PC-CARDS™
User's Manual
Printed Circuit Board Layout

September 1986

p·cad™
PERSONAL CAD SYSTEMS INC.

PREFACE

This manual describes PC-CARDS, P-CAD's program for printed circuit board layout. 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-CARDS and then read the *System Overview* manual to find out how PC-CARDS interacts with other P-CAD programs.

Chapter 1, INTRODUCTION, provides an overview of PC-CARDS, a program for printed circuit board (PCB) layout. This chapter also describes P-CAD conventions. Read this chapter first to become familiar with PC-CARDS.

Chapter 2, USING PC-CARDS, shows you how to use PC-CARDS to create and edit a PCB layout. 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-CARDS 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-CARDS 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-CARDS and how to change them. This appendix is intended for expert users.

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)

x

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

CONTENTS (Continued)

FIGURES

FIGURES (Continued)

FIGURES (Continued)

TABLES

CHAPTER 1. INTRODUCTION

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

OVERVIEW

PC-CARDS is a printed circuit board design editor that runs on personal computers. PC-CARDS is an integral part of P-CAD's electronic design automation systems.

With PC-CARDS, you can start a PCB layout by designing on-line, by packaging a schematic, or by digitizing an existing layout. To reduce routing congestion, components can be placed automatically by PC-PLACE, P-CAD's automatic placement program, or interactively using the ratsnest or the histogram and force vector analysis tools provided by PC-PLACE. Then PCBs can be routed manually using PC-CARDS or automatically by PC-ROUTE.

Using P-CAD's integrated family of utility programs, the PC-CARDS database can be used to check electrical continuity and physical design rules, extract and compare netlists, create fabrication drawings and photoplots, and back-annotate to schematics.

The key features of PC-CARDS include:

● **Intelligent database.** PC-CARDS is built around P-CAD's intelligent integrated database (IIDB) that continually keeps track of components and connectivities, even through multiple layers. The IIDB ensures that electrical connectivity is maintained throughout the design cycle.

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

P-CAD CONVENTIONS

We strongly recommend that you follow our filename, layer structure, and component 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.

.**PCB** for printed circuit board database files

.**PRT** for PCB component database files

.PLT for plot files

.PS for padstack files

.SSF for special symbol files (padstack 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 a design, you can assign each type of data to a different layer. For example, on a PCB you enter component side traces on the COMP layer; solder side traces on the SOLDER layer; and internal traces on the INT1 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-CARDS default layer screen using the VLYR command and changing and/or adding layers as desired.

2. Loading an existing .PCB or .PRT 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 or edit a PCB layout, we strongly recommend that you use the P-CAD layer names; some P-CAD programs look for data by layer name. For example, PC-ROUTE looks for the border outline on the layer named BRDOUT.

Also, when you create additional components, use the P-CAD layer structure for your library so that the components you create are compatible with P-CAD library components.

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

Table 1-1. PC-CARDS Default Layer Structure

Layer Position	Name	Pen	Status	Use
1	PADCOM	7	ON	Graphic component pads
2	FLCOMP	7	OFF	Flash component pads
3	PADSLD	8	OFF	Graphic solder pads
4	FLSOLD	8	OFF	Flash solder pads
5	PADINT	9	OFF	Graphic internal pads
6	FLINT	9	OFF	Flash internal pads

Table 1-1 Continued

Layer Position	Name	Pen	Status	Use
7	GNDCON	10	OFF	Graphic internal ground connections
8	FLGCON	10	OFF	Flash internal ground connections
9	CLEAR	12	OFF	Graphic universal clearance
10	FLCLER	12	OFF	Flash universal clearance
11	PWRCON	13	OFF	Graphic internal power connections
12	FLPCON	13	OFF	Flash internal power connections
13	SLDMSK	14	OFF	Graphic solder mask relief
14	FLSMSK	14	OFF	Flash solder mask
15	DRILL	15	OFF	Graphic drill template
16	FLDRLL	15	OFF	Flash drill template
17	PIN	4	ON	Graphic pin connections

Table 1-1 Continued

Layer Position	Name	Pen	Status	Use
18	BRDOUT	4	ON	Board outline
19	FLTARG	4	OFF	Flash alignment targets
20	SLKSCR	6	ON	Silkscreen paint
21	DEVICE	5	ON	Device names
22	ATTR	6	OFF	Attributes
23	REFDES	6	OFF	Reference designators
24 traces	COMP	1	ABL A	Component side
25	SOLDER	2	ABL	Solder side traces
26	INT1	3	OFF	Internal traces

Scale and Sizing

PCB layouts are 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 PCB layouts. In PC-CARDS, 1 DBU is 1 mil. The default grid is 50 DBUs in both axes, and the default wire width is 12 DBUs. You can change the grid and wire width to any increment of 1 DBU.

Libraries

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

Figure 1-1 shows the basic specifications that P-CAD uses to design components.
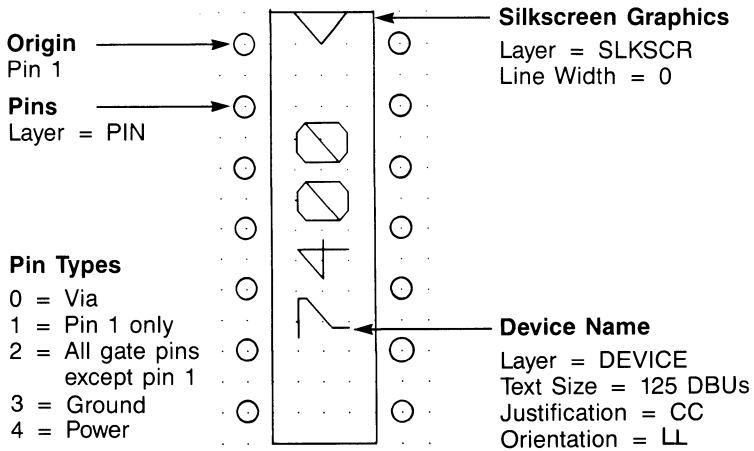
85124

Figure 1-1. P-CAD Component Specifications

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

P-CAD libraries also include padstack symbols, which
define the pad characteristics for pen plotting and
photoplotting. These symbols are created at standard
pad sizes: a 60-mil square pad for DIP pin 1 and a
60-mil round pad for all other DIP pads. All feedthrus
(vias) are 50-mil round. Clearance pads are created for
the solder mask and clearance layers. A drill symbol
for each different hole size is created for the drill layer.

You may create multiple padstack files for different manufacturing requirements. See Chapter 2, "Using PC-CARDS," for more information.

CHAPTER SUMMARY

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

CHAPTER 2. USING PC-CARDS

This chapter explains how to use PC-CARDS to lay out a PCB. It is divided into two parts.

Part A provides an overview to using PC-CARDS. 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 Layout
Example 2. Using Ratsnests
Example 3. Editing a Layout
Example 4. Creating a Padstack and a Special
            Symbol File
Example 5. Creating a P-CAD Compatible
            Component

PART A. OVERVIEW TO USING PC-CARDS

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

STARTING THE PROGRAM

Before starting PC-CARDS, 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-CARDS program file (PCCARDS.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
subdirectory you will be working in. At the DOS **C:\>**
prompt type:

   CD \PCAD\PROJ0 [Return]

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

PCCARDS [Return]

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

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

PC-CARDS

Options:

Configure PC-CARDS

>> Edit Database <<

Exit PC-CARDS

Press: [SPACE] for next option; [RETURN] to accept

Figure 2-1. PC-CARDS Opening Menu

This screen provides three options.

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

Edit database - Allows you to call the PC-CARDS program for creating and editing a layout.

Exit PC-CARDS - 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-CARDS** and the Configuration Screen is displayed as shown in Figure 2-2.

PC-CARDS Configuration

Database filename: None

Directory path:
   Current

Security Device port .... port 1
Mouse port .... port 1

Video save to disk .... Off

Enter the filename; Press: [RETURN] to accept

Figure 2-2. PC-CARDS Configuration Screen

The PC-CARDS Configuration Screen allows you to select the following parameters:

Database filename - The file that is automatically loaded when the PC-CARDS program is started. For example, you can load an existing PCB layout or you can load a layer structure different from the default layer structure here. 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 PCB database files. For example:

    \PCAD\PRT

    NOTE: You can enter more than one library directory. Use a semicolon to separate the libraries. For example:

    C:\PCAD\SYM;C:2PCAD\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.

To use PC-CARDS 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,** and **Video save to disk** 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-CARDS Configuration Screen. The system returns you to the PC-CARDS Opening Menu.

NOTE: If you save this configuration, the system saves it to a file called PCCARDS.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-CARDS environment.

THE DISPLAY

When you enter PC-CARDS, 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-CARDS
Opening Menu. See the *P-CAD Installation Guide* for
instructions on installing the security device.

Drawing Area   Main Menu

```
                                         SYMB [DETL]
                                         FILE  SYS
                                               SCMD
                                               ENTR
                                               DRAW
                                               NAME
                                               ATTR
                    Cursor                     EDIT
                        |        Submenu Area  MOVE
                        ↓                       ROT
                                               COPY
                                                DEL
                                               CLYR
                                               SWAP
                                         ZIN  ZOUT
                                         UWIN REDR
                                         PAN  LPAN
                                         STO  RCL
No Active Command...                     VLYR MASK
                        R   50:50   S G    0    0
     Message Line
                        Status Line
```
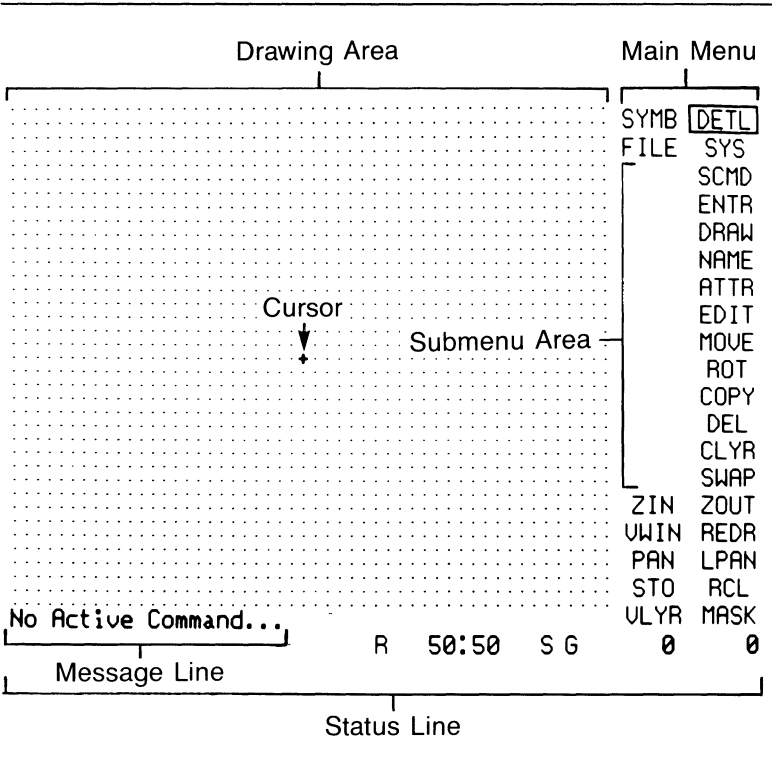
85125

Figure 2-3. Initial Screen Display

Commands

Commands are displayed in the two columns on the
right of the screen. Two sets of commands 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 component part to be added to the part library, and the DETL commands are used for editing a PCB layout.

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 component 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 again 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.

Button 3
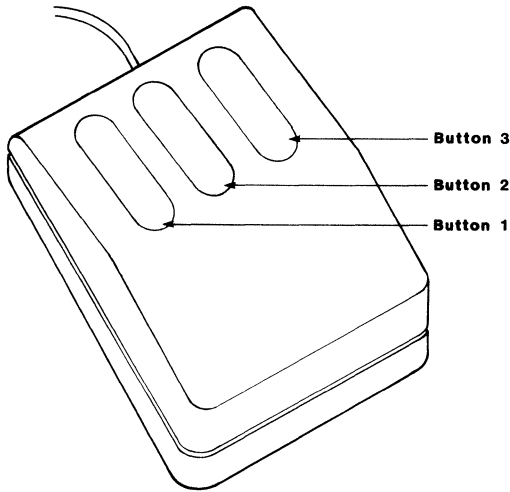
Button 2

Button 1

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
a new 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 moved the mouse.

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

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 (/). When you type 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-CARDS 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 the [F9] key 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 mil are user-definable and may represent any length, the P-CAD libraries are built assuming 1 DBU = 1 mil.

To give the drawing some real dimensions, if you use a P-CAD 14-pin DIP component in the drawing and plot your layout in 1:1 scale, the component will be 800-mils high and you can produce a layout measuring up to 60 by 60 inches.

The grid is a design aid to ensure that wires are straight and components 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 disappear automatically, even though the grid display is on. The point at which they disappear 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 50:50 DBUs or 50 mils in both axes.

The viewing commands PAN, LPAN (Long Pan), STO (Store), and RCL (Recall) commands 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 shown by the rectangle in Figure 2-5. This rectangle appears yellow on your display.


```
                                              SYMB ⌐DETL⌐
                                              FILE  SYS
                                                    SCMD
                                                    ENTR
                                                    DRAW
                                                    NAME
                                                    ATTR
                                                    EDIT
                                                    MOVE
                                                    ROT
                                                    COPY
                                                    DEL
                                                    CLYR
                                                    SWAP
                                               ZIN  ZOUT
                                              VWIN  REDR
                                               PAN ⌐LPAN⌐
                                               STO  RCL
  Long PAN: Select New Area...               VLYR  MASK
                        R   50:50    S G      6950 -5150
```
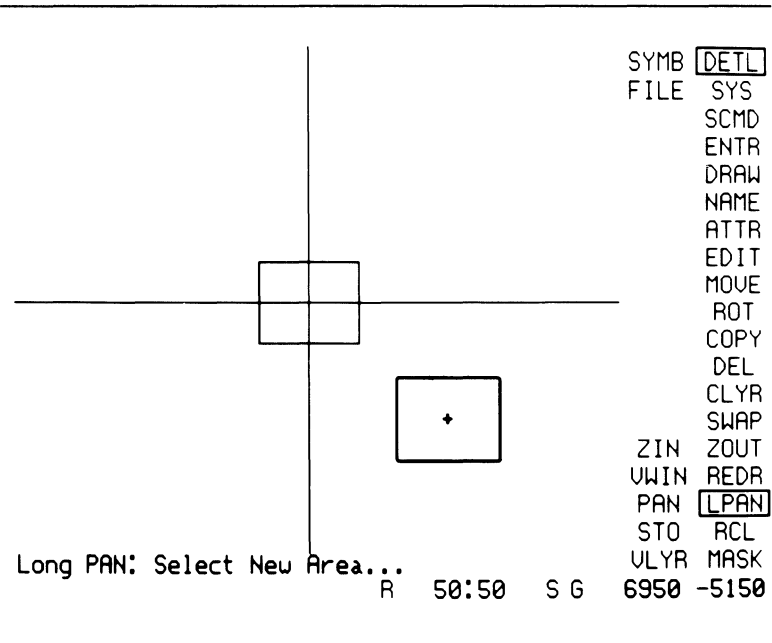
85126

Figure 2-5. LPAN Screen Display

Layer Screen

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

```
PADCOM  7  ON        BRDOUT  4  ON
FLCOMP  7  OFF       FLTARG  4  OFF
PADSLD  8  OFF       SLKSCR  6  ON
FLSOLD  8  OFF       DEVICE  5  ON
PADINT  9  OFF       ATTR    6  OFF
FLINT   9  OFF       REFDES  6  OFF
GNDCON 10  OFF       COMP    1  ABL A
FLGCON 10  OFF       SOLDER  2  ABL
CLEAR  12  OFF       INT1    3  OFF
FLCLER 12  OFF
PWRCON 13  OFF         ┌──────┐
FLPCON 13  OFF         │ QUIT │
SLDMSK 14  OFF         └──────┘
FLSMSK 14  OFF
DRILL  15  OFF
FLDRLL 15  OFF
PIN     4  ON
```

86101

Figure 2-6. PC-CARDS 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.

SAVING AND RETRIEVING DATABASE FILES

Drawings created with PC-CARDS may be saved as PCB database files and as plot files.

The FILE/SAVE command stores the entire database file. PC-CARDS always "remembers" the parameters when a drawing is saved. When you load an existing drawing to continue work on it, the parameters are 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 PCB layout, 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-CARDS environment.

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

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

The SYS/QUIT command exits PC-CARDS.

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 PCB layouts. 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 and Editing a Layout

PC-CARDS can accept a prepackaged database file generated from a schematic netlist by PC-PACK, or you can create a layout on-line.

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

Setting Up the Environment

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

Placing Components

If you are designing your own layout on-line, without a PC-PACK packaged schematic database as input, you will need to enter components into your layout. And, even if the layout is generated by PC-PACK, you may need to enter more components. Use the ENTR/COMP (Enter Component) command to call components from the library.

Components that have been placed may be duplicated with the COPY command, rotated with the ROT command, or repositioned for better placement with the MOVE command.

Using a Ratsnest

A ratsnest is a point-to-point display of lines between pins that represent electrical connections but do not specify the final PCB placement of the actual wires. The ratsnest display is an aid for component placement and for routing a circuit board.

Ratsnests can be added to a layout using the ENTR/RATN command to enter connectivity prior to final placement and routing. The ratsnests display can be viewed or suppressed. Toggle the R on the status line to green to turn on the ratsnest or to red to turn it off. When the ratsnest display is on, only the ratsnest of the remaining connections is displayed.

The ratsnest is also useful when moving and swapping components. The display lets you see the congestion of signal lines between devices.

Swapping Components

Pins, gates, and components can be swapped to improve
packaging and component placement for best routing.
The pins, gates, or components to be swapped are
selected, and the ratsnest display is automatically
turned on so you can see the changes in the circuit
connectivity.

Use the SWAP/COMP command to swap any component
with any other component.

Use the SWAP/GATE command to swap gates from one
device to another, or from one position in a device to
another position in the same device.

Use the SWAP/PIN command to swap pins of
components with logically equivalent pins of the same
component.

Wiring the Layout

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.) Place wires on
the COMP and SOLDER layers for two layer boards.
Use the INT1 layer and create other layers, for example
INT2, INT3, and so on, for internal wires on a
multilayer board.

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.

When you must change layers to avoid wires crossing,
you can enter the wire all at once, changing the active
layer as you go by moving the cursor to the layer
parameter on the status line and pressing Button 1, or
by selecting [F1], and then continuing with the wire.
PC-CARDS will automatically insert a via at the point
where the wire changes layers.

Components and nets may be identified with names
using the NAME/COMP (Name Component) and
NAME/NET commands. You can use [F3] to assign the
active net name while using the ENTR/WIRE command.
These names are used by other P-CAD programs such as
PC-NODES and PC-FORM to generate a netlist of the
components used in the PCB layout and the nets that are
connected to those parts. If you will be using
PC-ROUTE to route your board, all the nets must be
named.

Editing a Layout

You can edit wires in a PCB layout using the EDIT and
DELETE commands listed below. For a step-by-step
example on editing a PCB layout, see Example 3 in Part
B of this chapter.

EDIT/ADDV adds a vertex.
EDIT/AVIA adds a via.
EDIT/DELS deletes a segment.
EDIT/DELV deletes a vertex.
EDIT/DVIA deletes a via.
EDIT/LAYS changes the layer a segment is on.
EDIT/MOVA moves a wire segment or a vertex and
    all attached wire segments and vias on all layers.
EDIT/MOVS moves a segment.
EDIT/MOVV moves a vertex.
EDIT/MVIA moves a via.
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 may be deleted even if the layer that the
data is on is turned off. For more information, see
the command description of each DELETE
command in Chapter 3, "Commands."

NOTE: Vias are automatically added or deleted as
needed wherever an operation is done on a trace
using the ENTR/WIRE, EDIT/DELS, EDIT/DELV,
and EDIT/LAYS 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 or a ratsnest. When
editing wires, you may delete a wire with the EDIT
command. However, the ratsnest connection indicating
electrical connectivity in the database remains. To
remove the ratsnest connection from a pin, use the
ENTR/UCOM command.

Assigning Padstacks

After you create a PCB layout, you can link a special
symbol file that assigns padstack graphics to pin
connections in your layout for pen plotting and
photoplotting. Padstacks are described later in this
chapter in the section titled "Creating a Padstack."

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 padstack files (<filename>.PS) to
your layout as defined in the special symbol file at the
specified pin type locations. The SCMD/GSSF
command changes the display of pins to that of pads.
The pads are different shapes and sizes to distinguish
between different layers and pin types for pen plotting
and photoplotting.

To unlink the special symbol file, use the SCMD/GSSF
command and enter a [Return] instead of a filename.

Creating a Plot File

After you have created the layout and attached the
padstacks, you can create a plot file that is used by
PC-PRINT, PC-PLOTS, and PC-PHOTO to output plots
on a dot-matrix printer, a pen plotter, and a
photoplotter, respectively. Use the SYS/PLOT command
to create the plot file.

Saving the Layout

Use the FILE/SAVE command to save the PCB layout.

SYMBOL MODE

Symbol mode is used for creating components and
padstacks, or any other symbol you may use repeatedly
in your drawings, such as a logo or board outline. 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 Component

The following section provides a general description of
how to create a P-CAD compatible component. For
step-by-step instructions, see Example 6 in Part B of this
chapter.

When you create new components, P-CAD recommends
that you use its layer structure and relative sizes for
compatibility with P-CAD library components. See
Chapter 1, "Introduction," for P-CAD component
specifications under the section titled "Libraries."

Setting Up the Environment

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

Entering the Footprint

The arrangement of pins on a part is called its "footprint." Use the ENTR/PIN (Enter Pin) command to enter the pin connections.

For each pin you enter, you must assign a pin layer, pin name, pin type, and pin equivalence.

Place pins on the PIN layer.

The pin name is a number that identifies the pin.

The pin type determines what set of padstack graphics will represent the pin in the layout.

The pin equivalence number indicates logical equivalence. Pins with the same equivalence number (except 0) may be swapped. 0 means not swappable.

Set the layer (PIN), the type (TYPE:), and the logical equivalency (EQUIV:) parameters on the status line before entering the pin and the pin name.

Creating the Silkscreen Graphics

The DRAW commands are used to create the silkscreen
graphics. By convention, the SLKSCR layer is reserved
for creating the outline of the symbol graphics. Use the
DRAW/LINE and DRAW/RECT commands to draw
straight lines and boxes. Use the DRAW/ARC
command to draw arcs.

Entering the Device Name

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

Assigning the Origin

Use the ENTR/ORG (Enter Origin) command to assign
the origin (the reference point for placement). P-CAD's
standard is pin 1.

Adding Packaging Information

The SCMD (System) commands are used to add
packaging information. If you are using PC-PACK to
package schematic symbols into PCB components,
components must be assigned a Component Type ID.
PC-PACK uses the Component Type ID to subgroup
components, DIPS, discrete, and connector classes. Use
the SCMD/SCAT (System Command/Set Component
Attribute) command to assign the Component Type ID.
See Table 2-10 in "Example 5. Creating a P-CAD
Compatible Component," for a list of the P-CAD
Component Type ID numbers.

Use the SCMD/SPKG (System Command/Set Packaging) command to add logical packaging information into the PCB parts for use by PC-PACK. This command prompts for the number of gates, the number of pins for each gate, and the pin names for the package. This command installs the logic packaging and pin assignment information into the physical PCB device, such as a DIP.

The SCMD/SPAT (System Command/Set Pin Attributes) command can be used to check or change the pin types and equivalences entered into the footprint.

Saving the Component

Use the FILE/SAVE command to save the component.

Creating a Padstack

The following section provides a general description of what a padstack is and how to create one. For a step-by-step example on creating a padstack, see Example 4 in Part B of this chapter.

Padstacks are database files that define the pad characteristics for pen plotting and photoplotting flashes.

Each different item of graphics or flash must be located on a separate layer. In other words, the graphics that represent the component side pad should be on a different layer than the flash definition for the same pad. The graphics should be placed on different layers directly on top of each other with a common center and origin.

The P-CAD standard layer structure for PC-CARDS sets aside the first 17 layers for padstack definitions.

Setting Up the Environment

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

Because a padstack is smaller than a component or layout, it must be drawn on a smaller scale than that of parts or layouts. Adjust the current viewing scale so that the dot spacing is more convenient for this smaller-scale data creation. Use the ZIN command to zoom in, then select the X:Y cursor coordinates parameter on the status line, located in the right-most corner of the screen, to change the location to 0, 0.

On each layer that is to be used for the padstack (not all layers need to be used), turn the visibility status to ABL using the VLYR command.

Drawing the Graphics

Use the DRAW commands, such as DRAW/CIRC and DRAW/RECT, to make the graphic representations on each layer not reserved for a flash.

Drawing the Flashes

On the layers reserved for flashes, use the DRAW/FLSH command to draw the flashes. Each flash placed must be assigned the appropriate flash number (the appropriate aperture position number of the aperture wheel). Select the APER:<n> parameter on the status line to set this value before drawing the flash.

Assigning the Origin

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

Saving the Padstack File

Use the FILE/SAVE command to save the padstack. Use a filename with the extension .PS.

After the padstacks are created, the special symbol file that contains the padstack definitions can be created or edited. See Example 4 in Part B of this chapter for instructions on how to create the special symbol file.

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 on the screen may vary from the size of the text in your actual drawing, depending on the text dimensions specified for PC-PRINT, PC-PLOTS, and PC-PHOTO. You can change the text size by selecting the SIZ (Text Size) parameter on the status line. Or, you can use [F5] to enter a new text size when using the DRAW/TEXT command.

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 justification, 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 justification, 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). (This command is used mostly to put right reading text on the back of PCBs.)

CAUTION: Be sure to turn off the M parameter after you have entered the text you want mirrored.

PART B. EXAMPLES

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

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

To confirm the copies, at the DOS C:\> prompt, type:

    DIR \PCAD\PROJ0

Your directory listing should include the following files:

7400T.PRT	C60R32G.PS
7404T.PRT	N60S32G.PS
TUTORT.PCB	C60S32G.PS
TUTORT.SSF	N60R32P.PS
V50R28C.PS	C60R32P.PS
C60R32C.PS	C60S32P.PS
N60R32C.PS	N60S32P.PS
N60S32C.PS	C60S32C.PS
N60R32G.PS	

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 LAYOUT

Layouts are constructed from components. In this
example, you will lay out a simple circuit using library
parts. The steps in creating a layout are:

1. Set up the environment.
2. Place the components.
3. Wire the layout.
4. Get the special symbol file (which assigns
    padstack graphics to pin connections).
5. Create a plot file.
6. Save the layout.

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

86102
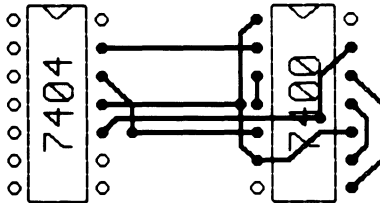
Figure 2-7. Circuit Layout

Step 1. Setting Up the Environment

Before you do any work, you must set up the working
environment to ensure a consistent starting point.

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.   Select VLYR to display the layer screen. Make sure
     the SLKSCRN layer is turned ON. Make sure that
     the COMP and SOLDER layers are set to ABL
     status and make SOLDER the active layer. You
     will be using these layers in "Step 3. Wiring the
     Layout" when you enter the wires.

3.   Select QUIT to exit the layer display.

Step 2. Placing the Components

1.   Examine the parts available.

     a. Select SYS/DOS. The system prompts:

         Enter DOS command (BUTTON 2 to quit)>

     b. Type DIR *.PRT and press [Return]. The system
         responds by displaying all the files in the PROJ0
         subdirectory with the .PRT extension. Notice in
         your listing the files:

            7404T.PRT
            7400T.PRT

You will use these parts 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

3. Type 7400T and press [Return].

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

   The system prompts:

   Select loc to place comp. (Orientation OK?)

4. Move the cursor around in the drawing area as
   shown in Figure 2-8. 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 box in Figure 2-8, you may
   need to zoom out with the ZOUT command or zoom
   in with the ZIN command before placing the
   component.

```
............................................  SYMB DETL
............................................  FILE SYS
............................................       SCMD
............................................  COMP ENTR
............................................  WIRE DRAW
         Cursor ──►  ┌─                        NAME
............................. │ .............  RATN ATTR
............................. │ .............  UCOM EDIT
............................. │ .............       MOVE
............................. │ .............        ROT
............................. │ .............       COPY
............................. │ .............        DEL
............................. │ .............       CLYR
............................. └─ ............        SWAP
............................................  ZIN ZOUT
............................................  UWIN REDR
............................................  PAN LPAN
............................................  STO  RCL
Select loc to place comp. (Orientation OK?)   ULYR MASK
                Fᴸ ⅃ ⊓ M R   50:50   S G   1100   850
```
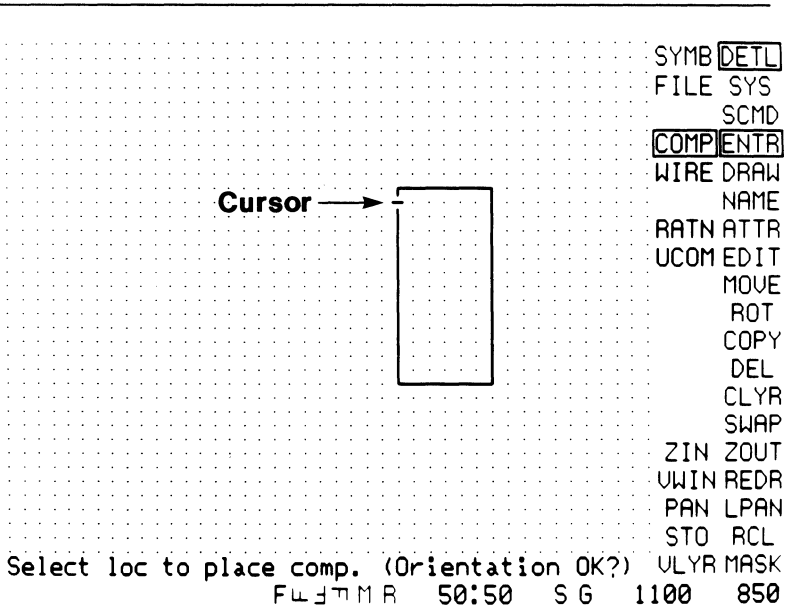
86103

Figure 2-8. Placing the First Component

Note the part of the prompt:

(Orientation OK?)

You can change the orientation of the component as it is placed by toggling through the four F's at the bottom of the screen.

