the system highlights the named net and prompts:

   Net <name> exists. Merge? YES NO.

   To merge the nets, select YES. The system responds:

   Nets merged.

   To place the net name, select the location.

   To choose a different name, select NO.

ERROR MESSAGES:

Message: No net found.

Cause: No net was found at the specified location.

Action: Check the net location using the REDR and ZIN commands and try to identify the net again.

Mode: SYMB
DETL

PAN

FUNCTION:

Moves, or pans, the current viewing window to a new location.

REMARKS:

The PAN command allows you to keep selecting viewing windows until you press Button 2 or select another command.

The PAN command is a nested command and can be used while one of the main operational commands is active.

OPERATION:

1. Select PAN. The system displays a crosshair cursor and prompts:

   Select view center.

2. Move the cursor and select a new view center, or specify a new view center by selecting the X Y coordinates on the status line and typing coordinates for the center of the view desired.

   The system redraws the new viewing window.

3. Press Button 2 or select a new command to end the command.

Mode: SYMB
DETL

RCL

(Recall)

FUNCTION:

Recalls numbered view windows previously stored with the STO command.

REMARKS:

The RCL command allows the recall of any of up to nine views that have been saved using the STO command. The nine numbered views are saved with the entire layout when you use the FILE/SAVE command.

The RCL command also allows the rapid recall of a RAM memory view referred to as MAP, which displays a screen view that is stored in memory as a bit map with IBM low resolution color graphics board.

The MAP view lasts only for the current session. The RCL command recalls any part of the design without requiring the system to completely redraw the saved view.

The RCL command is a nested command and can be used while one of the main operational commands is active.

RCL

OPERATION:

1. Select RCL. The system displays the status line menu:

   Restore View: 1 2 3 4 5 6 7 8 9 MAP

2. Select one of the numbered views or MAP. The system displays the view selected.

   If no view has been associated with the selected number or MAP, the system displays the following message:

   Undefined View

Mode: SYMB
DETL

REDR

(Redraw)

FUNCTION:

Redraws the current viewing window.

REMARKS:

The REDR command redraws the graphics in the current window to restore grid points or line segments that are partially displayed or not visible due to editing.

The REDR command is a nested command and can be used while one of the main operational commands is active.

OPERATION:

Select REDR to redraw the current viewing window. The system remembers the last menu command used before selecting REDR.

To halt the REDR command while it is in progress, press Button 1 or Button 2.

Mode: SYMB
DETL

Keyboard Command: /RESU

(Resume Execution)

Active during: /EXE

FUNCTION:

Resumes the execution of a command log file or macro after interruption.

REMARKS:

The file resumes running at the point where it was interrupted. This command is used with the /EXE (Execute) command. See the /EXE command description for more information.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type RESU and press [Return].

Mode: DETL
SYMB

ROT

(Rotate)

FUNCTION:

Rotates an object counterclockwise in 90 degree increments.

REMARKS:

If the object selected is a component, then all of its connections are rubberbanded.

Wires are not selectable with this command.

All objects except components are selectable only when they are on a currently visible layer. Components are selectable regardless of layer visibility status. Use the MASK command to prevent rotating components.

OPERATION:

1. Select ROT. The system prompts:

   Select the object(s)....

2. Select an object. The point you select on the object acts as the axis of rotation. The system highlights the object and prompts:

   BUT1 to rotate 90 degree CCW....

   (CCW stands for counterclockwise.)

ROT

3. Press Button 1 once for each 90 degree rotation.

4. To stop the operation, press Button 2.

Mode: DETL
SYMB

ROT/IDEN

(Rotate/Identify)

FUNCTION:

Rotates a group of individually selected objects. Rotation is counterclockwise in 90 degree increments.

REMARKS:

If the object selected is a component, then all of its connections are rubberbanded.

Wires are not selectable with this command.

All objects except for components are selectable only when they are on a currently visible layer. Components are selectable regardless of visibility status. Use the MASK command to prevent rotating components.

OPERATION:

1. Select ROT/IDEN. The system prompts:

   Select object. (BUT2 to terminate selection)

2. Select each item you want to include in the rotation. The system highlights each item selected.

ROT/IDEN

3. Press Button 2 to end selection. The system prompts:

   Select the center of rotation....

4. Choose a point around which you want all the selected objects to rotate. The system prompts:

   BUT1 to rotate 90 degrees CCW....

   (CCW stands for counterclockwise.)

5. Press Button 1 once for each 90 degree rotation.

6. To stop the operation, press Button 2.

ERROR MESSAGES:

Message: **New command or BUT1 expected.**

Cause: An invalid input was detected.

Action: Select a new command.

Message: **No object found.**

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: DETL
SYMB

ROT/WIN

(Rotate/Window)

FUNCTION:

Rotates object(s) enclosed and intersected by a window. Rotation is counterclockwise in 90 degree increments.

REMARKS:

All wire segments totally within the window rotate. The wires extending outside of the window rubberband.

The number of objects within a window that can be rotated depends on how much memory is available on your system after PC-CAPS is loaded, up to the operating system limit. Also, the time it takes to do a ROT/WIN operation depends on the number of objects within the window.

MASK does not protect wires from the ROT/WIN command.

> NOTE: We recommend that you not use the ROT/WIN command to rotate components with wires attached. Use the ROT command without WIN to rotate a component.

All objects except for components are selectable only when they are on a currently visible layer. Components are selectable regardless of visibility status. Use the MASK command to prevent rotating components.

ROT/WIN

OPERATION:

1. Select ROT/WIN. The system prompts:

   Corner 1....

2. Select the first corner of the window. The system prompts:

   Corner 2....

3. Select the diagonally opposite corner of the window. The system highlights the enclosed items and prompts:

   Select center of rotation....

4. Choose the point around which you want the entire window to rotate. The system prompts:

   BUT1 to rotate 90 degrees CCW....

   (CCW stands for counterclockwise.)

5. Press Button 1 once for each 90 degree rotation.

6. To stop the operation, press Button 2.

ROT/WIN

ERROR MESSAGES:

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Mode: SYMB
DETL

SCMD

(System Command)

FUNCTION:

Enables the submenu for the following system command operations: set component attribute, set pin attribute, pin number location, set net attribute, pin number, and get special symbol file.

OPERATION:

Select SCMD to enable the following submenu choices:

- In SYMB mode:

  SCAT (Set Component Attribute)
  SPAT (Set Pin Attribute)
  PNLC (Pin Number Location)

- In DETL mode:

  SNAT (Set Net Attribute)
  PNUM (Pin Number)
  GSSF (Get Special Symbol File)

Mode: DETL

SCMD/GSSF

(System Command/Get Special Symbol File)

FUNCTION:

Loads a special symbol file from disk. The special symbol file (<filename>.SSF) describes the correspondence between T-junctions and solder dots.

REMARKS:

This command changes the display from default solder dots at T-junctions to user-specified solder dots (<filename>.SDT). The special symbol file (<filename>.SSF) links the solder dots to the database until they are removed or changed by loading a different special symbol file.

OPERATION:

1. Select SCMD/GSSF. The system prompts:

   Enter filename:

2. Type the filename and press [Return] or Button 1.

   The file is loaded and all default solder dots are replaced with the one listed in the special symbol file.

3. Select REDR to view the new solder dot graphics.

SCMD/GSSF

Only one special symbol file can be attached to a database at any one time.

To remove a special symbol file from a database without adding another one in its place, answer the prompt in step 1 by pressing Button 2 to [Esc] instead of typing a filename. The existing special symbol file is removed from the database, and the default solder dot graphics are restored.

ERROR MESSAGES:

Message: **Can not open file <filename>.**

Cause: The attempt to open the named file has failed.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: **Database not compatible.**

Cause: The special symbol file specified is not in the correct database format.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of a filename.

SCMD/GSSF

Message: Error loading <filename>.

Cause: An error occurred in loading the specified special symbol file.

Action: Check the integrity of the special symbol file. Refer to "Example 7. Creating a Solder Dot and a Special Symbol File" in "Using PC-CAPS."

Message: Filename not found...

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command to verify the existence or spelling of a filename.

Message: Symbol has no picture data.

Cause: The symbol defined in the special symbol file contains no graphic data.

Action: Edit the symbol by entering the graphic data in the SYMB mode; then try using the SCMD/GSSF command again.

SCMD/GSSF

Message: Symbol origin not defined.

Cause: The symbol defined in the special symbol file has no origin defined.

Action: Edit the symbol and use the ENTR/ORG command to define the origin.

Mode: SYMB

SCMD/PNLC

(System Command/Pin Number Location)

FUNCTION:

Assigns pin information for later use in physical package design, including the following:

- Gates per package
- Pins per gate
- Location for reference designator
- Location for pin numbers
- Package pin numbers
- Section assignments

REMARKS:

Use this command only if you intend to translate your schematic into physical package information or desire package pin number references in the PC-FORM reports.

The allowable range for gates per package is from 1 to 255. The allowable range for number of pins per gate is from 1 to 255. The section assignment is a single letter.

This command prepares the locations that are used by the SCMD/PNUM command.

Status line for SCMD/PNLC:

Layer	Text Size	Text Justification	Text Orientation	Mirror
REFDES PINNUM	SIZ:125	L B	F	M

SCMD/PNLC

OPERATION:

1. Select SCMD/PNLC. The system sets the PINNUM layer active and prompts:

   Enter gates per package:

2. Type the number of gates per package, from 1 to 255, and press [Return] or Button 1. The system prompts:

   Enter number of pins per gate:

3. Type the number of pins per gate, from 1 to 255, and press [Return] or Button 1. The system sets the REFDES layer active and prompts:

   Select loc for ref designator....

4. The status line parameters are displayed. You may set the justification for the text and the text size according to your own standards.

SCMD/PNLC

5. Select the location with the cursor and press Button 1. (This location is reserved for the reference designators which will be assigned later.)

   The system sets the PINNUM layer active.

   As each pin is highlighted in turn, the system prompts:

   Select loc for pin number...

6. Select the location with the cursor and press Button 1. (This location is reserved for the pin number which will be assigned later.)

   The system then prompts:

   Enter package pin number for <pinname>:
   Gate assigned to section <x>

   Note that the pinname is the one that was assigned when the symbol was created and section <x> is the section identification that is automatically assigned.

7. Type the pin number, from 1 to 255, and press [Return] or Button 1. The system repeats the prompt in step 6 for each pin and section.

SCMD/PNLC

ERROR MESSAGES:

Message: **Invalid "<x>"**

Cause: A nonnumeric character was entered as the value.

Action: Enter a valid number within the range 1 to 255.

Message: **Out of range (1 to 255)**

Cause: The number entered for the gates per package is outside of the allowable range.

or

The number entered for the pins per gate is outside of the allowable range.

Action: Enter a number between 1 and 255.

Message: **PNLC: Available only in SYMB mode.**

Cause: The command SCMD/PNLC was input in DETL mode.

Action: Check the input (keyboard or command log file or macro file) for valid syntax.

Mode: DETL

SCMD/PNUM

(System Command/Pin Number)

FUNCTION:

Assigns a reference designator and section to a component.

REMARKS:

The reference designator and the pin numbers corresponding to the assigned section are displayed at the predetermined locations of the selected component. The locations are determined by the SCMD/PNLC command.

The reference designator can be an alphanumeric string from one to eight characters. The section assignment can be a single letter and/or a double letter string (a, b, c, d..., z, aa, ab, ac..., az, ba, bb, ...bz, ...ha, hb, ...hz) that corresponds to the number of gate sections created with the SCMD/PNLC command, or it can be a series of numbers that correspond to the pin numbers in the gate sections.

OPERATION:

1. Select SCMD/PNUM. The system prompts:

   Select a component....

SCMD/PNUM

2. Select the desired component. The system then highlights the selected component and prompts:

   =>
   Enter reference-designator/section

   and the previously assigned reference identity, if any.

3. Type the reference designator followed by a "/" and then the section for the component and press [Return] or Button 1.

   An example reference designator and section is given here:

   U1/B

   Or, type the reference designator followed by a "/" and then the pin numbers for the component. Begin the pin number sequence with a "(" and end the sequence with a ")". Separate each number with a space.

   An example of a reference designator and pin numbers for the component are given here:

   U1(1 2 3)

   NOTE: If the pin number sequence extends beyond 39 characters, type a ">" at the end of the line and continue typing the pin numbers on the next line. Be sure to type a space before the ">" and as the first character of the continuation line.

SCMD/PNUM

ERROR MESSAGES:

Message: Gate/part mismatch. Package gate into illegal part YES/NO

Cause: A different gate type has already been assigned to this part.

Action: To package gate into illegal part anyway, select YES.

To assign gate to a different physical part, select NO.

Message: Illegal section name.

Cause: An illegal section name has been specified in the SCMD/PNUM command.

Action: Refer to the command description for the SCMD/PNUM command for correct syntax or check your package definition library file (<filename>.FIL).

Message: Pin num location info missing.

Cause: The selected component has not been prepared using the SCMD/PNLC command.

Action: Use the SCMD/PNLC command and return to SCMD/PNUM.

SCMD/PNUM

Message: **PNUM: Available only in DETL mode.**

Cause: The command SCMD/PNUM was input in DETL mode.

Action: Check the input (keyboard or command log file or macro file) for valid syntax.

Message: **Section <reference designator/section> is occupied.**

Cause: The reference designator and section entered have already been used. The system highlights the existing component that already has the reference designator assigned.

Action: Enter another reference designator.

Message: **The specified section does not exist.**

Cause: An incorrect letter was entered for the section identification. For example, if you entered E as a section identification and the valid sections are A through D, then E does not exist and is not valid.

Action: Enter a valid letter for the section identification.

Mode: SYMB

SCMD/SCAT

(System Command/Set Component Attribute)

FUNCTION:

- Sets or modifies the Component Type ID for use by the PC-LOGS logic simulator.
- Indicates hierarchy in a nested symbol for PC-LINK.

REMARKS:

Every symbol you create except for nonhierarchical symbols that will not be simulated by PC-LOGS must be assigned a Component Type ID for use by other P-CAD programs. (Nonhierarchical symbols that will not be simulated are automatically assigned a Component Type ID of 255.)

The Component Type IDs supported by PC-LOGS are listed in the appendix titled "Primitives," in the *PC-LOGS User's Manual.*

For hierarchically designed symbols, assign a Component Type ID of 256. This enables "flattening" of the hierarchical database by PC-LINK.

For nonelectrical symbols, such as title blocks, logos, and solder dots, that you don't want listed in the netlist, assign a value of 0. A Component Type ID of 0 prevents PC-NODES from extracting the symbol into the netlist.

SCMD/SCAT

For symbols you create that cannot be packaged, such as power and ground, but that you want listed in the netlist, assign a value of -1. A Component Type ID of -1 prevents PC-PACK from packaging the symbol.

OPERATION:

1. Select SCMD/SCAT. The system prompts:

   Workcomp. Old type = <n>.New type=

   where **<n>** is the number of the type last used.

2. Type the number for the new type desired and press [Return] or Button 1.

ERROR MESSAGES:

Message: **Out of range (-32000 to 32000).**

Cause: The number entered for the Component Type ID is outside the allowable range.

Action: Enter a value between -32000 and 32000.

