p-cad

From ACCEL Technologies, Inc.

Master Designer

Version 8.5

Design Manager
Volume 1

October 1995

October 1995

Contents

Contents

About This Manual

The *Design Manager* manual describes the Design Manager module, one of the five major operating programs of P-CAD.

This manual is for use after P-CAD is installed. You need to understand basic CAD and CAE concepts and some commands in the DOS operating system to use P-CAD. As a starting point, the *P-CAD Tutorial* provides a good overview of the system.

Following this section is a conventions section that describes the terminology and symbols used throughout this manual.

The *Design Manager* manual consists of eight chapters, 3 appendixes, and an index. Each chapter contains conceptual and procedural information that explains how to use the Design Manager tools.

Chapter 1	"Overview" explains how the Design Manager module fits in the P-CAD design flow and briefly discusses the tools in the module.
Chapter 2	"Maintaining Design Files" explains how to create new directories and files that will contain the schematic sheets and PCB layouts.
Chapter 3	"Configuring the System" explains how to specify the text editor and the security device port and the printer port.
Chapter 4	"Monitoring Jobs" explains how to check the state of the current operations in your P-CAD account, your login access, and lists of locked files and other logged-in users.
Chapter 5	"Generating Reports" explains how to create text files such as the packaging, wire, component, and node lists and the bill of materials by using either the P-CAD graphical interface or the command line interface.

Chapter 6	"Comparing Netlists" explains how to compare the wires, signals, or pins on two schematic sheets, two PCB databases, or a schematic and a database, and how the *Netlist Comparison* tool reports the differences. You can compare netlists by using either the P-CAD graphical interface or the command line interface.
Chapter 7	"Annotating Engineering Changes" explains how to make engineering changes to a PCB database or a schematic database.
Chapter 8	"Editing Report Files" explains how to use a text editor to add, edit, delete, rename, copy, and print report files or other ASCII files.
Appendix A	"System Limits" describes name length limits; maximums for database items such as pins, components, and nets; and ranges for other items.
Appendix B	"Filename Extensions" lists the P-CAD filename extensions and the tools that produce the files.
Appendix C	"Reserved Words" lists DOS reserved device names and P-CAD attribute keywords.

Conventions

This manual uses the following conventions:

→	Connects commands. Commands following the arrow appear on submenus. For example *File*!*Load*
*	One or more characters can occupy the asterisk's position. Also known as a wildcard.
☞	In text, introduces a procedure that explains how to do a task.
[ ] – [ ]	Press the keys simultaneously. For example [Shift] – [F1]
[↵]	The return key. Press this key after typing data in a data entry box or on a message line or to accept a default. You can click left in many P-CAD tools instead of pressing [↵].
Enter	Indicates you need to press [↵] after typing data.
[Space]	The space bar. You can use this key to digitize a point within the drawing area.
italics	Indicates variable characters.
boldface	Indicates characters you enter from the keyboard; for example Enter **sheet7** in the data entry box.

boldface italics Indicates variable characters you enter from the keyboard; for example

Enter ***filename*** in the data entry box.

Click left Press and release the left button (button 1) on the mouse.

Click middle Press and release the middle button (button 2) on the mouse.

Click right Press and release the right button (button 3) on the mouse. If you are using a two-button mouse, press Ctrl and button 2 simultaneously to clickright.

Cycle Click left repeatedly on a cycle box until the item you want is selected. A cycle box is indicated by ◇.

Select Move the cursor to an item or point and press ↵, Space, or click left.

Overview 1

The Design Manager module provides file management tools that allow you to create and manage your design directories and files. It also provides tools that allow you to generate reports; view, edit, copy, delete, and print ASCII files; specify the default text editor; and implement engineering changes to schematics or PCB databases.

To understand how Design Manager works, you need to know the answers to these questions:

- How does Design Manager fit in the P-CAD design flow?

What tools does Design Manager provide?