5. Place the first component by moving the dashed box to the location shown in Figure 2-9 and pressing Button 1. The component will appear.

```
                                              SYMB DETL
                                              FILE  SYS
                                                    SCMD
                                              COMP ENTR
                                              WIRE DRAW
                                                    NAME
                                              RATN ATTR
                                              UCOM EDIT
                                                    MOVE
                                                     ROT
                                                    COPY
                                                     DEL
                                                    CLYR
                                                    SWAP
                                              ZIN  ZOUT
                                              VWIN REDR
                                              PAN  LPAN
                                              STO   RCL
Select loc to place comp. (Orientation OK?)   VLYR MASK
              FWJ⊓MR    50:50    S G    1350  1000
```
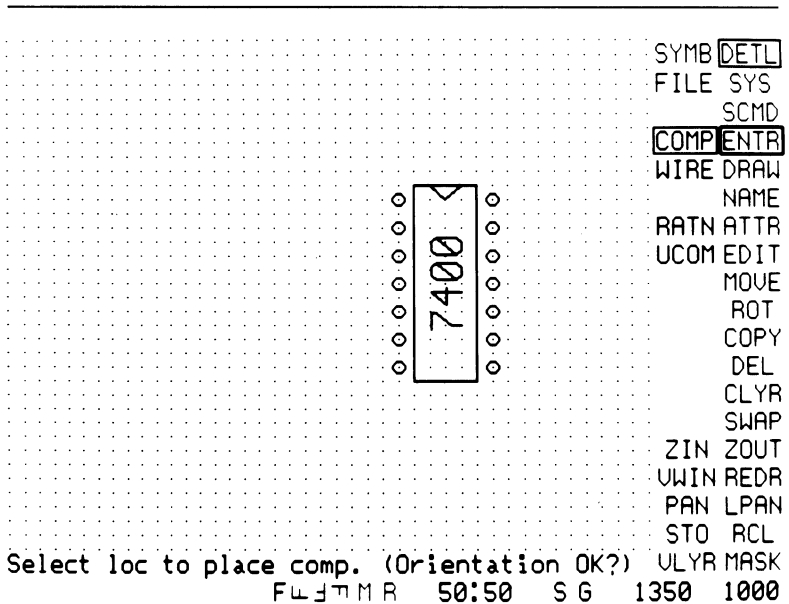
86104

Figure 2-9. First Component Placed

6.   Press Button 2 to enter a new part. The system
     prompts:

     :
     Comp-file-name

7.   Type 7404T and press [Return] to enter the new
     part name. The system prompts:

     Select loc to place comp. (Orientation OK?)

8.   Place the second component as shown in Figure
     2-10.

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

```
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   SYMB DETL
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   FILE SYS
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        SCMD
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   COMP ENTR
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   WIRE DRAW
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        NAME
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   RATN ATTR
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   UCOM EDIT
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        MOVE
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        ROT
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        COPY
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        DEL
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        CLYR
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .        SWAP
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   ZIN  ZOUT
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   VWIN REDR
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   PAN  LPAN
.  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .  .   STO  RCL
.                                                        VLYR MASK
Comp-file-name
```
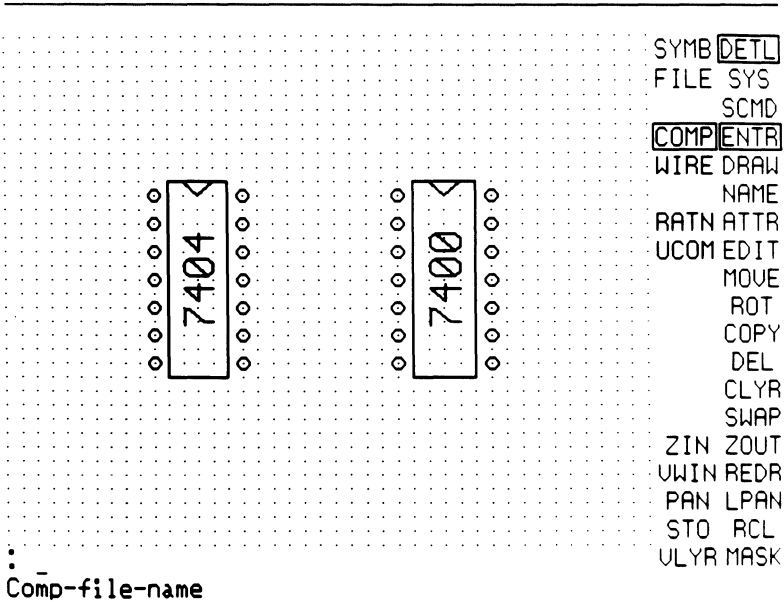
86105

Figure 2-10. Both Parts in Place

Step 3. Wiring the Layout

When the parts are in place, the next step is to enter the
wires between them to create the electrical connections
of the layout. You will enter the wires on the COMP
and SOLDER layers.

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

    Select start point....

2. Make sure the active layer is set to SOLDER on the
    status line. If it isn't, select the active layer
    parameter on the status line until SOLDER is
    displayed.

3. Make sure the wire width is set to 12. If it isn't, do
    the following:

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

       Enter line width:

    b. Type 12 and press [Return].

4. Make sure the line mode is set to ANGL on the
    status line.

    Figure 2-11 shows what your screen display should
    look like with SOLDER set to the active layer, line
    width set to 12, and line mode set to ANGL.

SYMB DETL
FILE SYS
     SCMD
COMP ENTR
WIRE DRAW
     NAME
RATN ATTR
UCOM EDIT
     MOVE
     ROT
     COPY
     DEL
     CLYR
     SWAP
ZIN ZOUT
UWIN REDR
PAN LPAN
STO RCL
ULYR MASK

Select start point....
SOLDER   ANGL W:12   R   50:50   S G

86106
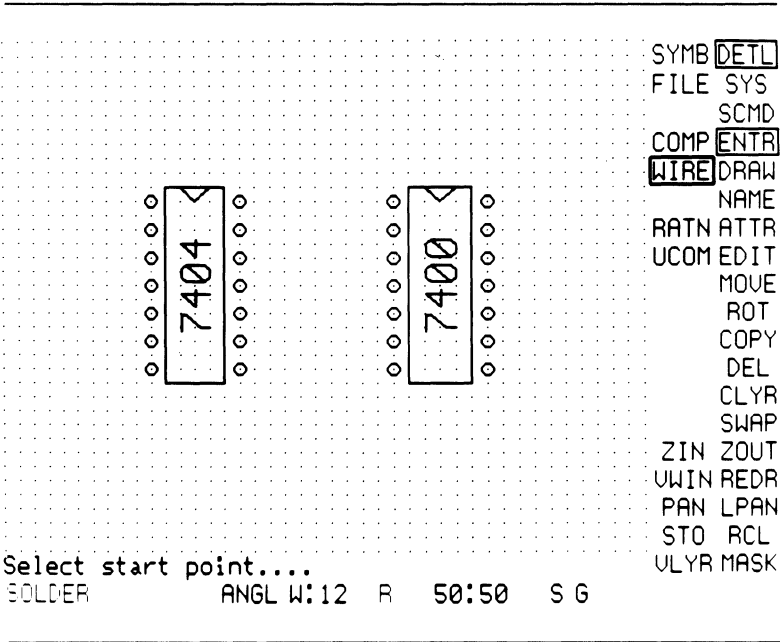
**Figure 2-11. Status Line Parameters Set
for Entering Wires**

5. Place the cursor over the point labeled A in Figure
    2-12 (Pin 1 of the 7400) and press Button 1. This
    marks the starting point of the wire. Then move
    the cursor to Point B and press Button 1 again.

    Following the same wire, continue this procedure
    with the remaining points (labeled C through G) in
    Figure 2-12.

NOTE: You can name the nets as you enter them if desired. For example, select the current net name parameter on the status line which is currently "un-named," or press [F3]. The system prompts:

Net name: un-named

Type N1 and press [Return]. The active net name is displayed on the status line, but not on the drawing. To assign and display the name of the net, use the NAME/NET command instead of [F3].

After you have selected the last point, press Button 2 to end the wire.
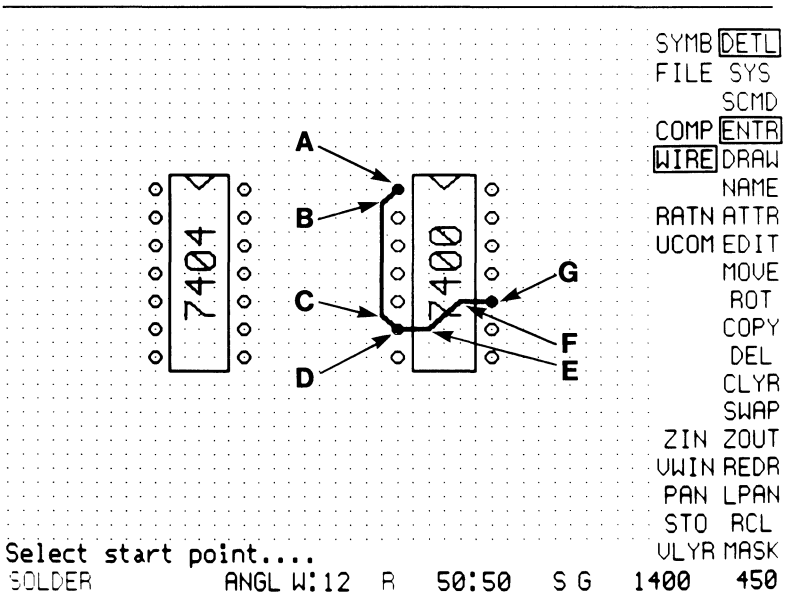
86107

Figure 2-12. First Wire in Place

6. The second wire to be entered contains a via. Add this wire by selecting the points indicated by letters A, B, and C in Figure 2-13, beginning with point A (Pin 12 of the 7404).

7. After you select point C, change the active layer to COMP by placing the cursor over the word SOLDER on the status line and pressing Button 1.

Select next point....
SOLDER un-named ANGL W: 12   R   50:50   S G   700   450

SYMB DETL
FILE SYS
      SCMD
COMP ENTR
WIRE DRAW
      NAME
RATN ATTR
UCOM EDIT
      MOVE
      ROT
      COPY
      DEL
      CLYR
      SWAP
ZIN ZOUT
VWIN REDR
PAN LPAN
STO RCL
VLYR MASK
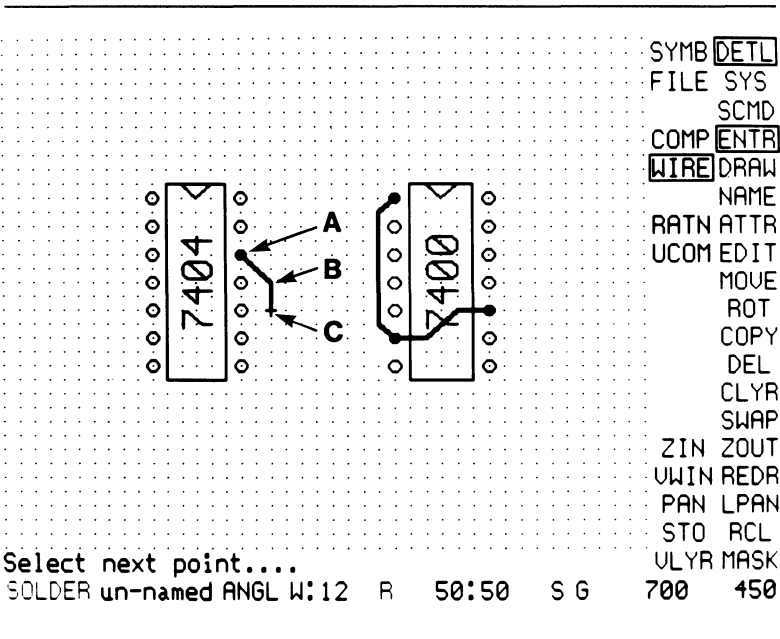
86108

Figure 2-13. First Part of Second Wire

NOTE: When you change layers to avoid hitting another wire, you enter the wire all at once, changing the active layer as you go, instead of entering separate wires on different layers. PC-CARDS automatically inserts a via at the point where the wire changes layers.

8. Complete the wire as shown in Figure 2-14 and press Button 2. Notice that the continuation of the wire is green, indicating that it is on the COMP layer. For the purpose of this example, we will enter horizontal wires on the COMP (green) layer and vertical wires on the SOLDER (red) layer.
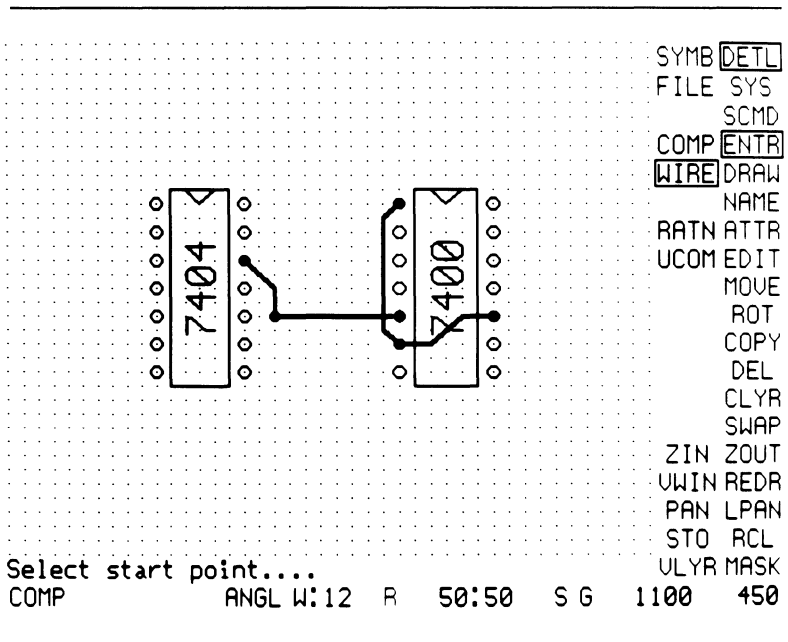
86109

Figure 2-14. Completion of Second Wire

9. Change the active layer back to SOLDER and begin a wire on the existing wire at the point indicated by the arrow in Figure 2-15. Notice that the existing wire is highlighted. This indicates that the wire you are starting is considered part of the same electrical connection, or net.
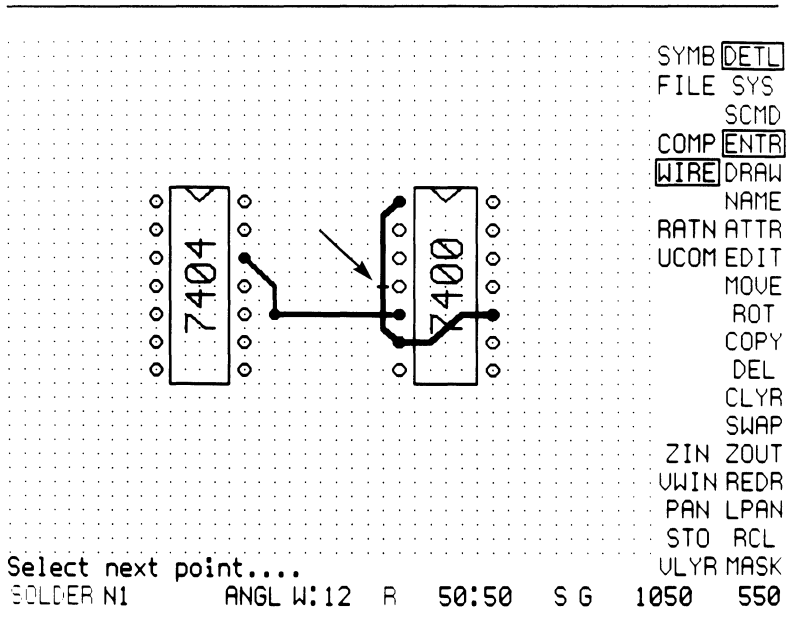
86110

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

10. Change the active layer to COMP. Complete the
     wire as shown in Figure 2-16. Again, PC-CARDS
     will insert a via at the point where you changed
     layers.
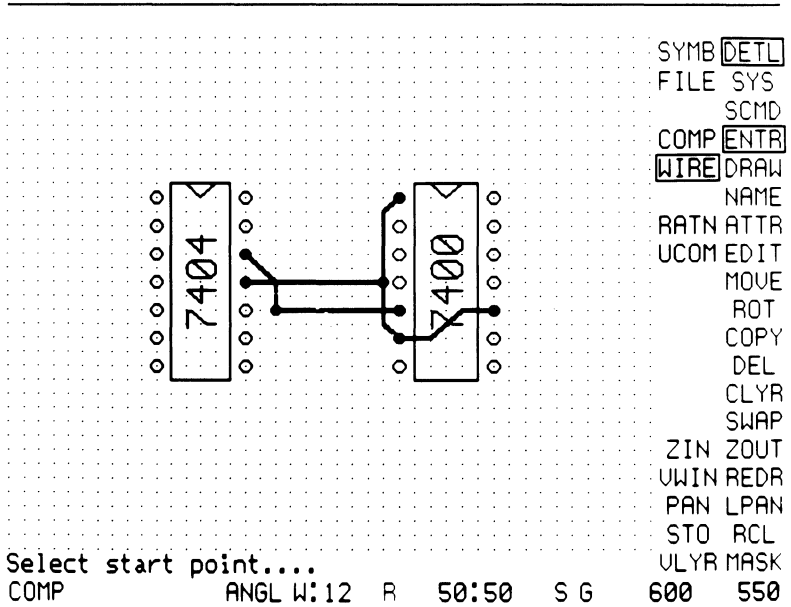
86111

Figure 2-16. Completion of Third Wire

11. Complete the wiring of the circuit as shown in
     Figure 2-17.   Remember, for the purposes of this
     example, enter horizontal wires on the COMP
     (green) layer and vertical wires on the SOLDER
     (red) layer. Select REDR to redraw the drawing.

NOTE: If you make a mistake while entering a wire:

a.   Select EDIT/DELS. The system prompts:

Select a segment

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

c.   Select REDR to redraw the drawing.

d.   Select ENTR/WIRE and enter the wire correctly.

You can also delete a portion of a line just drawn by tracing over it.

```
                                                    SYMB DETL
                                                    FILE SYS
                                                         SCMD
                                                    COMP ENTR
                                                    WIRE DRAW
                                                         NAME
                                                    RATN ATTR
                                                    UCOM EDIT
                                                         MOVE
                                                         ROT
                                                         COPY
                                                         DEL
                                                         CLYR
                                                         SWAP
                                                    ZIN  ZOUT
                                                    UWIN REDR
                                                    PAN  LPAN
                                                    STO  RCL
Select start point....                              ULYR MASK
SOLDER          ANGL W:12  R   50:50  S G
```
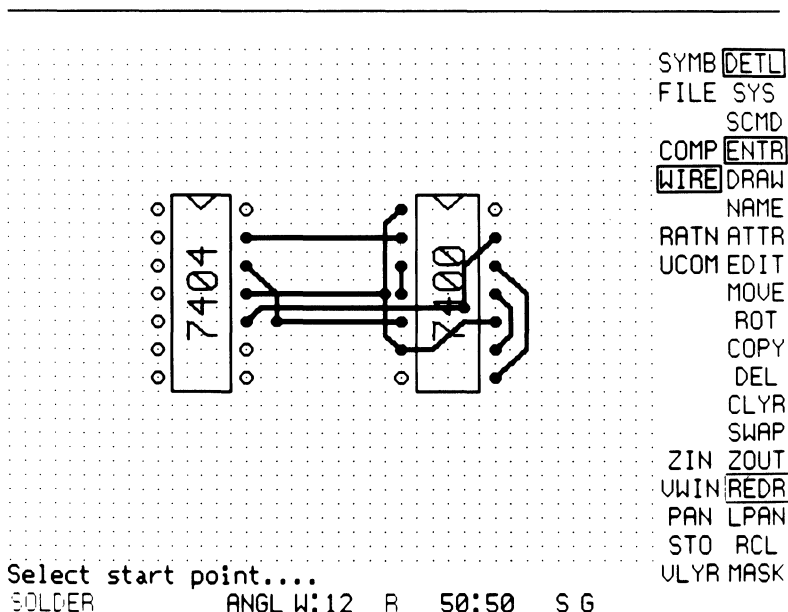
86112

Figure 2-17. Completed Wiring of Circuit

Step 4. Assigning the Padstacks

In this step, you will link a special symbol file of
padstack symbol definitions to your circuit layout.

The special symbol file that you will use in this step
contains a list of the pin types used in a layout and
assigns padstack graphics to each type. In Example 4
you can create a padstack and a special symbol file.

1. Select SCMD/GSSF to link the special symbol file to
    the layout. The system prompts:

    Enter file name:

2. Type TUTORT.SSF and press [Return].

3. Select REDR to redraw the circuit to show the special symbols. Figure 2-18 shows the result.
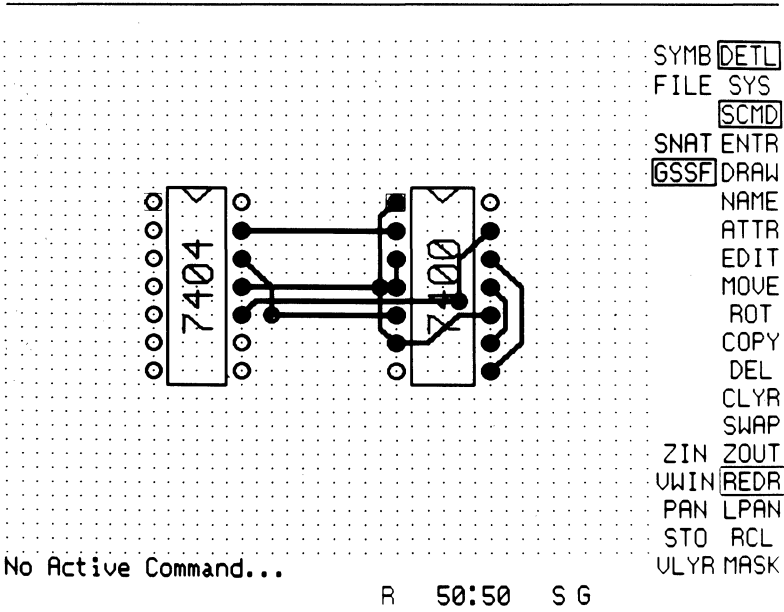
86113

Figure 2-18. Layout Showing Special Symbols

Step 5. Creating a Plot File

You have now completed a simple layout. Now you will create a plot file. PCB plot files are used by the P-CAD utilities PC-PRINT, PC-PLOTS, and PC-PHOTO to output plots on a dot-matrix printer, a pen plotter, and a photoplotter, respectively.

If you are using PC-PLOTS to plot your layout, the pen number assignments displayed on the 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, see the VLYR command description in Chapter 3, "Commands." To create a plot file:

1. Select SYS/PLOT. The system prompts:

    PLOT: Select Page Corner 1....

    NOTE: When you select the page corners for the plot window, make sure all the data you want in the plot file is displayed on the screen and inside the plot window. (The plot window cannot be larger than the screen display.) Use the ZOUT (Zoom Out) command, if needed, to position the data on the screen.

2. Position the cursor as shown in Figure 2-19 and press Button 1. The system prompts:

    PLOT: Select Page Corner 2....

SYMB DETL
FILE SYS
SCMD
ENTR
STAT DRAW
DOS NAME
PLOT ATTR
EDIT
QUIT MOVE
ROT
COPY
DEL
CLYR
SWAP
ZIN ZOUT
VWIN REDR
PAN LPAN
STO RCL
PLOT: Select Page Corner 1...   VLYR MASK
                    R   50:50   S G   200 1050
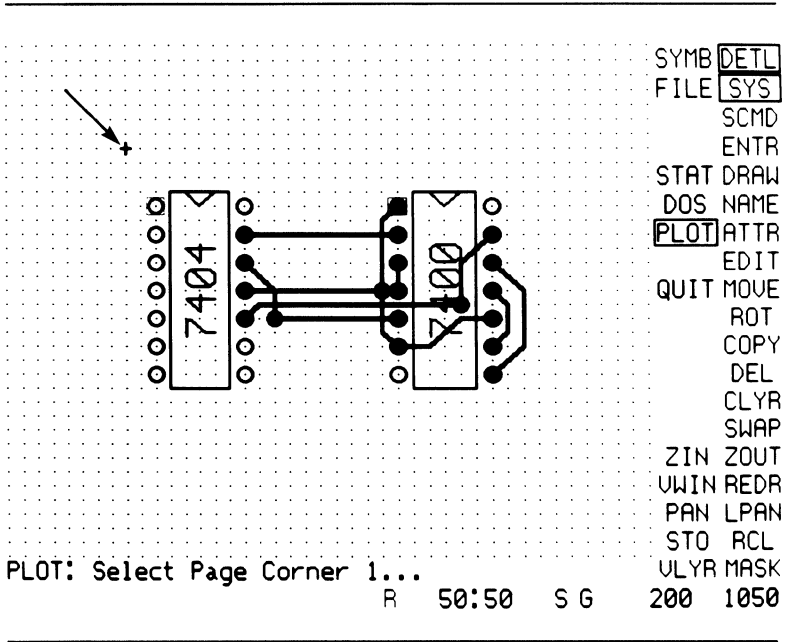
86114

Figure 2-19. Cursor Location for Page Corner 1

3.   Position the cursor as shown in Figure 2-20 and
     press Button 1.

```
                                            SYMB DETL
                                            FILE SYS
                                                 SCMD
                                                 ENTR
                                            STAT DRAW
                                            DOS  NAME
                                            PLOT ATTR
                                                 EDIT
                                            QUIT MOVE
                                                 ROT
                                                 COPY
                                                 DEL
                                                 CLYR
                                                 SWAP
                                            ZIN  ZOUT
                                            VWIN REDR
                                            PAN  LPAN
                                            STO  RCL
Plot file name: _                           VLYR MASK
                     R   50:50   S G   1550     50
```
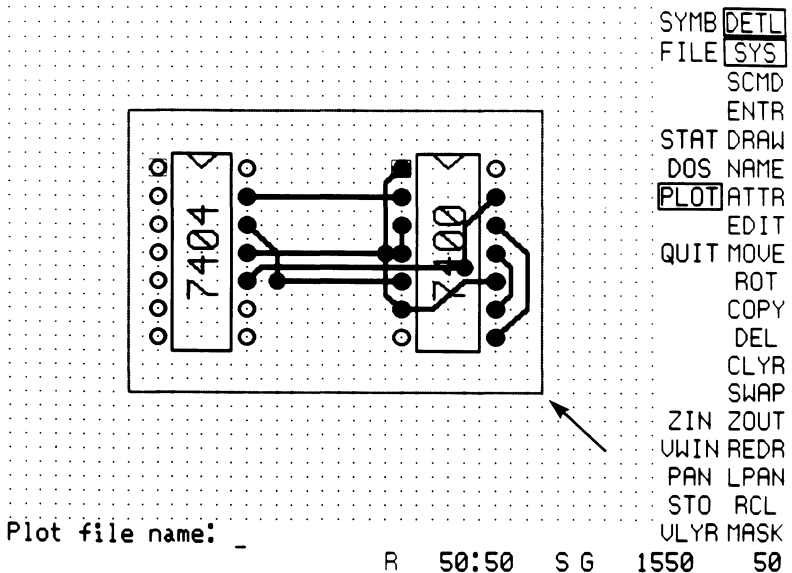
86115

Figure 2-20. Cursor Location for Page Corner 2

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

Plot file name:

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

NOTE: The .PLT extension is a P-CAD naming convention that is automatically added to the filename.

The system displays the following message:

Writing to plot file "Tutor.PLT."

and then

Plot file complete.

Step 6. Saving the Layout

Before you save the layout, unlink the special symbol file. A layout saved with the special symbols takes up more disk space than a layout without the special symbols.

1. Select SCMD/GSSF. This time instead of entering a file name, just press Button 2 or [Esc]. Select REDR to redraw the circuit. It should look like Figure 2-21.

```
                                               SYMB DETL
                                               FILE SYS
                                                    SCMD
                                               SNAT ENTR
                                               GSSF DRAW
                                                    NAME
                                                    ATTR
                                                    EDIT
                                                    MOVE
                                                    ROT
                                                    COPY
                                                    DEL
                                                    CLYR
                                                    SWAP
                                               ZIN  ZOUT
                                               VWIN REDR
                                               PAN  LPAN
                                               STO  RCL
No Active Command...                           VLYR MASK
                        R   50:50   S G   1750      0
```

86116
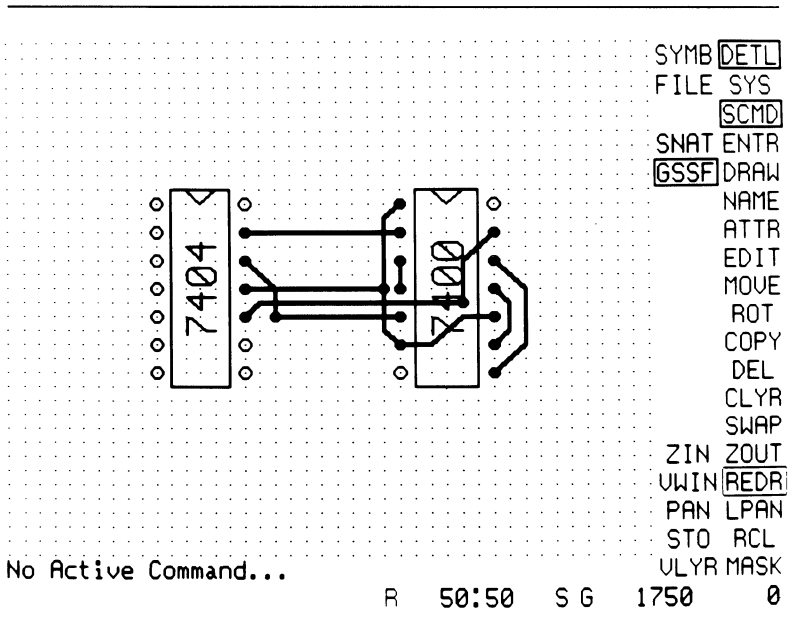
**Figure 2-21. Layout Showing Special
Symbol File Unlinked**

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

    Enter file name:

3. Type TUTOR and press [Return]. The .PCB
    extension is a P-CAD naming convention that is
    automatically added to the filename.

    The "official" version of the layout is in the file,
    TUTORT.PCB.

4. Select FILE/ZAP to erase the screen.

Summary

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

Table 2-1. Creating a Layout

Step	Commands Used	Layer/File
Setting up the environment	DETL	
Placing the components	SYS/DOS ENTR/COMP	7400T.PRT 7404T.PRT
Wiring the circuit	ENTR/WIRE	COMP SOLDER
Getting the special symbol file	SCMD/GSSF REDR	TUTORT.SSF
Creating a plot file	SYS/PLOT	TUTOR.PLT
Unlinking the special symbol file	SCMD/GSSF [Button 2]	
Saving the file	FILE/SAVE	TUTOR.PCB

EXAMPLE 2. USING RATSNESTS

A ratsnest shows the shortest electrical path connections between pins in a PCB layout.

Section 1 shows you how to use ratsnests to wire a circuit.

Section 2 shows you how to swap pins, gates, and components in a ratsnest.

In this example, you will do the following:

1. Use ratsnests to add to the circuit you laid out in Example 1.

2. Use the SWAP command to swap components, gates, and pins.

Section 1. Using the Ratsnest

A ratsnest is a point-to-point wiring diagram that represents electrical connections, but does not specify the final PCB placement of the actual wires. In this section, you will add another 7400 and some ratsnest segments to the layout you created in Example 1.

> NOTE: When the ratsnest display is turned on (the R on the status line is green), only the ratsnests of the unconnected pins are displayed; the ratsnests of the pins connected with the ENTR/WIRE command are not displayed. However, when you use the SWAP commands, the ratsnests of pins are displayed regardless of whether or not the pins have been connected.

1. Select FILE/LOAD to load TUTOR.PCB or TUTORT.PCB. The system prompts:

    Enter file name:

2. Type TUTOR.PCB or TUTORT.PCB and press [Return].

3. Select PAN to change the view center of the screen. The system prompts:

    PAN: Select View Center...