Mode: DETL

SCMD/SNAT

(System Command/Set Net Attribute)

FUNCTION:

Finds and highlights a named net and assigns a global attribute to a net.

REMARKS:

This command is used to assign a global net attribute to connect nets, such as power and ground, in hierarchical schematic designs. The global net attribute is not required to connect nets on different sheets in a nonhierarchical multisheet schematic. PC-LINK connects wires with the same net name from different sheets in a nonhierarchical schematic.

OPERATION:

1. Select SCMD/SNAT. The system prompts:

   Enter a net name:

2. Type the net name and press [Return] or Button 1. The system highlights the net and prompts:

   A global net? YES NO.

3. Select YES or NO.

SCMD/SNAT

ERROR MESSAGES:

Message: **Net <name> not found.**

Cause: The system cannot find a net with the name entered.

Action: Re-enter the net name and check the characters for accuracy. Make sure the proper layers are visible.

Mode: SYMB

SCMD/SPAT

(System Command/Set Pin Attributes)

FUNCTION:

Sets and/or modifies pin type and logic equivalence (LEQ) attributes.

REMARKS:

For pins, pin type indicates whether a pin is an input, output, or bidirectional (I/O) pin. This information is used by PC-LOGS or other simulators. The allowable integer range is from 0 to 2.

LEQ stands for logic equivalence. The default value for this attribute is LEQ = 0, where 0 means that each pin is logically unique and therefore not swappable with other pins. All pins are assumed to have an LEQ of 0 unless a different LEQ is set.

When LEQ = any integer greater than 0, it indicates an equivalence class for each pin. Each set of logically equivalent pins can then be given a unique number that marks them as logically equivalent. The allowable range is from 0 to 24.

The LEQ information is used by physical implementation applications such as a PCB layout system.

SCMD/SPAT

OPERATION:

1. Select SCMD/SPAT. The system highlights the first pin and prompts:

   Enter new type (0=I; 1=O; 2=IO.):
   Type of pin <pinname> is <n>

   The current pinname and number <n> are those previously assigned.

2. Press [Return] or Button 1 to accept the type displayed, or choose a new type (0, 1, or 2) in response to the prompt and press [Return]. The system prompts:

   Enter new code:
   Pin LEQ code is <n>

   where **<n>** is the last number used.

3. Press [Return] or Button 1 to accept the code displayed, or type a new code and press [Return] or Button 1.

   These prompts are displayed for each pin of the symbol.

SCMD/SPAT

ERROR MESSAGES:

Message: Out of range (0 to 2).

Cause: The value entered for pin type is outside the allowable range.

Action: Enter a value from 0 to 2.

Message: Out of range (0 to 24).

Cause: A value outside the allowable range for logic equivalence was entered.

Action: Enter a value from 0 to 24.

Mode: DETL **Keyboard Command: /SGAT**

(Set Global Attributes)

FUNCTION:

Specifies the snap-to-pin radius and specifies a global attribute to freeze or open nets to editing commands.

REMARKS:

Enter this command with the keyboard.

Currently, /SGAT is used to set the snap-to-pin radius associated with the ENTR/WIRE command.

The allowable range for the snap radius is from 1 to 1000 database units. This snapping action can be turned off by setting the tolerance region to 1.

To turn the snap-to-pin radius on, toggle the L on the status line to green. If this feature is on when you are entering wires, the wires snap to the nearest pin within the tolerance set with /SGAT. (See the status line parameter L description.)

The netlist can be frozen so that the EDIT/DELV and EDIT/DELS commands will not delete frozen nets and the pins remain committed. The /SGAT command does not protect nets from the ENTR/UCOM command or the DELETE commands.

/SGAT

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type SGAT and press [Return] or Button 1. The system prompts:

   Snap tolerance= <n>. New value=

   The previous tolerance setting <n> appears with the prompt.

3. Press [Return] to accept the setting, or type the desired snap radius and press [Return] or Button 1.

   The following prompt refers to freezing a netlist.

   If the netlist is frozen, the system prompts:

   Net list is Frozen. Free it? YES NO

   To unprotect, or open, the netlist, select YES. To keep a frozen netlist, select NO.

   If the netlist is open, the system prompts:

   Net list is Open. Freeze it? YES NO

   To protect, or freeze, the netlist, select YES. To keep the open netlist, select NO.

/SGAT

ERROR MESSAGES:

Message: Invalid "<xxx>"

Cause: A nonnumeric character was entered as the value.

Action: Enter a valid number within the range 1 to 1000.

Message: Out of range (1 to 1000).

Cause: The number entered for the snap-to-pin radius is outside of the range 1 to 1000.

Action: Re-enter a valid number.

Mode: SYMB
DETL

STO

(Store)

FUNCTION:

Saves view windows for later recall by the RCL command.

REMARKS:

The STO command allows the storage of up to ten views, including a MAP view. Any of the numbered views that are stored during a session are saved with the schematic when you select FILE/SAVE.

If the **Video save to disk** option on the PC-CAPS Configuration Screen is off, no bit map is stored. If the **Video save to disk** option is on, the bit map is saved for all graphic boards that support video saves.

The STO command is a nested command and can be used while one of the main operational commands is active.

OPERATION:

1. Select STO. The system displays the status line menu:

   SAVE VIEW: 1 2 3 4 5 6 7 8 9 MAP

2. Select one of the numbers or MAP. The system stores the current screen display as either a recallable view, referenced by the number chosen, or as a MAP view.

ERROR MESSAGES:

Message: **Already Defined. Overwrite? YES NO**

Cause: A stored view has already been assigned to the number selected.

Action: Select YES to overwrite the stored view with a new one. Select NO or press Button 2 to retain the original view and select another number for storing the view you want to save.

SYMB

(Symbol)

FUNCTION:

Sets the system to the SYMB (symbol) command mode.

REMARKS:

This mode is used for creating and editing graphic representations of symbols, symbolic views, or entities.

The main menu color is red when the system is in the SYMB command mode.

Data entered in the SYMB command mode can only be edited in the SYMB mode.

Symbols and components created in the SYMB mode can be entered in a schematic in DETL mode with the ENTR/COMP command.

OPERATION:

Select SYMB to change to the SYMB command mode. The main menu turns red.

Mode: SYMB
DETL

SYS

(System)

FUNCTION:

Enables the submenu for the following system operations: DOS, plot, statistics, or quit.

OPERATION:

Select SYS to enable the following submenu choices:

DOS
PLOT
STAT
QUIT

Mode: SYMB
DETL

SYS/DOS

(System/DOS)

FUNCTION:

Allows you to execute commonly used DOS commands from within the PC-CAPS environment.

REMARKS:

The commands you can enter from within the PC-CAPS environment include:

CHDIR (CD)	- Changes directory
CLS	- Clears the display screen
COPY	- Copies a file
DATE	- Sets the date
DEL	- Deletes a file
DIR	- Displays the directory list
ERASE	- Deletes a file
FIND	- Finds a pattern
MKDIR (MD)	- Makes a directory
PATH	- Sets the DOS search path
RENAME (REN)	- Renames a file
RMDIR (RD)	- Removes a directory
SET	- Sets environment parameters
TIME	- Sets the time
TYPE	- Displays the contents of a file
VER	- Displays version number
VERIFY	- Verifies data
VOL	- Displays volume I.D.

CAUTION: Do not try to change the volume from within the PC-CAPS environment.

SYS/DOS

OPERATION:

1. Select SYS/DOS. The system prompts:

   Enter DOS command (BUTTON 2 to quit) >

2. Type the DOS command and press [Return].

3. Type in another DOS command or press Button 2 or [Esc] to return to the drawing area.

ERROR MESSAGES:

Message: **DOS ERROR: <n>**

Cause: DOS failed to execute specified command. Not enough memory or misspelled command.

Action: Exit PC-CAPS and re-enter PC-CAPS to initiate memory or check spelling and re-enter command.

Mode: SYMB
DETL

SYS/PLOT

(System/Plot)

FUNCTION:

Creates a plot file for subsequently generating a plot.

REMARKS:

Plot files created with this command can be used by PC-PLOTS and PC-PRINT.

A plot window lets you enclose the area to be plotted. This area must include the entire drawing and the entire plot window must be visible on the screen.

Any data contained on layers that are turned off will not be included in the plot file.

You may enter up to 15 characters to specify the plot file to be created, including the directory, filename, and extension.

> CAUTION: Do not use the PAN command when creating a plot file or you may lose some data from your plot file.

OPERATION:

1. Position the viewing area on the screen to encompass the portion to be plotted. Once the SYS/PLOT command is selected, do not PAN or change the viewed area.

SYS/PLOT

2. Using the VLYR command to access the layer screen, turn on all the layers for data to be included in the plot file.

3. Select SYS/PLOT. The system prompts:

   PLOT: Select Page Corner 1...

4. Position the cursor and press Button 1. The system prompts:

   PLOT: Select Page Corner 2...

   NOTE: You can also enter the cursor coordinates for page corners 1 and 2 by selecting the X Y cursor coordinate parameter on the status line and typing in the X Y cursor coordinate position for corner 1 when prompted and then for corner 2 when prompted.

5. Position the cursor and press Button 1. The system prompts:

   Plot file name: <filename>

6. Type the filename and press [Return] or Button 1.

   NOTE: You don't need to type in the filename extension; the system automatically adds the .PLT filename extension. However, if you type a period (.) after the filename, with no extension, the system does not add an extension.

SYS/PLOT

The system displays:

Writing to plot file "<filename>"

and then

Plot file complete.

If a plot file has already been created during the current session, then the last plot filename appears as the default in step 5. You may replace it with a new one. Use caution when accepting the default filename because the old file will be overwritten.

If a new name is not used, the system prompts:

Replace '<filename>'? YES NO.

If you select YES, the existing plot file is replaced.

Mode: SYMB
DETL

SYS/QUIT

(System/Quit)

FUNCTION:

Exits the command menu and returns to the opening menu.

REMARKS:

If you choose to exit without saving, all work done since the last save is lost. The data cannot be recovered except by replaying your work using the command log file. (See the /EXE command description.)

OPERATION:

Select SYS/QUIT.

If the file has been changed since it was last saved, the following message appears:

Workfile modified. Exit still? YES NO.

To continue working, select NO or press Button 2. To preserve modifications you have made to the file, select FILE/SAVE.

To quit and return to the opening menu, select YES.

Mode: SYMB
DETL

SYS/STAT

(System/Statistics)

FUNCTION:

Displays a list of internal storage usage and capacity for the database file currently displayed on the screen.

REMARKS:

Following is an example of a statistics listing.

```
DATABASE STATISTICAL DATA

Db. Filename     :   \CIRCUITS\EZBOARD.SCH (24328 bytes)
Comps.               = 70        Max. = 500
Hidden Comps.        = 1
Unique Comps.        = 12        Max. = 300
Nets                 = 50        Max. = 600
Pins                 = 167       Max. = 2500/2100
PICture Group        = 152       Max. = 3500/3500
Picture ELEMent      = 2309      Max. = 48000/16000


                    Commands SYS/STAT
```

SYS/STAT

where:

Db. Filename is the name of the database currently displayed. If a pathname was specified when the file was loaded, the pathname is listed also.

For the rest of the items, the left column lists the item being described. The middle column lists the number of those items in the current database. The right column lists the maximum number of those items allowed.

The items listed are defined below:

Comps. is the number of components.

Hidden Comps. is the number of system and special symbols (for example, solder dots).

Unique Comps. is the number of unique component types.

Nets is the number of nets.

Pins is the number of pins. The right-hand column lists two numbers ("mmmm/bbbb") where "mmmm" represents the maximum number of items allowed and "bbbb" represents the number of items in the memory buffer.

SYS/STAT

PICture Group and **Picture ELEMent** are the graphical data. The right-hand column lists two numbers ("mmmm/bbbb") where "mmmm" represents the maximum number of items allowed and "bbbb" represents the maximum number of items allowed in the memory buffer.

If the memory buffer is full, the system will start paging; this is, the system will use the space on the disk drive for memory, slowing down the overall performance of your system. For example, if the number of Picture ELEMents exceeds 15000, the system will start paging.

OPERATION:

1. Select SYS/STAT. The system displays statistics for the database file last displayed.

2. To return to the drawing area, press any key or any button on the mouse.

Mode: SYMB
DETL

VLYR

(View Layer)

FUNCTION:

Enables the menu of layers and shows the status of each. Layers may be added or updated.

REMARKS:

Selection of the VLYR command displays a full-screen menu showing all layers currently available as well as their color/pen assignment numbers and status.

The layer screen allows the creation of new layers up to the system limit of 50.

When you turn on a layer that was previously not viewed, use the REDR command to display that layer. Some high resolution graphics cards may redraw automatically after exiting the layer screen.

OPERATION:

1. Select VLYR. The system displays all the available layer names, their color/pen assignment numbers for pen plotting, and their current status.

VLYR

2. You may make the following changes on this layer menu:

   - To change the name of an existing layer, select a layer name form the left column. The system prompts:

     New name: <xxx>

     where xxx is the original layer name.

     Type the name desired and press [Return].

   - To create a new layer, select the empty space directly below the last layer name displayed. The system prompts:

     New Layer name:

     Type the name desired, and press [Return].

     Respond to the prompts to select color/pen assignment and status for that layer.

   - To assign a new color to a layer, select its color/pen assignment number.

VLYR

The system displays a submenu of 15 colors. The color of the number is the viewing color. The number refers to the pen number when pen plotted.

Select a new color number.

- To change the status of a layer (ON, OFF, ABL), select the current status designation of that layer.

  Press Button 1 to toggle the status to ON, OFF, or ABL.

  Layer status is defined as follows:

  OFF - not visible
  ON - visible and editable, but not active
  ABL - visible and may become the active (accessible) layer
  ABL A - visible and active, meaning that any data created is recorded on this layer

  Only layers with ABL status can be made active.

  To make an ABL layer active, select the space directly to the right of the ABL designation.

  The active (A) status designation appears on the space selected.

VLYR

3. To return to the graphic display, select QUIT or press Button 2.

   When you return to the main menu, you may select the layer on the status line or press the [F1] key to change the active layer. The layer shown in the lower left corner of the status line is the current active layer.

Mode: SYMB
DETL

VWIN

(View Window)

FUNCTION:

Specifies a new viewing window and scales it to fit the total graphic display area.

REMARKS:

The VWIN command requires the selection of two corners to specify the rectangular viewing window. Select diagonally opposed corners, either lower-left and upper-right, or lower-right and upper-left. The corners may be selected in any order.

The VWIN command is a nested command and can be used while one of the main operational commands is active.

OPERATION:

1. Select VWIN. The system prompts:

   View Window: Corner 1...

VWIN

2. Position the cursor and select the first corner. The system prompts:

   View Window: Corner 2...

3. Select the second corner. The screen is redrawn with the new view.

Mode: SYMB
DETL

Keyboard Command: /WAIT

FUNCTION:

Inserts a pause and a message prompt into the recording of a macro file or a command log file (PCCAPS.CMD).

REMARKS:

During the recording of a macro, or any time during the recording of the current command log file, you may insert a waiting period with this command that results in a pause during the playback. A message prompt may be included and displayed during the pause.