How Design Manager Fits in the P-CAD Design Flow

The Design Manager module is one of five modules that make up the P-CAD system. It allows you to create and maintain project files. This is the first module that is displayed when you run P-CAD. The other four modules are as follows:

- The Schematic Tools module allows you to create and edit schematic databases.
- The PCB Tools module allows you to create and edit PCB databases.
- The Library Manager module allows you to set the search paths and libraries for each design. This module lets you create and edit parts and symbols and maintain the archived libraries in which they reside.
- The Interfaces module links P-CAD input and output to other schematic capture systems.

What Tools Design Manager Provides

The Design Manager tools support your work throughout the design process. You can create new design directories and define schematic sheets and PCB layout filenames at the beginning of a project. When you have a design to work with, you can run various reports about the design, then view the reports to provide an overview of the database contents. You can also implement engineering changes to the PCB database or schematic sheets.

The first time you start P-CAD (or select the *Design Manager* button), the screen shown in Figure 1-1 appears.The P-CAD opening screen with the *Design Manager* button selected contains the following tools:

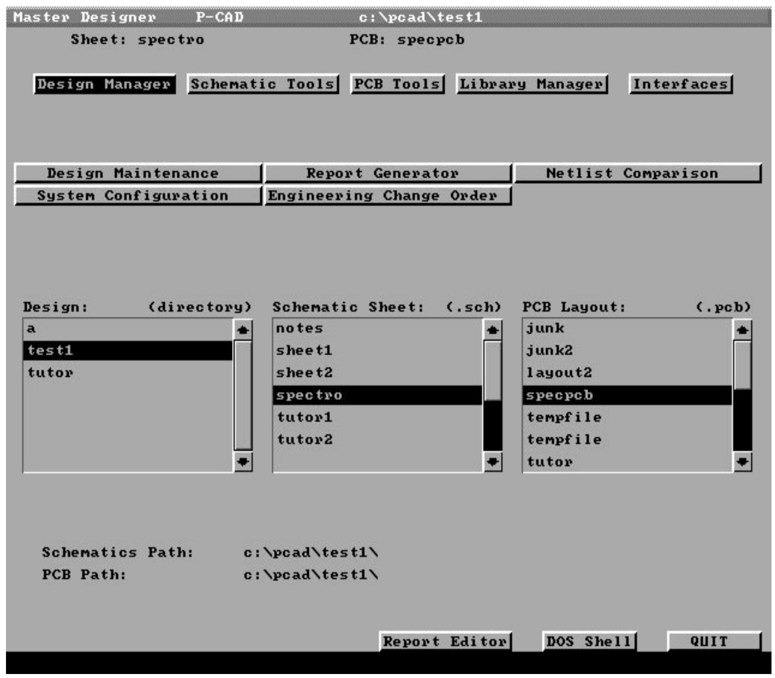

Figure 1-1. PCAD Opening Screen

- *Design Maintenance* allows you to add new design directories, schematic sheets, and PCB layouts. You can delete, copy, and rename existing directories, schematic sheets, or PCB layouts. You can specify paths to files in other directories or drives. You can also specify part cross-reference files and hierarchical symbol files that you use within the current design directory.
- *System Configuration* allows you to specify the text editor when you select the *EDIT* button on the Report Editor screen. It also

allows you to specify which port you are using for your security device and which port you are using for your printer.

- *Report Generator* creates ASCII-formatted report files that provide information such as packaging, wire, component, node, and materials lists.
- *Netlist Comparison* compares electrical connectivity between two schematic sheets, two PCB databases, or one or more schematic sheets and a PCB database, and reports the differences.
- *Engineering Change Order (ECO)* makes forward-annotation or backward-annotation between schematics and PCB databases and reports the changes.
- *Report Editor* lets you edit, add, delete, rename, copy, and print report files or other ASCII files associated with the design.

The rest of the chapters in this manual describe each of the tools in greater depth.

Maintaining Design Files 2

The *Design Maintenance* tool is an organizational tool that allows you to create and manage design directories, schematic sheets, and PCB layout files. Here you can select a part cross-reference file and hierarchical symbol files you can use while creating designs in the working directory. You use this tool to initiate a new design or to organize an existing directory.

The *Design Maintenance* tool is important because it sets up a configuration file that the P-CAD system uses to determine which files are associated with each design directory.