4. Select the center point of the 7404 as the view center.

    The screen display should look like Figure 2-22.

    NOTE: You may need to zoom out or zoom in, depending on the resolution of your monitor, to match the display shown.

```
                                                  SYMB DETL
                                                  FILE SYS
                                                       SCMD
                                                  SAVE ENTR
                                                  LOAD DRAW
                                                  ZAP  NAME
                                                       ATTR
                                                       EDIT
                                                       MOVE
                                                       ROT
                                                       COPY
                                                       DEL
                                                       CLYR
                                                       SWAP
                                                  ZIN  ZOUT
                                                  VWIN REDR
                                                 [PAN] LPAN
                                                  STO  RCL
PAN: Select View Center...                        VLYR MASK
                         R   50:50    S G    450    550
```
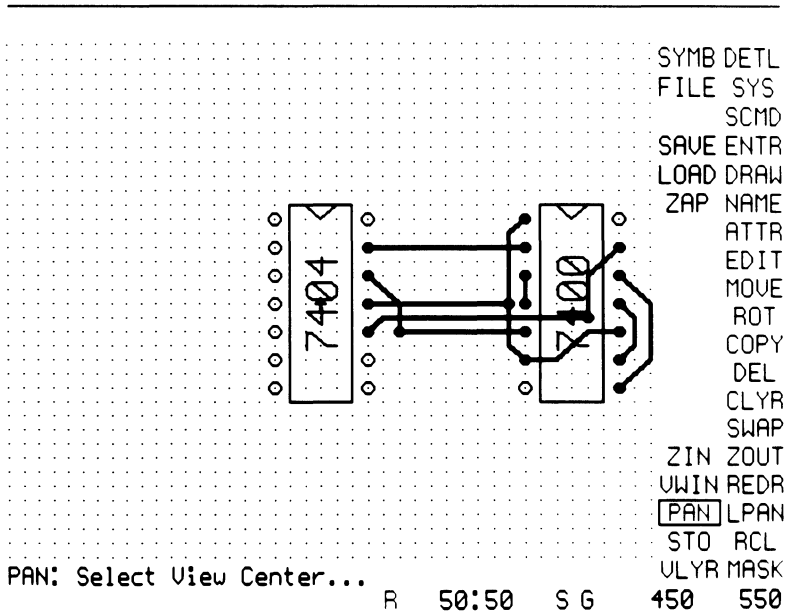
86117

**Figure 2-22. Screen Display After TUTORT.PCB
Is Loaded and View Center Is Changed**

5. Place another copy of the 7400 part to the left of
    the 7404 as shown in Figure 2-23. An easy way to
    do this is to:

    a. Select the COPY command.
    b. Select a point on the existing copy of the 7400
        part.
    c. Select a point in the left third of the screen as
        the location for the new copy of the 7400 part.
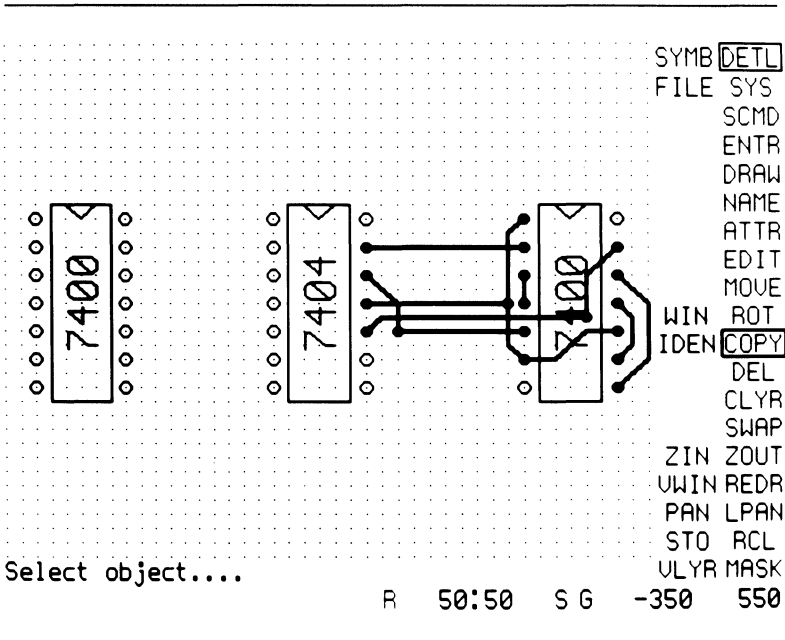
86118

Figure 2-23. Placement of the Second 7400 Part

6. Turn the ratsnest display on by moving the cursor
    to the red R on the status line and pressing Button
    1. The R will turn green, indicating the ratsnest
    display is on.

7. Select ENTR/RATN to begin the ratsnest. The
    system prompts:

    Select a pin....

8. First select the pin labeled A and then the pin
    labeled B as shown in Figure 2-24. A ratsnest
    segment appears between the two pins.

    NOTE: A ratsnest segment connects only the two
    pins which are its endpoints; it does not connect
    any of the pins in between.
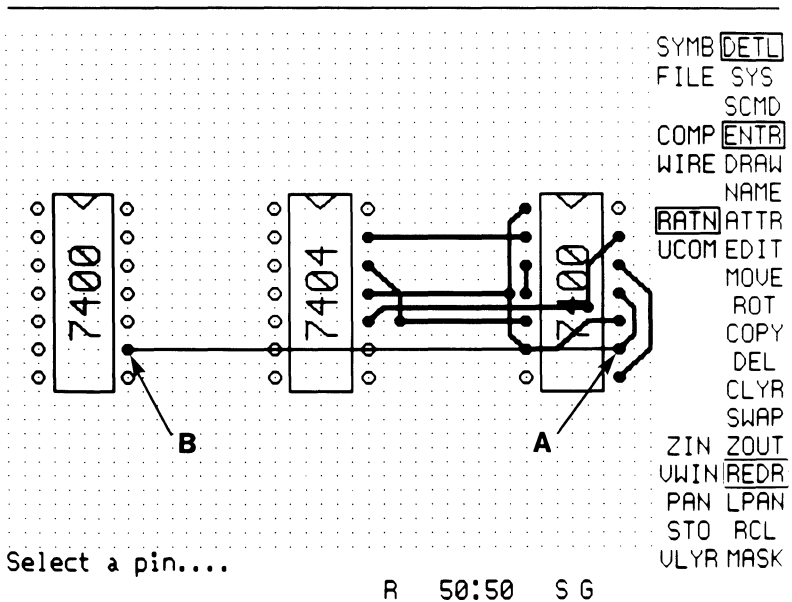
86119

Figure 2-24. First Ratsnest Segment Placed

9. Press Button 2 to complete the ratsnest segment.
    The segment will disappear. It is not erased; it is
    only hidden and can be redisplayed using the
    REDR command.

10. Add two more ratsnest segments as indicated in
    Figure 2-25. Remember to press Button 2 after
    entering each segment. Redraw the screen.

86120
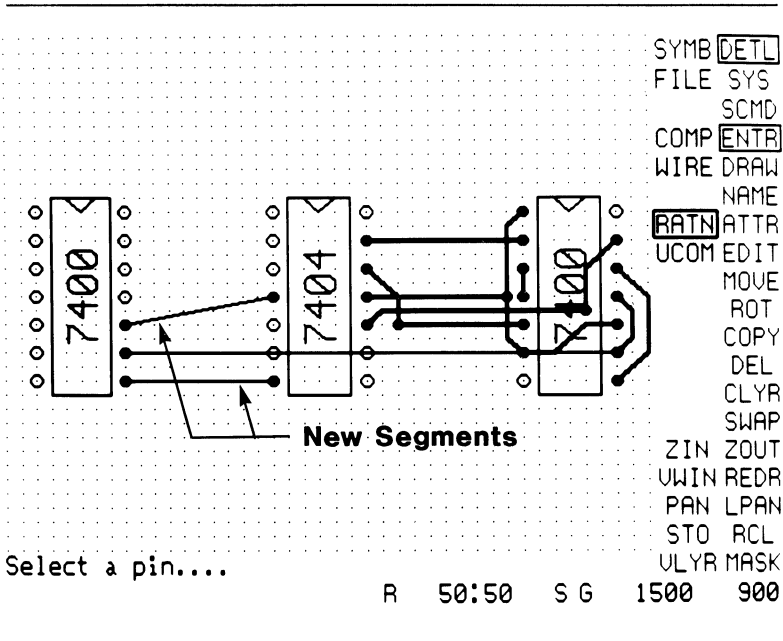
Figure 2-25. Completed Ratsnests

Notice that pins with ratsnest connections are shown
filled, whereas pins with no ratsnest connections are not
filled.

Section 2. Using the Swap Commands

1. Select SWAP/COMP. The system prompts:

    Select first comp....

2. Using VLYR, set the DEVICE layer to ABL and active. Turn the COMP and SOLDER layers to OFF. Redraw the screen. The circuit will now look like Figure 2-26.
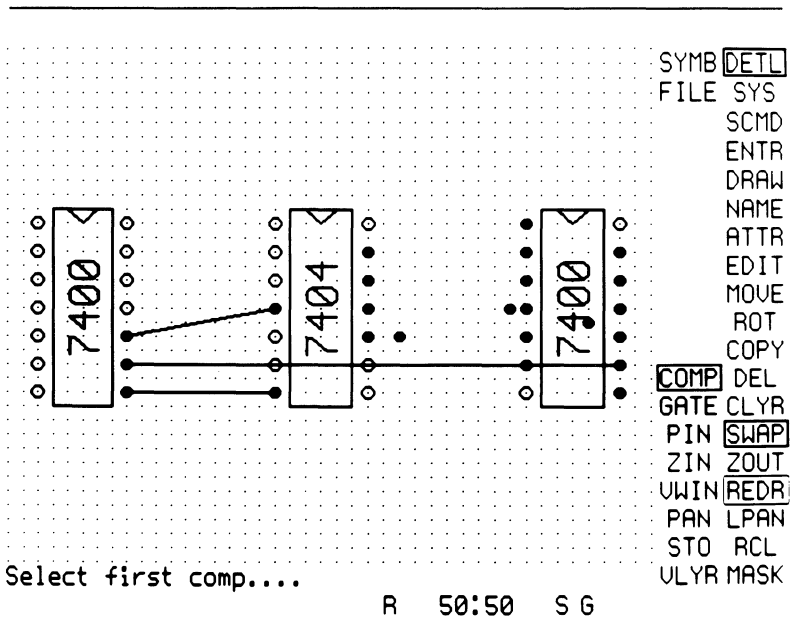
86121

Figure 2-26. COMP and SOLDER Layers Turned Off

3. Select the two 7400 parts. The two parts will change places, and the ratsnest will rearrange itself as shown in Figure 2-27.
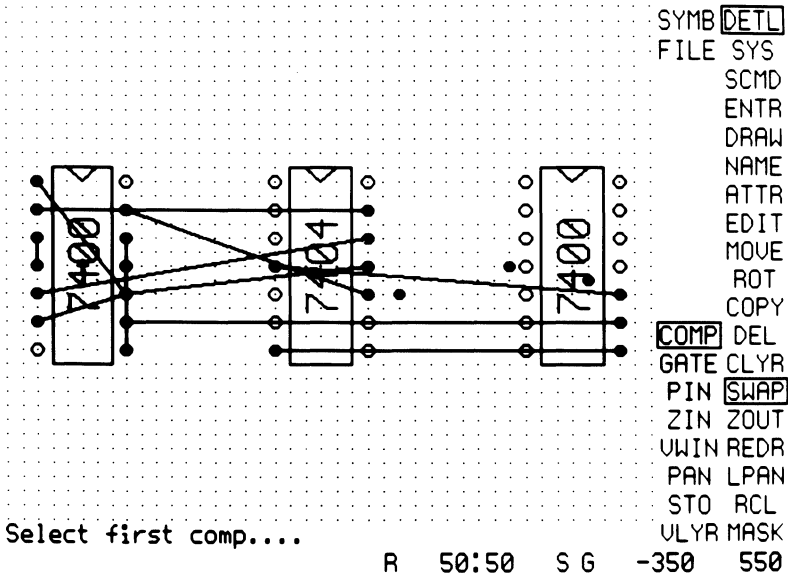
Figure 2-27. Result of SWAP/COMP

4. Select REDR.

    NOTE: The SWAP/GATE and SWAP/PIN
    commands will work only if packaging information
    has been entered. This step is described in Example
    5.

5. Select SWAP/GATE. The system prompts:

    Select first gate....

    The ratsnest will disappear.

6. Select the two pins labeled A and B (select A first,
    then B) in Figure 2-28 and redraw the screen.
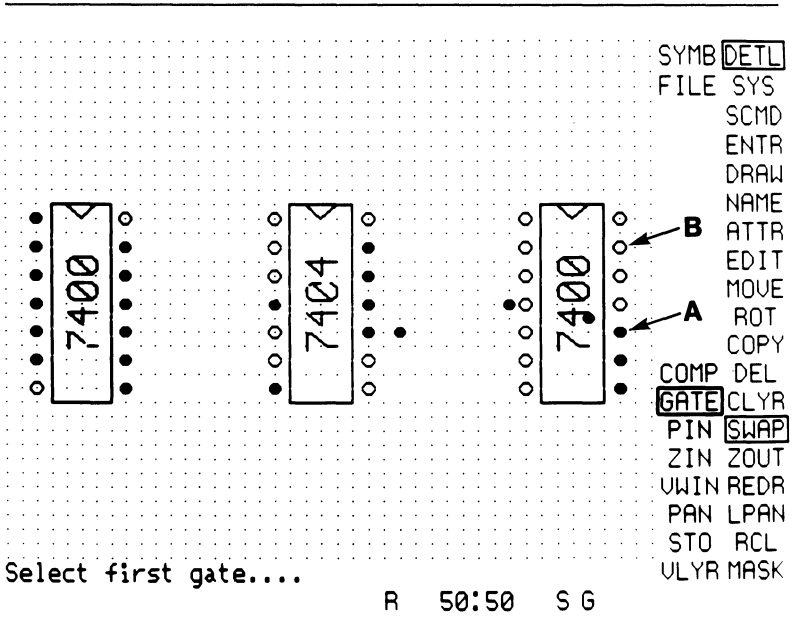
Figure 2-28. SWAP/GATE Command Prompt

The ratsnest that appears now shows a different
gate of the 7400 part wired into the circuit. The
new ratsnest is shown in Figure 2-29.
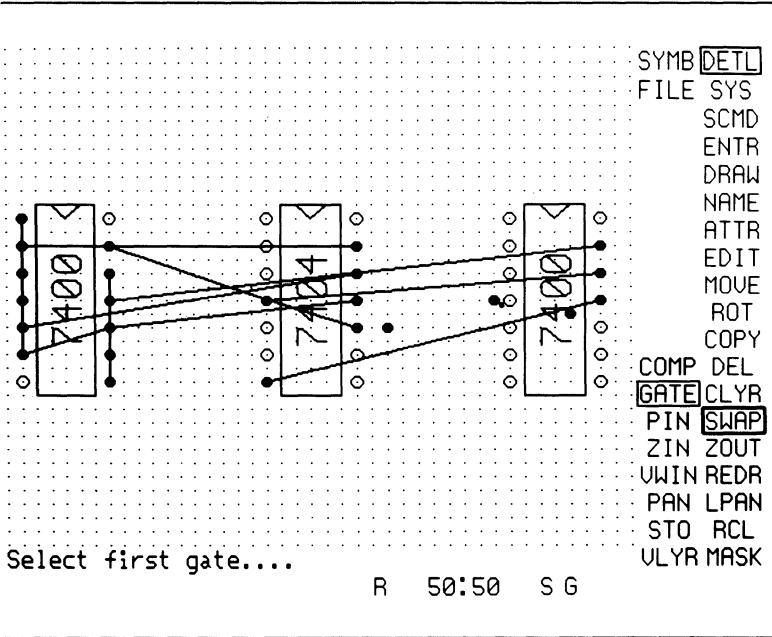
86124

Figure 2-29. Result of SWAP/GATE

7.   Select SWAP/PIN.

8. Select the two pins labeled A and B in Figure 2-30 and redraw the screen.
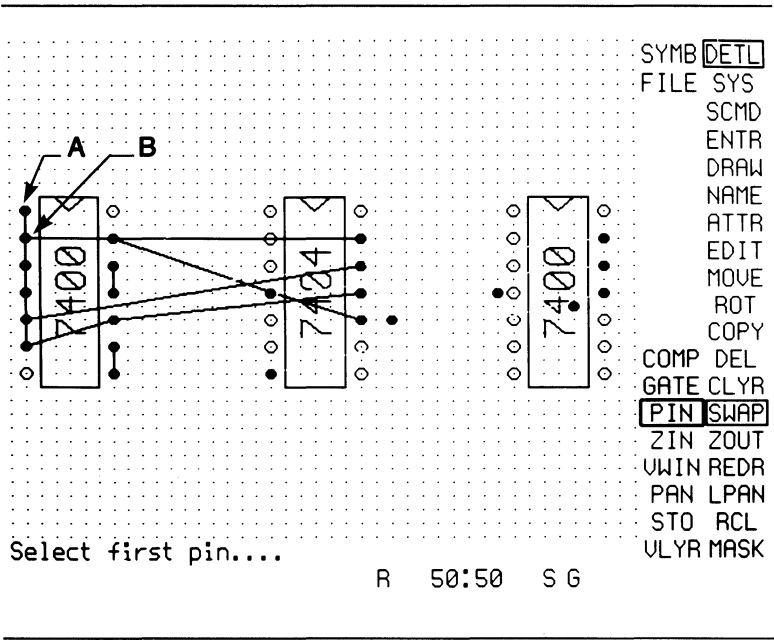
Figure 2-30. SWAP/PIN Command Prompt

In the ratsnest that appears now, pins 1 and 2 of the
7400 part on the left are reversed, as shown in
Figure 2-31.
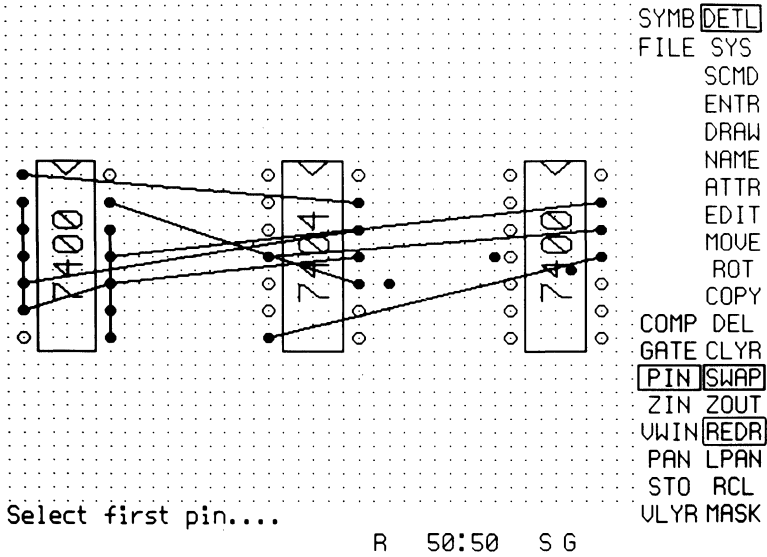
Figure 2-31. Result of SWAP/PIN

9. Select FILE/ZAP to clear the screen.

Summary

Table 2-2 summarizes the methods you learned in this part and the commands used in each method.

Table 2-2. Using Ratsnests

Step	Commands Used	Layer/File
Using a ratsnest	FILE/LOAD PAN COPY ENTR/RATN	TUTOR.PCB or TUTORT.PCB
Using the SWAP commands	SWAP/COMP SWAP/GATE SWAP/PIN FILE/ZAP	

EXAMPLE 3. EDITING A LAYOUT

Wires consist of vertices, segments, and vias. Each point you enter when creating a wire is a vertex. A section of wire between two adjacent vertices is a segment. An electrical connection connecting two wires on different layers is a via. When you edit a wire in a layout, you do so by changing parts of the wire instead of deleting the entire wire.

> CAUTION: Some EDIT commands are not as "intelligent" as the data entry commands. For example, if you enter a wire that crosses another wire on the same layer using the ENTR/WIRE command, the system asks you if you want to merge the nets. However, if you change the layer of a wire to the same layer of a wire that crosses it using the EDIT/LAYS command, no message is displayed asking you if you want to merge the nets.

In this example you will learn to use the EDIT commands to make changes to a layout by moving a segment and the attached segment and via, changing the layer of a segment, and entering a new wire. The EDIT commands are shown in Table 2-3.

Table 2-3. EDIT Commands

Command	Function
EDIT/ADDV	Adds vertex
EDIT/DELV	Deletes vertex
EDIT/MOVV	Moves vertex
EDIT/LAYS	Changes the layer of a segment
EDIT/DELS	Deletes segment
EDIT/MOVS	Moves segment
EDIT/AVIA	Adds via
EDIT/DVIA	Deletes via
EDIT/MVIA	Moves via
EDIT/MOVA	Moves segment or vertex and all attached segments and vias on all layers

Figure 2-32 shows the circuit you will be working with. When you are finished, it will look like Figure 2-33.
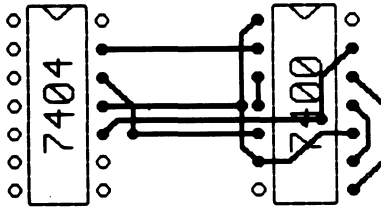
86127

Figure 2-32. Example Layout Before Editing
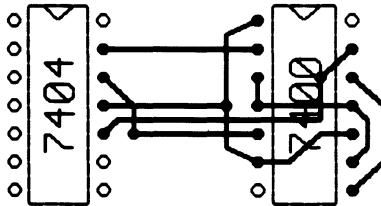
86128

Figure 2-33. Example Layout After Editing

Load the file TUTORT.PCB. Your screen should look like Figure 2-34.

> NOTE: You may need to zoom out or zoom in to match the screen shown.

Moving a Segment and the Attached Segment and Via

1. Select EDIT/MOVA to begin editing the circuit. The system prompts:

    Select a point....

2. Select a point on the segment at the via indicated in Figure 2-34.

Figure 2-34. Segment to Be Moved

3.   Move the segment to the left, as shown in Figure
      2-35, and press Button 1. Notice that the attached
      segment and via moves with the segment. Redraw
      the screen.
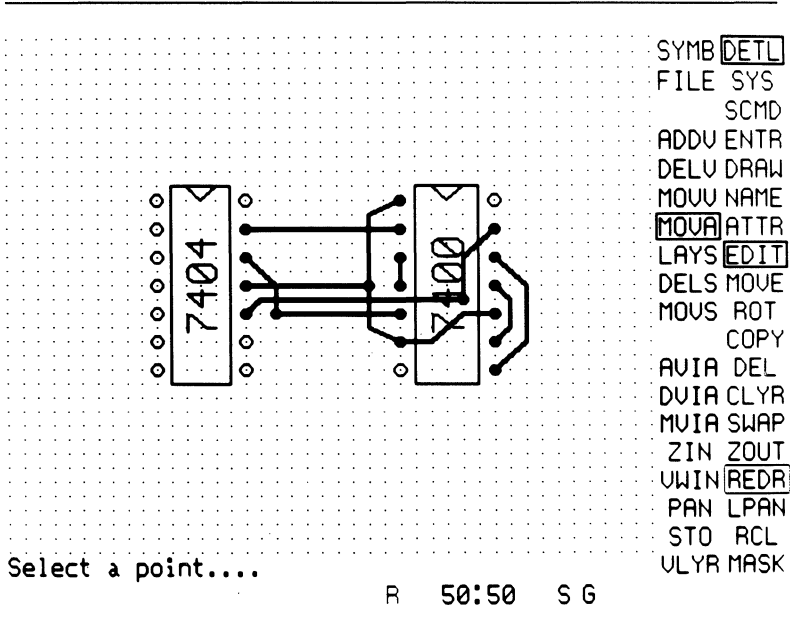
Figure 2-35. Segment Moved

Changing the Layer of a Segment

1. Select EDIT/LAYS. The system prompts:

   Select a segment....

2. Change the active layer to COMP.

3. Select any point on the segment indicated by the
   arrow in Figure 2-36. The segment changes from
   red to green indicating that it has changed from the
   SOLDER layer to the currently active COMP layer.
   Notice also that vias were automatically added and
   deleted where needed, as shown in Figure 2-37.
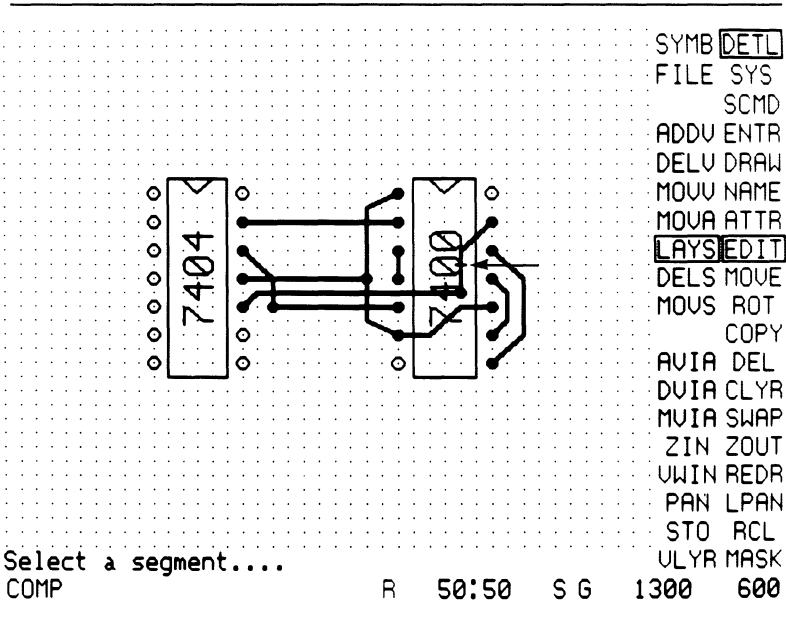
86131

**Figure 2-36. Segment to Be Changed
to a Different Layer**

Figure 2-37. Segment Layer Changed
from SOLDER to COMP

Entering a New Wire

1. Select ENTR/WIRE to enter a new wire.

2. Change the active layer from COMP to SOLDER
    and draw the new wire from the point labeled A in
    Figure 2-38 to the point labeled B. The system
    prompts:

    Merge the nets? YES NO

3. Select YES. Compare your results with Figure
    2-39.
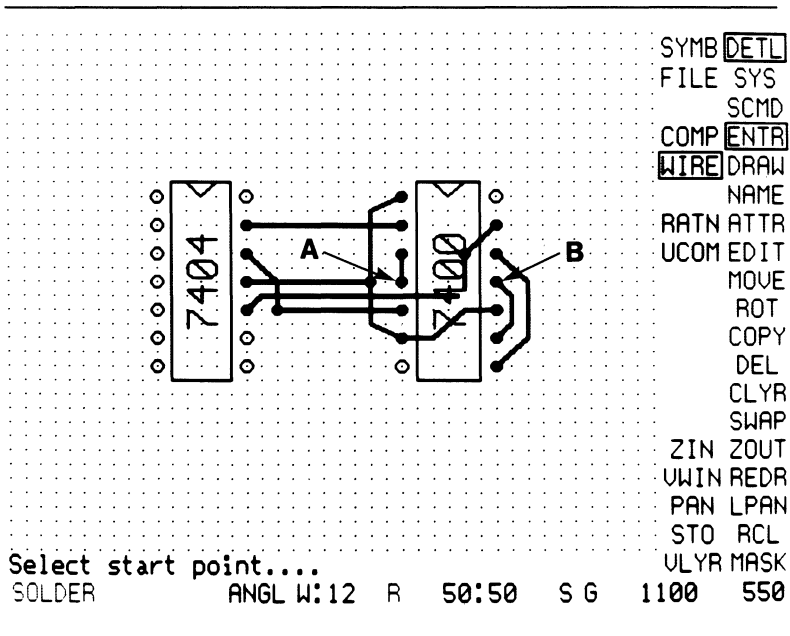
Figure 2-38. Endpoints of Wire to Be Entered

SYMB DETL
FILE SYS
     SCMD
COMP ENTR
WIRE DRAW
     NAME
RATN ATTR
UCOM EDIT
     MOVE
     ROT
     COPY
     DEL
     CLYR
     SWAP
ZIN ZOUT
VWIN REDR
PAN LPAN
STO RCL
ULYR MASK

Select start point....
SOLDER   ANGL W:12   R   50:50   S G   1400   550
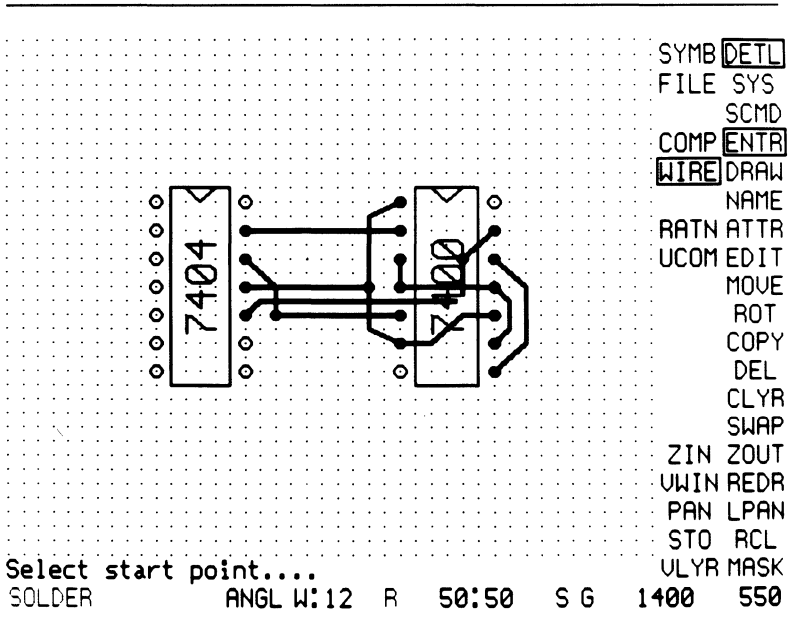
86134

Figure 2-39. New Wire Entered

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

     This completes the wiring corrections.

5.   Redraw the screen and compare it to Figure 2-33.

6.   Select FILE/ZAP to clear the screen.

Summary

Now that you have completed this example, you are familiar with the EDIT commands and know how to use them to make minor changes in a layout. If you worked through all the examples this far, you have learned how to:

- Create and save a circuit layout

- Use ratsnests, and use the SWAP commands

- Edit a layout

If you would like to learn how to create and use padstacks and special symbol files, go to Example 4.

EXAMPLE 4. CREATING A PADSTACK AND A SPECIAL SYMBOL FILE

In "Example 3. Creating a Layout," you linked a special symbol file to the layout. The special symbol file assigns padstack graphics according to the pin types used in the layout. In this example, you will create a padstack and its associated special symbol file.

A pad is a drawn or plotted terminal point at which wires are interconnected in a PCB layout. The pad may take virtually any shape that can serve as a connection point, for example, circular, square, oval, or diamond. The pen plots and other graphics for a PCB often require different pad shapes for the same point on different layers of the board.

A padstack is a database file which contains the set of pad shapes which are required for the various layers of a certain pin type. Padstacks are linked to a pin in a layout according to a special symbol file.