OPERATION:

1. Type /. The system prompts:

   Menu command:

2. Type WAIT and press [Return]. The system prompts:

   Wait time (0 = wait for a key):

/WAIT

3. Type a zero or a number that specifies in seconds how long the program is to pause and press [Return]. If you enter 0 (zero), the program pauses until a key is pressed during replay. The system then prompts:

   Message:

4. Type the message that you want to display during the wait when the file plays back and press [Return]. The message is limited to 43 characters.

   If you do not want a message, press [Return] only.

Mode: SYMB
DETL

X

FUNCTION:

Toggles the cursor between normal and full-screen crosshairs.

REMARKS:

The default cursor is a small plus sign.

The full-screen cursor extends to the horizontal and vertical edges of the display area.

OPERATION:

Press [X] on the keyboard.

The cursor display mode toggles to the opposite state, either from full-screen to the small plus, or from the small plus to full-screen mode.

Mode: SYMB
DETL

ZIN

(Zoom/In)

FUNCTION:

Increases the current display magnification by a factor of two and centers the screen around the cursor.

REMARKS:

There are nine steps between the minimum and maximum display magnifications. This command allows easy viewing of small details on a large drawing.

This command accepts multiple selects to allow you to increase the display magnification by multiple factors of two in one redraw.

For example, if you press Button 1 twice quickly, the display magnification increases by a factor of four in one redraw. If you press Button 1 three times quickly, the display magnification increases by a factor of six in one redraw.

To halt the zoom and subsequent redraw, press Button 1 or Button 2 while the command is in progress. Press Button 1 again for a second level of magnification without waiting for a complete redraw of the screen.

The ZIN command is a nested command and can be used while one of the main operational commands is active.

ZIN

OPERATION:

1. Select ZIN. The system prompts:

   Select view center...

2. Move the cursor to the desired location and press Button 1.

   The system increases the display magnification by a factor of two and centers the display around the point selected.

3. To continue zooming in, press Button 1. To exit, press Button 2.

   If you zoom in all the way, the system displays:

   At zoom in limit

Mode: SYMB
DETL

ZOUT

(Zoom/Out)

FUNCTION:

Decreases the display magnification by a factor of two and centers the screen around the cursor.

REMARKS:

There are nine steps between the maximum and minimum display magnifications. This command allows easy viewing of details on a large drawing.

This command accepts multiple selects to allow you to decrease the display magnification by multiple factors of two in one redraw.

For example, if you press Button 1 twice quickly, the display magnification decreases by a factor of four in one redraw. If you press Button 1 three times quickly, the display magnification decreases by a factor of six is one redraw.

To halt the zoom and subsequent redraw, press Button 1 or Button 2 while the command is in progress.

The ZOUT command is a nested command and can be used while one of the main operational commands is active.

ZOUT

OPERATION:

1. Select ZOUT. The system prompts:

   Select view center...

2. Move the cursor to the desired location and press Button 1.

3. To continue zooming out, press Button 1.

   If you zoom out all the way, the system displays:

   At zoom out limit.

STATUS LINE PARAMETERS

This section describes the parameters available on the status line. Many of these parameters are only active during specific menu and submenu command operations. Others are active during most commands. In either case, all status line parameters can be selected without exiting from a command function.

Moving the cursor to the status line allows you to select and alter the parameters with Button 1.

Some parameters are toggles that alternate between two settings. Others prompt you for input or let you select from among several options.

This section is organized alphabetically. You can look up most parameters by the item as it shows on the status line. For example, if you see F's displayed on the status line, look under "F" in this section for a description.

However, the status line displays various layer names (for example, IEEE, WIRES, BUS, ...) that do not have an alphabetical entry unto themselves in this section. This slot on the status line is the layer parameter and is described as the LAYER parameter in this section. Cursor coordinates are covered under X Y and grid spacing under X:Y.

Note also that there are two entries for L. One entry is for left justification and the other is for toggling the snap-to-pin feature.

The following status line parameters are available with most commands:

Example:

GRID UNITS	GRID TOGGLES	X Y COORDINATES
10:10	S G	150 200

Mode: SYMB
DETL

ANGL

(Any Angle)

Active during: DRAW/LINE and ENTR/WIRE

FUNCTION:

Specifies that lines or wires may be entered at any angle.

REMARKS:

The alternatives are ORTH (orthogonal), which limits the drawing of lines or the entry of wires to 90 degree angles only, or 45D, which limits the drawing of lines or the entry of wires at 45 or 90 degree angles.

OPERATION:

1. Select DRAW/LINE or ENTR/WIRE. The system displays the last mode selected (ORTH, ANGL, or 45D) on the status line.

2. Toggle to ANGL mode.

3. Continue with the DRAW/LINE or ENTR/WIRE command.

Mode: SYMB
DETL

B

(Bottom Justified)

Active during: ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies that text is bottom justified relative to the cursor location.

REMARKS:

Two fields are displayed on the status line for horizontal and vertical text justification.

Each field provides three choices for text justification. These fields lie between the text size and orientation fields. The horizontal field is to the left of the vertical field.

The choices for horizontal justification of text are: L, C, and R, for left, center, and right, respectively.

The choices for vertical justification of text are: T, C, and B, for top, center, and bottom, respectively.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, or SCMD/PNLC.

Status Line Parameter: B

2. To specify bottom vertical justification, select B.

3. To specify horizontal justification, select L, C, or R for the desired justification.

4. Proceed with a menu command.

Mode: SYMB
DETL

C

(Center Justified)

Active during: ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies that text is center justified relative to the cursor location.

REMARKS:

Two fields are displayed on the status line for horizontal and vertical text justification. Each field provides three choices for text justification. These fields lie between the text size and orientation fields. The horizontal field is to the left of the vertical field.

The choices for horizontal justification of text are: L, C, and R, for left, center, and right, respectively.

The choices for vertical justification of text are: T, C, and B, for top, center, and bottom, respectively.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, or SCMD/PNLC.

Status Line Parameter: C

2. To specify center horizontal justification, select C.

3. To specify center vertical justification, select C in the respective field.

4. Proceed with a menu command.

Mode: SYMB
DETL

DASHED

Active during: DRAW/LINE, CIRC, ARC, and RECT

FUNCTION:

Draws dashed line segments during DRAW/LINE, CIRC, ARC, or RECT.

REMARKS:

This field has three options: SOLID, DOTTED, and DASHED.

OPERATION:

1. Select DRAW/LINE, CIRC, ARC, or RECT. The system displays the last line type selected (SOLID, DASHED, DOTTED) on the status line.
2. Select DASHED from the line type field.
3. Proceed with the current DRAW command.

Mode: SYMB
DETL

DOTTED

Active during: DRAW/LINE, CIRC, ARC, and RECT

FUNCTION:

Draws dotted line segments during DRAW/LINE, CIRC, ARC, or RECT.

REMARKS:

This field has three options: SOLID, DOTTED, and DASHED.

OPERATION:

1. Select DRAW/LINE, CIRC, ARC, or RECT. The system displays on the status line the last line type selected (SOLID, DASHED, DOTTED).
2. Select DOTTED from the line type field.
3. Proceed with the current DRAW command.

Mode: SYMB
DETL

F

(Orientation)

Active during: ATTR/ACOM, DRAW/TEXT, ENTR/COMP, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies text or component orientation equal to that of the position of the green F on the status line.

REMARKS:

The four choices for orientation are represented by the four F's displayed on the status line. They are:

- Normal, left to right text
- 90 degrees rotated counterclockwise
- 180 degrees rotated counterclockwise
- 270 degrees rotated counterclockwise

The current orientation is specified by the F that is highlighted in green. The other three available orientations are displayed in red.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, ENTR/COMP, NAME/COMP, NAME/NET, or SCMD/PNLC.

Status Line Parameter: F

2. Select the group of four F's specifying the four available orientations.

3. Press Button 1 to highlight the F with the desired orientation. The F that turns green represents the orientation for entering text or components.

4. Proceed with a menu command.

Mode: SYMB
DETL

G

(Grid Lock)

Active during: All Commands

FUNCTION:

Specifies whether the grid lock is on or off.

REMARKS:

If the G on the status line is green, then grid lock is on. When the grid lock is on, the cursor is forced to move in the increments specified by the X and Y grid spacing assignment on the status line.

If the G on the status line is red, then grid lock is off. When the grid lock is off, the cursor is free to move in increments of 1 database unit.

The [F8] key also toggles the grid lock.

OPERATION:

1. Select G with the mouse.
2. Toggle to the opposite state, turning the cursor grid lock on (green) or off (red).
3. Proceed with a menu command.

Mode: SYMB **I/O**

Active during: ENTR/PIN

FUNCTION:

Specifies the pin type as an I/O (input/output) pin during ENTR/PIN.

REMARKS:

When you create symbols, you must enter each pin with the ENTR/PIN command.

The pin type options are INPUT, OUTPUT, and I/O.

OPERATION:

1. Select ENTR/PIN. The system displays the last active pin type on the status line.
2. Move the cursor to the pin type parameter on the status line.
3. Select I/O.
4. Proceed with the ENTR/PIN command.

Mode: SYMB

INPUT

Active during: ENTR/PIN

FUNCTION:

Specifies the pin type as an input pin during ENTR/PIN.

REMARKS:

When you create symbols, you must enter each pin with the ENTR/PIN command.

The pin type options are INPUT, OUTPUT, and I/O.

OPERATION:

1. Select ENTR/PIN. The system displays the last active pin type on the status line.
2. Move the cursor to the pin type parameter on the status line.
3. Select INPUT.
4. Proceed with the ENTR/PIN command.

Mode: SYMB
DETL

L

(Left Justified)

Active during: ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies that text is left justified relative to the cursor location.

REMARKS:

Two fields are displayed on the status line for horizontal and vertical text justification.

Each field provides three choices for text justification. These fields lie between the text size and orientation fields. The horizontal field is to the left of the vertical field.

The choices for horizontal justification of text are: L, C, and R, for left, center, and right, respectively.

The choices for vertical justification of text are: T, C, and B, for top, center, and bottom, respectively.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, or SCMD/PNLC.

Status Line Parameter: L (Left Justified)

2. To specify left horizontal justification, select L.

3. To specify vertical justification, select T, C, or B for the desired justification.

4. Proceed with a menu command.

Mode: DETL **L**

(Lock Snap to Pin)

Active during: ENTR/WIRE

FUNCTION:

Specifies that wires snap to nearest pin location within the tolerance radius set with the /SGAT command.

REMARKS:

The status line indicator for snap to pin is an L that appears between the W:<n> and X:Y grid spacing features.

This toggle locks the snap to pin feature on (green) or turns it off (red).

OPERATION:

1. Select ENTR/WIRE.
2. Select the L on the status line.
3. Toggle the L on (red) or off (green).
4. Proceed with the ENTR/WIRE command.

Mode: SYMB
DETL

LAYER

Active during: All commands that add graphics to the database

---

FUNCTION:

Selects the currently active layer from among the enabled layers.

REMARKS:

The status line layer parameter allows you to change layers "on the fly." The [F1] key also selects among the enabled layers. Only layers that have an ABL status can be chosen on the status line or with [F1].

Use the VLYR command to change layer status. When VLYR is used to view the layer status, ABL indicates which layer(s) are enabled, meaning that they may become the active destination layer for new data.

Every layer has a current status. The four possible states are:

OFF - not visible
ON - visible and editable but not accessible
ABL - visible and may become active (accessible)
ABL A - visible and active

The active layer (ABL A) is the current destination for data.

Status Line Parameter: LAYER

Only an ABL layer can become an ABL A layer. Only one layer is the ABL A layer at any time. A layer must have the active assignment removed before it can change status to OFF or ON.

OPERATION:

1. Select a command. The system displays the name of the current ABL A layer selected on the status line.

2. Select the layer name displayed. The system switches to the next ABL layer, if any. The layer shown on the status line is the active layer.

Mode: DETL **M**

(Mirror)

Active during: ATTR/ACOM, DRAW/TEXT, ENTR/COMP, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies a mirror image orientation for the placement of text or a component.

REMARKS:

The status line indicator for mirror image orientation is the M on the status line.

If the M is green, then mirror image is on. If the M is red, then mirror image is off, or normal.

When you enter text with the Mirror feature turned on, the text displays mirrored on the screen and is mirrored when the text is plotted.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, ENTR/COMP, NAME/COMP, NAME/NET, or SCMD/PNLC.
2. Select M on the status line.
3. Toggle mirror image on (green), or off (red).
4. Proceed with a menu command.

Mode: SYMB
DETL

ORTH

(Orthogonal)

Active during: DRAW/LINE and ENTR/WIRE

FUNCTION:

Specifies that lines or wires may be entered at 90 degree angles only.

REMARKS:

The alternatives are ANGL (Angle), which allows the drawing of lines or entry of wires at any angle, or 45D, which limits the drawing of lines or the entry of wires at 45 or 90 degree angles only.

OPERATION:

1. Select DRAW/LINE or ENTR/WIRE. The system displays the last mode selected (ORTH, ANGL, or 45D) on the status line.

2. Toggle to ORTH mode.

3. Continue with the DRAW/LINE or ENTR/WIRE command.

Mode: SYMB

OUTPUT

Active during: ENTR/PIN

FUNCTION:

Specifies the pin type as an output pin during ENTR/PIN.

REMARKS:

When you create symbols, you must enter each pin with the ENTR/PIN command.

The pin type options are INPUT, OUTPUT, and I/O.

OPERATION:

1. Select ENTR/PIN. The system displays the last active pin type on the status line.
2. Move the cursor to the pin type parameter on the status line.
3. Select OUTPUT.
4. Proceed with the ENTR/PIN command.

Mode: SYMB
DETL

R

(Right Justified)

Active during: ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies that text is right justified relative to the cursor location.

REMARKS:

Two fields are displayed on the status line for horizontal and vertical text justification.

Each field provides three choices for text justification. These fields lie between the text size and orientation fields. The horizontal field is to the left of the vertical field.

The choices for horizontal justification of text are: L, C, and R, for left, center, and right, respectively.

The choices for vertical justification of text are: T, C, and B, for top, center, and bottom, respectively.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, or SCMD/PNLC.

Status Line Parameter: R

2. To specify right horizontal justification, select R.

3. To specify vertical justification, select T, C, or B for the desired justification.

4. Proceed with a menu command.

Mode: SYMB
DETL

S

(Show Grid)

Active during: Most Commands

FUNCTION:

Specifies whether the screen display of grid points is on or off.

REMARKS:

If the S on the status line is green, then the screen display is on. If the S is red, then the screen display is off.

When the screen display is on, the system displays grid reference points in the increments defined by the X and Y grid setting.

The screen displays a reference grid as long as the S indicator on the status line is green, provided the current viewing window scale allows for adequate display of the dots without cluttering the graphics.

When the view is zoomed out, the system doubles the display increments set by the X and Y Grid setting to provide a visual reference. If zooming out continues, the system discontinues the grid display.

The [F7] key also toggles this feature.

Status Line Parameter: S

OPERATION:

1. Select S on the status line or press [F7]. The system toggles the grid display on (green) or off (red).

2. Proceed with a menu command.

Mode: SYMB
DETL

SIZ:<n>

(Text Size)

Active during: ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies the current text size.

REMARKS:

Format is SIZ:<n>, where <n> is an integer from 2 to 5000, specifying the height of any text to be placed in database units.

The width of text is proportionate to the selected height.

OPERATION:

1. Select SIZ:<n> on the status line. The system prompts:

   Enter text size:

2. Type the new text height and press [Return].

   The new text size is displayed on the status line.

3. Proceed with a menu command.

Status Line Parameter: SIZ:<n>

ERROR MESSAGES:

Message: Out of range (2 to 5000).

Cause: The number entered is less than 2 or greater than 5000.

Action: Enter a valid number for text size.

SOLID

Mode: SYMB
DETL

Active during: DRAW/LINE, CIRC, ARC, and RECT

FUNCTION:

Draws solid line segments during DRAW/LINE, CIRC, ARC, or RECT.

REMARKS:

This field has three options: SOLID, DOTTED, and DASHED.

OPERATION:

1. Select DRAW/LINE, CIRC, ARC or RECT. The system displays on the status line the last line type selected (SOLID, DASHED, DOTTED).

2. Select SOLID from the line type field.

3. Proceed with the current DRAW command.

Mode: SYMB
DETL

T

(Top Justified)

Active during: ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, and SCMD/PNLC

FUNCTION:

Specifies that text is top justified relative to the cursor location.

REMARKS:

Two fields are displayed on the status line for horizontal and vertical text justification. Each field provides three choices for text justification.

The choices for vertical justification of text are: T, C, and B, for top, center, and bottom, respectively.

The choices for horizontal justification of text are: L, C, and R, for left, center, and right, respectively.

OPERATION:

1. Select ATTR/ACOM, DRAW/TEXT, NAME/COMP, NAME/NET, or SCMD/PNLC.

2. To specify top vertical justification, select T.

Status Line Parameter: T

3. To specify horizontal justification, select L, C, or R for the desired justification.

4. Proceed with a menu command.

Mode: SYMB
DETL

W:<n>

(Width)

Active during: DRAW/LINE, RECT, CIRC, ARC, and ENTR/WIRE

FUNCTION:

Specifies the current width for line segments, including lines, wires, rectangles, circles, and arcs.

REMARKS:

Specify an integer from 0 to 250 to represent the width of line graphics or wires in database units. A setting of 0 (zero) is the minimum width available.

OPERATION:

1. Select DRAW/LINE, RECT, CIRC, ARC, or ENTR/WIRE. The status line shows the current width setting as W:<n>.

2. Select the W:<n> feature on the status line. The system prompts:

   Enter line width:

Status Line Parameter: W:<n>

3. Type the new width and press [Return]. The new width setting is displayed on the status line.

ERROR MESSAGES:

Message: **Out of range (0 to 250)**

Cause: An invalid number was entered.

Action: Enter a number within the range.

Mode: SYMB
DETL

X Y

(Cursor Coordinates)

Active during: Most Commands

FUNCTION:

Displays the current cursor position in the lower right corner of the screen and can be selected to change the coordinates.

REMARKS:

The X Y display is constantly updated when the cursor is moved so that the current X and Y location of the cursor can be determined.

Typing X and Y coordinates on the keyboard allows you to enter lines, wires, and other graphics at locations that are offscreen.

The keyboard entry of coordinates can specify the location of the viewing window during the use of the PAN and LPAN commands.

Status Line Parameter: X Y

OPERATION:

1. Select the X Y coordinates on the status line. The system prompts:

   Enter X position:

2. Type the new X coordinate and press [Return] or Button 1. The system prompts:

   Enter Y position:

3. Type the new Y coordinate and press [Return] or Button 1. The system prompts you to enter more coordinates if more coordinates are required,

   or

   Carries out the desired command at the coordinates just specified.

   NOTE: If the coordinates are not visible and your cursor is free to move, you may still select the slot on the status line for the X Y coordinates.

   You may also enter both the X and the Y coordinates in response to the first prompt. Type the X coordinate and press the space bar. Type the Y coordinate and press [Return].

Mode: SYMB
DETL

X:Y Display

(Grid Spacing)

Active during: Most Commands

FUNCTION:

The numeric display on the status line that indicates in database units the intervals between grid points.

REMARKS:

The X:Y display is used to assign the X and Y intervals desired between the grid points.

When the grid lock is on, the cursor moves from grid point to grid point in increments specified by the X:Y display.

The grid reflects the increments specified by the X:Y feature.

OPERATION:

1. Select the X:Y slot on the status line. The system prompts:

   Enter x grid size:

Status Line Parameter: X:Y Display

2. Type the new grid setting for the X axis and press [Return] or Button 1. The system prompts:

   Enter y grid size:

3. Type the new grid setting for the Y axis and press [Return] or Button 1.

   NOTE: You can also enter both the X and Y settings to the first prompt by separating them with a space.

Mode: SYMB
DETL

45D

(45 Degree Angles)

Active during: DRAW/LINE and ENTR/WIRE

FUNCTION:

Specifies that lines or wires may be entered at 45 or 90 degree angles only.

REMARKS:

The alternatives are ORTH (orthogonal), which limits the drawing of lines or the entry of wires to 90 degree angles only, or ANGL, which allows the drawing of lines or the entry of wires at any angle.

OPERATION:

1. Select DRAW/LINE or ENTR/WIRE. The system displays the last mode selected (ORTH, ANGL, or 45D) on the status line.

2. Toggle to 45D line mode.

3. Continue with the DRAW/LINE or ENTR/WIRE command.

CHAPTER 4. DOS REFERENCE

DOS is the Disk Operating System for your personal computer. DOS starts your computer, controls the disk drives, and allocates file storage. This chapter covers what you need to know about DOS to use P-CAD software. It applies to versions 2.0, 2.1, and 3.0 of DOS. The commands and other information in this chapter are arranged alphabetically.

This reference assumes the use of an IBM PC (or compatible computer) with a hard disk and an IBM PC style keyboard. If you have another type of computer or a different keyboard, there may be a difference in the keys, the hard disk, or other parts of the system. Check your computer manual if you are not sure about your system configuration.

This chapter is a reference to DOS as DOS applies to the P-CAD system. For comprehensive information on DOS commands, see your DOS manual.

AUTOEXEC.BAT FILE

To run P-CAD programs, P-CAD recommends that you use an AUTOEXEC.BAT file in the root directory for drive C: (your hard disk). A letter such as A, B, or C, followed by a colon is the drive specifier; it tells DOS on which drive a file is located.

The AUTOEXEC.BAT file contains DOS commands that are automatically executed when you boot your system. (To boot a computer system means to start and load the operating system.) You can create this file with or without a text editor.

NOTE: If you used the P-CAD AUTOLOADER, you don't need to create the AUTOEXEC.BAT file. The AUTOLOADER creates the AUTOEXEC.BAT file with the appropriate commands. However, you can edit the file (if desired) to add more commands.

You can use the AUTOEXEC.BAT file to perform many functions. For example, it can automatically display a message or copy files each time you start your system.

To create an AUTOEXEC.BAT file with a text editor:

1. Create a new nondocument file named AUTOEXEC.BAT and type:

   ```
   PATH C:\DOS;C:\PCAD\EXE [Return]
   PROMPT $P$G [Return]
   ```

   NOTE: The first command line sets the search path to two places: the DOS directory and the directory where the P-CAD program files are located.

   The second command line instructs the system to display the current path as part of the " >" prompt.

2. If your text editor is not in the root directory, copy the AUTOEXEC.BAT file you created to the root directory by typing:

   ```
   COPY AUTOEXEC.BAT C:\ [Return]
   ```

   NOTE: With a text editor, you can edit the AUTOEXEC.BAT file to add other command lines that the system will execute when it is booted. See your DOS manual for more information.

3. After you create or edit the AUTOEXEC.BAT file, press [Ctrl]-[Alt]-[Del] to restart the system.

To create an AUTOEXEC.BAT file without a text editor:

1. At the root directory, type:

   ```
   COPY CON: AUTOEXEC.BAT [Return]
   PATH C:\DOS;C:\PCAD\EXE [Return]
   PROMPT $P$G [Return]
   ```

2. Press [Ctrl]-[Z] or [F6], and then press [Return] to save the information you entered. The system responds with the message:

   1 file(s) copied

3. After you create the AUTOEXEC.BAT file, press [Ctrl]-[Alt]-[Del] to restart the system.

BACKUP COMMAND

Use the BACKUP command to make a backup of your hard disk in case your hard disk crashes. (Backups are usually made on floppy diskettes.) To backup a single file from your hard disk, use the following format:

```
BACKUP C:FILE.1 A: [Return]
```

To backup a hard disk directory named \PCAD\PROJ0, first change to that directory, then type:

```
BACKUP C:\PCAD\PROJ0 A: [Return]
```

This command causes each file from the PROJ0 directory to be backed up on the diskette in drive A.

To backup the \PCAD\PROJ0 directory and all of its subdirectories (and all of their subdirectories), include the parameter "/S." For example:

BACKUP C:\PCAD\ A:/S [Return]

backs up all the \PCAD\PROJ0 files and all of the files in subdirectories from your hard disk onto backup diskettes.

To backup your entire hard disk, type:

BACKUP C:\ A: /S [Return]

NOTE: To copy backed up files to the hard disk, you must use the RESTORE command. See "RESTORE COMMAND."

BATCH FILES

Batch files enable you to use your system more efficiently by setting up sequences of frequently used DOS commands. A batch file contains one or more DOS commands and requires .BAT as the filename extension. Once you create a batch file, you can type the name of that file with or without the .BAT extension. The file causes commands to execute sequentially, as if you had entered them one after another at the keyboard.

You can use the COPY CON: command or a line or text editor such as EDLIN to write batch files. In the following example, COPY CON: is used to write a batch file that causes your system to change to another directory and to load a program. To create this type of batch file, type:

```
COPY CON: PROJ1.BAT [Return]
CD C:\PCAD\PROJ1 [Return]
PCPLOTS [Return]
[Ctrl]-[Z] [Return]
```

Once this batch file is created, you can type PROJ1 and press [Return] to execute the commands in the PROJ1.BAT file. DOS then changes directories to the \PCAD\PROJ1 directory and runs the PC-PLOTS program.

P-CAD recommends that you place batch files in the root directory or in another directory that has been included in the PATH command of the AUTOEXEC.BAT file.

BOOTING AND REBOOTING

To boot the system when its power is off, simply turn its power switch on. To reboot when the power is on, press [Ctrl]-[Alt]-[Del]. Keep the door on drive A open when you boot or reboot to assure that DOS accesses the hard disk.

> CAUTION: Rebooting stops your current operation and starts the system from scratch. Any work that you have not saved on disk is lost.

CHECKING DISK SPACE

You can check disk space by using either the DIR command or the CHKDSK command. The DIR command is explained later in this chapter. It provides a list of all files on the current directory and reports the remaining disk space.

The CHKDSK command displays a summary of disk space in use and disk space available. It also reports total and free RAM. To use CHKDSK to check your A disk, type:

CHKDSK A: [Return]

CHKDSK corrects certain file errors if you specify the /F parameter in the CHKDSK command. For more information, refer to your DOS manual.

CONFIG.SYS FILE

The CONFIG.SYS file is a system configuration file that allocates memory space for files and for temporary storage areas called buffers. DOS uses the CONFIG.SYS file only when you boot or reboot your system.

If your system does not already have a CONFIG.SYS file containing the following information, you must create one. You can create this file with or without a text editor.

> NOTE: If you used the P-CAD AUTOLOADER, you don't need to create the CONFIG.SYS file. The AUTOLOADER creates the CONFIG.SYS file with the appropriate commands. However, you can edit the file (if desired) to add more commands.

To create a CONFIG.SYS file with a text editor:

1. Create a file named CONFIG.SYS and type:

   BUFFERS=12 [Return]
   FILES=15 [Return]

NOTE: If you have an existing CONFIG.SYS file, add the above information for buffers and for files. The first command line specifies the blocks of memory that DOS allocates when you start it. The second command line specifies the maximum number of files that DOS allows to be open at one time.

2. If your text editor is not in the root directory, copy the file you created to the root directory by typing:

   COPY CONFIG.SYS C:\ [Return]

3. Press [Ctrl]-[Alt]-[Del] to restart the system.

To create a CONFIG.SYS file without a text editor:

1. At the root directory, type:

   COPY CON: CONFIG.SYS [Return]
   BUFFERS=12 [Return]
   FILES=15 [Return]

   NOTE: The first command line directs DOS to copy what you enter from the keyboard into a file named CONFIG.SYS. The command lines for buffers and files are described earlier in this section.

2. Press [Ctrl]-[Z] or [F6], and then press [Return] to save the information that you entered. The system responds with the message:

   1 file(s) copied

3. After you create the CONFIG.SYS file, press [Ctrl]-[Alt]-[Del] to restart the system.

COPY CON: COMMAND

The COPY CON: command creates files by copying whatever you type on the keyboard (console) to a new file. COPY CON: is useful for creating batch (<filename>.BAT) and other short files.

To use the COPY CON: command, type:

COPY CON: <filename> [Return]

where **<filename>** is the name of the file you want to create.

The cursor blinks on the line below. The system waits for you to type the commands you want the new file to include.

Type one command per line and press [Return] after each command. On the final line, press [Ctrl]-[Z] or [F6], and then press [Return] to end the file.

For example, to set up an AUTOEXEC.BAT file with the COPY CON: command, start from the root directory and type:

```
COPY CON: AUTOEXEC.BAT [Return]
PATH C:\DOS;C:\PCAD\EXE [Return]
PROMPT $P$G [Return]
[F6] [Return]
```

The AUTOEXEC.BAT file in this example executes two commands when the system is started. The PATH command instructs DOS to search two directories for commands or programs. The PROMPT command causes the on-screen prompt to display the path to the current directory.

COPYING FILES

With the COPY command you can copy files onto diskettes and onto your hard disk. The format of this command is shown as follows:

COPY <file1> <file2>

where **<file1>** is the name of the source file (the file from which you are copying) and **<file2>** is the name of the destination file (the file to which you are copying).

If you do not use a second filename, DOS assigns the name of the source file to the new file, unless you are in the directory of the file being copied. In that case, DOS gives the following error message:

File cannot be copied onto itself.

The following options can precede a filename used in the copy command:

- Disk drive designator (A: or C:)
- Path specifier such as \PCAD\PROJ1

If you use both options, the disk drive designator must come before the path specifier, as in this example:

COPY C:\PCAD\PROJ3\DESIGN.1 A:

This command copies the DESIGN.1 file from the \PCAD\PROJ3 directory on drive C to the current directory for drive A.

Copying From a Diskette

To copy all of the files on a diskette in drive A to the current directory of drive C, change to the destination directory (the directory to which you will copy) on the hard disk and type:

COPY A:*.* C: [Return]

Each filename is listed on the screen as it is copied. To copy one file from a diskette in drive A to the current directory of the hard disk, type:

COPY A:LAYS.PCB C: [Return]

CAUTION: Before copying, you may want to write-protect your source diskette to prevent accidentally overwriting a program or data that you want to keep. To write-protect your source diskette, cover the notch on the diskette with one of the gummed tabs usually provided with new diskettes.