Padstacks are also used to specify flash information for photoplotting. A flash consists of an aperture setting and a place on the film which is to be exposed. A photoplotter aperture, unlike a standard camera aperture, has a fixed size and shape which must match that of the pad or trace to be produced on film. Instead of having a single adjustable aperture, a photoplotter has a number of apertures.

The aperture settings used by PC-CARDS, which are shown in Table 2-4, correspond to the 24 positions on the aperture wheel of the Gerber photoplotter.

> NOTE: Check with your photoplotting service to make sure these settings are available.

Table 2-4. P-CAD Aperture Settings

Setting	Size	Shape	Type
1	5	Round	Line
2	10	Round	Line
3	12	Round	Line
4	25	Round	Line
5	50	Round	Line
6	75	Round	Line
7	100	Round	Line
8	0	Donut	Flash
9	60	Round	Flash
10	60	Square	Flash
11	70	Round	Flash
12	70	Square	Flash
13	75	Round	Flash
14	75	Square	Flash
15	50	Round	Flash
16	55	Round	Flash
17	80	Round	Flash
18	85	Round	Flash
19	90	Round	Flash
20	100	Round	Flash
21	125	Round	Flash
22	0	Bracket	Flash
23	50	*	Flash
24	100	**	Flash

*   Drill crosshair, 5 mil
**   Bowtie target, quadrants 2 and 4

P-CAD's PC-PHOTO utility program uses these aperture settings to convert a PC-CARDS database into instructions for a photoplotter, which in turn produces the finished tooling film from which the actual PCB is fabricated. You can transfer these instructions to the photoplotter either by means of a disk or magnetic tape or by using PC-PHOTO to drive the photoplotter directly. Before running PC-PHOTO, make sure the settings listed in Table 2-4 correspond to the PC-PHOTO aperture list. If they don't, edit the list in PC-PHOTO to match the list in Table 2-4.

The P-CAD standard layer structure for PC-CARDS reserves the first 17 layers of the layout for padstack definitions. These layer assignments are shown in Table 2-5. Notice that the first 16 layers consist of 8 pairs of layers. For each graphics layer, there is a corresponding flash aperture layer.

You must conform to these layer conventions in order to keep your padstack database files compatible with P-CAD libraries.

You create PCB layouts and padstacks in independent databases. The PC-CARDS SCMD/GSSF command links the two databases.

Table 2-5. P-CAD Padstack Layer Structure

Layer Name	Contents of Layer
PADCOM	Component side pad graphics
FLCOMP	Component side pad flash
PADSLD	Solder side pad graphics
FLSOLD	Solder side pad flash
PADINT	Internal layer pad graphics
FLINT	Internal layer pad flash
GNDCON	Ground plane connection graphics
FLGCON	Ground plane connection flash
CLEAR	Universal clearance graphics
FLCLER	Universal clearance flash
PWRCON	Power plane connection graphics
FLPCON	Power plane connection flash
SLDMSK	Soldermask graphics
FLSMSK	Soldermask flash
DRILL	Drill symbol graphics
FLDRLL	Drill symbol flash
PIN	Connectivity pin (graphics only)

The following examples illustrate P-CAD naming conventions for padstacks.

Example 1:

N60R32G.PS

where:

N - Noncommitted
60 - 60 mils
R - Round
32 - 0.032-inch drill
G - Ground tie

Example 2:

C70S45P.PS

where:

C - Committed
70 - 70 mils
S - Square
45 - 0.045-inch drill
P - Power tie (If "C" is used in this position, it indicates a universal clearance, which clears both ground and power.)

Table 2-6 shows the Gerber photoplotter aperture chart, which describes the padstack shapes and sizes for pin types 0 through 10.

Table 2-6. Gerber Photoplotter Aperture Chart
(Sheet 1 of 3)

LAYER	TYPE 0 V50R28C.PS	TYPE 2 (N/C) 60R32C.PS	TYPE 3 (N/C) 60R32G.PS	TYPE 4 (N/C) 60R32P.PS
PADCOM	.050 Circle	.060 Circle	.060 Circle	.060 Circle
FLCOMP	Aperture 15	Aperture 9	Aperture 9	Aperture 9
PADSLD	.050 Circle	.060 Circle	.060 Circle	.060 Circle
FLSOLD	Aperture 15	Aperture 9	Aperture 9	Aperture 9
PADINT	.050 Circle	.060 Circle	.060 Circle	.060 Circle
FLINT	Aperture 15	Aperture 9	Aperture 9	Aperture 9
GNDCON	.020 Ring .060 Inner Diam .100 Outer Diam	.020 Ring .060 Inner Diam .100 Outer Diam	Aperture 9 .025 Width X .100 Outer Diam	.020 Ring .060 Inner Diam
FLGCON	Aperture 8 ·	Aperture 8	Aperture 22	Aperture 8
CLEAR	.100 Circle Solid Circle	.125 Circle Solid Circle	.125 Circle Solid Circle	.125 Circle Solid Circle
FLCLER	Aperture 20	Aperture 21	Aperture 21	Aperture 21
PWRCON	.020 Ring .060 Inner Diam .100 Outer Diam	.020 Ring .060 Inner Diam .100 Outer Diam	.020 Ring .060 Inner Diam .100 Outer Diam	Aperture 9 .025 Width X
FLPCON	Aperture 8	Aperture 8	Aperture 8	Aperture 22
SLDMSK	.060 Circle	.070 Circle	.070 Circle	.070 Circle
FLSMSK	Aperture 9	Aperture 11	Aperture 11	Aperture 11
DRILL	+28	+32	+32	+32
FLDRLL	Aperture 23 Text 28	Aperture 23 Text 32	Aperture 23 Text 32	Aperture 23 Text 32
PIN*	.050	.050	.050	.050

* The pin layer reflects connectivity (C) with a solid circle or no
    connectivity (N) with a hollow circle.

Table 2-6. Gerber Photoplotter Aperture Chart
(Sheet 2 of 3)

LAYER	TYPE 1 (N/C) 60S32C.PS	TYPE 5 (N/C) 60S32P.PS	TYPE 6 (N/C) 60S32G.PS
PADCON	.060 Square	.060 Square	.060 Square
FLCOMP	Aperture 10	Aperture 10	Aperture 10
PADSLD	.060 Square	.060 Square	.060 Square
FLSOLD	Aperture 10	Aperture 10	Aperture 10
PADINT	.060 Circle	.060 Circle	.060 Circle
FLINT	Aperture 9	Aperture 9	Aperture 9
GNDCON	.020 Ring .060 Inner Diam .100 Outer Diam	.020 Ring .060 Inner Diam .100 Outer Diam	Aperture 9 .025 Width X
FLGCON	Aperture 8	Aperture 8	Aperture 22
CLEAR	.125 Circle Solid Circle	.125 Circle Solid Circle	.125 Circle Solid Circle
FLCLER	Aperture 21	Aperture 21	Aperture 21
PWRCON	.020 Ring .060 Inner Diam .100 Outer Diam	Aperture 9 .025 Width X .100 Outer Diam	.020 Ring .060 Inner Diam
FLPCON	Aperture 8	Aperture 22	Aperture 8
SLDMSK	.070 Square	.070 Square	.070 Square
FLSMSK	Aperture 12	Aperture 12	Aperture 12
DRILL	+32	+32	+32
FLDRLL	Aperture 23 Text 32	Aperture 23 Text 32	Aperture 23 Text 32
PIN*	.050	.050	.050

* The pin layer reflects connectivity (C) with a solid circle or no connectivity (N) with a hollow circle.

Table 2-6. Gerber Photoplotter Aperture Chart
(Sheet 3 of 3)

LAYER	TYPE 8 (N/C) 80R50C.PS	TYPE 9 (N/C) 90R60C.PS	TYPE 10* 210R182C.PS	TYPE 7 (N/C) 100R70C.PS
PADCOM	.080 Circle	.090 Circle	.210 Circle	.100 Circle
FLCOMP	Aperture 17	Aperture 19	.210 Solid Circle	Aperture 20
PADSLD	.080 Circle	.090 Circle	.210 Circle	.100 Circle
FLSOLD	Aperture 17	Aperture 19	.210 Solid Circle	Aperture 20
PADINT	.080 Circle	.090 Circle	.210 Circle	.100 Circle
FLINT	Aperture 17	Aperture 19	.210 Solid Circle	Aperture 20
GNDCON	.025 Ring .080 Inner Diam .130 Outer Diam	.025 Ring .090 Inner Diam .140 Outer Diam	.025 Ring .210 Inner Diam .260 Outer Diam	.025 Ring .100 Inner Diam .150 Outer Diam
FLGCON	.025 Ring .080 Inner Diam .130 Outer Diam	.025 Ring .090 Inner Diam .140 Outer Diam	.025 Ring .210 Inner Diam .260 Outer Diam	.025 Ring .100 Inner Diam .150 Outer Diam
CLEAR	.160 Solid Circle	.180 Solid Circle	.420 Solid Circle	.200 Solid Circle
FLCLER	.160 Solid Circle	.180 Solid Circle	.420 Solid Circle	.200 Solid Circle
PWRCON	.025 Ring .080 Inner Diam .130 Outer Diam	.025 Ring .090 Inner Diam .140 Outer Diam	.025 Ring .210 Inner Diam .260 Outer Diam	.025 Ring .100 Inner Diam .150 Outer Diam
FLPCON	.025 Ring .080 Inner Diam .130 Outer Diam	.025 Ring .090 Inner Diam .140 Outer Diam	.025 Ring .210 Inner Diam .260 Outer Diam	.025 Ring .100 Inner Diam .150 Outer Diam
SLDMSK	.090 Circle	.100 Circle	.220 Circle	.110 Circle
FLSMSK	Aperture 19	Aperture 20	.220 Solid Circle	.110 Solid Circle
DRILL	+50	+60	+182	+70
FLDRLL	Aperture 23 Text 50	Aperture 23 Text 60	Aperture 23 Text 182	Aperture 23 Text 70
PIN**	.070	.080	.200	.090

**   Type 10 is a connect only pin

*   The pin layer reflects connectivity (C) with a solid circle or no connectivity (N) with a hollow circle.

Table 2-7 provides an example of the P-CAD standards
for creating an uncommitted, 60-mil round, 0.032-inch
drill, universal clearance pad (N60R32C.PS). This table
lists the layers, commands, and the suggested
parameters, including circle size and line width for
graphics and aperture size for the flashes when using
P-CAD's aperture list.

Table 2-7. Padstack Symbol Standards for a N60R32C.PS

Entity	Layer	Command Used	Suggested Parameters
Component side pads	PADCOM	DRAW/CIRC	60-mil circle 0 line width
	FLCOMP	DRAW/FLSH	APER: 9
Solder side pads	PADSLD	DRAW/CIRC	60-mil circle 0 line width
	FLSOLD	DRAW/FLSH	APER: 9
Internal trace layer pads	PADINT	DRAW/CIRC	60-mil circle 0 line width
	FLINT	DRAW/FLSH	APER: 9
Ground plane connections	GNDCON	DRAW/CIRC	20-mil ring 60-mil inner diameter 100-mil outer diameter
	FLGCON	DRAW/FLSH	APER: 8
Universal clearance	CLEAR	DRAW/CIRC	125-mil solid circle
	FLCLER	DRAW/FLSH	APER: 21

Table 2-7 Continued

Entity	Layer	Command Used	Suggested Parameters
Power plane connections	PWRCON	DRAW/CIRC	20-mil ring 60-mil inner diameter 100-mil outer diameter
	FLPCON	DRAW/FLSH	APER: 8
Solder mask relief	SLDMSK	DRAW/CIRC	70-mil circle 0 line width
	FLSMSK	DRAW/FLSH	APER: 11
Drill	DRILL	DRAW/LINE	Crosshair (+) showing drill center 0 line width
		DRAW/TEXT	60 Text size
	FLDRLL	DRAW/FLSH	APER: 23
Pin connections	PIN	DRAW/CIRC	50-mil circle 0 line width
Center of padstack		ENTR/ORG	

NOTE: Aperture parameters are for the Gerber photoplotter only.

In this example, you will create an uncommitted, 60-mil round, 32-mil drill padstack and a special symbol file, and you will link the special symbol file to a layout. The steps in this process are:
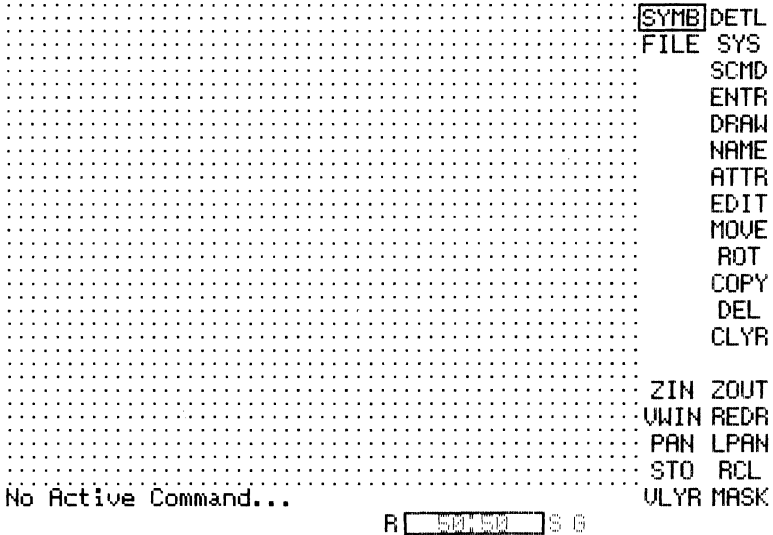
1. Set up the environment.
2. Create the padstack.
3. Create the special symbol file.
4. Link the special symbol file.

Step 1. Setting Up the Environment

1. Set the drawing mode to SYMB.

2. Select the X:Y grid spacing parameter on the status line as shown in Figure 2-40. The system prompts:

   Enter x grid size:

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

```
                                                    SYMB DETL
                                                    FILE SYS
                                                         SCMD
                                                         ENTR
                                                         DRAW
                                                         NAME
                                                         ATTR
                                                         EDIT
                                                         MOVE
                                                         ROT
                                                         COPY
                                                         DEL
                                                         CLYR

                                                    ZIN  ZOUT
                                                    VWIN REDR
                                                    PAN  LPAN
                                                    STO  RCL
No Active Command...                                VLYR MASK
                            R[  50:50  ]S G
```

86135

Figure 2-40. 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 the center of the screen and press Button 2 four times.

6. Select the X Y cursor coordinate parameter on the status line as shown in Figure 2-41. The system prompts:

    Enter x position:

```
                                              SYMB DETL
                                              FILE SYS
                                                   SCMD
                                                   ENTR
                                                   DRAW
                                                   NAME
                                                   ATTR
                                                   EDIT
                                                   MOVE
                                                   ROT
                                                   COPY
                                                   DEL
                                                   CLYR

                                              ZIN  ZOUT
                                              VWIN REDR
                                              PAN  LPAN
                                              STO  RCL
Select view center...                         VLYR MASK
                           R    5:5    S 6    275    25
```
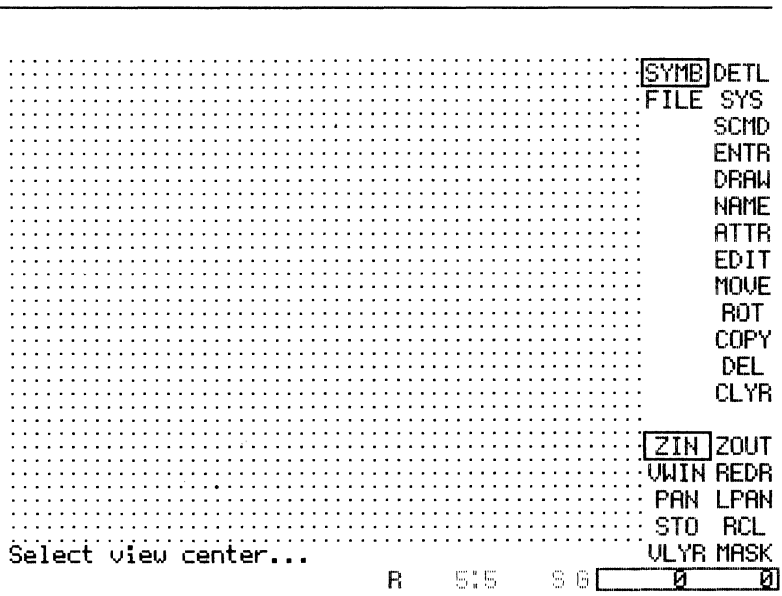
86136

Figure 2-41. Selecting X Y Cursor Location Status Line Parameter

7. Type 0 0 and press [Return] to set the cursor location to 0. This allows you to enter the coordinates for the location of a circle with a minimum of calculation.

The result is shown in Figure 2-42.

```
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::  :[SYMB]DETL
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::  :FILE  SYS
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         SCMD
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         ENTR
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         DRAW
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         NAME
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         ATTR
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         EDIT
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         MOVE
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         ROT
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         COPY
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         DEL
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::         CLYR
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::  :[ZIN]ZOUT
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::  :VWIN REDR
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::  : PAN  LPAN
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::  : STO  RCL
Select view center...                                     VLYR MASK
                         R    5:5      S G[      0       0]
```

86137

**Figure 2-42. View After Selecting ZIN
and Entering the Cursor Location**

8. Using VLYR, set the following layers to ABL
    status: PADCOM, FLCOMP, PADSLD, FLSOLD,
    PADINT, FLINT, GNDCON, FLGCON, CLEAR,
    FLCLER, PWRCON, FLPCON, SLDMSK, FLSMSK,
    DRILL, FLDRILL, and PIN. Make PADCOM the
    active layer. Turn the rest of the layers off.

Step 2. Creating the Padstack

In this step you will create the padstack described in Table 2-7 and shown in Figure 2-43.
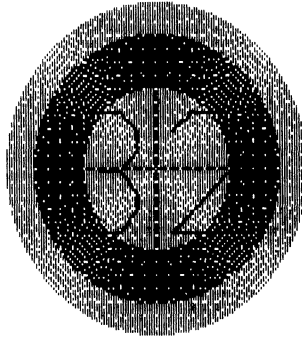
86138

Figure 2-43. Padstack to Be Drawn in Step 2

The procedure you will follow in creating a padstack is:

1. Draw the graphics.
2. Draw the flashes.
3. Assign the origin and save the padstack file.

Drawing the Graphics

1. Select DRAW/CIRC to begin drawing the pad graphics. The system prompts:

    Center point....

2. Before you select the center point, change the line width (W: on the status line) to 0.

    NOTE: A line width of 0 is the smallest width that can be seen on the screen, and it is the smallest size that the pen plotter permits.

3. Draw a circle 60 mils in diameter in the center of the screen as follows: Select the XY cursor coordinate parameter on the status line. The system prompts:

    Enter x position:

4. Type 0 0 [Return] to indicate the location of the center of the circle. The system prompts:

    Point on circumference....

5. Select the X Y cursor coordinate parameter again on the status line. The system prompts:

    Enter x position:

6. Type 0 30 [Return] to indicate the location of the circumference of the circle.

    This circle represents a 60-mil pad. Your pad should look like the one in Figure 2-44.

```
                                                      :SYMB DETL
                                                      :FILE SYS
                                                      :     SCMD
                                                      :LINE ENTR
                                                      :RECT DRAW
                                                      :FREC NAME
                                                      :CIRC ATTR
                                                      : ARC EDIT
                                                      :TEXT MOVE
                                                      :FLSH ROT
                                                      :     COPY
                                                      :     DEL
                                                      :     CLYR

                                                       ZIN ZOUT
                                                      :VWIN REDR
                                                       PAN LPAN
                                                       STO  RCL
Center point....                                      VLYR MASK
PADCOM  SOLID        W:0   R    5:5    S G[    0     30]
```
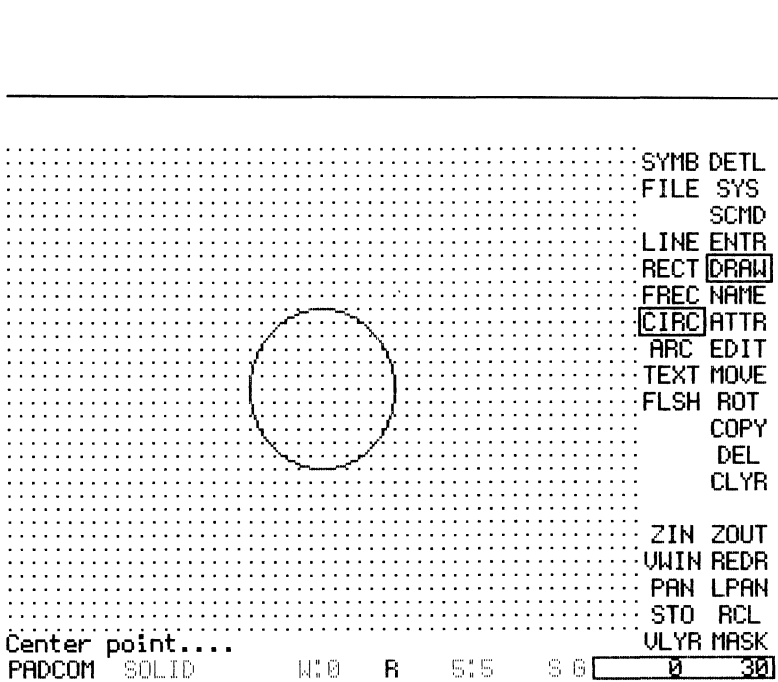
86139

Figure 2-44. Pad Drawn on PADCOM Layer

7. Change the active layer to PADSLD.

8. On this layer, draw a circle 60 mils in diameter as
    you did for the component side pad. Use 0 0 for
    the cursor location for the center of the circle and
    use 0 30 for the circumference. Your screen should
    look like Figure 2-45.

```
........................................:SYMB DETL
........................................:FILE SYS
........................................:     SCMD
........................................:LINE ENTR
........................................:RECT DRAW
........................................:FREC NAME
........................................:CIRC ATTR
........................................: ARC EDIT
........................................:TEXT MOVE
........................................:FLSH ROT
........................................:     COPY
........................................:     DEL
........................................:     CLYR
........................................:
........................................: ZIN ZOUT
........................................:UWIN REDR
........................................:PAN LPAN
........................................:STO  RCL
Center point....                        :ULYR MASK
PADSLD  SOLID        W:0    R    5:5   S G[    0    30]
```
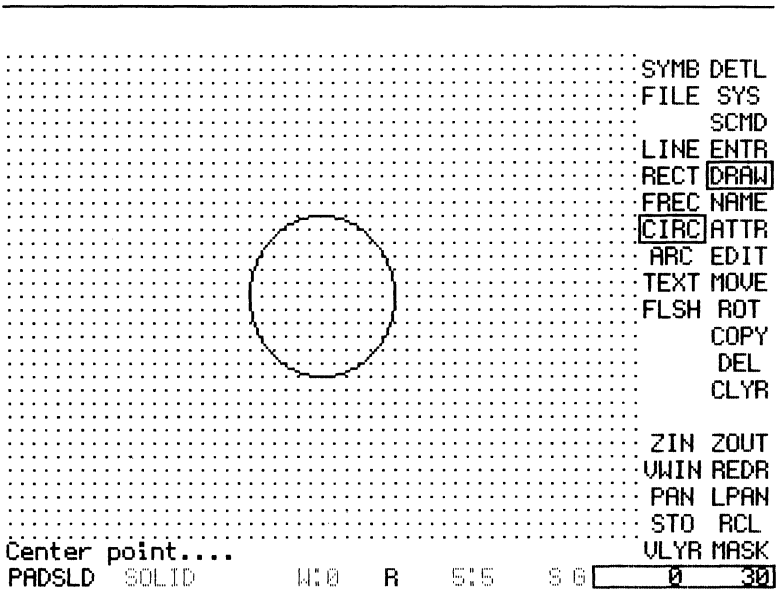
86140

Figure 2-45. Pad Drawn on PADSLD Layer

9.   Change the active layer to PADINT.

10. On this layer, draw a 60-mil circle. Use 0 0 for the
     cursor location for the center of the circle and use 0
     30 for the circumference. Your screen should look
     like Figure 2-46.

```
                                                    SYMB DETL
                                                   :FILE SYS
                                                         SCMD
                                                   :LINE ENTR
                                                   :RECT DRAW
                                                   :FREC NAME
                                                   :CIRC ATTR
                                                    ARC EDIT
                                                   :TEXT MOVE
                                                   :FLSH ROT
                                                         COPY
                                                         DEL
                                                         CLYR

                                                    ZIN ZOUT
                                                   :VWIN REDR
                                                    PAN LPAN
                                                    STO RCL
Center point....                                    VLYR MASK
PADINT   SOLID          W:0   R    S:5    S 6       0      30
```
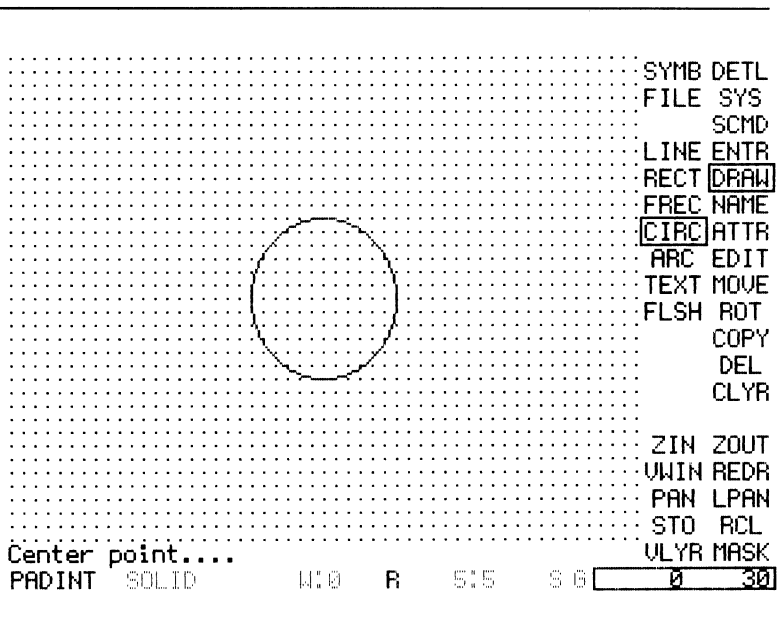
86141

Figure 2-46. Pad Drawn on PADINT Layer

11. Change the active layer to GNDCON.

12. Change the line width to 20 (W:20).

13. On this layer, draw a ring with a 60-mil inner diameter and a 100-mil outer diameter. Use 0 0 for the cursor location for the center of the circle and use 0 40 for the circumference. Your screen should look like Figure 2-47.
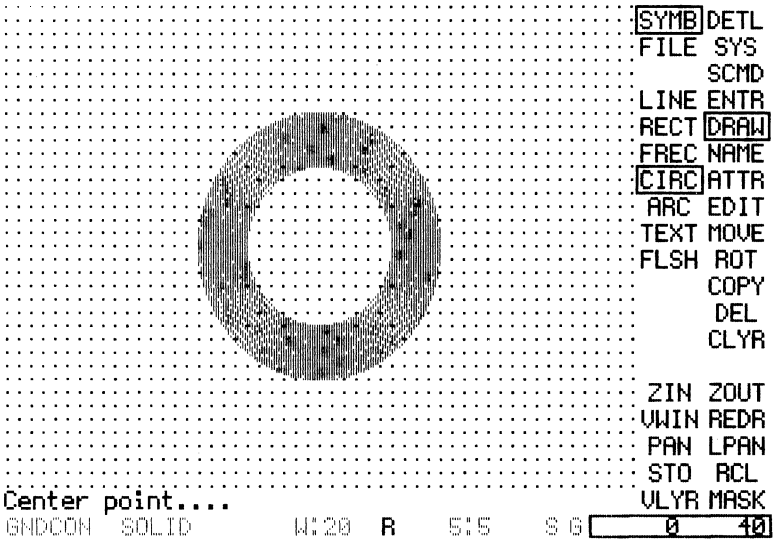
86142

Figure 2-47. Pad Drawn on GNDCON Layer

14. Change the active layer to CLEAR.

15. Change the line width to 63.

16. On this layer, draw a solid circle 125-mils in diameter. Use 0 0 for the cursor location for the center of the circle and use 0 31 for the circumference. Your screen should look like Figure 2-48.