Copying to a Diskette

When copying files to a blank diskette, you must first format the diskette with the FORMAT command (see "Formatting a Diskette").

To copy a single file from a hard disk directory to a diskette, type the specific filename and filename extension. For example, to copy a file named PWR.SYM from a directory named MODEM to the diskette in drive A, change to the MODEM directory by typing:

CD \PCAD\MODEM [Return]

Then type:

COPY C:PWR.SYM A: [Return]

The PWR.SYM file is copied to the diskette in drive A.

Copying a Diskette

To copy all the files on a diskette to another diskette, use the DISKCOPY command. This command is useful for making backup copies of any working diskettes you use.

To use this command, type:

DISKCOPY A: A: [Return]

The system prompts you to insert the source diskette (the original) and then the destination diskette (the copy) into drive A.

NOTE: The DISKCOPY command formats the destination diskette while copying it. If the diskette is not blank, all the data is erased.

Copying Between Directories

Sometimes it is necessary to copy between directories on your hard disk. For example, you might have a schematic or board layout in a directory named PROJ1 on the hard disk and need to move it to another hard disk directory, PROJ2. First, change to the PROJ1 directory by typing:

```
CD \PCAD\PROJ1 [Return]
```

Then, to copy all the PROJ1 files to PROJ2, type:

```
COPY *.* \PCAD\PROJ2 [Return]
```

DOS copies all of the files in PROJ1 to the PROJ2 directory.

To copy a single file from one directory to another, substitute the filename for the *.* in the example above.

DIRECTORIES

Directories help keep your files organized and easily accessible. A hard disk is usually organized into a system consisting of a root directory and several subdirectories. Floppy diskettes can also be organized in this fashion.

The number of subdirectories and files that any directory can have is based upon the version of DOS that you have.

This section includes explanations of the root directory, directory commands, and directory listings.

For information on specifying paths between directories, see "Specifying a Path" in this chapter.

The Root Directory

The root directory is the topmost directory in the directory structure. It contains essential files for starting the computer.

For example, the AUTOEXEC.BAT file is located in the root directory. The **C:\>** prompt indicates the root directory.

Directory Commands

There are three essential commands for managing hard disk directories.

- **MD** Make a new directory
- **CD** Change to a directory
- **RD** Remove a directory

Making a Directory

To make a directory, type MD followed by a space and the name you want to give the directory. Each directory must have a unique name of no more than eight characters. A backslash (\) after the space in the MD command instructs DOS to create the new directory under the root directory.

To create a directory named PCAD under the root directory, type:

MD \PCAD [Return]

A new directory named PCAD is created under the root directory on your hard disk.

To make a directory named PROJ1 under the PCAD directory, type:

MD \PCAD\PROJ1 [Return]

Changing to Another Directory

To change to another directory, type CD followed by the path to the directory to which you are changing. For example, to change to the \PCAD\PROJ1 directory, type:

CD \PCAD\PROJ1 [Return]

Removing a Directory

Before removing a directory, copy the files that you want to keep into another directory or onto a backup diskette. Removing a directory requires that you delete all of the files in that directory.

To remove a directory, first change to that directory and erase all of its files. Then change to its parent directory and use the RD command to remove the unwanted directory.

For example, to remove a directory named \PCAD\PROJ9, erase all of its files, change to its parent directory, and then type:

RD \PCAD\PROJ9 [Return]

If you're currently in the \PCAD directory, type:

RD PROJ9 [Return]

Directory Listings

The DIR command tells the system to display a listing of the files on a specified diskette or hard disk directory. For each file, the directory listing shows:

- The filename and its extension
- The file length in bytes
- The date and time the file was last edited
- The names of the subdirectories and the amount of free space on a specified diskette or hard disk.

To list a directory of a diskette in drive A, use the DIR command. Type:

DIR A: [Return]

For a directory listing of the root directory of the hard disk, start at the C:\> prompt and type:

DIR [Return]

To check a subdirectory, such as the \PCAD\EXE directory, change to that directory with the CD command and type:

DIR [Return]

Or, to check the \PCAD\EXE directory without changing to that directory, type:

DIR \PCAD\EXE [Return]

To cause the system to pause after listing each page of filenames, use the /P option. Type:

DIR/P [Return]

DOS prompts you to press any key for the next page. To view a multicolumn directory with only the filenames listed, use the /W option. Type:

DIR/W [Return]

EDLIN

EDLIN is the line editor that comes with DOS. For the P-CAD system, you can use EDLIN to create and to edit certain batch and text files. (You can also use any word processing program in nondocument mode to create and edit these files.)

Using EDLIN instead of the COPY CON: command to create your AUTOEXEC.BAT and CONFIG.SYS files allows you to edit and to update these files without the use of another editor.

For instructions on using EDLIN, refer to your DOS manual.

FILES AND FILENAMES

All the programs and data in the system are stored in files. This section explains how to use filenames and filename extensions, how to change a filename, and how to erase a file.

Filenames

Each file requires a unique name of up to eight characters. The name can be followed by a period and an extension of up to three characters. The following characters are valid in a filename:

- The letters of the alphabet
- The numbers 0 through 9
- Special characters: ! @ # $ % & ( ) { } _ ' '

  NOTE: Although DOS accepts the special characters listed above, avoid using them in P-CAD schematic symbol and PCB part filenames (<filename>.SYM, <filename>.PRT). Use only letters and numbers in symbol and part filenames.

Filename Extensions

Certain filename extensions are used by P-CAD software. Many of these extensions are added automatically to the filename as a default.

For instance, netlists include the .NLT default extension with the filename.

The list below gives several typical P-CAD filenames and extensions:

PCNODES.EXE
AND2.SYM
LAYS.SCH
7400.PRT

Reserved Filenames

Certain device names are reserved for use by DOS. Do not use these names for your files:

CON	AUX	COM1	COM2	NUL
LPT1	LPT2	LPT3	PRN	

For example, if you use the filename CON.20 when making a 20 pin connector, you will be forced to reboot when you try to save the file.

Changing Filenames

The RENAME (REN) command changes the name and/or extension of a file.

To rename a symbol named BUFR3.SYM to BUFFER3.SYM, for example, type:

REN BUFR3.SYM BUFFER3.SYM [Return]

To verify that the file has been renamed, use the DIR command. For example, type:

DIR BUF* [Return]

DOS lists all files beginning with BUF.

Erasing Files

You can erase files with either the DEL or the ERASE command.

> CAUTION: Before erasing a file, be sure that you really want to delete that file permanently. Once erased, a file cannot be recovered.

To erase a file, start in the directory in which the file is located. Type DEL or ERASE, followed by a space, the filename, and [Return]. For example:

ERASE ANDGATE4.SYM [Return]

The file to be erased does not have to be on the active drive. By using the drive specifier, you can delete a file on any drive in the system. To erase a file on drive A, for example, type:

DEL A:SHEET3.PCB [Return]

FORMATTING A DISKETTE

Formatting a diskette prepares it to store files by organizing it into tracks and sectors. While formatting a diskette, DOS creates a directory for the files.

> CAUTION: The FORMAT command erases the disk being formatted.

- **Do NOT type FORMAT and press [Return].** Using the FORMAT command without a disk drive letter results in formatting the default drive, which is usually the hard disk.

- **Do NOT type FORMAT C:.** This command erases the entire contents of your hard disk.

- **Do NOT format a diskette that contains files that you want to keep.** Use the FORMAT command to prepare new or blank diskettes only.

To format a blank diskette, insert it into drive A. Then, from the **C:\>** prompt, type:

FORMAT A:/V [Return]

The /V is optional. It allows you to add a volume label to the diskette directory.

After formatting with the /V parameter, DOS prompts for the volume label of the diskette. The volume label is displayed each time the DIR command is used.

A diskette label can have up to 11 characters. You might name a diskette of your data after the project you are working on, for example, MODEM PCB.

When formatting is complete, DOS asks if you want to format another diskette. Type Y for YES to repeat the procedure. Type N for NO to exit.

GLOBAL FILENAME CHARACTERS

You can use question marks (?) and asterisks (*) in filenames and in filename extensions to make it easier for you to give certain commands.

A ? can be used to represent any character within a filename or filename extension. For example, if you're in the root directory and you type:

DIR PC???.BAK [Return]

DOS will list all the files on your hard disk that begin with PC followed by any three characters and have a .BAK extension.

The * can be used to represent any character or number of characters within a file or filename extension. For example, if you're in the root directory and you type:

DIR PC*.* [Return]

DOS will list all the files on your hard disk that begin with PC and are followed by an extension or no extension. For more information on global filename characters, see the DOS manual.

PATH COMMAND

Rather than searching just the current directory, the PATH command instructs DOS to automatically search specified directories for programs and batch files to execute. The path command can be included in the AUTOEXEC.BAT file so that you do not have to re-enter it each time you start the system.

The P-CAD recommended AUTOEXEC.BAT file contains the following PATH command:

PATH C:\DOS;C:\PCAD\EXE

This command instructs DOS to search:

1. The DOS directory (C:\DOS)
2. The \PCAD\EXE directory (C:\PCAD\EXE)

Semicolons separate the directories in the command.

To check the path that is currently active, type:

PATH [Return]

PCADDRV.SYS FILE

The PCADDRV.SYS file is the P-CAD system driver file that configures the P-CAD software to your computer hardware.

If your system does not already have a PCADDRV.SYS file containing the following information, you must create one. You can create this file with or without a text editor.

> NOTE: If you used the P-CAD AUTOLOADER, you don't need to create the PCADDRV.SYS file. The INSTALL program creates the PCADDRV.SYS file with the appropriate commands. However, if desired, you can edit the PCADDRV.SYS file.

To create the PCADDRV.SYS file with a text editor:

1. Create a file named PCADDRV.SYS and type:

   SYSTEM\PCAD\DRV\<filename>.DRV
   DISPLAY\PCAD\DRV\<filename>.DRV
   INPUT\PCAD\DRV\<filename>.DRV

The first command line specifies the name of the file which contains the driver information for your hardware computer type.

The second command line specifies the name of the file which contains the driver information for your graphics display type.

The third command line specifies the name of the file which contains the driver information for your input device type.

The filenames with the .DRV extension are the device driver files which the INSTALL program loads into the \PCAD\DRV directory. Refer to your *System Overview Manual* for a list of the current driver files provided by P-CAD.

2. If your text editor is not in the root directory, copy the file you created to the root directory by typing:

   COPY PCADDRV.SYS C:\ [Return]

3. Press [Ctrl]-[Alt]-[Del] to restart the system.

To create a PCADDRV.SYS file without a text editor:

1. At the root directory, type:

   COPY CON: PCADDRV.SYS
   SYSTEM \PCAD\DRV\<filename>.DRV
   DISPLAY \PCAD\DRV\<filename>.DRV
   INPUT \PCAD\DRV\<filename>.DRV

   and press [Return] after each line.

2. Press [Ctrl]-[Z] or [F6], and then press [Return] to save the information that you entered. The system responds with the message:

   1 file(s) copied

3. After you create the PCADDRV.SYS file, press [Ctrl]-[Alt]-[Del] to restart the system.

PIPING INPUT/OUTPUT

With the DOS piping function, you can use the output of one program or command as input to another program or command. For example, the following command:

```
DIR ¦ SORT
```

causes the the output of the DIR command (an unalphabetized list of files) to be piped as input to the SORT command where the file is sorted. For more information on piping, see the DOS manual.

PRINTING A DIRECTORY LISTING

You can print a directory listing using any one of three methods.

- The [CTRL]-[P] method toggles the print function on and off. The print function causes the system to print data at the same time it is displayed.
- The [PrtSc] method prints the current screen display.
- Typing DIR > LPT1 prints a listing of the current directory.

To print a directory using [Ctrl]-[P], first make sure the printer is ready and the screen is displaying the DOS prompt for the directory you want to print (for example, C:\PCAD\PROJ1). Press:

[Ctrl]-[P]

Then type:

DIR [Return]

The system prints the directory as it is listed on the screen. When the directory is printed, press [Ctrl]-[P] again to turn the print function off.

To print a directory using [PrtSc], first make sure the printer is ready, then type DIR to display the directory. If the directory is longer than one screen, use the /P option. Then, hold down [Shift] and press [PrtSc]. The printer prints the contents of the screen. Press [Shift]-[PrtSc] again for each additional screen of the directory.

PRINTING A FILE

To print a text file, change to the appropriate drive or hard disk directory and press:

[Ctrl]-[P]

Then type:

TYPE <filename> [Return]

where **<filename>** is the name of the file you want to print.

[Ctrl]-[P] turns on the print function and the TYPE command displays the specified file.

When the entire file is printed, press [Ctrl]-[P] again to turn the print function off.

PRINTING THE SCREEN

To print on-screen information, first check that the printer is ready, then press [Shift]-[PrtSc].

All information on the screen is printed.

PROMPT (DOS)

The DOS prompt consists of the characters the system displays to show that it is ready to accept commands from the keyboard. The default prompt is the letter of the current disk drive followed by an angle bracket; for example, **C>**. You can specify another prompt using the PROMPT command.

The drive letter in the DOS prompt indicates the active drive. DOS will assume that all commands and files you use are on this drive unless you specify another drive.

The P-CAD recommended DOS prompt displays the path to the current directory. For example, the **C:\>** means that DOS is currently accessing the root directory. A **C:\PCAD\PROJ1>** prompt means that DOS is currently accessing the PROJ1 directory under the PCAD directory.

PROMPT COMMAND

Use the PROMPT command to specify the on-screen DOS prompt. If you do not use this command, the prompt is the disk drive letter without a directory name. You can include the PROMPT command in the AUTOEXEC.BAT file.

The P-CAD recommended AUTOEXEC.BAT file contains the following PROMPT command:

```
PROMPT $P$G
```

This command instructs DOS to display the current directory name in the prompt; for example:

```
C:\PCAD\EXE>.
```

There are several options available with the PROMPT command. For more information, refer to your DOS manual.

RESTORE COMMAND

With the RESTORE command you can restore files to your hard disk provided these files were previously backed up with the BACKUP command.

For example, to restore all the files in a directory named \PCAD\PROJ0, type the following:

```
RESTORE A:\PCAD\PROJ0 [Return]
```

The first parameter, "A:," is the backup diskette drive. The second parameter, "PCAD\PROJ0," is the hard disk directory you want to restore.

Files are restored to the current directory if you do not specify a path. If you do not specify a filename extension, then all files backed up from the directory will be restored.

To restore a specified directory and all backed up files in all subdirectories, include the /S parameter in the RESTORE command. For example:

```
RESTORE A: C:\PCAD /S [Return]
```

restores the \PCAD directory and all backed up subdirectories.

SPECIFYING A PATH

If you give a command involving a file that is located in a directory other than the one you're currently in, you must specify a path for DOS to use to access that file.

For example, if you're currently in the PCAD directory and you want a directory listing for a file named LOG.1 in the PROJO directory, type:

```
DIR \PCAD\PROJO\LOG.1 [Return]
```

The first backslash in this command tells DOS to begin its search at the root.

When specifying a path, you can instruct DOS to move up one level from the current directory by including two periods in the path. For example, if you're in the PRT directory and you want to access a file named LOG.2 located in the PROJ1 directory, type:

```
..\PROJ1\LOG.2 [Return]
```

DOS backs up one level to PCAD, the parent directory of PRT, and continues its path down to the PROJ1 directory and to the LOG.2 file.