SYMB DETL
FILE SYS
SCMD
LINE ENTR
RECT DRAW
FREC NAME
CIRC ATTR
ARC EDIT
TEXT MOVE
FLSH ROT
COPY
DEL
CLYR

ZIN ZOUT
VWIN REDR
PAN LPAN
STO RCL
VLYR MASK
0   31

Center point....
CLEAR   SOLID   W:63 R   5:5   S G   0   31
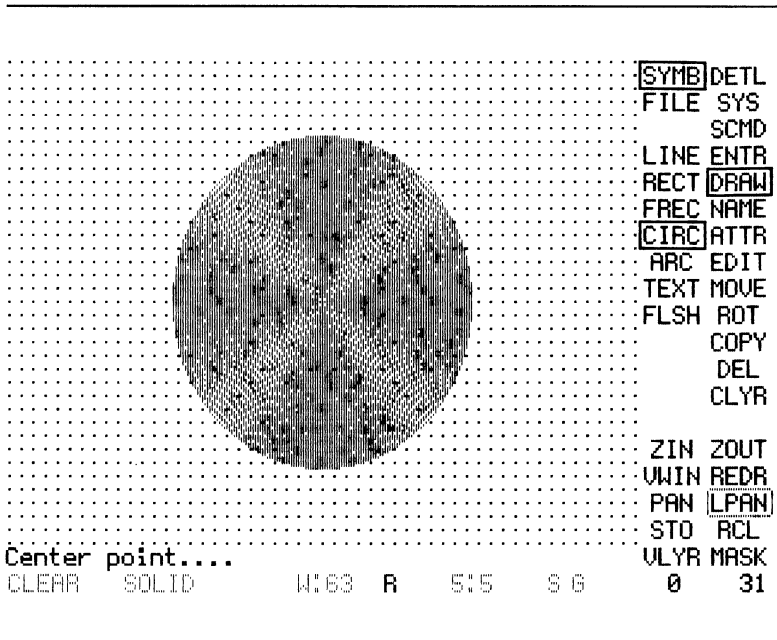
86143

Figure 2-48. Pad Drawn on CLEAR Layer

17. Change the active layer to PWRCON.

18. Change the line width to 20.

19. On this layer, draw a ring with a 60-mil inner
     diameter and a 100-mil outer diameter. Use 0 0 for
     the cursor location for the center of the circle and
     use 0 40 for the circumference. Your screen should
     look like Figure 2-49.
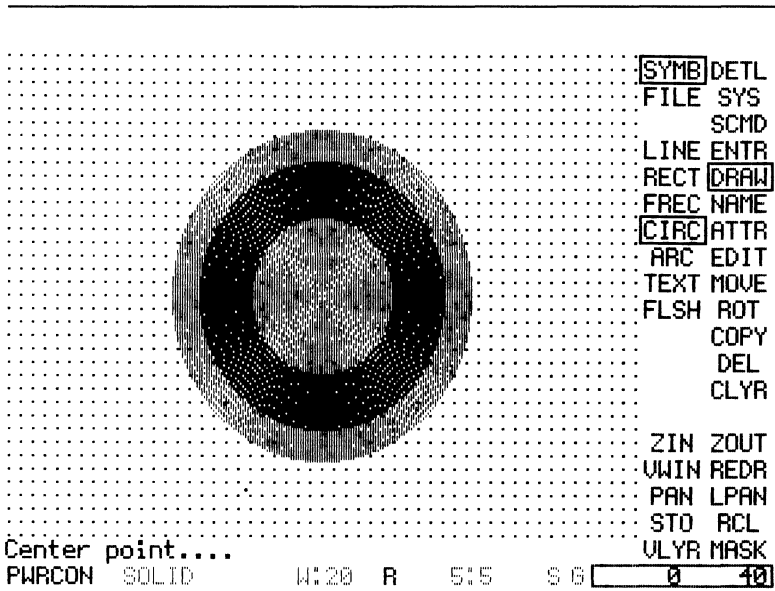
86144

Figure 2-49. Pad Drawn on PWRCON Layer

20. Change the active layer to SLDMSK.

21. Change the line width to 0.

22. On this layer, draw a circle 90 mils in diameter
    with the same center point as the 60-mil circle. Use
    0 0 for the cursor location for the center of the
    circle and use 0 45 for the circumference of the
    circle. Compare your drawing with Figure 2-50.
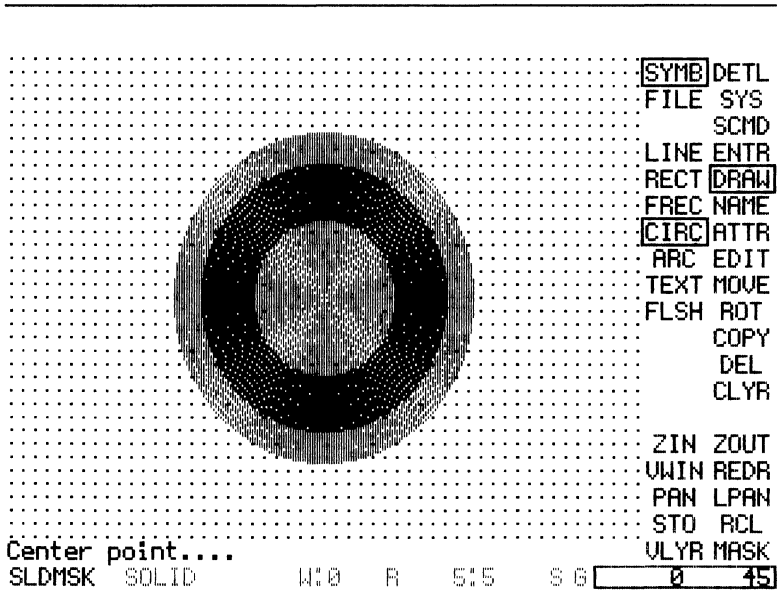
86145

Figure 2-50. Pad Drawn on SLDMSK Layer

23. Using VLYR, change the color of the DRILL layer
    by placing the cursor over the number 15 following
    the word DRILL. Select a new color for the DRILL
    layer by moving the cursor over the number 11 as
    shown in Figure 2-51 and pressing Button 1.

```
PADCOM 7  ABL     BRDOUT 4  OFF
FLCOMP 7  ABL     FLTARG 4  OFF
PADSLD 8  ABL     SLKSCR 6  OFF
FLSOLD 8  ABL     DEVICE 5  OFF
PADINT 9  ABL     ATTR   6  OFF
FLINT  9  ABL     REFDES 6  OFF
GNDCON 10 ABL     COMP   1  OFF
FLGCON 10 ABL     SOLDER 2  OFF
CLEAR  12 ABL     INT1   3  OFF
FLCLER 12 ABL
PWRCON 13 ABL         QUIT
FLPCON 13 ABL
SLDMSK 14 ABL A
FLSMSK 14 ABL
DRILL  15 ABL
FLDRLL 15 ABL
PIN    4  ABL

Color:  1 2 3 4 5 6 7 8 9 10 [11] 12 13 14 15
```

86146

Figure 2-51. Changing the Color of the DRILL Layer

24. Change the active layer to DRILL.

25. Select DRAW/LINE. The system prompts:

    Select start point....

26. Select points to draw lines through the center of the padstack that represent the center of the drill hole as shown in Figure 2-52.


```
............................................:SYMB DETL
............................................:FILE SYS
............................................:     SCMD
............................................:LINE ENTR
............................................:RECT DRAW
............................................:FREC NAME
............................................:CIRC ATTR
............................................: ARC EDIT
............................................: TEXT MOVE
............................................:FLSH ROT
............................................:     COPY
............................................:     DEL
............................................:     CLYR
............................................:
............................................: ZIN ZOUT
............................................: UWIN REDR
............................................: PAN LPAN
............................................: STO RCL
Select start point....                        VLYR MASK
DRILL    SOLID   ORTH W:0   R   S:S   S G      95  -70
```
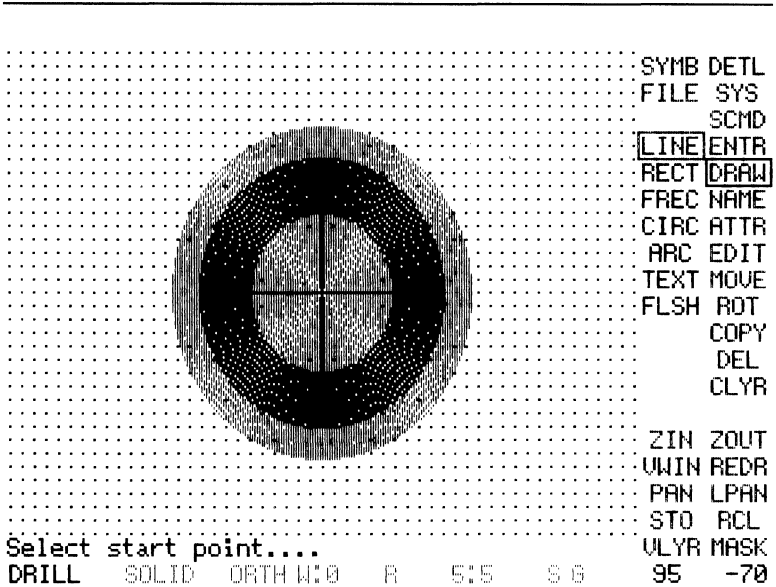
                                                        86147

**Figure 2-52. Lines Drawn to Represent
Center of Drill Hole**

Now you will add the drill size to the drawing.

27. Select DRAW/TEXT to add the drill size to the DRILL layer. The system prompts:

Select location. (Attributes OK?)...

28. First set the text parameters on the status line.

    a.   Set the text size (SIZ: on the status line) to 60.

    b.   Set the text justification and horizontal text alignment parameters on the status line to C for center.

    c.   Set the text orientation to the leftmost F on the status line. Compare your screen with Figure 2-53.
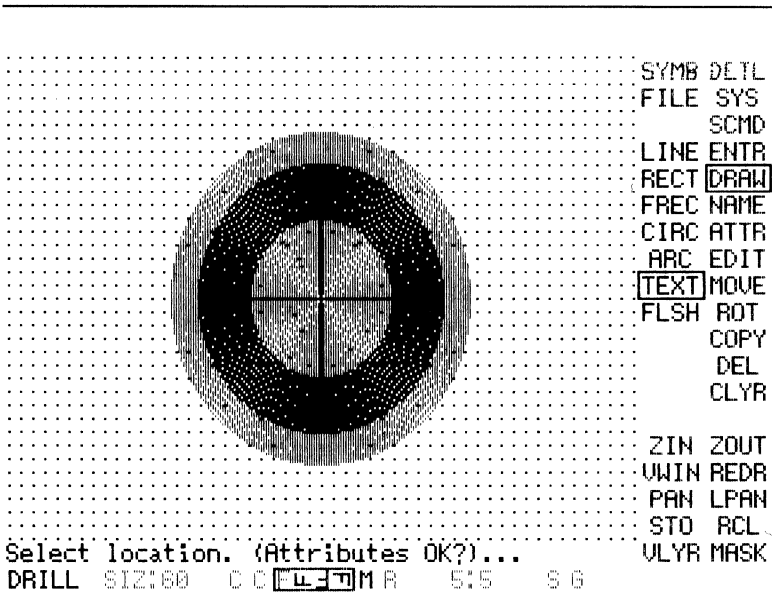
86148

Figure 2-53. Attributes of DRILL Layer Text Set

29. Select the center of the padstack for the location of the drill size text.

30. Type 32 and press [Return].

    The text will appear inside the circle. Redraw the screen and compare your screen to Figure 2-54.

SYMB DETL
FILE SYS
SCMD
LINE ENTR
RECT DRAW
FREC NAME
CIRC ATTR
ARC EDIT
TEXT MOVE
FLSH ROT
COPY
DEL
CLYR

ZIN ZOUT
VWIN REDR
PAN LPAN
STO RCL
VLYR MASK

Select location. (Attributes OK?)...
DRILL SIZ:60 C C F L J ¬ M R   5:5   S G
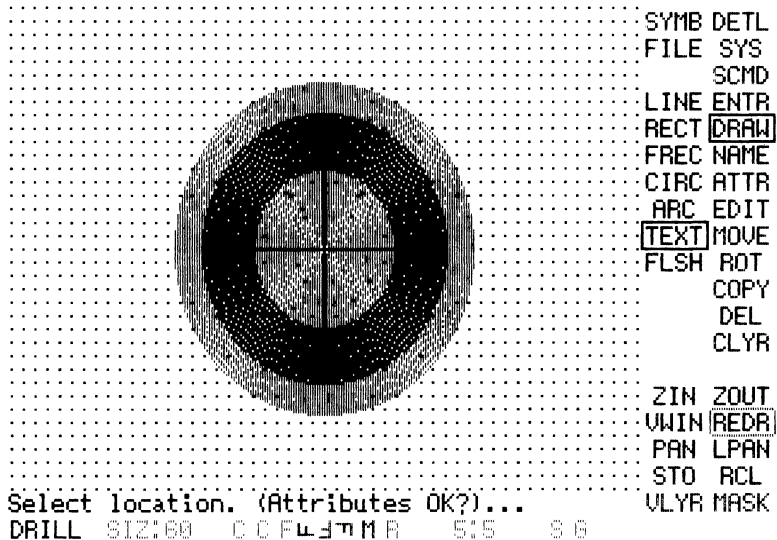
86149

Figure 2-54. Drill Size Added to Drill Layer

31. Select DRAW/CIRC.

32. Change the active layer to PIN.

33. Make sure the line width is set to 0.

34. Draw a circle 60 mils in diameter. Use 0 0 for the cursor location for the center of the circle and use 0 30 for the circumference. Your screen should look like Figure 2-55.
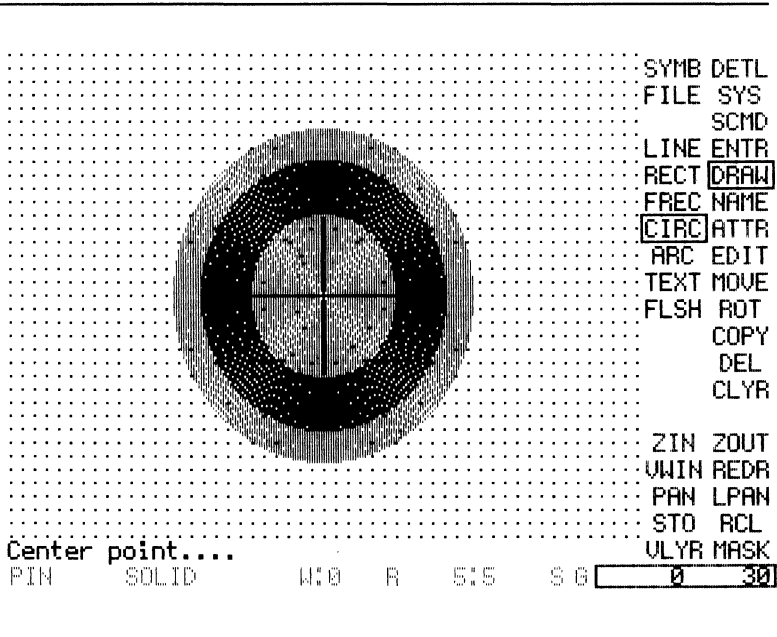
86150

Figure 2-55. Completed Padstack Graphics

You have now completed the graphics for the padstack. Next, you will enter the padstack flashes.

Drawing the Flashes

1. Select DRAW/FLSH to draw the flash. The system prompts:

   Select location....

2. Change the active layer to FLCOMP.

3. Select the aperture for the flash by placing the cursor over the APER: parameter on the status line and pressing Button 1. The system prompts:

   Enter aperture number:

4. Type 9 and press [Return]. The system prompts:

   Select location...

5. Place the cursor over the center of the padstack and press Button 1. The flash will appear in the center of the padstack as shown in Figure 2-56.

```
............................................................:SYMB DETL
............................................................:FILE SYS
............................................................:     SCMD
............................................................:LINE ENTR
............................................................:RECT DRAW
............................................................:FREC NAME
............................................................:CIRC ATTR
............................................................: ARC EDIT
............................................................:TEXT MOVE
............................................................:FLSH ROT
............................................................:     COPY
............................................................:     DEL
............................................................:     CLYR
............................................................:
............................................................:ZIN  ZOUT
............................................................:UWIN REDR
............................................................:PAN  LPAN
............................................................:STO  RCL
Select location....                                          ULYR MASK
FLCOMP   APER:9        FLJ┐    R    5:5    S 6         0        0
```
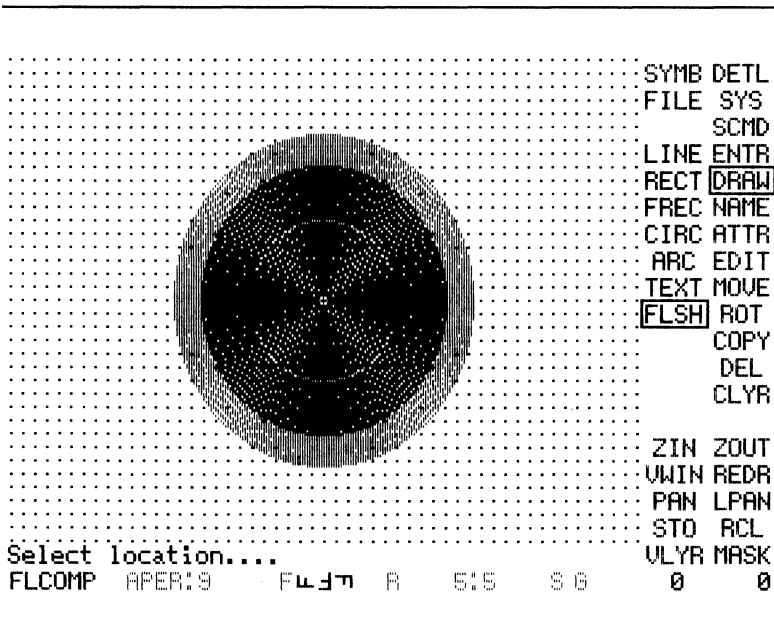
86151

Figure 2-56. Entering Flash on FLCOMP Layer

6. Change the active layer to FLSOLD.

7. Select an aperture setting of 9. Place a flash in the center of the padstack as you did on the FLCOMP layer.

8. Change the active layer to FLINT.

9. Select an aperture setting of 9. Place a flash in the center of the padstack.

10. Change the active layer to FLGCON.

11. Select an aperture setting of 8. Place a flash in the center of the padstack.

12. Change the active layer to FLCLER.

13. Select an aperture setting of 21. Place a flash in the center of the padstack.

14. Change the active layer to FLPCON.

15. Select an aperture setting of 8. Place a flash in the center of the padstack.

16. Change the active layer to FLSMSK.

17. Select an aperture of 11. Place a flash in the center of the padstack.

18. Change the active layer to FLDRILL.

19. Select an aperture of 23. Place a flash in the center of the padstack.

20. Redraw your screen.

    Compare your completed padstack with Figure 2-57.

SYMB DETL
FILE SYS
SCMD
LINE ENTR
RECT DRAW
FREC NAME
CIRC ATTR
ARC EDIT
TEXT MOVE
FLSH ROT
COPY
DEL
CLYR

ZIN ZOUT
VWIN REDR
PAN LPAN
STO RCL
VLYR MASK

Select location....

86152
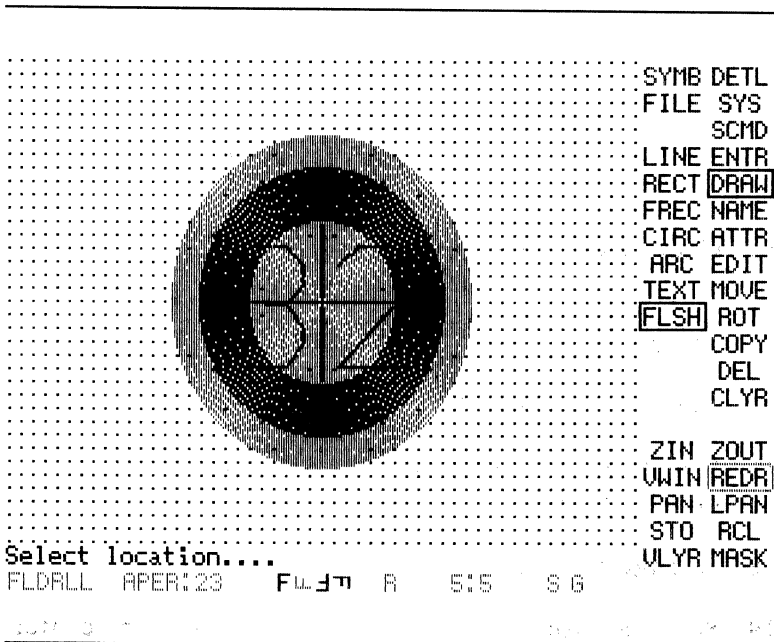
Figure 2-57. Completed Padstack

Assigning the Origin and Saving the Padstack File

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

    Select the origin....

2. Select the center point of the padstack.

3. Select FILE/SAVE to save your completed padstack in a file. Name the file N60R32CT.PS to represent the noncommitted, 60-mil round, 32-mil drill pad you just created. The "T" is for this tutorial.

The special symbol file you create in "Step 3. Creating the Special Symbol File" will insert the padstack graphics into a layout when linked by the SCMD/GSSF command.

4. Select FILE/ZAP to clear the screen.

5. Select SYS/QUIT to exit from PC-CARDS.

Step 3. Creating the Special Symbol File

In this step you will create the special symbol file shown in Figure 2-58. You can do this with or without a text editor. In "Step 4. Linking the Special Symbol File" you will link the special symbol file to your layout. The special symbol file will insert padstack graphics for all pins and vias according to their pin types.

```
%Special Symbol File
%
0    *    V50R28C.PS          %Feedthru Clearance
1    N    N60S32C.PS          %Noncommitted Square Clearance
1    C    C60S32C.PS          %Committed Square Clearance
2    N    N60R32C.PS          %Noncommitted Round Clearance
2    C    C60R32C.PS          %Committed Round Clearance
3    N    N60R32G.PS          %Noncommitted Round Ground
3    C    C60R32G.PS          %Committed Round Ground
4    N    N60R32P.PS          %Noncommitted Round Power
4    C    C60R32P.PS          %Committed Round Power
5    N    N60S32P.PS          %Noncommitted Square Power
5    C    C60S32P.PS          %Committed Square Power
6    N    N60S32G.PS          %Noncommitted Square Ground
6    C    C60S32G.PS          %Committed Square Ground
```

Figure 2-58. Special Symbol File

Figure 2-58 shows the format of the special symbol file.

The first column lists the pin types which have padstacks assigned to them. Pin type 0 is reserved for the default via padstack.

The second column lists the pin's connectivity status. A "C" indicates that the pin is committed to a net; an "N" indicates that the pin is not committed. An "*" indicates that the pin may be either committed or uncommitted. You can assign different padstacks to committed and uncommitted pins.

The third column lists the padstack file which is assigned to each combination of pin type and connectivity status.

> 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-CARDS.

If your system has a text editor or word processing program that you are familiar with, use it to create a file named TUTOR.SSF, and then type the information shown in Figure 2-58 into it.

When creating the special symbol file, you must enter the space(s) shown in Figure 2-58 using the space bar. Although several spaces are shown in the figure between type and status entries and between status and padstack entries, 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: TUTOR.SSF [Return]

2.   Type in the information exactly as shown in Figure 2-58.

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

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

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

Step 4. Linking the Special Symbol File

1.   Load the TUTORT.PCB file.

2.   Select SCMD/GSSF to link the special symbol file
to the layout. The system prompts:

      Enter file name:

3.   Type TUTORT.SSF and press [Return].

4.   Select REDR to redraw the screen. The screen
display should look like Figure 2-59.

```
SYMB DETL
FILE SYS
     SCMD
SNAT ENTR
GSSF DRAW
     NAME
     ATTR
     EDIT
     MOVE
     ROT
     COPY
     DEL
     CLYR
     SWAP
ZIN  ZOUT
VWIN REDR
PAN  LPAN
STO  RCL
VLYR MASK
```

No Active Command...

R   50:50   S G
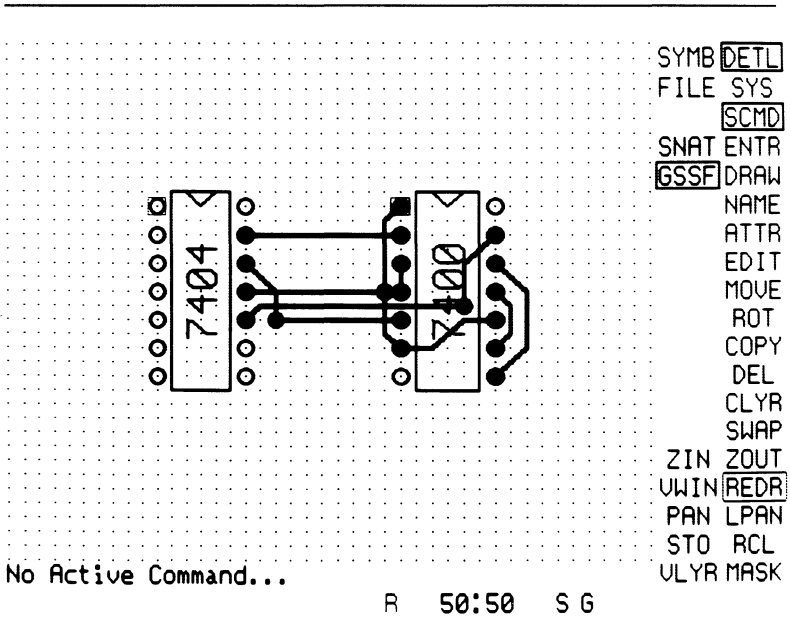
85182

Figure 2-59. Linking the Special Symbol File

5.   Select SCMD/GSSF again to unlink the special
     symbol file. Press [Return] this time instead of
     entering a filename.

Summary

You have now learned the essential steps in creating and using padstacks and special symbol files. Review the steps listed in Table 2-8 to check your understanding of the process.

Table 2-8. Creating a Padstack and a Special Symbol File

Step	Commands Used	Layer/File
Setting up the environment	SYMB ZIN VLYR	
Creating the padstack		
a. Drawing the graphics	DRAW/CIRC	PADCOM PADSLD PADINT GNDCON CLEAR PWRCON SLDMSK PIN
	DRAW/LINE	DRILL
	DRAW/TEXT	DRILL

Table 2-8 Continued

Step	Commands Used	Layer/File
b. Drawing the flashes	DRAW/FLSH	FLCOMP FLSOLD FLINT FLGCON FLPCON FLSMSK FLDRILL FLCLER
c. Assigning the origin	ENTR/ORG	
d. Saving the file	FILE/SAVE FILE/ZAP	N60R32C.PS
Creating the special symbol file		TUTOR.SSF
Linking the special symbol file	FILE/LOAD SCMD/GSSF	TUTORT.PCB TUTOR.SSF

Using PC-CARDS 2-123

EXAMPLE 5. CREATING A P-CAD COMPATIBLE COMPONENT

Components are the building blocks of circuit layouts designed with PC-CARDS. In this example, you will learn to create a TTL 7400 PCB part. The steps in the process are:

1. Set up the environment.
2. Enter the footprint.
3. Create the silkscreen graphics.
4. Enter device name.
5. Assign origin.
6. Add packaging information.
7. Save the component.

NOTE: For more information about the 7400 PCB part, refer to any TTL Data Book.

Step 1. Setting Up the Environment

Creating any drawing in PC-CARDS requires that you set up a working environment.

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

2. Select VLYR to display the layer screen and to set the layer assignments.

   Set PIN, SLKSCR, and DEVICE to ABL status and make PIN the active layer. Turn the rest of the layers off.

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

000-0052-06

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

    Select view center...

5. Move the cursor to the center of the screen and press Button 1 twice.

Step 2. Entering the Footprint

To use your part in a PCB layout, you must indicate where its pins are to be placed. The arrangement of pins on a part is called its "footprint."

1. Select ENTR/PIN to enter the pin connections. The system prompts:

    Select pin location:...

    Figure 2-60 shows the screen display.

```
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   SYMB  DETL
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   FILE  SYS
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         SCMD
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .    PIN  ENTR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   ORG   DRAW
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         NAME
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         ATTR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         EDIT
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         MOVE
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         ROT
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         COPY
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         DEL
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         CLYR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   ZIN   ZOUT
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   VWIN  REDR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   PAN   LPAN
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   STO   RCL
Select pin location: ...                                               VLYR  MASK
PIN     TYPE:0  EQUIV:0    R    50:50    S G
```

85183

**Figure 2-60. Screen Display After
Selecting ENTR/PIN Command**

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

PIN   PIN is the active layer.

TYPE:0   Any pins entered will be of type 0.
Table 2-9 lists the pin types used in
this example.

EQUIV:0   Any pins entered will be assigned a
logical equivalence of 0. 0 means not
swappable. Pins with the same
equivalence number, for example, 1
and 1 or 2 and 2, may be swapped.

R   Ratsnest display mode is turned off
(red).

50:50   Interval between working grid points
and cursor access interval is set to 50
DBUs (mils).

S   Screen grid is turned on (green).

G   Grid lock for the cursor is turned on
(green). When grid lock is on, the
cursor is forced to move in increments
specified by the X:Y grid spacing
assignment discussed above. When grid
lock is off, the cursor is free to move
in increments of 1:1 DBUs.

<mmm,   The cursor is located at the
nnn>   point <x=mmm, y=nnn>.

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

Table 2-9. Pin Types

Type	Meaning
0	Via (an electrical connection between layers)
1	Pin 1 only (usually square)
2	All gate pins except pin 1, ground, and power pins
3	Ground (#7)
4	Power (#14)

2. For each pin, you will be entering information from Table 2-10. The pin type determines what set of padstack graphics will represent the pin in the layout. The pin equivalence number indicates logical equivalence. A 7400 part consists of four 2-input NAND gates. The two inputs of each gate are logically equivalent. To indicate that pins are swappable, you must assign pins the same equivalence number. Pins with the same equivalence number (except 0) may be swapped.

Table 2-10. Pin Information

Pin	Type	Equivalence
1	1	1
2	2	1
3	2	0
4	2	2
5	2	2
6	2	0
7	3	0
8	2	0
9	2	3
10	2	3
11	2	0
12	2	4
13	2	4
14	4	0

3. Before you select the pin location, set the status line parameters for entering pins.

  a. Set the pin type for pin 1 by placing the cursor over the words TYPE:0 and pressing Button 1. The system prompts:

   Enter pin type number:

   Type 1 and press [Return].

b. Set the pin's logical equivalence by placing the cursor over the words EQUIV:0 on the status line and pressing Button 1. The system prompts:

Pin equivalence number:

Type 1 and press [Return].

Now that you have set pin layer, type, and equivalence, enter the first pin.

4. Select the point shown in Figure 2-61.


```
                                              SYMB  DETL
                                              FILE   SYS
                                                    SCMD
                                              PIN  ENTR
                     ⊕                        ORG  DRAW
                                                   NAME
                                                   ATTR
                                                   EDIT
                                                   MOVE
                                                    ROT
                                                   COPY
                                                    DEL
                                                   CLYR

                                              ZIN   ZOUT
                                              VWIN  REDR
                                              PAN   LPAN
                                              STO   RCL
Select pin name location. (Attrb OK?)...      VLYR  MASK
PIN    SIZ:125  C C FⅬ⅃⊓M R    50:50   S G     750   850
```

85184

Figure 2-61. Entering the First Pin

The system prompts:

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

The message "(**Attrb OK?**)" refers to the text
attributes on the status line, but for this example,
do not display the pin names. To prevent the pin
names from being displayed with the part, press
Button 2 or [Esc]. The system prompts:

Enter pin name:

5.   Type 1 and press [Return]. The pin name is
     recorded internally but is not displayed.

6.   Enter the other 13 pins in a U configuration,
     changing pin type and pin equivalence as needed
     using the pin type and equivalence information in
     Table 2-9, until all pins are entered as shown in
     Figure 2-62.

     NOTE: The order in which you enter pins is very
     important. The system recognizes the pin numbers
     by the order in which you enter the pins. If you
     make a mistake when you enter a pin, delete it
     using the DEL command and enter it correctly
     using the ENTR/PIN command if it was the last
     pin entered. If it wasn't the last pin entered, you
     must delete all the pins and start over.

Figure 2-62. 7400 PCB Part Footprint

Step 3. Creating the Silkscreen Graphics

In this step you will add the silkscreen graphics for the
7400 PCB part by creating an outline on the silkscreen
layer that looks like Figure 2-63. The outline consists
of a box with a V-shaped line at one end. P-CAD's
standard is to use a V-shaped line instead of an arc,
because an arc takes more time to plot.

85186

Figure 2-63. Graphics for 7400 PCB Part

Drawing the Box

1. Select DRAW/LINE. The system prompts:

    Select start point....

2. First set the status line parameters.

    a. Change the active layer to SLKSCR (silkscreen).

    b. Change the line width to 0 (W: on the status line).

    c. Change the line mode to ORTH.

Figure 2-64 shows the screen after status line parameters are set.


```
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  SYMB  DETL
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  FILE  SYS
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·        SCMD
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  LINE  ENTR
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  RECT  DRAW
· · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · · ·  FREC  NAME
· · · · · · · · · · · · ● · · · · · ● · · · · · · · · · · · · · · ·  CIRC  ATTR
· · · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · ·  ARC   EDIT
· · · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · ·  TEXT  MOVE
· · · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · ·  FLSH  ROT
· · · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · ·        COPY
· · · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · ·        DEL
· · · · · · · · · · · · · · ● · · · · · · ● · · · · · · · · · · · ·        CLYR
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  ZIN   ZOUT
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  VWIN  REDR
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  PAN   LPAN
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  STO   RCL
Select start point....                                             VLYR  MASK
SLKSCR  SOLID  ORTH A:0    R    50:50    S G  1050    250
```
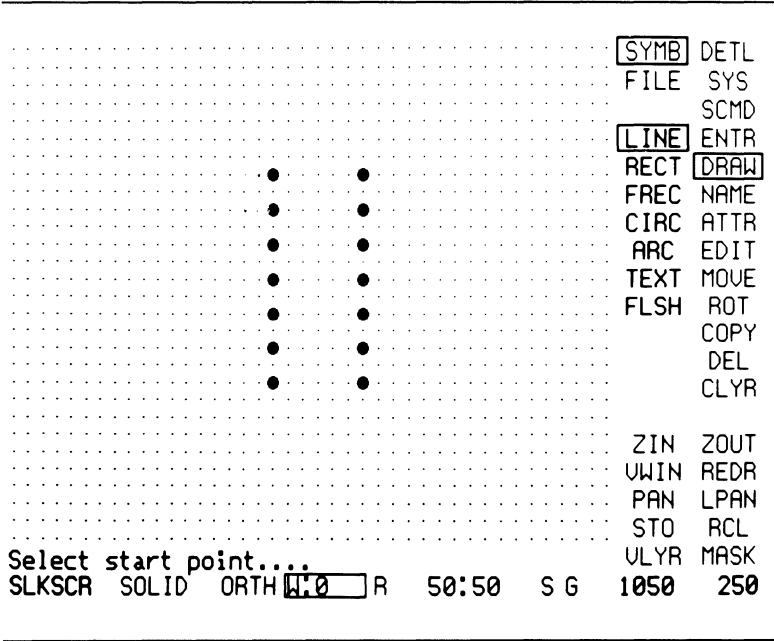
85187

Figure 2-64. Status Line Parameters Set

3. Select the point shown in Figure 2-65 to begin the
    line.

Figure 2-65. Starting Point of Line Selected

4. Select the next point shown in Figure 2-66 to draw
    the line. Notice that the line is drawn as you move
    the cursor.
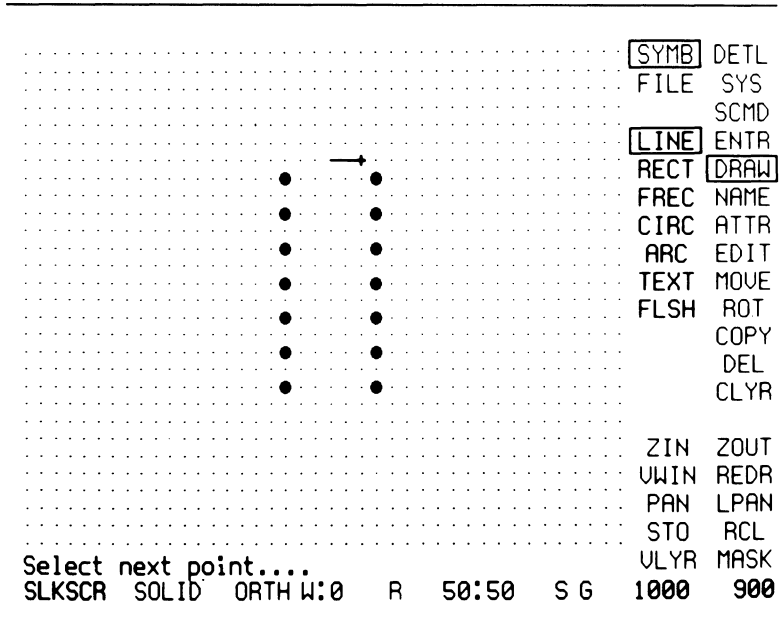
85189

Figure 2-66. First Line Segment of 7400 PCB Part

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

a. Select DEL.

b. Select a point on the line. The line will be
    deleted.

c. Select REDR to redraw the drawing.

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

5. Select the point shown in Figure 2-67 to draw the second line segment.

```
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   SYMB  DETL
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   FILE  SYS
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         SCMD
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   LINE  ENTR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   RECT  DRAW
. . . . . . . . . . . . . . . . . ●    ┐    ● . . . . . . . . . . . . . . .     FREC  NAME
. . . . . . . . . . . . . . . . . ●    │    ● . . . . . . . . . . . . . . .     CIRC  ATTR
. . . . . . . . . . . . . . . . . ●    │    ● . . . . . . . . . . . . . . .     ARC   EDIT
. . . . . . . . . . . . . . . . . ●    │    ● . . . . . . . . . . . . . . .     TEXT  MOVE
. . . . . . . . . . . . . . . . . ●    │    ● . . . . . . . . . . . . . . .     FLSH  ROT
. . . . . . . . . . . . . . . . . ●    │    ● . . . . . . . . . . . . . . .           COPY
. . . . . . . . . . . . . . . . . ●    ↓    ● . . . . . . . . . . . . . . .           DEL
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .         CLYR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   ZIN   ZOUT
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   UWIN  REDR
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   PAN   LPAN
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .   STO   RCL
Select next point....                                                          ULYR  MASK
SLKSCR  SOLID  ORTH W:0    R    50:50    S G   1000    200
```
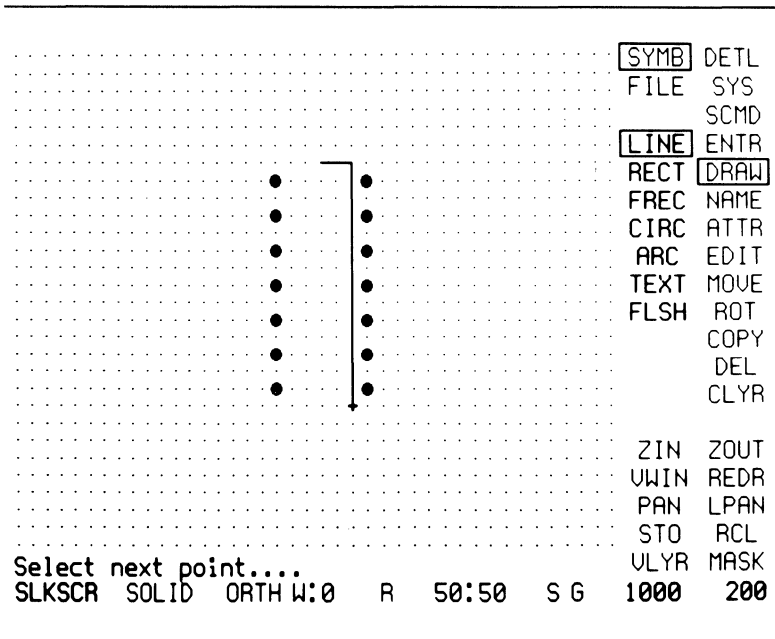
85190

Figure 2-67. Second Line Segment of 7400 PCB Part

6. Select the next three points to complete the
    rectangle as shown in Figure 2-68.

85191

Figure 2-68. Completed Box Part of 7400 PCB Part

7. Press Button 2 to end the line.

    NOTE: The box could also have been created with
    the DRAW/RECT (Draw/Rectangle) command.

Drawing the V-Shaped Line

1. Change the line mode to ANGL on the status line.

2. Select the point shown in Figure 2-69.
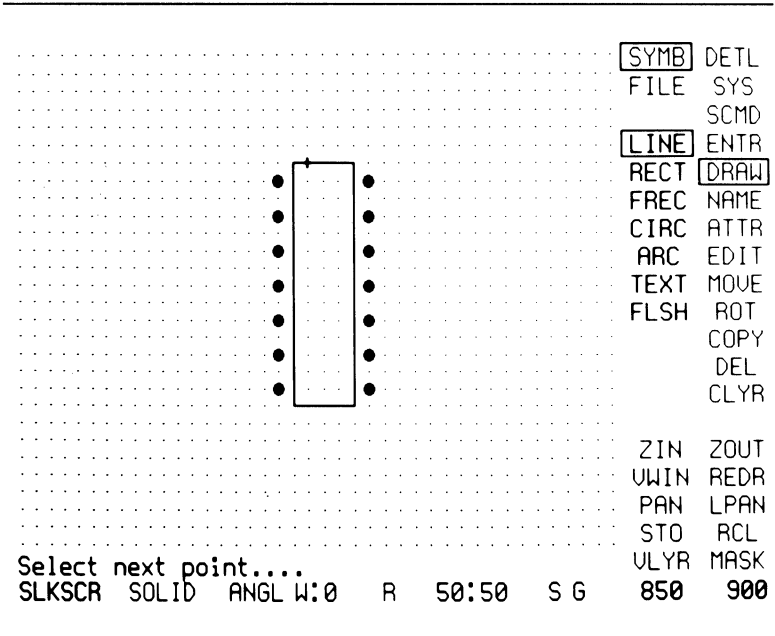
85192

Figure 2-69. Point for Start of V-Shaped Line

3. Select the next two points to complete the V-shaped
    line as shown in Figure 2-70.
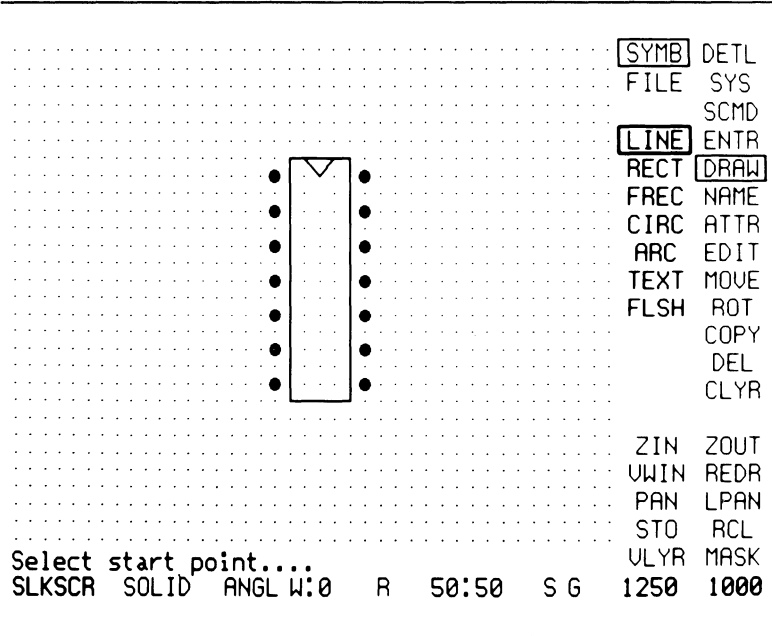
85193

**Figure 2-70. Completed V-Shaped Line
for 7400 PCB Part**

4. Select REDR to redraw the screen.

You have completed the symbol graphics for the 7400
PCB part.

Step 4. Entering the Device Name

1. Select DRAW/TEXT to add the device
    identification text.

2. First set the parameters for entering text on the status line:

    a. Change the active layer to DEVICE.

    b. Set the text size to 125.

    c. Set the text orientation to the second F on the status line.

    d. Set the text justification and horizontal text alignment parameters on the status line to C for center.

3. Select the point for placing text in the center of the component as shown in Figure 2-71.
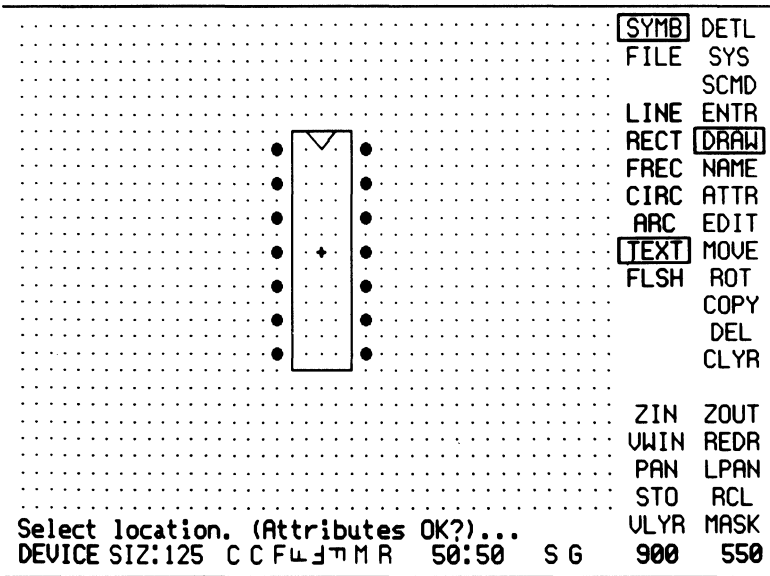
Figure 2-71. Point for Entering Text

4.   Type 7400 and press [Return]. Notice that the text
     enters at the center point and is automatically
     centered as shown in Figure 2-72.
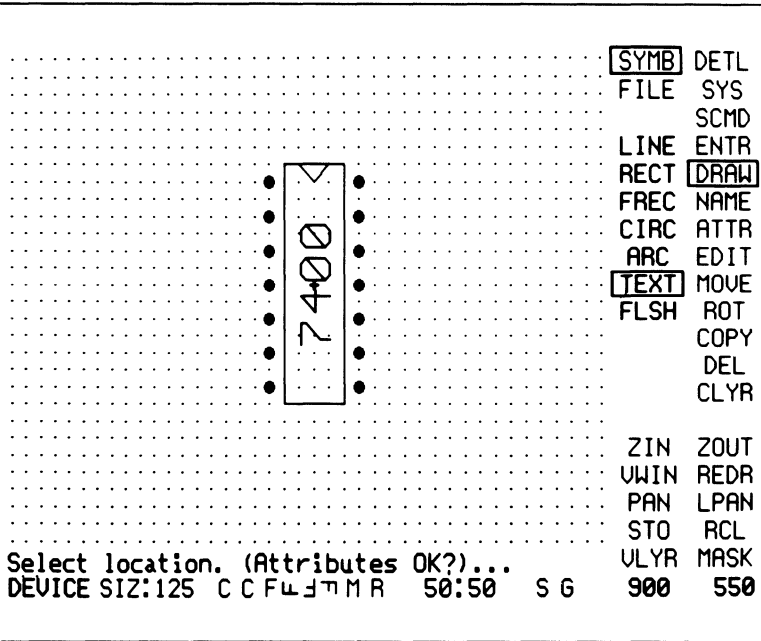
85195

Figure 2-72. 7400 Part With Text

Step 5. Assigning the Origin

The origin is the reference point used when you place
the component in a circuit layout. You can choose any
point in the drawing as an origin. P-CAD's convention
is to use pin number 1.

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

   Select the origin

2. Select pin number 1 as shown in Figure 2-73.


```
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·  [SYMB]  DETL
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·   FILE   SYS
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·           SCMD
· · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · · ·    PIN   [ENTR]
· · · · · · · · · · · · · · · · ·  ▽  · · · · · · · · · · · · · · · · · · ·   [ORG]   DRAW
· · · · · · · · · · · · · · · →•       •· · · · · · · · · · · · · · · · · ·           NAME
· · · · · · · · · · · · · · · ·•       •· · · · · · · · · · · · · · · · · ·           ATTR
· · · · · · · · · · · · · · · ·•  ⦸    •· · · · · · · · · · · · · · · · · ·           EDIT
· · · · · · · · · · · · · · · ·•  ⦸    •· · · · · · · · · · · · · · · · · ·           MOVE
· · · · · · · · · · · · · · · ·•  ᘜ    •· · · · · · · · · · · · · · · · · ·           ROT
· · · · · · · · · · · · · · · ·•  ᗡ    •· · · · · · · · · · · · · · · · · ·           COPY
· · · · · · · · · · · · · · · ·•       •· · · · · · · · · · · · · · · · · ·           DEL
· · · · · · · · · · · · · · · ·•       •· · · · · · · · · · · · · · · · · ·           CLYR
```

Select the origin....   R   50:50   S G

ZIN	ZOUT
VWIN	REDR
PAN	LPAN
STO	RCL
VLYR	MASK
750	850

85196

**Figure 2-73. Screen Display After
Selecting ENTR/ORG**

Step 6. Adding Packaging Information

In "Step 2. Entering the Footprint," you entered pin
types and equivalences when you entered the pins. In
this step, you will add packaging information. The
packaging information and the pin types and
equivalences add "intelligence" to the 7400 PCB part.

1. Select ZIN to zoom in. (Place the cursor in the
    center of the screen.) The screen should look like
    Figure 2-74.
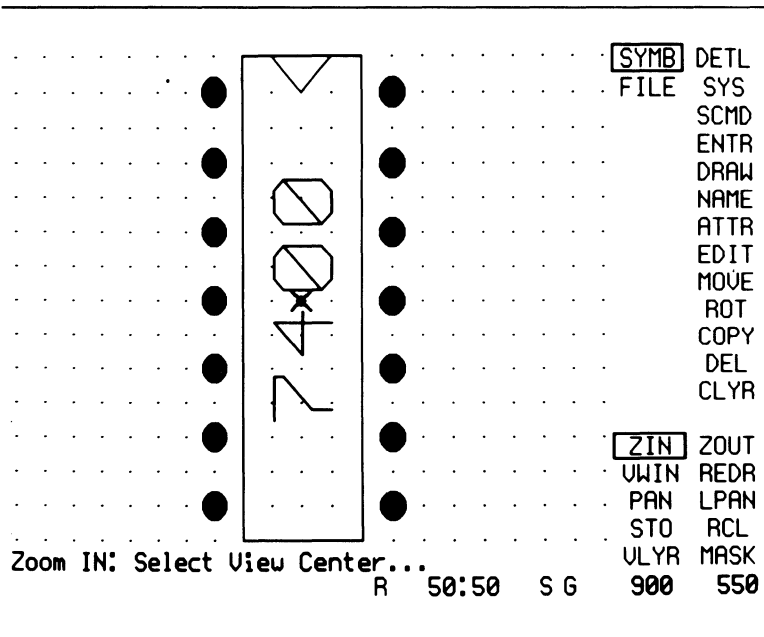
85197

Figure 2-74. 7400 Part After Selecting ZIN

First, you will assign a Component Type ID to the
7400 part. This information is used by utility
programs such as PC-PACK, which groups
components for placement on the PCB. Table 2-11
lists P-CAD's standard Component Type IDs.

Table 2-11. P-CAD Component Type IDs

Type	Number
DIP	10000
Resistors	11100
Capacitors	11200
Inductors	11300
Transistors	11400
Other Discretes	11000
Connectors	12000
Miscellaneous	13000

2. Select SCMD/SCAT (System Command/Set
   Component Attribute) to assign the Component
   Type ID.

   The screen will display the present Component
   Type ID assigned to your 7400 PCB part. Enter the
   new type.

3. Type 10000 and press [Return].

   If you would like to check the pin types and
   equivalences you entered in "Step 2. Entering the
   Footprint," select SCMD/SPAT (System
   Command/Set Pin Attribute).

PC-CARDS will highlight each pin in turn starting with pin 1, display the pin type and equivalence of each pin, and prompt for a new pin type and equivalence.

If you worked through Step 2, the pin types and equivalences should be correct. You can verify this by comparing the values displayed with those given in Table 2-10. If the pin type or pin equivalence values are incorrect, enter the correct values from Table 2-10. If correct, select Button 2 or [Esc] to see the next value.

NOTE: Once you start using the SPAT command, there is no way to exit except by going all the way through every pin in the component.

4. Select SCMD/SPKG (System Command/Set Packaging) to enter packaging information about the 7400 part. The system prompts:

    Enter number of gates:

5. Since a TTL 7400 PCB part consists of 4 NAND gates, type 4 and press [Return]. The system prompts:

    Enter number of pins per gate:

6. Since each gate in a TTL 7400 PCB part consists of 2 input pins and 1 output pin, type 3 and press [Return]. The system prompts:

    Enter name of gate pin 1:

7. Type INA and press [Return].

    NOTE: If you are using PC-CARDS in conjunction
    with P-CAD's schematic capture program, PC-CAPS,
    when you assign packaging, the pin names assigned
    to the component pin must correspond to the pin
    names of the symbols used in the schematic.

8. Repeat step 7 for pins 2 and 3. Assign them the
    names INB and OUTY, respectively. PC-CARDS
    then prompts:

    Select gate 1 pin INA....

9. Place the cursor on pin 1 and press Button 1. The
    system prompts:

    Select gate 1 pin INB....

10. Repeat the step above for pins INB and OUTY of
    gate 1 and pins INA, INB, and OUTY of gates 2, 3,
    and 4, selecting the appropriate pins as given in
    Table 2-12.

Table 2-12. Gate Pin Locations

Gate	Pin Name	Component Pin
1	INA	1
1	INB	2
1	OUTY	3
2	INA	4
2	INB	5
2	OUTY	6
3	INA	9
3	INB	10
3	OUTY	8
4	INA	12
4	INB	13
4	OUTY	11

NOTE: Pins 7 and 14 are the ground and power pins, respectively. Ground and power pin connections were defined by the pin type (see Table 2-10).

Step 7. Saving the File

Adding the packaging information was the final step in building the 7400 PCB part. Now you will save and name the component.

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

    Enter file name:

2. Type 7400 and press [Return].

   The system creates the file 7400.PRT containing the PCB part that you created in this example.

3. Select FILE/ZAP to clear the screen.

Summary

You have now learned some basic steps in creating and saving a component. Review the steps below to check your understanding of the process.

Table 2-13. Creating a Component

Step	Commands Used	Layer/File
Setting up the environment	SYMB VLYR	
Entering the footprint	ENTR/PIN	PIN
Creating the symbol graphics	DRAW/LINE REDR	SLKSCR
Entering the device name	DRAW/TEXT	DEVICE
Assigning the origin	ENTR/ORG	
Adding component information	SCMD/SCAT SCMD/SPKG	
Saving the file	FILE/SAVE FILE/ZAP	7400.PRT

CHAPTER SUMMARY

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

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

CHAPTER 3. COMMANDS

This chapter is a command reference for PC-CARDS.
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-CARDS 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
descriptions 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-CARDS 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 on keyboard commands, refer to the specific command description in this chapter.

FUNCTION KEYS

The ten function keys on the keyboard provide quick access to 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 while using the ENTR/WIRE or the ENTR/RATN 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 while using a text command.

Key	Function

[F6]   **TEXT ORIENTATION** - Cycles through the text orientation choices. The current text orientation is shown by the green F on the status line.

[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 on and off. 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**   **ATTR/ACOM**
     DETL
             (Attribute/Add Component Attribute)

FUNCTION:

Adds an attribute to a component.

REMARKS:

Attributes (or parameters) can be added to a component in the SYMB mode, or to a single instance of a component in the DETL mode. Attributes added in one mode can only be edited in the same mode.

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

    GND_PIN=7

    DIST="(AMPLE Electronics)"

A keyword must start with a letter and must be no more than 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.

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

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

Status line for ATTR/ACOM:

Layer	Text Size	Text Justifi- cation	Text Orien- tation	Mirror
ATTR	SIZ:125	L B	F	M

ATTR/ACOM

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.

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 selecting another command, the attribute may appear to move.

ATTR/ACOM

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 " or (, but does not end with " or).

Action: End the value with " 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: Re-enter 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 can only be changed in the same mode. For example, an attribute added to a component 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 for details on entering values.

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

OPERATION:

1. Select ATTR/SCHG. The system prompts:

    Select one attribute....

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.

ATTR/SCHG

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 mode (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.

                                                        (

ATTR/SCHG

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   Keyboard Command: /CFIL
       DETL
                                    (Command Log File)

_____

FUNCTION:

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

REMARKS:

PCCARDS.CMD is the filename used by the system to
record your commands during the current session. The
PCCARDS.CMD file is created automatically when you
start PC-CARDS.

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-CARDS, the command log file is
renamed from PCCARDS.CMD to PCCARDS.CM$. Any
existing PCCARDS.CM$ 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 to 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 number of objects within a window that can be copied depends on how much memory is available on your system after PC-CARDS 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.

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 active layer parameter is active.

3. Select the desired object. The object transfers to the active layer, taking on the color of the new layer. 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.

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 parameters 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 transfer 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 the other objects you want to transfer in the
   same manner.

5. Press Button 2 to finish selecting objects. 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 depends on how much memory is available
on your system after PC-CARDS 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.

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 parameters 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.

000-0052-06

CLYR/WIN

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/WIN:

Layer
DEVICE

OPERATION:

1. Select CLYR/WIN. The system prompts:

    Corner 1....

2. Make sure that the desired layer is active.

3. Move the cursor to the desired location and 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 to be transferred 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   **COPY**
      SYMB

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
subcommands WIN and IDEN 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
are selectable regardless of layer visibility status.

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.

Use the MASK command to prevent copying components
or wires.

COPY

OPERATION:

1. Select COPY. The system prompts:

    Select the object(s)....

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 make 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.

COPY

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

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: The database limit for the maximum number of components or pins has been reached.

Action: Redesign the PCB so that not as many components are used in the database.

COPY/IDEN

Message: **Exceeded limit for number of nets.**

Cause: The database limit for the maximum number of nets has been reached.

Action: Redesign your PCB so that not as many nets are used in the database.

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/WIN

(Copy/Window)

FUNCTION:

Copies a number of objects intersected by and/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-CARDS 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.

COPY/WIN

When copying a net, the net name is copied only if the net name and the entire net is 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.

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....

COPY/WIN

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.

ERROR MESSAGES:

Message: Can not copy net.

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

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: The database limit for the maximum number
          of components or pins has been reached.

COPY/WIN

Action: Redesign the PCB so that not as many
components are used in the database.

Message: **Exceeded limit for number of nets.**

Cause: The database limit for the maximum number
of nets has been reached.

Action: Redesign your PCB so that not as many nets
are used in the database.

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 object is not masked, and make
sure you are in the same mode in which the
object was entered, position the cursor, and
try again.

COPY/WIN

Message: No source net.

Cause: Program error detected in copying net.

Action: Contact P-CAD customer support and save the database, the PC-CARDS.CMD file, and the PCCARDS.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 corresponding pin(s) in the copied window.

Action: None.

Message: Shorting more than one net.

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

Action: Uncommit the pins that are hit before copying the wire, or enter the wire with the ENTR/WIRE command.

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 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. However, 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 SYMB 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.

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

DEL

After deleting an item, redrawing the screen highlights the deleted item until another command is selected. Selecting a different command causes the deleted item to 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.

    CAUTION: Deleting a wire deletes the entire net and its associated ratsnest connection, including any portions of the net that are not directly connected to the wire. Use the EDIT/DELS command to delete individual wire segments. ENTR/UCOM uncommits pins from a net without deleting the net itself.

DEL

ERROR MESSAGES:

Message: No object found.

Cause: The system cannot find an object.

Action: Make sure you are in the correct 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 command, 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.

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/UNDO

(Delete/Undo)

FUNCTION:

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

REMARKS:

DEL/UNDO restores items deleted with the DEL
command alone, provided 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 DEL/WIN command deletes ratsnest connections that are completely enclosed 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-CARDS 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.

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

The MASK command does not protect wires from DEL/WIN.

DEL/WIN

OPERATION:

1. Select DEL/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 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.

DEL/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, position the cursor,
and try again.

Mode:

DETL

(Detail)

FUNCTION:

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

REMARKS:

This mode is used for creating and editing layout designs and documentation drawings or charts.

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

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

Components created in SYMB command mode can be entered as components in DETL command mode. Once a component is entered, it is treated as a single entity.

OPERATION:

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

Mode: SYMB   **DRAW**
      DETL

FUNCTION:

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

OPERATION:

Select DRAW and one of following submenu choices:

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

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=12. You can specify from 0 to 250 database units.

Status line for DRAW/ARC:

Layer	Line Type	Line Width
SLKSCR	SOLID	W:2

OPERATION:

1. Select DRAW/ARC. The system prompts:

   Center point:...

DRAW/ARC

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

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 from the starting angle to the ending angle in a counter-clockwise direction.

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=12. You can specify from 0 to 250 database units.

Status line for DRAW/CIRC:

Layer	Line Type	Line Width
SLKSCR	SOLID	W:2

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/FLSH

(Draw Flash)

FUNCTION:

Places photoplot aperture flashes on a component, part, or in a database.

REMARKS:

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

- The layer the flash will be drawn on.

- The aperture to flash (1 - 65). The system allows the specification of up to 65 different apertures for the GTCO photoplotter. For the Gerber photoplotter, use flash apertures from 1 to 24.

Status line for DRAW/FLSH:

Layer	Flash Aperture	Text Orientation
SLKSCR	APER:65	F

DRAW/FLSH

OPERATION:

1. Select DRAW/FLSH. The system prompts:

    Select location....

2. Change the status line parameters as needed.

3. Position the cursor and press Button 1. The system places the flash.

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-PRINT, PC-PLOTS,* and *PC-PHOTO User's Manuals* 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
SLKSCR	SOLID	W:2

DRAW/FREC

OPERATION:

1. Select DRAW/FREC. The system prompts:

   Corner 1...

   Corner 1 is the first of two diagonally opposite corners.

2. Select the first corner. The system prompts:

   Corner 2...

3. Select the second corner. The system draws the filled rectangle and prompts again for corner 1.

4. To draw another rectangle, repeat steps 2 and 3.

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° angles), ANGL (for any angle), or 45D (for 45° angles).

- The line width in effect. The default is W=12. You can specify from 0 to 250 database units.

Status line for DRAW/LINE:

Layer	Line Type	Line Mode	Line Width
SLKSCR	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 your 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 selecting points by pressing Button 1, or press Button 2 to terminate selection of points and end the line.

    NOTE: To delete part of a line just drawn, trace the cursor backwards along the line 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=12. You can specify from 0 to 250 database units.

Status line for DRAW/RECT:

Layer	Line Type	Line Width
SLKSCR	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 drawn on.

- The current text size in effect. [F5] allows you to
  change the text size.

- The horizontal justification in effect. L, R, or C
  stands for left, right, or center justification.

- The vertical justification in effect. B, T, or C
  stands for bottom, top, or center justification.

- The current text orientation as indicated by the
  green F.

- The current status of the mirror parameter. A
  green M indicates that mirroring is on; red indicates
  off. 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,
  PC-PLOTS, and PC-PHOTO.

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	Size	Text Justifi- cation	Orien- tation	Mirror
SLKSCR	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].

Mode: SYMB   **EDIT**
       DETL

FUNCTION:

Enables the submenu for editing vertices and segments
of lines and wires as well as for editing vias (feedthrus).

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:

● In SYMB mode:

   ADDV (Add Vertex)
   DELV (Delete Vertex)
   MOVV (Move Vertex)
   LAYS (Change Segment Layer)
   DELS (Delete Segment)
   MOVS (Move Segment)

EDIT

- In DETL mode:

    ADDV (Add Vertex)
    DELV (Delete Vertex
    MOVA (Move All)
    MOVV (Move Vertex)
    LAYS (Change Segment Layer)
    DELS (Delete Segment)
    MOVS (Move Segment)
    AVIA (Add Via)
    DVIA (Delete Via)
    MVIA (Move Via)

Mode: DETL   **EDIT/ADDV**
      SYMB
                                  (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 T-junctions are 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 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   **EDIT/AVIA**

                                    (Edit/Add Via)

FUNCTION:

Adds a via to a vertex of an existing wire.

REMARKS:

Vias can be added to wires only. Vias cannot be added
to lines drawn with the DRAW/LINE command.

OPERATION:

1. Select EDIT/AVIA. The system prompts:

    Select one vertex....

2. Select a vertex on the desired wire. The system
    highlights the wire and places a circle representing
    a via.

3. Continue to select new vertices for vias until you
    are done.

ERROR MESSAGES:

Message: No vertex found.

Cause: The system cannot add a vertex where the
          cursor was positioned.

Action: Reposition the cursor, and try again. If
          necessary, zoom in for greater accuracy.

Mode: DETL   **EDIT/DELS**
       SYMB
                         (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.

When deleting a wire, any attached via will be automatically deleted if fewer than two different layers are associated with the via.

Vertices are automatically deleted when T-junctions are broken.

> NOTE: Although you may delete a wire or a segment of a wire, the ratsnest connection is not deleted if the netlist is frozen. To uncommit a pin, you can use either the /SGAT command to open the netlist before deleting the segment leading to the committed pin, or you can use the ENTR/UCOM command to remove the connectivity of a pin after deleting the segment. The ENTR/UCOM command can uncommit a pin whether the netlist is frozen or not.