For more information on specifying a path, see your DOS manual.

STOPPING A COMMAND

As soon as you press [Return] after typing a command, DOS carries out the command. To stop the command in progress, press either [Ctrl]-[C] or [Ctrl]-[Break].

STOPPING THE SCREEN TO READ

When information on the screen is scrolling by too quickly to be read, press [Ctrl]-[S]. The screen stops scrolling. Press any key to continue. This feature is useful when you are looking at lengthy files or directories.

APPENDIX A. ERROR MESSAGES

This appendix describes the error messages that may appear on the screen, the cause of each error, and the appropriate user action.

Message: Already at the root. Can not POP.

Cause: The top level in the hierarchy is already displayed.

Action: Select another command.

Message: Already Defined. Overwrite? YES NO

Cause: A stored view has already been assigned to the number selected.

Action: Select YES to overwrite the stored view with a new one. Select NO or press Button 2 to retain the original view and select another number for storing the view you want to save.

Message: Another macro definition in progress.

Cause: The system is currently recording a macro definition and the /MAC command was entered. One macro cannot be nested within another.

Action: Finish the current macro in progress before starting another one. Type /MEND to end the current macro.

Message: Both nets named. Merge not allowed.

Cause: An attempt was made to merge two nets that have been assigned different names.

Action: Do not merge the nets, or unname one of the two nets and then merge the nets.

Message: Can't open "<filename>"

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command to verify the existence or spelling of a filename.

Message: Can not copy net.

Cause: An attempt to copy a net was unsuccessful. Either the maximum number of nets has been reached, or the new net would have shorted one or more named nets.

Action: Use the SYS/STAT command to determine if the maximum number of nets has been reached, or use ENTR/WIRE to add the new net.

Message: Can not link pin to net.

Cause: Database error occurred when attempting to add a new pin to a net.

Action: Verify integrity of database and/or call P-CAD Customer Support.

Message: Can not open file <filename>.

Cause: The attempt to open the named file has failed.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: Command disabled or not found.

Cause: The SYS/DOS command is not recognized.

Action: Check spelling of command and check legal commands in command reference.

Message: Component has no symbol.

Cause: The specified component has no graphic data entered in SYMB mode.

Action: Check the spelling of the component filename. Edit the symbol file and/or check the symbol creation process against "Example 5. Creating a P-CAD Compatible Symbol" in Chapter 2, "Using PC-CAPS."

Message: Component <name> exists. Another name?

Cause: An existing component name has been entered. The system highlights the named component and prompts for a different name.

Action: Since component names must be unique, enter a different name or press Button 2 to terminate the command.

Message: Database error, contact customer support.

Cause: A fatal error condition has been detected.

Action: Contact P-CAD Customer Support and save the database, the PCCAPS.CMD file, and the PCCAPS.DBG file for further analysis.

Message: Database format not compatible.

Cause: An incompatible database format was detected upon opening the specified file name.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of the filename.

Message: Database not compatible.

Cause: The special symbol file specified is not in the correct database format.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command, then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: Depth limit exceeded. Can not PUSH.

Cause: The number of supported hierarchy levels in a schematic design has been exceeded. The LEVL/PUSH command allows only 16 levels of hierarchy.

Action: Redesign the schematic to have fewer hierarchical levels.

Message: DISK FULL!!! FILE SAVED NO GOOD!!!

Cause: The destination disk for a FILE/SAVE operation is full.

Action: Use SYS/DOS command and then use the delete command to free up space on the destination disk or insert a new diskette with available space.

Message: DOS ERROR: <n>

Cause: DOS failed to execute specified command. Not enough memory or misspelled command.

Action: Exit PC-CAPS and re-enter PC-CAPS to initiate memory or check spelling and re-enter command.

Message: Error loading <filename>.

Cause: An error occurred in loading the specified special symbol file.

Action: Check the integrity of the special symbol file. Refer to "Example 7. Creating a Solder Dot and a Special Symbol File" in "Using PC-CAPS."

Message: Exceeded limit for number of components or pins.

Cause: An attempt was made to exceed the maximum number of allowable components or pins. The component maximum is 499. The pin maximum is 2500.

Action: Redesign the schematic using hierarchy or multiple sheets.

Message: Exceeded limit for number of nets.

Cause: An attempt was made to exceed the database limit for the maximum number of nets (600).

Action: Redesign your schematic using hierarchy or multiple sheets.

Message: Filename not found...

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command to verify the existence or spelling of a filename.

Message: Gate/part mismatch. Package gate into illegal part YES/NO

Cause: A different gate type has already been assigned to this part.

Action: To package gate into illegal part anyway, select YES.

To assign gate to a different physical part, select NO.

Message: Hit more than one net.

Cause: The starting point of a new wire is at the intersection of two or more wires.

Action: Choose a different start point.

Message: Illegal section name.

Cause: An illegal section name has been specified in the SCMD/PNUM command.

Action: Refer to the command description for the SCMD/PNUM command for correct syntax or check your package definition library file (<filename>.FIL).

Message: Invalid pin name.

Cause: The specified pin name is illegal.

Action: Specify a nonblank pin name.

Message: Invalid "<x>"

Cause: A nonnumeric character was entered as the value.

Action: Enter a valid number within the valid range.

Message: Invalid "<xxx>"

Cause: A nonnumeric character was entered as the value.

Action: Enter a valid number within the range 1 to 1000.

Message: Keyword must start with a letter.

Cause: The keyword does not start with a letter.

Action: Begin the keyword with a letter.

Message: Macros nested too deep.

Cause: Nested log files exceeded the system limit of five.

Action: Do not nest the macros deeper than five.

Message: Matching " or) expected.

Cause: The value begins with a " or (, but does not end with a " or).

Action: End the value with a " where appropriate or a) where appropriate.

Message: More than one named net. No merge.

Cause: An attempt to merge more than one named net was made.

Action: Reroute the net so that only one named net is merged at a time, or unname one of the nets.

Message: Net hit is named <netname>. Change? YES NO

Cause: The new wire that you are starting to enter is named, and you have hit a named net.

Action: You have three options.

1. Select YES to change the existing net name to the net name of the wire you are currently entering.
2. Select NO to merge the wires and have the new wire acquire the name of the existing wire.
3. Press Button 2 twice and enter the wire at different coordinates.

Message: Net <name> not found.

Cause: The system cannot find a net with the name entered.

Action: Re-enter the net name and check the characters for accuracy. Make sure the proper layers are visible.

Message: New command or BUT1 expected.

Cause: An invalid input was detected.

Action: Select a new command.

Message: New database size too small.

Cause: The specified database has been built in the "large" (640K) database format and cannot be processed by the PC-CAPS (512K) program on a TI portable computer.

Action: Contact P-CAD Customer Support or use a version of PC-CAPS configured for 640K databases.

Message: No attribute found.

Cause: The cursor was not in the right location or the wrong mode is active.

Action: Change modes (SYMB or DETL) or reposition the cursor and try again.

Message: No attribute spec present.

Cause: An empty attribute specification was detected.

Action: Re-enter the attribute.

Message: No component found.

Cause: The system cannot find a component where the cursor was positioned, or the object selected is not a component.

Action: Make sure you are in the same mode in which the object was entered, reposition the cursor, and try again.

Message: No component named <filename>.

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: No database named <filename>.

Cause: The system cannot find the filename entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS command and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: No DB exists for the comp selected.

Cause: No database could be found when attempting to push into the specified component using the LEVL/PUSH command.

Action: Check the path specified on the PC-CAPS Configuration Screen. Verify that the component database file is present in the path or the current directory using the SYS/DOS command.

Message: No DB name for current design.

Cause: No name has been assigned to the current database in which a LEVL/PUSH command was attempted.

Action: Save the file using the FILE/SAVE command.

Message: No net found.

Cause: No net was found at the specified location.

Action: Check the net location using the REDR and ZIN commands and try to identify the net again.

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Message: No pin found.

Cause: No pin was found at the specified location.

Action: Use the REDR and the ZIN commands to check the pin location and try again.

Message: No segment found.

Cause: The system cannot find a segment.

Action: Make sure you are in the same mode in which the segment was entered, position the cursor, and try again.

Message: No segment/vertex found.

Cause: The system cannot find a segment or vertex.

Action: Make sure you are in the same mode in which the object was entered, position the cursor, and try again.

Message: No source net.

Cause: Program error detected in copying net.

Action: Contact P-CAD Customer Support and save the database, the PCCAPS.CMD file, and the PCCAPS.DBG file for further analysis.

Message: No symbol named <filename>.

Cause: No symbol file was found for the filename defined in the specified special symbol file.

Action: Check the special symbol file for spelling and then use the SYS/DOS command and do a DIR (Directory) to verify that the symbol file exists.

Message: Not allowed to short more than one net.

Cause: An attempt was made to merge more than one named net.

Action: Reroute the net so that only one named net is merged at a time.

Message: No vertex found.

Cause: The system cannot find a vertex.

Action: Make sure you are in the same mode in which the wire was entered, position the cursor, and try again.

Message: No wire to copy.

Cause: When copying a window, a net was enclosed that contained no wires (only connectivity). The net was not copied and no new net was created for the parallel pin(s) in the copied window.

Action: None.

Message: Obj not implemented.

Cause: Program error detected.

Action: Contact P-CAD customer support and save the database, the PCCAPS.CMD file, and the PCCAPS.DBG file for further analysis.

Message: Other subcmd from CLYR main menu entry?

Cause: A subcommand that does not pertain to the CLYR command was entered.

Action: Check the input (keyboard or command log file or macro file) for correctness and try again.

Message: Other subcmd from COPY main menu entry?

Cause: A subcommand that does not pertain to the COPY command was entered.

Action: Check the input (keyboard or command log file or macro file) for correctness and try again.

Message: Other subcmd from DEL main menu entry?

Cause: A subcommand that does not pertain to the DEL command was entered.

Action: Check the input (keyboard or command log file or macro file) for correctness and try again.

Message: Other subcmd from MOVE main menu entry?

Cause: A subcommand that does not pertain to the MOVE command was entered.

Action: Check the input (keyboard or command log file or macro file) for correctness and try again.

Message: Out of range (0 to 2).

Cause: The value entered for pin type is outside the allowable range.

Action: Enter a value from 0 to 2.

Message: Out of range (0 to 24).

Cause: A value outside the allowable range for logic equivalence was entered.

Action: Enter a value from 0 to 24.

Message: Out of range (0 to 127).

Cause: An invalid number was entered.

Action: Enter a number within the range.

Message: Out of range (0 to 250).

Cause: An invalid number was entered.

Action: Enter a number within the range.

Message: Out of range (1 to 255).

Cause: The number entered for the gates per package is outside of the allowable range.

or

The number entered for the pins per gate is outside of the allowable range.

Action: Enter a number between 1 and 255.

Message: Out of range (1 to 1000).

Cause: The number entered for the snap-to-pin radius is outside of the range 1 to 1000.

Action: Re-enter a valid number.

Message: Out of range (2 to 5000).

Cause: The number entered is less than 2 or greater than 5000.

Action: Enter a valid number for text size.

Message: Out of range (-32000 to 32000).

Cause: The number entered for the Component Type ID is outside the allowable range.

Action: Enter a value between -32000 and 32000.

Message: Out of space.

Cause: Insufficient memory to complete the command execution.

Action: Save the database using the FILE/SAVE command and re-enter PC-CAPS to initialize memory. Then try again.

Message: Overlapping component. OK to enter? YES NO

Cause: The location you selected for the new component is already occupied.

Action: You have two options.

1. Select YES to enter the component in the occupied location anyway.
2. Select NO to select another location.

Message: Overlapping pins. OK to enter? YES NO

Cause: The location you selected for the new pin is already occupied.

Action: You have two options.

1. Select YES to enter the pin in the occupied location anyway.
2. Select NO to select another location.

Message: Path too long.

Cause: The path and filename for the specified database overflows the internal filename buffer.

Action: Shorten the path in the PC-CAPS Configuration Screen.

Message: Pin connected. Can't uncommit.

Cause: When attempting to uncommit a pin, the identified pin was connected and the pin was not allowed to be uncommitted.

Action: Unconnect the pin using the EDIT/DELS command and use the ENTR/UCOM command again.

Message: Pin is not committed.

Cause: When attempting to uncommit a pin, the identified pin was already uncommitted.

Action: None.

Message: Pin num location info missing.

Cause: The selected component has not been prepared using the SCMD/PNLC command.

Action: Use the SCMD/PNLC command and return to SCMD/PNUM.

Message: PNLC: Available only in SYMB mode.

Cause: The command SCMD/PNLC was input in DETL mode.

Action: Check the input (keyboard or command log file or macro file) for valid syntax.

Message: PNUM: Available only in DETL mode.

Cause: The command SCMD/PNUM was input in SYMB mode.

Action: Check the input (keyboard or command log file or macro file) for valid syntax.

Message: Program error, contact customer support.

Cause: Program error detected.

Action: Contact P-CAD customer support and save the database, the PCCAPS.CMD file and the PCCAPS.DBG file for further analysis.

Message: Section <reference designator/section> is occupied.

Cause: The reference designator and section entered have already been used. The system highlights the existing component that already has the reference designator assigned.

Action: Enter another reference designator.

Message: Shorting more than one net.

Cause: An attempt was made to copy a wire onto pins that were owned by more than one named net.

Action: Re-enter the net so that it doesn't short any uncommitted pin or uncommit the pins that are hit before copying the wire.

Message: Symbol has no picture data.

Cause: The symbol defined in the special symbol file contains no graphic data.

Action: Edit the symbol by entering the graphic data in the SYMB mode; then try using the SCMD/GSSF command again.

Message: Symbol origin not defined.

Cause: The symbol defined in the special symbol file has no origin defined.

Action: Edit the symbol and use the ENTR/ORG command to define the origin.

Message: The specified section does not exist.

Cause: An incorrect letter was entered for the section identification. For example, if you entered E as a section identification and the valid sections are A through D, then E does not exist and is not valid.

Action: Enter a valid letter for the section identification.

Message: Valid point or new command expected.

Cause: Invalid input was detected.

Action: Enter a point or select a new command.

Message: Value part expected.

Cause: The value part of an attribute was not found for the given entry.

Action: Re-enter the attribute. Refer to the ATTR/ACOM command description for an example.

Message: = expected.

Cause: The keyword before the equal sign (=) has a space in it.

Action: Re-enter the keyword without pressing the space bar.

APPENDIX B. EXECUTABLE FILES

An executable file is a keystroke-by-keystroke recording of a series of command operations done during a PC-CAPS program session. In a later program session, you can use the /EXE keyboard command to automatically rerun the sequence of commands in the file.

Executable files are in ASCII format. You can use a text editor, such as DOS EDLIN, to redesign any portions of an executable file.

There are two types of executable files. The command log file is created automatically when you start program operation. Macro files are created during program operation when you specify that the program is to create them. You can also use a text editor to create a macro file.

The following sections describe executable files, give instructions on how to create and replay them, outline special executable file commands, and show a sample file.