> The MASK command does not protect wires from the EDIT/DELS command.

EDIT/DELS

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.

   In the case of wires, this operation 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.

EDIT/DELS

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.

When deleting a vertex, any attached via will be automatically deleted if fewer than two different layers are associated with the via.

Vertices are automatically deleted when T-junctions are 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 where the cursor was positioned.

Action: Reposition the cursor, and try again. If necessary, zoom in for greater accuracy.

Mode: DETL   **EDIT/DVIA**

                                            (Edit/Delete Via)

___

FUNCTION:

Deletes a via.

OPERATION:

1. Select EDIT/DVIA. The system prompts:

    Select one via....

2. Select the via to delete. The via is deleted.

ERROR MESSAGES:

Message: No via found.

Cause: The system cannot find a via where the cursor
          was positioned.

Action: Reposition the cursor in the center of the via,
          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.

When changing the layer a wire is on, vias will be automatically inserted and deleted as needed.

Status line for EDIT/LAYS:

Layer
DEVICE

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 vias 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 vias. The system prompts:

    Select new position....

3. Move the cursor to the desired new location. The affected segments and vias will be redrawn at the new location.

4. Select the final 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 part of a wire connected to a pin, then the pin can become graphically unconnected; however, the pin stays connected to the net internally. To uncommit a pin, use ENTR/UCOM.

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....

EDIT/MOVV

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.

ERROR MESSAGES:

Message: No vertex found.

Cause: The system cannot add a vertex where the cursor was positioned.

Action: Reposition the cursor and try again. If necessary, zoom in for greater accuracy.

Mode: DETL   **EDIT/MVIA**

(Edit/Move Via)

FUNCTION:

Moves a via.

OPERATION:

1. Select EDIT/MVIA. The system prompts:

    Select a via....

2. Select the via to move. The system prompts:

    Move the via to....

3. Move the cursor to the new via position and press
    Button 1. The system moves the via to the new
    location and continues to prompt for the selection
    of a via until another command is selected.

ERROR MESSAGES:

Message: No via found.

Cause: The system cannot find a via where the cursor
           was positioned.

Action: Reposition the cursor in the center of the via,
           and try again.

Mode: SYMB
      DETL

ENTR

(Enter)

FUNCTION:

Enables the submenu for entering pins, origins, components, wires, ratsnests, and for uncommitting pins.

OPERATION:

Select ENTR to enable the following submenu choices:

- In SYMB Mode:

    PIN
    ORG   (Origin)

- In DETL Mode:

    COMP   (Component)
    WIRE
    RATN   (Ratsnests)
    UCOM   (Uncommit a Pin)

Mode: DETL   **ENTR/COMP**

(Enter/Component)

FUNCTION:

Enters components into a design.

REMARKS:

A component is an entity that has been previously
created in SYMB mode and saved in a library.

A component that is entered mirrored will have any
associated text mirrored also.

A DOS path up to 39 characters long may be specified
for the component filename. For example:

C:\PCAD\PRT\TTL\7404.PRT

Each component may be given a unique name (from 1 to
8 characters) with the [F4] key at the time it is entered.
This name can uniquely identify the component to
distinguish it from other components, or placements, of
the same graphic entity.

A component may also be named later with the
NAME/COMP command.

Only one version of a component with the same
filename can be used in a PCB layout.

ENTR/COMP

Status line for ENTR/COMP:

Comp Name	Orien- tation	Mirror
|___|	F	M

OPERATION:

1. Select ENTR/COMP. The system prompts:

    :
    Comp-file-name

2. Type the component filename, and press [Return] or Button 1 to confirm the filename.

    NOTE: You don't need to type a filename extension; the system automatically adds the .PRT filename extension. However, if you type a period (.) after the filename with no extension, the system assumes the filename has no extension.

    The system prompts:

    Select loc to place comp. (Orientation OK?)

    To name the component, press [F4] or move the cursor to the status line and to the left of the text orientation until you see an empty rectangle. Press Button 1. The system prompts for a component name.

ENTR/COMP

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 is not visible until you use the NAME/COMP command.

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 that location.

5. To place 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 5.

ERROR MESSAGES:

Message: Component has no symbol.

Cause: The specified component has no data defined in SYMB mode.

Action: Check the spelling of the component filename. Edit the symbol file and check the data definition against "Example 5. Creating a P-CAD Compatible Component" in Chapter 2, "Using PC-CARDS."

ENTR/COMP

Message: **Database format not compatible.**

Cause: An incorrect 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 and then do a DIR (Directory) to verify the existence or spelling of a filename.

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

Cause: The database limit for the maximum number of components or pins has been reached.

Action: Redesign the PCB so that not as many components are used in the database.

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.

ENTR/COMP

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: **Path too long.**

Cause: The path and filename for the specified database overflows the internal filename buffer.

Action: Shorten the path in the PC-CARDS Configuration Screen.

Message: **Scale Factor = 0.**

Cause: Invalid filename entered.

Action: Enter a valid filename.

Mode: SYMB

ENTR/ORG

(Enter/Origin)

FUNCTION:

Enters or sets the origin point of a component.

REMARKS:

When a component is entered into a design, the cursor shows at the origin. The origin serves as the 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 pins line up on grid points when a component is placed.

> NOTE: All P-CAD supplied library parts have the origin at pin 1.

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 at this time until you enter the component with the ENTR/COMP command.) The origin is entered.

ENTR/ORG

If Button 2 is pressed instead of 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 an origin point by pressing 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.

REMARKS:

A pinname must be assigned in order to enter the pin into the component. However, you may make the pin name invisible.

> CAUTION: Leaving the pinname blank by pressing the space bar is not allowed by the system.

Certain programs, such as the PC-PACK utility program, may require that the pins be entered in a particular sequence.

When entering pins, the status line lets you specify the pin type, which is the cross-reference necessary for correct padstack selection and placement. (See the SCMD/GSSF command description). A pin type of 0 is reserved for the system default for vias and should not be used in components.

In addition, the logical equivalence (EQUIV) on the status line specifies logical pin equivalence for swapping. (See the SWAP/PIN command description).

The SCMD/SPAT command can be used to verify the pin sequence and pin type. This command can also be

ENTR/PIN

used to correct the pin type. If the pin sequence is wrong, delete and re-enter the pins in the correct sequence.

Grid spacing should be compatible with that of the part or component dimensions. Grid lock should also be on. Grid lock ensures that pins are entered on grid points and that wires can be connected to them.

The maximum number of pins allowed per part is 255.

Status line for ENTR/PIN:

Layer	Pin Type	Equivalence
PIN	TYPE:24	EQUIV:127

OPERATION:

1. Select ENTR/PIN. The system prompts:

    Select pin location...

ENTR/PIN

2. To change the pin type, move the cursor to the
    status line and select TYPE:<n>. The system
    prompts:

    Enter pin type:

    Type a number for the pin type. The range is from
    0 to 25.

    To change the logic equivalence of the pin, select
    EQUIV:<n>. The system prompts:

    Pin equivalence number

    Type a number for the equivalence. The range is
    from 0 to 127.

3. Set the active layer as needed.

4. Select the pin location. The system places a circle
    at the pin location and prompts:

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

    The status line changes to provide text features.

5. Select text parameters on the status line as needed.

ENTR/PIN

6. Select the pinname location with the cursor and press Button 1 to make the pin name visible. To make the pinname invisible, press Button 2.

    When you press Button 2, the pinname is associated with the pin connection even though the name is not visible. The system prompts:

    Enter pin name.

7. Type the pinname and press [Return] or Button 1.

    Text entered in lower case changes to upper case after redrawing the screen.

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: Exceeded limit for number of nets.

Cause: The database limit for the maximum number of nets has been reached.

Action: Redesign your PCB so that not as many nets are used in the database.

ENTR/PIN

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

Cause: The database limit for the maximum number of components or pins has been reached.

Action: Redesign the PCB so that not as many components are used in the database.

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/RATN

(Enter/Ratsnest)

FUNCTION:

Enters ratsnests into a design, connecting 2 or more component pins together by the shortest "airline" distance between them. This command establishes connectivity between the selected pins.

REMARKS:

Ratsnests are considered to have electrical properties. They are handled differently than lines or wires and cannot be edited as graphics.

Connectivity is established when one pin connects to another pin.

Connecting component pins with ratsnests establishes nets.

When a ratsnest is connected to a pin, the pin graphics change from a hollow pad to a filled pad, thus indicating an electrically committed pin.

Ratsnests may be given explicit net names or left unnamed.

ENTR/RATN

OPERATION:

1. Select ENTR/RATN. The system prompts:

   Select a pin....

2. Select the first pin.

   The active net name displays on the status line. If the first pin is unconnected, the status line shows "unnamed" for the net name.

3. To change the active net name, place the cursor over the net name on the status line and press Button 1.

   The active net name can also be changed with the [F3] key. The system prompts:

   Net name:

   and displays the active net name following the prompt.

   To assign a new name, type the new name and press [Return] or Button 1. To unname, press Button 2.

4. Move the cursor to the next pin for the connection. Press Button 1. A ratsnest segment is established. The system prompts:

   Select a pin....

ENTR/RATN

5.   To add additional pins to the ratsnest, repeat step 4.
     To end the ratsnest, press Button 2.

     When an existing ratsnest is picked, the system
     highlights it and displays the active net name.

     The new ratsnest segment then merges with the
     existing wire and acquires the active net name.

     When you attempt to connect one existing ratsnest
     with another existing ratsnest and either of them is
     unnamed, the system highlights the first and then
     the second. The system prompts:

     Merge the nets? YES NO.

ENTR/RATN

To merge the ratsnests, select YES. To prevent a merge, select NO or press Button 2.

If both ratsnests are named, the system responds:

More than one named net. No merge.

ERROR MESSAGES:

Message: **Both nets named. Merge not allowed.**

Cause: An attempt was made to merge two nets that are assigned names.

Action: Do not merge the nets, or unname one of the two nets and then merge the nets.

Message: **Net exists. Name change not allowed.**

Cause: An attempt was made to change the net name for a ratsnest to a name of an existing net.

Action: Use a different name when changing the net name or merge the two nets using the ENTR/RATN or ENTR/WIRE command.

ENTR/RATN

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.

.

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 or a ratsnest.

REMARKS:

Committed pins show as filled circles. Uncommitted
pins appear as empty circles.

The ENTR/UCOM command uncommits pins from nets
and ratsnests. This command overrides and removes
connectivity established from a schematic netlist as
assigned through the PC-PACK utility program.

This command also overrides the protection afforded by
freezing the netlist with the /SGAT command. (See the
/SGAT command description.)

When editing wires, you may delete a wire with the
EDIT command. However, the ratsnest connection
remains. To remove a pin from a net, use the
ENTR/UCOM command.

OPERATION:

1.   Select ENTR/UCOM. The system prompts:

     Select a pin....

ENTR/UCOM

2. Position the cursor on the pin to uncommit and press Button 1. The pin changes to an empty circle and becomes uncommitted. The ratsnest is redrawn.

3. The system prompts again:

   Select a pin....

4. Continue selecting pins to uncommit, if appropriate.

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.

ENTR/UCOM

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 design.

REMARKS:

Wires provide the electrical connectivity. Lines entered
with the DRAW/LINE command, however, provide
graphic information only.

Connectivity is established when one wire forms a "T"
junction with another or when a wire connects to a pin.
Crossing wires on the same layer causes connectivity
errors.

Connecting component pins with wires establishes nets.

Nets may be given alphanumeric names or left
unnamed. If you want the net name to be visible, you
must use the NAME/NET command.

When entering wires, if the wire engages a ratsnest, the
ratsnest is displayed only for the pins that are not
connected by wires.

When a wire is connected to a pin, the pin image
changes from an open circle to a filled circle.

When entering wires, if you begin a wire and press
Button 2 to abort the wire and the ratsnest has already
been entered with the ENTR/RATN command, the
ratsnest will not be deleted.

ENTR/WIRE

To insert a via at the beginning of a trace, change layers, press Button 1 at the same point (at the end of a segment on another layer) and then press Button 2. A via will be added at the beginning of the segment.

To insert a via at the end of a trace, press Button 1, change layers, press Button 1 again, and then press Button 2. A via will be added at the end of the segment.

When entering wires, if you begin a wire from an uncommitted pin and press Button 2 to abort the wire, the pin will be left uncommitted.

Status line for ENTR/WIRE:

Layer	Net Name	Line Mode	Line Width
COMP		ORTH	W:15

OPERATION:

1. To enter a new wire or create a net, select ENTR/WIRE. The system prompts:

    Select start point....

2. Position the cursor and select. The system prompts:

    Select next point....

ENTR/WIRE

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 then prompts:

    Select next point....

4. To enter additional wire segments, repeat step 3. To end a wire, press Button 2.

5. To delete part of a wire just entered, trace the cursor backwards along the wire to erase and press Button 1. The segment that is traced over disappears.

OPTIONS:

Naming a New Net - To name a wire or net upon creating it, press [F3]. The system prompts:

    Net name:

Type the net name and press [Return]. The active net name displays on the status line.

Changing a Net Name - To change the net name, or other parameters (layer, width, angle), position the cursor over the item on the status line and press Button 1.

ENTR/WIRE

Renaming a Net - To rename a wire/net, select ENTR/WIRE and then select the wire. When an existing wire is picked, the system highlights it and displays the active net name on the status line.

Then, select the net name on the status line or press [F3]. The system prompts for the new name.

Type the name and press [Return]. Move the cursor back to the drawing area and 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. (See also NAME/NET for viewing net names.)

Adding to a Net - If adding a wire to an existing net, the new wire segment merges with the existing wire/net and acquires the same net name.

Crossing Two Nets on the Same Layer - When you cross two wires on the same layer the system responds with a message. If either wire is unnamed, the system highlights the wires and prompts:

Merge the nets? YES NO.

ENTR/WIRE

To merge the wires into one net, select YES. To prevent a merger, select NO or press Button 2.

If both wires are named, the system responds:

More than one named net. No merge.

You cannot merge two named nets. Change layers with [F1], the status line, or VLYR. Then continue with ENTR/WIRE. A via is automatically inserted at the point of the layer change, and the remainder of the wire is added on the new layer.

ERROR MESSAGES:

Message: Both nets named. Merge not allowed.

Cause: An attempt was made to merge two nets that are assigned names.

Action: Do not merge the nets, or unname one of the two nets and then merge the nets.

ENTR/WIRE

Message: **Exceeded limit for number of nets.**

Cause: The database limit for the maximum number
of nets has been reached.

Action: Redesign your PCB so that not as many nets
are used in the database.

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

Cause: The database limit for the maximum number
of components or pins has been reached.

Action: Redesign the PCB so that not as many
components are used in the database.

Message: **More than one named net. No merge.**

Cause: An attempt to short more than one named net
was made.

Action: Reroute the net so that fewer than two named
nets are hit.

Message: **Net hit is named <netname>. Change? Yes No**

Cause: The wire that you are entering has a net name
and the starting point of the wire has hit a
named net on the same layer.

ENTR/WIRE

Action: You have three options:

1. Select YES to change the existing net name to the new one being entered.

2. Select NO to preserve the net name of the existing net. The new net acquires the net name of the existing net.

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 short more than one named net.

Action: Reroute the wire path so that named nets are not shorted.

Mode: SYMB   **Keyboard Command: /EXE**
DETL

(Execute)

FUNCTION:

Replays macro files and command log files.

REMARKS:

After a session when you have exited to DOS, the system assigns the filename PCCARDS.CMD to the session recording, or command log file.

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

Use the PCCARDS.CM$ 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. 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 specified begins execution.

To pause a macro or a command log file, press [Ctrl]-[S]. To resume execution, type two slashes (//).

To pause a macro or a command log file and allow user input, press the space bar. The system returns control to the user. Type /RESU to continue automatic execution.

/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 is encountered during playback. This message serves as a prompt or reminder.

> NOTE: You may edit the file after the recording is made and you exit to DOS. Use a word processor in the nondocument mode to make modifications.

One command you might want to add with a word processor is the /INTR command. 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.

Refer to Appendix B for more information on executable files.

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: Can't open "<filename>"

Cause: The system cannot find the filename as entered.

Action: Check the spelling and re-enter the filename. Check the directory path, if necessary. Use the SYS/DOS and then do a DIR (Directory) command to verify the existence or spelling of a filename.

Mode: **SYMB**   **FILE**
        DETL

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.PCB

    \PCAD\PROJ3\TESTFILE.PCB

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 name of the file and press [Return] or
    Button 1.

    NOTE: You don't need to type a filename
    extension for .PCB files, but you do for .PRT files.
    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 incorrect 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 and do a DIR
          (Directory) to verify the existence or spelling
          of a 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-CARDS (512K) program
          on a TI portable computer.

Action: Contact P-CAD customer support or use a
          version of PC-CARDS configured for 640K
          databases.

FILE/LOAD

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: **Path too long.**

Cause: The path and filename for the specified
           database overflows the internal filename
           buffer.

Action: Shorten the path in the PC-CARDS
           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 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 recall a previously saved file

- SYS/QUIT to exit the system

   NOTE: Although DOS accepts various characters in
   filenames (for example, #), avoid using them in
   part filenames (<filename>.PRT). Use only
   alphanumeric characters in part 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 .PCB
    filename extension in detail mode and the .PRT
    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 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 the SYS/DOS command and then use the
           DOS delete command to free up space on the
           destination disk or insert a new disk with
           available space.

Mode: SYMB   **FILE/ZAP**
       DETL

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   Keyboard Command: **/INTR**
      DETL
                        (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   **Keyboard Command: /LANG**
DETL

(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**
      DET

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 of 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 axes. It also displays the stored views and shows the size and position relative to the axes.

2. Move the cursor to relocate the reference rectangle.

3. To select the new location and view the new graphics display, press Button 1. To restore the original viewing window unchanged, press Button 2.

Mode: **SYMB**   Keyboard Command: **/LSTY**
       DETL

(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   Keyboard Command: **/LYRN**
       DETL
                                          (Layer Name)
Active during: Most Commands

FUNCTION:

Toggles between the active layers.

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   **Keyboard Command: /MAC**
DETL
                                            (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 component parts

●   The storage and replay of consistent board outlines

●   The wiring of multiple bus structures

Macros may be recorded any time during a session.

Any macro recorded is also recorded within the current
command log file (PCCARDS.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.

/MAC

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, if the current status line settings differ from those when you made the macro, objects may be drawn on different layers, or with different grid spacing than originally specified.

OPERATION:

1. Type /. The system prompts:

    Menu command:

2. Type MAC [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].

    The macro recording begins and continues until you enter the keyboard command /MEND (Macro end).

/MAC

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 5.

Action: Do not nest macros deeper than 5.

Mode: **SYMB**
     DETL
                                                 MASK

FUNCTION:

Enables a menu for protecting (masking) or unprotecting components or wires from editing commands.

REMARKS:

The MASK command prevents editing of components and wires; that is, MASK protects components and wires from commands like ROT, MOVE, and DEL by making the masked data unselectable by editing commands.

> NOTE: MASK does not protect wires from the /WIN subcommand.

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   **Keyboard Command: /MEND**
       DETL

(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   **MOVE**
      SYMB

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 can be used with the subcommands WIN and IDEN 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 grid 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 the 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 for you 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 the amount of memory available on your system after PC-CARDS 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.

All objects except components and wires 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.

For correct rubberbanding of the wires attached to a component, include part of the connecting wire in the window.

MASK does not protect wires from MOVE/WIN.

This command moves net names that are partially enclosed in the window.

MOVE/WIN

OPERATION:

1. Select MOVE/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, 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 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 assigned a 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. (See also ENTR/COMP.)

If a board has been created with PC-PACK, the reference designator names are already assigned. Use NAME/COMP only to make the text visible.

Status line for NAME/COMP:

Layer	Text Size	Text Justifi- cation	Text Orien- tation	Mirror
SLKSCR	SIZ:125	L B	F	M

NAME/COMP

OPERATION:

1. Select NAME/COMP. The system 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 ('). 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.

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 layer parameter setting on the status line to change the active layer, or use the VLYR command to access the layer screen to change the active layer. Then select NAME/COMP. 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 end the
         command.

Mode: DETL   **NAME/NET**

FUNCTION:

Names a net or makes visible the name of a net that has
been previously entered into a design.

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 lines 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 Justifi-cation	Text Orien-tation	Mirror
SLKSCR	SIZ:125	L B	F	M

NAME/NET

OPERATION:

1. Select NAME/NET. The system prompts:

    Select a net....

2. Select the desired net. The system highlights the net, and if no net name has been assigned, prompts:

    Enter net name:

    If the net has already been named, the system skips ahead to the prompt in 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. Set the text attributes on the status line, if necessary, and 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.

NAME/NET

5. Press Button 1. The name displays at that location.

6. To display multiple copies of the name continue to position the cursor and press Button 1.

    If you enter an existing name, the system highlights the named net and prompts:

    Net <name> exists. Merge? YES NO.

    Select YES or NO.

    Select YES to merge the nets. The system responds:

    Nets merged.

    Select the location to place the net name.

    Select NO to choose a different name.

ERROR MESSAGES:

Message: Net <name> exists Merge YES NO.

Cause: An existing net name was entered.

Action: Enter a new name.

NAME/NET

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. 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 the
    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
file 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 is retained 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 numbers 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   Keyboard Command: **/RESU**
      DETL
                                (Resume Execution)

Active during: /EXE

FUNCTION:

Resumes the execution of a command log file or macro
after an interrupt.

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 ratsnested connections are rubberbanded. The wires are also rubberbanded.

> NOTE: You should not use any of the Rotate commands if wires are attached to the component.

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 the rotation of components.

OPERATION:

1. Select ROT. The system prompts:

    Select the object(s)....

ROT

2. Select an object. The point that you choose also becomes the center of the rotation for the object. The system highlights the object and prompts:

    BUT1 to rotate 90 degree CCW....

    (CCW stands for counterclockwise.)

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 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/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:

The number of objects within a window that can be rotated depends on how much memory is available on your system after PC-CARDS 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.

All wire segments totally within the window rotate. The wire segments extending outside of the window do rubberband.

MASK does not protect wires from the ROT/WIN command.

All objects except components and wires are selectable when they are on a currently visible layer. 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. Stop the operation by pressing 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, net, and pin attributes, set packaging assignments', and get special symbol file.

OPERATION:

Select the SCMD command to enable the following submenu choices:

● In SYMB mode:

        SCAT (Set Component Attribute)
        SPAT (Set Pin Attributes)
        SPKG (Set Packaging)
        GSSF (Get Special Symbol File)

● In DETL mode:

        SNAT (Set Net Attribute)
        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 pin types and padstacks
(<filename>.PS).

REMARKS:

This command changes the display from default pins to
that of the pads which are required for pen or
photoplotting. The special symbol files
(<filename>.SSF) link padstacks to the database until
they are removed or changed by loading a different
special symbol file.

Since the system can redraw pins more quickly
than pads, it is more efficient to add special symbol
files only to plot files created with SYS/PLOT.

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 pin types which are
     named in the file have their graphic characteristics
     updated to the padstack configuration.

SCMD/GSSF

3. Select the REDR command to view the padstack graphics.

   Only one special symbol file can be attached to a database at any one time. Consequently, the system removes existing padstack data before prompting for a new special symbol filename. You may notice a slight delay while the existing padstack data is removed.

   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 or [Esc] instead of typing a filename. The existing special symbol file is removed from the database, and the default pin graphics are restored.

   NOTE: There must be a one-to-one correspondence between all pin type and padstacks or the system responds with error messages indicating that it was not able to find the referenced items. If error messages of this type occur, the padstack does not link to the database.

ERROR MESSAGES:

Message: Can not open file.

Cause: An attempt to open named file for I/O failed.

SCMD/GSSF

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: 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 and then do a DIR
          (Directory) to verify the existence or spelling
          of a filename.

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 4. "Creating a Padstack
          and a Special Symbol File" in Chapter 2,
          "Using PC-CARDS."

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

SCMD/GSSF

the SYS/DOS command and then do a DIR
(Directory) to verify the existence or spelling
of a filename.

Message: No symbol named.

Cause: No symbol 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: Symbol has no picture data.

Cause: The symbol specified in the special symbol
         file contains no graphic data in the SYMB
         mode.

Action: Edit the symbol, entering the graphic data in
         the SYMB mode. Then try using the
         SCMD/GSSF command again.

Message: Symbol origin not defined.

Cause: The symbol specified 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/SCAT**

(System Command/Set Component Attribute)

FUNCTION:

Sets or modifies the Component Type ID which the
PC-PACK utility program uses to identify components,
DIPS, discrete, and connector classes.

REMARKS:

The Component Type ID that PC-PACK expects are:

DIPS	--	10000
Resistors	--	11100
Capacitors	--	11200
Inductors	--	11300
Transistors	--	11400
Other Discretes	--	11000
Connectors	--	12000
Miscellaneous	--	13000

These types are already included in the P-CAD
part libraries. If you create your own parts with
PC-CARDS and plan to package schematics from
PC-CAPS into your parts, use SCMD/SCAT to include
the appropriate Component Type ID.

For more detail, refer to the "Input" section of the
PC-PACK User's Manual.

SCMD/SCAT

OPERATION:

1. Select SCMD/SCAT. The system prompts:

    Workcomp. Old type = <n>. New type=

    where **<n>** is the number of the current type and
    Workcomp means that you are specifying a type for
    the component you are currently editing.

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: An invalid Component Type ID was entered.

Action: Enter a number within the range.

Mode: DETL   **SCMD/SNAT**

(System Command/Set Net Attribute)

FUNCTION:

Finds and highlights a named net.

REMARKS:

This command adds a global attribute to a net.
Currently, the global attribute has no effect on the data.

OPERATION:

1. Select SCMD/SNAT. The system prompts:

    Enter a net name:

2. Type the net name and press [Return] or Button 1.
    If the net does not exist, the system displays:

    <name> not found.

    If the net does exist, the system highlights the net
    and prompts:

    A global net? YES NO.

3. Since the global attribute is not fully implemented
    at this time, press Button 2. The net is
    unhighlighted.

Mode: SYMB   **SCMD/SPAT**

(System Command/Set Pin Attributes)

FUNCTION:

Sets and/or modifies pin type and logic equivalence
(LEQ) attributes.

REMARKS:

The pin type attribute assigns a particular numerical
value to a pin. The allowable range is 0-24. The pin
type of 0 is reserved for the system default for vias and
should not be used for components. The pin type
attribute is used later to define padstacks in
conjunction with the special symbol files. (See the
SCMD/GSSF command description.)

LEQ stands for logic equivalence. The default value
for this attribute is LEQ = 0, where 0 means that each
pin is logically unique and cannot be swapped. All pins
are assumed to have this setting unless a different LEQ
is set.

When the LEQ is any integer greater than 0, it indicates
an equivalence class for each pin. Each set of logically
equivalent pins can be given a unique number to mark
them as logically equivalent and therefore as swappable.
The allowable range is from 0 to 24.

Pins with the same equivalence number (other than 0)
can be swapped.

SCMD/SPAT

OPERATION:

1. Select SCMD/SPAT. The system highlights the first pin and prompts:

    Enter new type:
    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 enter a new type in response to the prompt. Next, the system prompts:

    Enter new code:
    Pin LEQ code is <n>

    where <n> is the last number used. 0 is the default.

3. Press [Return] or Button 1 to accept the number displayed, or type a new number and press [Return] or Button 1.

    These prompts are repeated for each pin of the component.

SCMD/SPAT

ERROR MESSAGES:

Message: **Out of range (0 to 24).**

Cause: The value entered is outside the allowable range.

Action: Enter pin types and LEQ values as a number between 0 and 24.

Mode: SYMB   **SCMD/SPKG**

(System Command/Set Packaging)

FUNCTION:

Sets the logical packaging assignments into the PCB
parts.

REMARKS:

This command installs the logic packaging and pin
assignment information into the physical PCB device
(DIP, etc.). This information is used by PC-PACK to
make the packaging assignments as defined on the
schematic.

OPERATION:

1. Select SCMD/SPKG. The system prompts:

    Enter number of gates:

2. Enter the number of gates or elements to be
    packaged into the device. The maximum number of
    elements is 255. The system prompts:

    Enter number of pins per gate:

3. Type the number of pins or connection points for
    one gate and press [Return]. The system prompts:

    Enter name of gate pin <n>:

4. Type the name of the pin as it was assigned on the

SCMD/SPKG

schematic symbol not the printed circuit part. The system continues to prompt for each gate pin in turn. The system asks you to identify each pin on the part with this prompt:

Select gate <n> pin <x>...

Select the indicated gate pin. The system highlights the pin to indicate acceptance and continues to the next gate pin until all specified gates and pins have been identified.

NOTE: All PCB parts must be packaged using SCMD/SPKG before they can be used with PC-PACK.

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: SPKG: Available only in SYMB mode.

Cause: The SCMD/SPKG command was entered in DETL mode.

Action: Check the input (keyboard or command log file or macro file) for the correct syntax.

Mode: **SYMB**   Keyboard Command: **/SGAT**
       DETL

                        (Set Global Attributes)

FUNCTION:

Specifies a global attribute to freeze or open nets to editing commands.

REMARKS:

Enter this command with the keyboard. Skip the first prompt regarding the snap-to-pin tolerance by pressing Button 2, as this feature is not functional in PC-CARDS.

Databases created by PC-PACK default to a frozen netlist, whereas all others default to open.

The netlist can be frozen so that the EDIT commands do not delete frozen nets and the pins remain committed. The /SGAT command does not protect nets from the ENTR/UCOM command.

/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=

3. Press [Return] or Button 2 to skip this feature because it is currently only active in PC-CAPS.

    Answer the next message, which is a toggle for protecting the netlist connectivity. The system prompts:

    Net list is open. Freeze it? YES NO

    To protect, or freeze, the ratsnests connections regardless of editing the wires, select YES. Select NO to keep an open netlist.

    The reverse of the above prompt is:

    Net list is frozen. Free it? YES NO

    To unprotect, or open, the ratsnests connections and thus allow editing of the connections, select YES.

    Select NO to protect the netlist as is.

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 nine views, including a MAP view. Any of the numbered views that are stored during a session are saved with the board layout when you select FILE/SAVE.

If the **Video save to disk** option on the PC-CARDS 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

STO

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.

Mode: DETL   **SWAP**

(Swap)

FUNCTION:

Enables the submenu for the following swap operations:
components, gates, and pins. With each option, one type
may be swapped with another of the same type (gates
with gates, pins with pins, etc.).

OPERATION:

Select SWAP to enable the following submenu choices:

    COMP
    GATE
    PIN

    NOTE: You should not use any of the Swap
    commands if wires are attached to the components.

Mode: DETL   # SWAP/COMP

(Swap/Component)

_____

FUNCTION:

Swaps any component with any other component.

REMARKS:

Use this command to improve on the placement of devices. It does not add or delete components from the PCB.

The SWAP/COMP command swaps components regardless of the types of components involved. For example, a resistor may be swapped with a microprocessor.

OPERATION:

1. Select SWAP/COMP. The system prompts:

    Select first component....

2. Select the desired component with Button 1. The system highlights it, and shows the ratsnest of all connections to that component. Use Button 2 to release a component from selection. The system prompts:

    Select second component....

SWAP/COMP

3. Pick the second component with Button 1. The system highlights the component, including all of its connections, and swaps the first one with the second, showing the new ratsnest.