> NOTE: The <filename>.CMD and <filename>.BAK files created by PC-PACK and PC-BACK are also executable files and have the same format as the command log file. PC-LOGS also produces an executable command record file, _PCLOGS.OUT, and allows you to create executable command files. These files are not described here, but are explained in the appropriate user's manuals.

THE COMMAND LOG FILE

Each time you start PC-CAPS, the program opens the PCCAPS.CMD command log file and stores in it all the commands you use during the program session.

When you run PC-CAPS again, the system renames the command log file from PCCAPS.CMD to PCCAPS.CM$. Any existing PCCAPS.CM$ file is overwritten with the new file. To save a command log file, rename it with a new filename.

There are two uses for the command log file:

- If you do not save a file in which you were working (for example, if you accidentally use the FILE/ZAP command), you can recover the lost data by renaming and then replaying the command log file.

- You can rename the command log file and use it to duplicate a series of operations for re-use (for example, you might need to draw the same circuit several times) or for demonstration purposes.

MACRO FILES

A macro file is similar to the command log file, except that you specify, during program operation, when you want the file to begin and end.

To create a macro, follow the steps below.

1. During program operation, type /MAC and press [Return].

   The program prompts for a starting coordinate position.

2. Use the mouse or the status line cursor coordinates parameter to specify the location for the origin of the macro. Each time you replay the macro, the program will use this origin point to determine the position of the entries made in the macro.

   For convenience in subsequent replaying of the macro, you may want to use 0,0 for this position. For example, you might specify the starting position as 0,0 and enter a component at 5,10. If you then replay the macro at a location of 100,100, the component's new location will be 105,110.

   The program prompts for the filename.

3. Enter the macro filename and press [Return]. For your convenience, a .MAC filename extension is recommended for all macros.

   Continue performing the program operations. The program records all the commands in a macro file with the name you specified.

4. To terminate the macro recording session, type /MEND and press [Return]. The program saves the macro file. You can continue working in the program or quit.

RUNNING AN EXECUTABLE FILE

Use the following steps to run a macro or a command log file.

1. During program operation, type /EXE and press [Return].

   The program asks for the name of the file to execute.

2. Type the name of the file and press [Return].

   If the file is a command log file, the program replays the file.

   If the file is a macro, the system prompts:

   Macro location...

   Use the mouse or the cursor coordinate status line parameter to enter the location for the macro. This point determines the new location of the origin point you specified when creating the macro. As soon as you enter the location, the program replays the macro.

When you rerun an executable file, some program conditions may not be the same as when the file was created. For example, a different layer may be active, a different viewing window may be displayed, or the line width or text orientation may be different. In cases like these, when you replay the file, the results will be different. To avoid this, you may want to save a database file that has the appropriate starting features, and load this database before replaying the file.

SPECIAL FUNCTIONS

There are several commands that perform special functions for executable files. These commands are listed below.

/EXE [Return] begins execution of a macro or a command log file.

[Ctrl]-[S], pressed while the file is executing, produces a pause. The pause lasts until you type //.

/WAIT <n> <message> [Return] produces a pause when the file is executing. You can enter this command while recording a file or add it to a file using a text editor. <n> specifies the number of seconds the pause is to last. If you do not enter a value for <n>, the pause lasts until you press the space bar. You can enter text to be displayed during the pause by using the <message> option.

/INTR [Return] interrupts the execution of the file. You can enter this command while recording a file or add it to a file using a text editor. When you execute the file, the program stops execution when it encounters this command, and returns you to normal program operation. The execution of the file resumes when you use the /RESU command.

/RESU [Return] causes a file interrupted by the /INTR command to resume.

/CFIL [Return] turns off and restarts the command log file currently being recorded. This command allows you to record only the operations you want in the command log file. For example, if you execute a macro, the /CFIL command enables you to not record all the macro file commands in the command log file.

> NOTE: This command has no effect on macro files; you must use the /MEND command to turn off a macro file.

EXECUTABLE FILE STRUCTURE

Each line in an executable file consists of one command. Menu commands and some status line commands are preceded by the word "Command." Text entered from the keyboard is preceded by the word "Text." Cursor coordinates are represented numerically. B1 signifies Button 1 of the mouse or the space bar. B2 signifies Button 2 or [Esc].

Several status line changes and other commands are represented by codes called opcodes. These codes and their meaning are listed in the next section.

A sample macro file is shown below. The command log file for the session would be the same except that the system uses a standard command log file first line of "LOGFILE" instead of "MACRO" and the last command in the command log file would be "SYS/QUIT" instead of "END_MACRO."

> NOTE: Commands must be entered in uppercase as shown below.

Sample Macro File:

```
MACRO 0 0
Command DRAW
Command RECT
B1 -180 20
B1 -50 130
B1 -30 20
B1 90 130
Command DRAW
Command LINE
B1 -180 -10
B1 90 -10
Opcode 192
Text wait
Wait 0 Press [Space] to continue...
Command ENTR
Command COMP
Text a:7400.sym
B1 -180 -120
Command WIRE
B1 -230 -80
B1 -180 -80
B2
B1 -230 -120
B1 -180 -120
B2
B1 -10 -100
B1 160 -100
B1 160 100
B2
Opcode 192
Text mend
END_MACRO
```

OPCODES

The following list shows all the opcodes and the operations they represent.

51	Set net name
60	Set component name
62	Component orientation toggle
63	Component mirror toggle
107	Set grid size
109	Cursor step size toggle
110	Grid display toggle
111	Grid lock toggle
112	Cursor coordinate keyboard entry
115	Snap-to-pin lock toggle
120	Set pin type
121	Set pin logical equivalence
125	Force vector display toggle
140	Set text size
141	Text orientation toggle
142	Text mirror toggle
145	Text horizontal justification toggle
146	Text vertical orientation toggle
190	[Esc]
192	/ (keyboard command initiation)
196	Backspace (display last message)
197	Cursor cross-hair display toggle
198	Resume execution from file
199	Interrupts in com file

APPENDIX C. USING PC-CAPS WITH A DIGITIZER

Starting with release 1.26, most P-CAD programs began using loadable device drivers to interface to the unique hardware devices such as computer hardware type, graphics hardware type, and input device type. The loadable device drivers allow you to use a digitizer with the PC-CAPS program in addition to a mouse.

WHAT IS A DIGITIZER?

A digitizer is a data input device which allows you to digitize or trace an existing drawing or layout into the system rather than creating it on-line with a mouse. The digitizer consists of two parts, a digitizer tablet on which the drawing or layout is affixed and a digitizing puck consisting of a sightpiece and function buttons. Figure C-1 shows the parts of the digitizer.

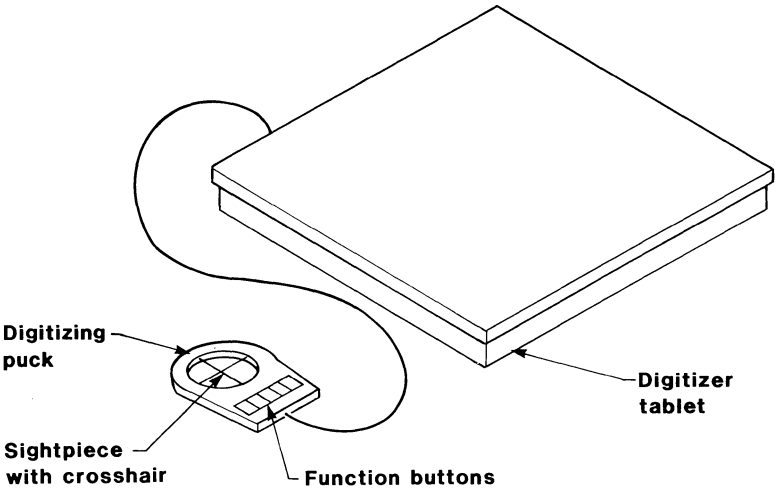


Figure C-1. Digitizer

Digitizer tablets come in sizes ranging from 12 inches by 12 inches to 36 inches by 48 inches and larger, and some have backlit surfaces.

To digitize, you affix a drawing or layout to the digitizing tablet and then trace over or "digitize" each point or vertice with the puck. The difference between this mode of input and the mouse is that with a digitizer each point on the digitizer tablet surface has a direct relationship to the screen cursor location. With a mouse the cursor location is relative.

As the puck is moved from one location on the tablet to another and back, the cursor will return to the same coordinate location within the resolution of the digitizer tablet. With the mouse, the location or position of the mouse on the surface on which it rests and any database location are relative to each other; that is, the screen cursor location and the mouse surface location have no direct relationship.

Digitizers are designed to provide a quick means of entering design drawings into the system or entering an existing design. We do not recommend that you use a digitizer in an editing mode where the database is to be zoomed and panned frequently and/or where the menus will be used continually. To edit the database, we recommend that you use the mouse. Switching back and forth from digitizer input mode to mouse input mode is explained in the next section.

INSTALLING THE LOADABLE DEVICE DRIVERS

To install the device driver for digitizing, edit the INPUT line of the loadable driver reference file (PCADDRV.SYS) in the root directory. This file is created and placed in the root directory when you run the INSTALL program. This file specifies the name and location of the device driver files. Figure C-2 shows a sample of the loadable driver reference file configured for use with a mouse.

```
SYSTEM \PCAD\DRV\SIBMPC.DRV
DISPLAY \PCAD\DRV\DIBMEGA.DRV
INPUT \PCAD\DRV\IMOUSYS.DRV
```

Figure C-2. Sample of a Loadable Driver Reference File Configured for Mouse

Filenames with a .DRV extension are device driver files, which the AUTOLOADER program copies into the \PCAD\DRV directory.

The first line specifies the path and the name of the system driver file for your computer system type.

The second line specifies the path and name of the display driver file for your graphics monitor.

The third line specifies the path and name of the input device driver file for your input device type.

Figure C-3 shows a sample of a loadable driver reference file configured for use with a GTCO digitizer.

```
SYSTEM \PCAD\DRV\SIBMPC.DRV
DISPLAY \PCAD\DRV\DIBMEGA.DRV
INPUT \PCAD\DRV\IGTCO.DRV
```

Figure C-3. Sample of a Loadable Driver Reference File Configured for a GTCO Digitizer

Because the digitizer is not used for editing sessions, the loadable driver reference file must be changed back and forth when switching from digitizer to mouse mode. To do this, you must create a batch file called CHGSYS.BAT with any text editor. This batch file lets you toggle back and forth from digitizer mode to mouse mode by entering CHGSYS at the DOS prompt.

Before you create the batch file, create an alternate loadable driver reference file configured for your digitizer.

To create the alternate loadable driver reference file, follow the steps below.

1. In the root directory, copy the PCADDRV.SYS file that is configured for a mouse to an alternate file called PCADDRV.DIG.

2. Edit the PCADDRV.DIG file to look like the PCADDRV.SYS file that is configured for the digitizer as shown in Figure C-3. If you are using a digitizer other than the GTCO, refer to Appendix C, "Loadable Device Drivers," in your *System Overview* manual for a list of the digitizer input driver files provided by P-CAD.

To create the batch file, in the root directory create a file named CHGSYS.BAT and type the following, pressing [Return] after each line.

```
echo off
cd c:\
if exist PCADDRV.DIG goto dig
if exist PCADDRV.MOU goto mouse
echo ERROR: Alternate .DIG or .MOU file does not exist
goto end
:dig
ren PCADDRV.SYS  PCADDRV.MOU
ren PCADDRV.DIG  PCADDRV.SYS
echo System now configured for a DIGITIZER....
goto end
:mouse
ren PCADDRV.SYS  PCADDRV.DIG
ren PCADDRV.MOU  PCADDRV.SYS
echo System now configured for a MOUSE....
:end
```

Now, when you want to toggle back and forth from digitizer to input mode, type CHGSYS at the DOS prompt. The program will toggle the PCADDRV.SYS file back and forth from digitizer to input mode automatically. The system will display a message indicating the current mode.

After you type CHGSYS at the DOS prompt, you will need to change the **Digitizer port** option on the Configuration Screen from "port 2" to "port 1" for mouse input mode and from "port 1" to "port 2" for digitizer input mode since the digitizer is connected to port 2.

After you enter the last option, the program displays the following message:

Save this configuration? No

To save this configuration, type Y for YES and press [Return]. The system saves the modified configuration as a file (PCCAPS.CFG) and returns to the Opening Menu. Now, when PC-CAPS is executed, the configuration file is configured for the mouse.

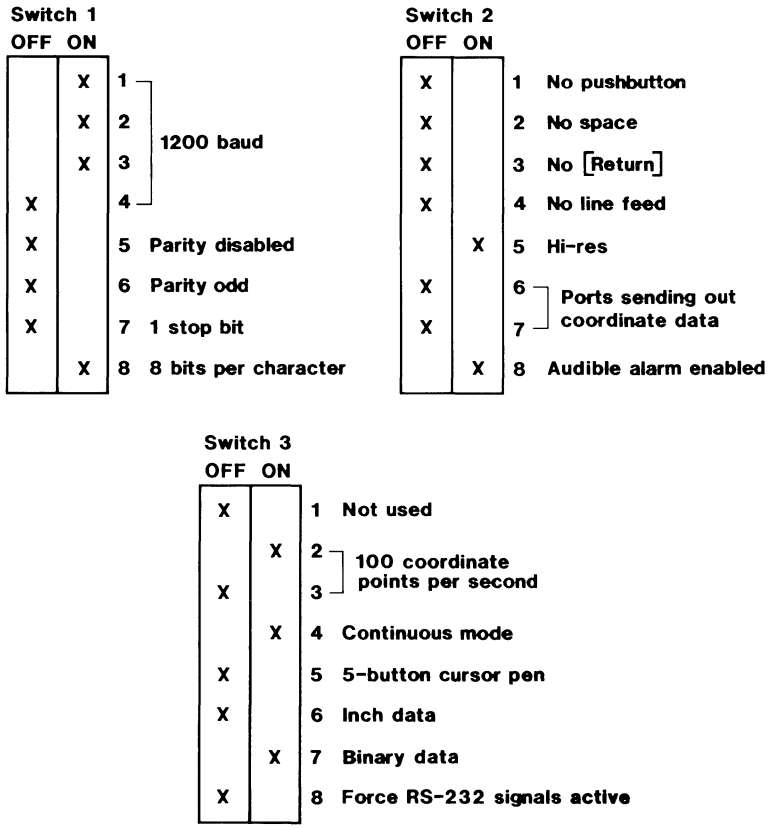
To use this configuration for the current session only, press [Return] for NO. This configuration will be used for the next session and will not be saved after you exit PC-CAPS.

SETTING UP THE DIGITIZER

When you start PC-CAPS, the input mode will be set for digitizing.

NOTE: The mouse will not operate when the system is configured for the digitizer and the digitizer will not operate when the system is configured for the mouse.

Before starting PC-CAPS, make sure the digitizer is located in an area where it can be used comfortably and is close to the computer terminal to which it is connected. Make sure the power supply is plugged in and the switches on the digitizer and the cable configuration are set to the manufacturer's specifications. P-CAD supports the GTCO Digi-pad 5 (Model No. 71D4). Figure C-4 shows the switch settings required on the GTCO digitizer to interface to PC-CAPS. Table C-1 shows the cable configuration.