Mode: DETL   SWAP/GATE

(Swap/Gate)

FUNCTION:

Swaps gates from one device to another, or from one position in a device to another position in the same device.

REMARKS:

This command only allows swapping of like devices. An inverter may be swapped with another inverter, but not with an AND gate. Any gate may be swapped with another in its package, or swapped to another like package.

All SWAP/GATE operations are recorded automatically in a report file (<filename>.RPT) on disk for later use with the PC-BACK utility program in back-annotation. (See the *PC-BACK User's Manual.)*

OPERATION:

1.   Select SWAP/GATE. The system prompts:

     Select first gate....

SWAP/GATE

2. Position the cursor on one of the pins of the desired
    gate and press Button 1. All pins of that gate are
    highlighted, as are their ratsnest connections. Use
    Button 2 to release a gate from selection. The
    system prompts:

    Select second gate....

3. Position the cursor on one of the pads of the second
    gate and press Button 1. Its pads and connections
    are also highlighted, and the two gates are swapped.

ERROR MESSAGES:

Message: **Different package types.**

Cause: When attempting to swap gates, the second
         gate specified is part of a package with a type
         different from the first gate's package type.

Action: Select a gate from a package with the same
         type as the first gate's package.

Message: **Gates not compatible.**

Cause: The second gate is not the same type as the
         first.

Action: Select a different gate.

SWAP/GATE

Message: **No gate found.**

Cause: No gate pin was found at the specified location using the SWAP/GATE command.

Action: Check the pin location by using REDR and ZIN, and try again.

Message: **No packaging info. Can not swap.**

Cause: The gate selected using the SWAP/GATE command is in a package that has no packaging information.

Action: Use the SCMD/SPKG command to assign packaging information for the associated <filename>.PRT file.

Message: **Report file record not generated.**

Cause: Using the SWAP/GATE command, a swap was not recorded in the <filename>.RPT file because the support limit for pins per gate in the report file was exceeded. Only 16 pins per gate can be recorded in the report file.

Action: Pre-assign gates with more than 16 pins. Then use manual back-annotation to update the schematics for swapping gates with more than 16 pins.

SWAP/GATE

Message: **Swap report file exists. Append? YES NO**

Cause: Swap report file already exists for the current
database.

Action: You have two options:

Select YES to add data to the current swap
report file.

Select NO to start a new swap report file.

Mode: DETL   **SWAP/PIN**

                                                (Swap/Pin)

FUNCTION:

Swaps pins of components with logically equivalent pins
of the same component.

REMARKS:

This command only swaps logically equivalent pins
within a specific gate. (See LEQ under the
SCMD/SPAT command.)

All SWAP/PIN operations are recorded automatically in
a report file (<filename>.RPT) on disk for later use
with the PC-BACK utility program in back annotation.
(See the *PC-BACK User's Manual.)*

OPERATION:

1.   Select SWAP/PIN. The system responds:

     Select first pin....

2.   Select the first pin for swapping. The system
     highlights the pin and its connections. Use Button 2
     to release a pin from selection. The system
     responds:

     Select second pin....

SWAP/PIN

3. Select the second pin to be swapped. The system highlights the pin and its connections. The two pins are swapped.

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: **Pins not swappable.**

Cause: The pins are not from the same gate or are not of the same logic equivalency (LEQ).

Action: Select different pins.

SWAP/PIN

Message: **Swap report file exists. Append? YES NO**

Cause: Swap report file already exists for the current database.

Action: You have two options:

Select YES to add data to the current swap report file.

Select NO to start a new swap report file.

Message: **The two pins are not swappable.**

Cause: The second pin identified using the SWAP/PIN command is not from the same gate as the first pin, or the second pin is not swappable.

Action: Check the LEQ information for the package in question using the SCMD/SPAT command, and try again.

Mode: **SYMB**

(Symbol)

FUNCTION:

Sets the system to the SYMB (symbol) command mode.

REMARKS:

SYMB is used for creating and editing components and padstacks.

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 same mode.

Components created in SYMB mode can be entered as components in the DETL (detail) mode with the ENTR/COMP command.

OPERATION:

Select SYMB to activate 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 the SYS command and one of the following
submenu choices:

DOS
PLOT
STAT
QUIT

Mode: **SYMB**   **SYS/DOS**
       DETL

(System/DOS)

FUNCTION:

Allows you to execute commonly used DOS commands
from within the PC-CARDS environment.

REMARKS:

The commands you can enter from within the
PC-CARDS 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 ID

CAUTION: You cannot change the volume from
within the PC-CARDS environment.

000-0052-06

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 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 the specified command.
         Not enough memory or misspelled command.

Action: Exit PC-CARDS and re-enter PC-CARDS 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, PC-PHOTO, 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.

Link special symbol files (<filename>.SSF) with the SCMD/GSSF command before making 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.

SYS/PLOT

OPERATION:

1. Position the viewing area on screen to encompass the portion to be plotted. Once you choose the SYS/PLOT command, do not PAN or change the viewed area.

2. Select the layers to be included in the plot file. Only viewed layers are placed 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 corner 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 and then corner 2 when prompted.

5. Position the cursor and press Button 1. The system prompts:

   Plot file name: <filename>

SYS/PLOT

6. Type the filename and press [Return] or Button 1.

    NOTE: You don't need to type a filename
    extension; the system automatically adds the .PLT
    extension. However, if you type a period (.) after
    the filename, with no extension, the system does not
    add an extension.

    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 loaded database 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.

    Select YES or NO.

    If you select YES, the existing plot file is replaced.

SYS/PLOT

ERROR MESSAGES:

Message: **Disk is full. Database corrupted.**

Cause: Insufficient space on disk to create file.

Action: Use the SYS/DOS command and then use the
           DOS delete command to free up space on the
           destination disk or insert a new disk with
           available space.

Mode: **SYMB**
DETL

SYS/QUIT

(System/Quit)

FUNCTION:

Exits the command menu and returns to the opening menu.

REMARKS:

If you 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.

OPERATION:

Select the SYS/QUIT command. 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. Select the FILE/SAVE command to preserve modifications you have made to the file.

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 allowed 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; that 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 an alternate screen showing all layers currently available as well as their color/pen assignment numbers and viewing status.

You may create new layers, change the color or viewing status, and change the names of existing layers. Layer names can be up to six characters. Once a layer is created, it cannot be deleted, but you may alter any of its settings.

The layer screen also 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 REDR automatically after exiting VLYR.

VLYR is a nested command and can be used while one of the main operational commands is active.

VLYR

OPERATION:

1. Select VLYR. The system displays all the available layer names, their color/pen assignment numbers for pen plotting, and their current status.

2. You may make the following changes on this layer menu:

   ● To change the name of an existing layer, select layer name from 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].

VLYR

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. The system displays a submenu of the 15 assignment numbers available at the bottom of the screen. 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, or ABL), select the 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   accessible
    ABL - visible and may become the active
      accessible layer.
    ABL A - visible and active, meaning that any
      data created is recorded on this layer.

VLYR

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.

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**   **VWIN**
      DETL

(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 the VWIN command. The system prompts:

   View Window: Corner 1...

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   **Keyboard Command: /WAIT**
       DETL

FUNCTION:

Inserts a pause and a message prompt into the recording of a macro file or a command log file (PCCARDS.CMD).

REMARKS:

During the recording of a macro, or any time during the recording of the current command log file, you may insert a wait with this command that results in a pause during the playback. A message may also be added 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 44 characters. If you do not want a message, press [Return] only.

Mode: **SYMB**   **X**
        DETL

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 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 in 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 in one redraw.

To halt the zoom out 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 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 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 an F on the status line,
look under F in this section for an explanation.

The status line also displays various layer names (for
example, WIRES, COMP, SOLDER,...) that do not have
their own alphabetical entry 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 Display.

Note also that there are two entries for R. One entry is
for right justification and the other is for turning the
ratsnest display on or off.

The following status line parameters are available with
most commands:

Example:

RATS NEST	X:Y GRID UNITS	GRID TOGGLES	COORDINATES X	Y
R	10:10	S G	150	200

Mode: **SYMB**   **ANGL**
       DETL

(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

APER: < n >

(Aperture)

Active during: DRAW/FLSH

FUNCTION:

Specifies the aperture as a number from 1 to 65 during DRAW/FLSH.

The Gerber photoplotter supports up to 65 aperture positions. The GTCO photoplotter supports only 24 aperture positions.

REMARKS:

When you enter a flash aperture, specify the number of the aperture wheel position represented by the flash.

OPERATION:

1. Select DRAW/FLSH. The system displays the last active flash aperture as a number on the status line.

2. Move the cursor to the aperture field on the status line.

3. The system prompts:

    Enter aperture number:

4. Type a number from 1 to 65 and press [Return]. Proceed with the DRAW/FLSH command.

Mode: SYMB   **B**
        DETL

<div align="right">(Bottom Justified)</div>

Active during: ATTR/ACOM, DRAW/TEXT, ENTR/PIN, NAME/COMP, NAME/NET

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 justification of text.

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, ENTR/PIN, NAME/COMP, or NAME/NET.

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   **C**
       DETL

(Center Justified)

**Active during: ATTR/ACOM, DRAW/TEXT,
ENTR/PIN, NAME/COMP, and NAME/NET**

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, ENTR/PIN,
      NAME/COMP, or NAME/NET.

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   **DASHED**
       DETL

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**   **DOTTED**
       DETL

**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   **EQUIV: < n >**

                                           (Pin Equivalence)

Active during: ENTR/PIN

___

FUNCTION:

Specifies the pin equivalence as a number from 0 to 127
during ENTR/PIN.

REMARKS:

When you enter pins, you may specify the pin
equivalence for swapping using the EQUIV:<n>
command. You can also assign pin equivalence after
you enter the pin using the SCMD/SPAT command. The
EQUIV:<n> command limits numbers to 0 to 127;
SCMD/SPAT limits numbers to 0 to 24. You can accept
a higher number (25 to 127) during SCMD/SPAT by
pressing Button 1 or [Return].

OPERATION:

1.   Select ENTR/PIN. The system displays the last
     active pin equivalence as a number on the status
     line.

2.   Move the cursor to the pin equivalence field on the
     status line. The system prompts:

        Pin equivalence number:

Status Line Parameter: EQUIV: < n >

3. Type the number for pin equivalence and press
    [Return]. Proceed with the ENTR/PIN command.

Mode: SYMB   **F**
        DETL

                                     (Orientation)

Active during: ATTR/ACOM, DRAW/TEXT, ENTR/COMP, ENTR/PIN, NAME/COMP, and NAME/NET

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 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, ENTR/PIN, NAME/COMP, or NAME/NET.

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   **G**
       DETL

                                          (Grid Lock)

Active during: Most Commands

FUNCTION:

Specifies whether the grid lock is on or off.

REMARKS:

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.

When the grid lock is off, the cursor is free to move in increments of 1 database unit.

If the G on the status line is green, then grid lock is on. If the G on the status line is red, then grid lock is off.

The [F8] key also toggles the grid lock.

OPERATION:

1. Select G on the status line.

2. Toggle to the opposite state, turning the cursor grid lock on (green) or off (red).

3. Proceed with a menu command.

000-0052-06

Mode: SYMB   **L**
      DETL

<div align="right">(Left Justified)</div>

**Active during: ATTR/ACOM, DRAW/TEXT,
ENTR/PIN, NAME/COMP, and NAME/NET**

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 one of these commands: ATTR/ACOM,
    DRAW/TEXT, ENTR/PIN, NAME/COMP, or
    NAME/NET.

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: SYMB   **LAYER**
        DETL

Active during: Most Commands

FUNCTION:

Selects the currently active layer from among the
enabled layers.

REMARKS:

The status line LAYER parameter allows you to change
layers while a menu command is active. The [F1] key
also selects among the enabled layers.

Another method is to use the VLYR command. Only
layers that have an ABL status can be chosen.

To change layer status, use the menu command VLYR.
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
graphic 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

Status Line Parameter: **LAYER**

If a layer is "OFF," it is not visible. If a layer is "ON," it is visible.

If a layer is "ABL," it is visible and may be turned to active. If it is "ABL A," then it is visible and active.

The active layer is the current destination for creating graphics.

Only an ABL layer can become an ABL A layer. Further, a layer must have the active assignment removed before it can be turned off or on again.

One layer is the ABL A, or active, layer.

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, ENTR/PIN, NAME/COMP, and NAME/NET

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 is displayed mirrored on the screen and is mirrored when the text is plotted.

OPERATION:

1. Select one of these commands: ATTR/ACOM, DRAW/TEXT, ENTR/COMP, ENTR/PIN, NAME/COMP, or NAME/NET.

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, 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: **DETL**
      SYMB

R

(Ratsnest)

Active during: Most Commands

FUNCTION:

Specifies whether ratsnests are viewed or suppressed.

REMARKS:

When the R is green, the ratsnest is displayed for the unconnected pins. When it is red, the ratsnest is not visible.

OPERATION:

1. Select the R to the left of the X:Y grid units.

2. Toggle the R to green to turn on the ratsnest view or red to turn it off.

3. Select REDR to redraw the display to show the ratsnest view or to suppress it.

Mode: SYMB   **R**
          DETL

(Right Justified)

Active during: ATTR/ACOM, DRAW/TEXT,
ENTR/PIN, NAME/COMP, and NAME/NET

FUNCTION:

Specifies that text is right justified relative to the
cursor location.

REMARKS:

Two fields are displayed on the status line for the
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 one of these commands: ATTR/ACOM,
     DRAW/TEXT, ENTR/PIN, NAME/COMP, or
     NAME/NET.

Status Line Parameter: R (Right Justified)

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**   **S**
       DETL

                                        (Show Grid)

Active during: Most Commands

FUNCTION:

Specifies whether the screen display of grid points is on
or off.

REMARKS:

When the screen display is ON, the system displays grid
reference points in the increments decreed 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.

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.

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, ENTR/PIN, NAME/COMP, and NAME/NET

FUNCTION:

Specifies the current size for the creation of text.

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 any placed 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].

    If the number is less than 2 or greater than 5000, the system displays:

    Out of range (2 to 5000).

Status Line Parameter: SIZ: < n >

The new text size is displayed on the status line.

3.   Proceed with a menu command.

Mode: **SYMB**
DETL

SOLID

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**   **T**
             DETL

(Top Justified)

**Active during: ATTR/ACOM, DRAW/TEXT,
ENTR/PIN, NAME/COMP, and NAME/NET**

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 one of these commands: ATTR/ACOM,
      DRAW/TEXT, ENTR/PIN, NAME/COMP, or
      NAME/NET.

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   TYPE: < n >

Active during: ENTR/PIN

FUNCTION:

Specifies the pin type as a number from 0 to 25 during
ENTR/PIN.

REMARKS:

When you enter pins, you must specify each pin with
the ENTR/PIN command.

OPERATION:

1. Select ENTR/PIN. The system displays the last
    active pin type as a number on the status line.

2. Move the cursor to the pin type field on the status
    line. The system prompts:

    Enter pin type number:

Status Line Parameter: TYPE: < n >

3. Type the number of the pin type. If the number
    you specify is not between 0 and 25, the system
    responds:

    Out of range (0 to 25)

    Type a number within the proper range.

4. Proceed with the ENTR/PIN command.

Mode: SYMB $\qquad$ W: <n>
      DETL

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:

The format is W:<n>, where <n> is an integer from 0 to 250 specifying the width of line graphics or wires in database units. A setting of 0 (zero) is the minimum width available.

OPERATION:

1. Select one of these commands: 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.

    If the number is greater than 250, the system
    responds:

    Out of range (0 to 250)

    Type a number within the range.

Mode: **SYMB**   **X Y**
       DETL

                       (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 enters
lines, wires, and other graphics at locations that are
offscreen.

The keyboard entry of coordinates can also specify the
location of the viewing window as with the PAN and
LPAN commands.

This feature is useful any time when it is more suitable
to type the numbers rather than to locate coordinates
with the mouse.

To specify X and Y coordinates by typing them instead
of using the mouse, follow the procedure below.

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 for 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 select the slot
    for the X Y coordinates and specify their values.

    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 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 display reflects the increments specified by the X:Y feature.

When zoomed out, the grid points may not show on the screen.

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**   **45D**
        DETL

(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 non-document file named AUTOEXEC.BAT and type:

    PATH C:\DOS;C:\PCAD\EXE [Return]
    PROMPT PG [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 PG [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 PG [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 non-document 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 that 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 loadable driver
reference 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 PG

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 PC-CARDS error messages that may appear on the screen, the cause of each error, and the appropriate user action.

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 are assigned names.

Action: Do not merge the nets, or unname one of the two nets and then merge the nets.

Message: Can not copy net.

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

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.

Cause: An attempt to open named file 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: 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 and then do a
            DIR (Directory) command 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 the spelling of the command and the list of DOS
            commands that can be used within PC-CAPS in the SYS/DOS
            command description.

Message: Component has no symbol.

Cause: The specified component has no data defined in SYMB mode.

Action: Check the spelling of the component filename. Edit the symbol
            file and check the data definition against "Example 5. Creating a
            P-CAD Compatible Component" in "Using PC-CARDS."

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 end 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
            PCCARDS.CMD file, and the PCCARDS.DBG file for further
            analysis.

Message: Database format not compatible.

Cause: An incorrect 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 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 and
            then do a DIR (Directory) to verify the existence or spelling of a
            filename.

Message: Different package types.

Cause: When attempting to swap gates, the second gate specified is part
           of a package with a type different from the first gate's package
           type.

Action: Select a gate from a package with the same type as the first
           gate's package.

Message: DISK FULL!!! FILE SAVED NO GOOD!!!

Cause: The destination disk for a FILE/SAVE operation is full.

Action: Use the SYS/DOS command and then use the DOS delete
           command to free up space on the destination disk or insert a new
           disk with available space.

Message: DOS ERROR: <n>.

Cause: DOS failed to execute the specified command. Not enough
           memory or misspelled command.

Action: Exit PC-CARDS and re-enter PC-CARDS to intiate 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
           4. Creating a Padstack and a Special Symbol File" in "Using
           PC-CARDS."

Message: Exceeded limit for number of components or pins.

Cause: The database limit for the maximum number of components or
           pins has been reached.

Action: Redesign the PCB so that not as many components are used in
           the database.

Message: Exceeded limit for number of nets.

Cause: The database limit for the maximum number of nets has been reached.

Action: Redesign your PCB so that not as many nets are used in the database.

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 and then do a DIR (Directory) to verify the existence or spelling of a filename.

Message: Gates not compatible.

Cause: The second gate is not the same type as the first.

Action: Select a different gate.

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 starting point.

Message: Invalid pin name.

Cause: The specified pin name is illegal.

Action: Specify a nonblank pin name.

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 " or (, but does not end with " or).

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

Message: More than one named net. No merge.

Cause: An attempt to short more than one named net was made.

Action: Reroute the net so that fewer than two named nets are hit.

Message: Net exists. Name change not allowed.

Cause: An attempt was made to change the net name for a ratsnest to a
           name of an existing net.

Action: Use a different name when changing the net name or merging the
           two nets using the ENTR/RATN or ENTR/WIRE command.

Message: Net <name> exists. Merge YES NO

Cause: An existing net name was entered.

Action: Enter a new net name.

Message: Net hit is named <netname>. Change? YES NO

Cause: The wire that you are entering has a net name and the starting point of the wire has hit a named net on the same layer.

Action: You have three options:

1. Select YES to change the existing net name to the new one being entered.

2. Select NO to preserve the net name of the existing net. The new net acquires the net name of the existing net.

3. Press Button 2 twice and enter the wire at different coordinates.

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-CARDS (512K) program on a TI portable computer.

Action: Contact P-CAD Customer Support or use a version of PC-CARDS 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 mode (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, then 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 gate found.

Cause: No gate pin was found at the specified location using the
            SWAP/GATE command.

Action: Check the pin location by using REDR and ZIN, and try again.

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 packaging info. Can not swap.

Cause: The gate selected using the SWAP/GATE command is in a
            package that has no packaging information.

Action: Use the SCMD/SPKG command to assign packaging information
            for the associated <filename>.PRT file.

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
            PC-CARDS.CMD file, and the PCCARDS.DBG file for further
            analysis.

Message: No symbol named.

Cause: No symbol was found for the filename defined in the specified
            special symbol file.

Action: Check the special symbol file for spelling, and use the SYS/DOS
            command and then 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 short more than one named net.

Action: Reroute the wire path so that named nets are not shorted.

Message: No vertex found.

Cause: The system cannot add a vertex where the cursor was positioned.

Action: Reposition the cursor, and try again. If necessary, zoom in for
            greater accuracy.

Message: No via found.

Cause: The system cannot find a via where the cursor was positioned.

Action: Reposition the cursor in the center of the via, 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 corresponding 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
           PCCARDS.CMD file, and the PCCARDS.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 24)

Cause: The value entered is outside the allowable range.

Action: Enter pin types and LEQ values as a number between 0 and 24.

Message: Out of range (-32000 to 32000)

Cause: An invalid Component Type ID was entered.

Action: Enter a number within the range.

Message: Out of space.

Cause: Insufficient memory to complete the command execution.

Action: Save the database using the command and re-enter PC-CARDS
           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-CARDS 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: Pins not swappable.

Cause: The pins are not from the same gate or are not of the same logic
            equivalency (LEQ).

Action: Select different pins.

Message: Program error, contact PCAD.

Cause: Program error detected.

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

Message: Report file record not generated.

Cause: Using the SWAP/GATE command, a swap was not recorded in
            the <filename>.RPT file because the support limit for pins per
            gate in the report file was exceeded. Only 16 pins per gate can be
            recorded in the report file.

Action: Pre-assign gates with more than 16 pins. Then use manual
            back-annotation to update the schematics for swapping gates
            with more than 16 pins.

Message: Scale Factor = 0.

Cause: Invalid filename entered.

Action: Enter a valid filename.

Message: SPKG: Available only in SYMB mode.

Cause: The SCMD/SPKG command was entered in DETL mode.

Action: Check the input (keyboard or command log file or macro file) for
           the correct syntax.

Message: Shorting more than one net.

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

Action: Uncommit the pins that are hit before copying the wire, or enter
           the wire with the ENTR/WIRE command.

Message: Swap report file exists. Append? YES NO

Cause: Swap report file already exists for the current database.

Action: You have two options:

           Select YES to add data to the current swap report file.

           Select NO to start a new swap report file.

Message: Symbol has no picture data.

Cause: The symbol specified in the special symbol file contains no
           graphic data in the SYMB mode.

Action: Edit the symbol, entering the graphic data in the SYMB mode.
           Then try using the SCMD/GSSF command again.

Message: Symbol origin not defined.

Cause: The symbol specified 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 two pins are not swappable.

Cause: The second pin identified using the SWAP/PIN command is not
             from the same gate as the first pin, or the second pin is not
             swappable.

Action: Check the LEQ information for the package in question using the
             SCMD/SPAT command, and try again.

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-CARDS or PC-PLACE 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>.BKA files created by PC-PACK and PC-BACK are also executable files and have the same format as the command log file. These files are not described here, but are explained in the appropriate user's manuals.

THE COMMAND LOG FILE

Each time you start PC-CARDS, the program opens the
PCCARDS.CMD file and stores in it all the commands
you use during the program session.

When you run PC-CARDS again, the system renames the
command log file from PCCARDS.CMD to
PCCARDS.CM$. Any existing PCCARDS.CM$ 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

To run a macro or a command log file, follow the steps below.

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 meanings 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 upper case 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	Ratsnest display toggle
120	Set pin type
121	Set pin logical equivalence
122	Set aperture number
124	Histogram display toggle
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 crosshair display toggle
198	Resume execution from file
199	Interrupts in com file

APPENDIX C. USING PC-CARDS 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-CARDS 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.
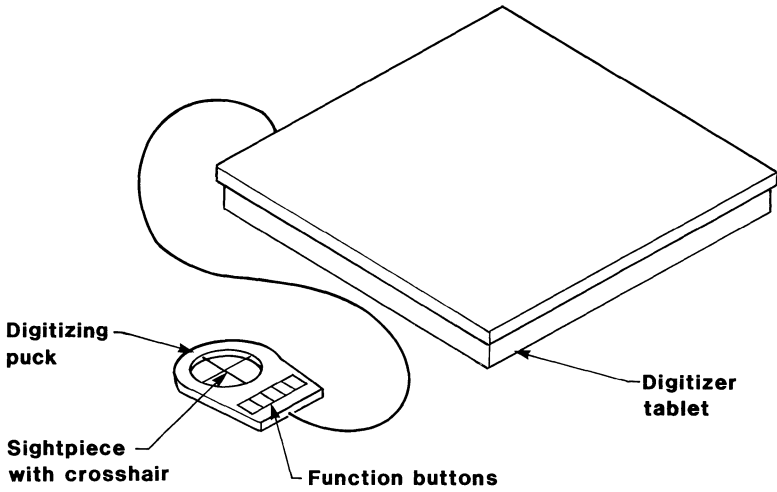
Figure C-1. Digitizer

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 (PCCARDS.CFG) and returns to the Opening Menu. Now, when PC-CARDS 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-CARDS.

SETTING UP THE DIGITIZER

When you start PC-CARDS, 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-CARDS, 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.

Switch 1
OFF ON

OFF	ON		
	X	1	⌉
	X	2	⌐ 1200 baud
	X	3	
X		4	⌋
X		5	Parity disabled
X		6	Parity odd
X		7	1 stop bit
	X	8	8 bits per character

Switch 2
OFF ON

OFF	ON		
X		1	No pushbutton
X		2	No space
X		3	No [Return]
X		4	No line feed
	X	5	Hi-res
X		6	⌉ Ports sending out
X		7	⌋ coordinate data
	X	8	Audible alarm enabled

Switch 3
OFF ON

OFF	ON		
X		1	Not used
	X	2	⌉ 100 coordinate
X		3	⌋ points per second
	X	4	Continuous mode
X		5	5-button cursor pen
X		6	Inch data
	X	7	Binary data
X		8	Force RS-232 signals active

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.

SW1

OFF (↑)	X	X	X		X	X
ON (↓)				X		
	1	2	3	4	5	6

SWITCHES	FUNCTION
1,2,3,4,5,6	Baud rate = 4800

SW2

OFF (↑)	X	X	X		X	X
ON (↓)				X		
	1	2	3	4	5	6

SWITCHES	FUNCTION
1	No parity
2	Bit 8 =1
3	ASCII data
4	String
5,6	Stream

86190

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-CARDS 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-CARDS.

STARTING THE PROGRAM IN DIGITIZER INPUT MODE

When you enter PC-CARDS in the digitizer input mode, the PC-CARDS Configuration Screen will appear as shown in Figure C-6.

PC-CARDS Configuration

Database filename: None

Directory path:
   Current

Security Device port... port 1
Digitizer port.. port 1
Digitizer bell... On
Video save to disk... Off
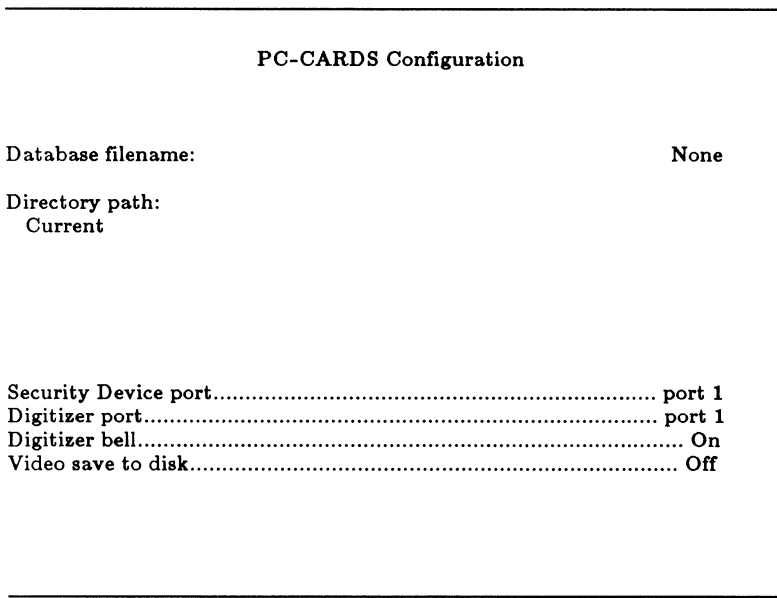
Figure C-6. PC-CARDS Configuration Screen in the Digitizer Input Mode

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-CARDS."

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 PCB layout 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.

Reference point 2

Measured distance between
point 1 and 2:
    X = 10 inches
    Y = 8 inches

Coordinate values to enter at reference point 2
to obtain a 1:1 scaled database:
    X_2 = 500
    Y_2 = 400

2 to 1 Scale Layout

0,0
Reference point 1
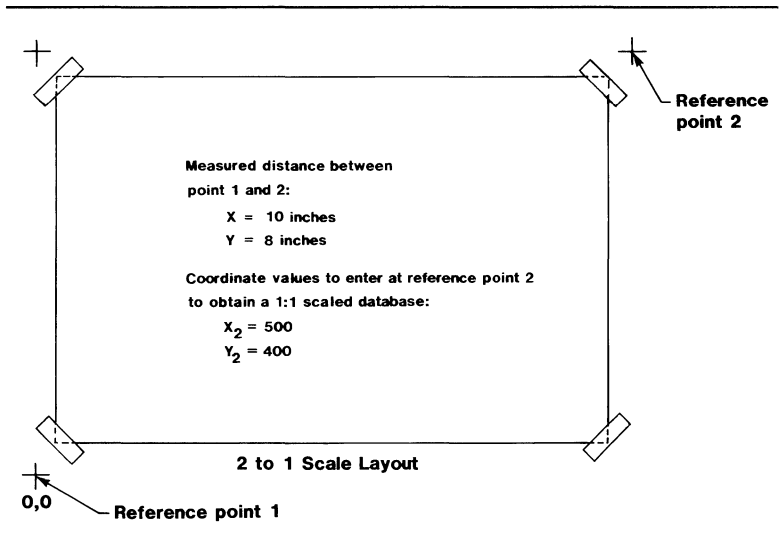
86089

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-CARDS
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-CARDS Configuration Screen.

APPENDIX D. COLOR SELECTION

This appendix is intended for expert users. It describes the default colors for PC-CARDS 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.

```
Bits: 11   10   9   8   7   6   5   4   3   2   1   0
       x    x   x   x   x   x   x   x   x   x   x   x
       |    |   |   |   |   |   |   |   |   |   |   |
       \    |   |   /   \   |   |   /   \   |   |   /
            Red              Green           Blue
```
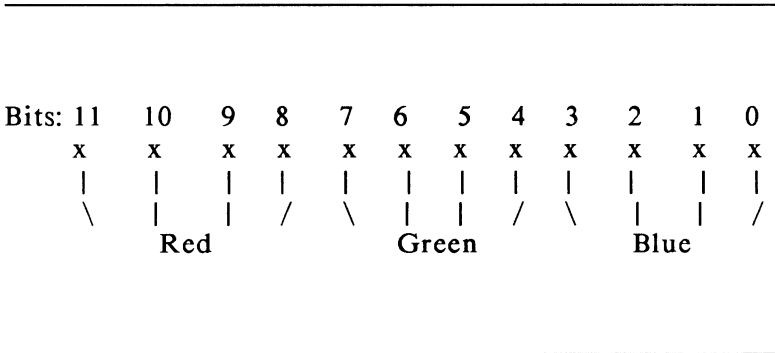
Figure D-1. Color Fields

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 the 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.

INDEX