86088

Figure C-4. GTCO Digi-pad 5 (Model No. 71D4) Switch Settings. The X's represent the switch settings.

Table C-1. GTCO Digi-pad 5 Cable Configuration

Computer			Digitizer	
Signal	**Pin**		**Pin**	**Signal**
AC Gnd	1	<---------->	1	AC Gnd
Trans	2	<---------->	2	Trans
Recv	3	<---------->	3	Recv
RTS	4	<---------->	4	RTS
CTS	5	<---------->	5	CTS
DSR	6	<---------->	6	DSR
DC Gnd	7	<---------->	7	DC Gnd
DTR	20	<---------->	20	DTR

Baud rate: 1200

P-CAD also supports the Houston Instruments True Grid 1011 and HIPAD DT-11 digitizers. Figure C-5 shows the switch settings for the True Grid 1011.

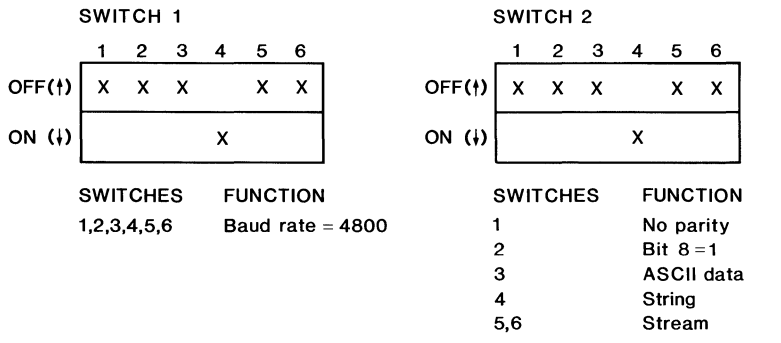


Figure C-5. Houston Instruments True Grid 1011 Switch Settings. The X's represent the switch settings.

The HIPAD digitizer does not utilize switch settings but requires the cable configuration shown in Table C-2.

Table C-2. Houston Instruments HIPAD Digitizer (Model No. DT-11) Cable Configuration

Computer			Digitizer	
Signal	**Pin**		**Pin**	**Signal**
Recv	3	<----------	22	RS-232
DC Gnd	7	<----------	20	DC Gnd
RTS	4	<┐		
CTS	5	─┘		
DTR	20	┌─		
DSR	6	<┤		
CD	8	<┘		

Also, make sure the digitizer is correctly connected to the appropriate serial communications port on the computer. If your system is configured with two communications ports, we recommend that you connect the digitizer tablet to communications port 2 and connect the security device with the mouse attached to communications port 1.

Affix the drawing which is to be digitized to the digitizing tablet by taping the corners of the drawing down securely, positioning the horizontal and vertical drawing axis to the digitizer. You do not need to precisely align the drawing and tablet; PC-CAPS will automatically compensate for any misalignment between the drawing and the digitizer in both the horizontal and vertical axes.

Now you are ready to start PC-CAPS.

STARTING THE PROGRAM IN DIGITIZER INPUT MODE

When you enter PC-CAPS in the digitizer input mode, the PC-CAPS Configuration Screen will appear as shown in Figure C-6.

PC-CAPS Configuration

Database filename: None

Directory path:
Current

Security Device port..... port 1
Digitizer port..... port 1
Digitizer bell..... On
Video save to disk..... Off

Figure C-6. PC-CAPS Configuration Screen in the Digitizer Input Mode

If your personal computer is configured for two serial communication ports, set the **Security Device port**

option to "port 1" and the **Digitizer port** option to "port 2."

The digitizer has a bell that beeps each time you digitize a point. If you don't want the beep, the bell can be turned off by selecting "Off" for the **Digitizer bell** option on the Configuration Screen. The default is "On."

The rest of the options on this screen are described in Part A of Chapter 2, "Using PC-CAPS."

Setting the Scaling Factor

After the configuration screen options have been set, select the **Edit Database** option on the main menu. If the security device is correctly installed and the system is correctly configured for digitizing, the system prompts:

DIGITIZE Reference Point 1

Digitize a known reference point in the lower left-hand corner of the drawing. Then the system prompts:

Enter X Coordinate and
Enter Y Coordinate

Enter the value for each of these coordinates in the appropriate database units (for example, type 10 10 [Return]). If you press [Return] without entering a value for either of these two prompts, the system will assume the X,Y coordinates are 0,0 for the lower left-hand reference point. (0,0 is the recommended value for this point.)

NOTE: If the system does not accept a digitized point, make sure the digitizer is on-line and connected to the specified communications port. Also, make sure that the two lights on the digitizer puck are on, indicating that the system power is on. If the system still does not respond, check the digitizer switch settings and the system interconnect cable for the correct pin configuration.

Next, the system prompts:

DIGITIZE Reference Point 2

Digitize a known reference point in the upper right-hand corner of the drawing. Then the system prompts:

Enter X Coordinate and
Enter Y Coordinate

Enter the X and Y coordinates. The values entered here must be the real (1 to 1) distance between the first reference point and the second reference point, not the measured scale distance on the drawing. The system uses these values to calculate the scale factor. The scale factor is determined by the ratio of the measured distance between the reference points and the specified values.

For example, if a schematic has been designed at a scale of 2 to 1 (twice actual size), and the measured scale distance between the two reference points is 10 inches in the X direction and 8 inches in the Y direction, you would enter one half of the measured distance for reference point 2, or X = 500 and Y = 400. (This example assumes the default 0,0 coordinates were specified for reference point 1.) The system then

computes the scale based on the measured distance between the reference points and the specified coordinates. Each subsequently digitized point has a value of one half of the measured distance on the layout producing a finished database of the correct size.

The scale of the original drawing can be any scale desired including less than 1 to 1 scaling (1/2 size), as shown in Figure C-7.

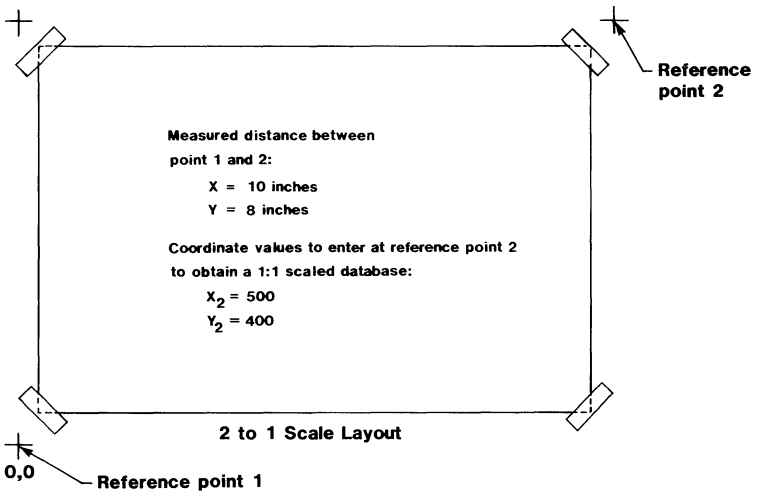


Figure C-7. Scaling Factor Example

In addition to the automatic scaling feature, the system also automatically adjusts the position of the drawing axis to the tablet axis. You can affix the drawing to the tablet in any horizontal or vertical orientation and the system automatically squares the drawing to the tablet axis.

Because of this feature, you don't need to precisely position the drawing axis to the tablet axis.

Setting the Viewing Window

After the alignment and scaling has been set, the system prompts you to set a viewing window for digitizing. For a maximum viewing screen, set the window to extend slightly outside the defined reference points.

> NOTE: Once the digitizing window has been set, we recommend that you not change it during the digitizing session. If the database is zoomed out beyond the limits of the digitizing surface, you may not be able to access the menus and status line commands due to the absolute mapping of the digitizer to the viewing screen.
>
> If you need to change the viewing window, use the VWIN command to return to the original window size. If you can't access the menu commands due to the viewing window size, you can use the [F10] key to enter the menu area and select a command. (Be sure to remove the digitizer puck from the tablet surface.) You can also enter the commands from the keyboard by typing a slash (/), then the command and [Return].

USING THE DIGITIZER TO ENTER A DRAWING

After you have set up the digitizer, you are ready to enter a drawing into the system. The PC-CAPS commands will work just as they do when using the

mouse, except for the control of the cursor on the screen. Each cursor position on the digitizing tablet will produce a unique position on the viewing screen and in the database.

As the puck is moved away from a position on the tablet and back, the screen cursor will return to exactly the same absolute position on the screen. In this manner the drawing on the digitizer can be precisely duplicated or traced into the system.

During the digitizing session, you should frequently check the progress of the job on the computer monitor to be sure that the data was entered correctly.

If you are using the digitizer bell, the system will beep each time a digitized point has been accepted. If a point is digitized and you do not hear the beep, check the status on the screen. The system may be waiting for a response to a "Merge Net" or similar request.

If a single job requires more than one digitizing session to complete, the digitizer reference points and X,Y coordinate values must be respecified each time the database is loaded. It is not important that the drawing be located in the exact same spot on the digitizer tablet, but it is important to use the same reference points and coordinate specifications that were used in the previous session(s). Once the database has been loaded, check the alignment of the drawing or layout to the database to be sure all points have been re-entered correctly before continuing.

> NOTE: If the same job is to be loaded into the database each time the program is started, the database name may be entered as the default database name on the PC-CAPS Configuration Screen.

APPENDIX D. COLOR SELECTION

This appendix is intended for expert users. It describes the default colors for PC-CAPS and how to change them. We do not recommend that you change the default colors; however, this appendix explains some of the things you need to know if you are an expert user and would like to change the default colors.

The color entries for the color map are values ranging from 0 to 4095. The color is determined by the percentage of the three primary colors, which is specified by setting the appropriate bits in the red, green, and blue fields as shown in Figure D-1.

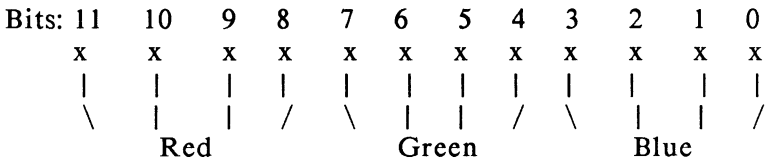


Figure D-1. Color Fields

The default color map entries are listed in the following table. The zero entry in the map index column is the background color, which is black. The number seven entry is for the cursor. The entries 8-15 match up to pen numbers 7-14 and color 15 is always the same as color 1.

Table D-1. Default Color Map

Map Index	Color	Color Value	
		Hex	Binary
0	Black	000	0000 0000 0000
1	Green	0F0	0000 1111 0000
2	Red	F00	1111 0000 0000
3	Yellow	FF0	1111 1111 0000
4	Blue	00F	0000 0000 1111
5	Cyan	0FF	0000 1111 1111
6	Magenta	F0F	1111 0000 1111
7	White	FFF	1111 1111 1111
8	Gray	444	0100 0100 0100
9	Light Green	4B4	0100 1011 0100
10	Light Red	B44	1011 0100 0100
11	Light Yellow	BB4	1011 1011 0100
12	Light Blue	44B	0100 0100 1011
13	Light Cyan	4BB	0100 1011 1011
14	Light Magenta	B4B	1011 0100 1011
15	Light Gray	BBB	1011 1011 1011

Not all of the colors are available on all graphics display devices. For example, if a device is only capable of 64 colors instead of 4096, the two most significant bits for each of the red, green, and blue fields will be used to determine the colors on that device.

CHANGING THE COLORS

To change the default colors, you must edit the PCADDRV.SYS file located in the root directory. This file is created when you run the INSTALL program on the SYSTEM AUTOLOADER diskette. You can use any text editor to edit this file.

Following is an example of the format of the line you would add to the PCADDRV.SYS file to change the color map:

```
COLORS 000 0F0 F00 FF0 00F 0FF F0F FFF 444
4B4 B44 BB4 44B 4BB B4B BBB
```

The colors are represented in hexadecimal notation with the first hex digit in a set representing the amount of red, the second hex digit representing the amount of green, and the third hex digit representing the amount of blue in the color.

Follow these guidelines when you edit the PCADDRV.SYS file:

1. Make sure you have 16 three-digit hex numbers listed after the word COLORS.

2. Make sure that at least one space separates the word COLORS from the first hex set and that there is at least one space between each hex set.

3. All 16 hex entries must be on the same line in the file.

APPENDIX E. THE ADD_DOTS PROGRAM

This appendix describes the P-CAD ADD_DOTS program and how to use it to add solder dots to a schematic.

OVERVIEW

ADD_DOTS is a utility program that automatically adds solder dots at T-junctions to a schematic that was created using an earlier than 1.3 version of PC-CAPS.

USING ADD_DOTS

To start ADD_DOTS, type:

ADD_DOTS [Return]

The Help Screen shown in Figure E-1 is displayed.

```
ADD_DOTS Database Utility.
Copyright (C) 1984, 1985 - Personal CAD Systems, Inc.

This utility adds solder dots to the schematic databases.
Usage Is: ADD_DOTS <inputfile>  [<outputfile>]
<inputfile>    :   Input Database file name.
<outputfile>   :   Output database file name.
                   If not specified the output overwrites the input.
```

Figure E-1. ADD_DOTS Help Screen

Then type:

ADD_DOTS <inputfile> <outputfile>

The outputfile is optional. If you do not specify the outputfile, the outputfile will overwrite the inputfile.

Also, if desired, you can bypass the Help Screen by typing:

ADD_DOTS <inputfile> <outputfile>

directly from the DOS prompt.

ERROR MESSAGES:

The following list describes error messages that may appear on the screen, the cause of each error, and the appropriate user action.

Message: Illegal number of arguments.

Cause: There are more than three command line arguments.

Action: Rerun the program as follows: ADD_DOTS <input filename>[<output filename>]

Message: Failed to get database type of <input filename>.

Cause: Database corrupted.

Action: Correct the database or call P-CAD.

Message: <input filename> does not exist.

Cause: Program could not find the input database file.

Action: Make sure the database exists or the filename is correct.

Message: <input filename> format is incompatible.

Cause: The input database is not a P-CAD schematic type database.

Action: Make sure the database is a schematic created with PC-CAPS.

Message: <input filename> is not a schematic database.

Cause: The input database is not a schematic database.

Action: Make sure the input database is a schematic.

Message: Not enough memory.

Cause: The program needs at least 640K of memory.

Action: Make sure 640K of memory is available.

Message: Not enough space, can not load the database.

Cause: Not enough space is available on the disk.

Action: Clean up the disk.

Message: Can not save <output filename>, DISK IS FULL.

Cause: Not enough space is available on the disk.

Action: Make sure enough disk space is available.

Message: Can not open debug file.

Cause: The file for debugging purposes may not be opened.

Action: Check ANSI.SYS for FILES=15 statement.

Message: Saving the <output filename> output database failed.

Cause: This is an internal error.

Action: Call P-CAD. Check for ADD_DOTS.DBG file.

Message: Program or Database error.

Cause: This is an internal error.

Action: Call P-CAD. Check for ADD_DOTS.DBG file.

NOTE: A debug file called ADD_DOTS.DBG may be created by the program. This file is used by the programmer for debugging.

INDEX

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Z