This chapter presents information about the following topics:

- opening the Design Manager screen
- maintaining design directories
- maintaining PCB layout files
- maintaining schematic sheets
- selecting a cross-reference file
- selecting hierarchical symbols associated with the design
- exiting the Design Manager screen

To organize the files that make up a final PCB design, you start by creating a design directory, which is your current working directory. Within this design directory, you name the schematic sheets and PCB layout databases and store other files required for building the design. One or more schematic sheets form the logical, symbolic representation of the PCB design. When the schematic sheets are approved for the design, you package them into a PCB database.

You can also indicate the name of a cross-reference file used to package a schematic database into a PCB database and you can list hierarchical symbols stored in your current design directory for use in design simulation. A cross-reference file (*filename*.fil) is an ASCII file that contains information for mapping logic symbols to component part outlines.

Using the Design Maintenance tool updates the filenames listed in the P-CAD opening screen. When you select a design name in the Designs list box, the system automatically displays the filenames of any associated schematic sheets and PCB layouts in the other two list boxes. The list boxes work the same way throughout the P-CAD modules.

The *Design Maintenance* tool also determines which paths and filenames appear in the title bar of all top-level screens. The path and filename show which design you are currently working in and along which path the design is located.

Opening the Design Maintenance Screen

To open the *Design Maintenance* tool from the P-CAD opening screen, select the *Design Manager* button. The button is highlighted and the Design Manager tool buttons are available for selection.

Note: When you first use P-CAD (and no configuration file exists), the Design Maintenance screen is automatically displayed. A decision box asks if you want to create a configuration file. You must add at least one design directory to create the configuration file.

To open the Design Maintenance screen

- Select *Design Maintenance*. The Design Maintenance screen appears as shown in Figure 2-1.

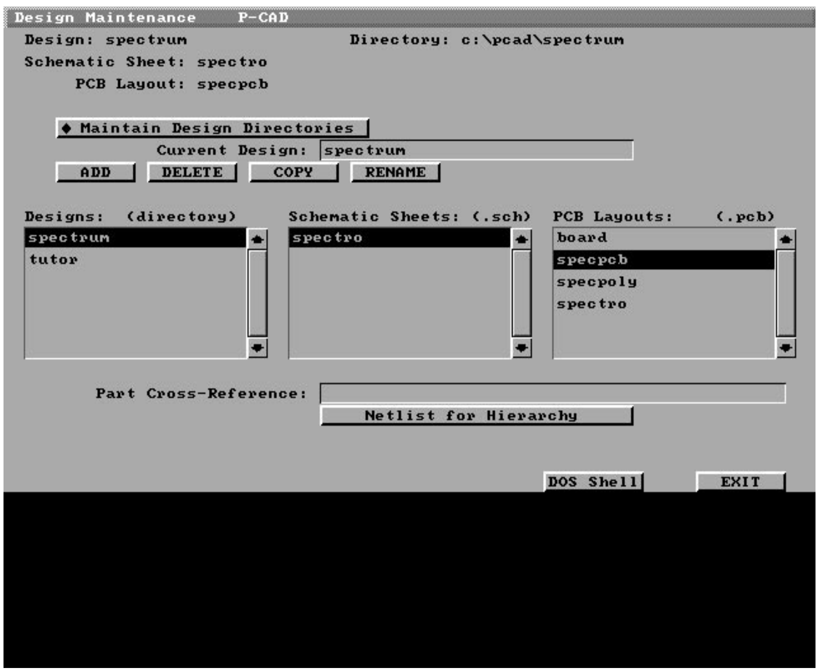

Figure 2-1. Design Maintenance Screen

You can exit the screen without entering a design name, but you lose the automatic file selection features that Design Manager provides to other tools and modules. If you don't enter at least one design name, many features of the P-CAD system are not accessible to you when you select buttons to activate the features.

Maintaining Design Directories

A design directory contains all files associated with one or more PCB databases. You can create a design directory to hold the files associated with creating a PCB before you begin designing the PCB.

The Design Maintenance tool allows you to add, delete, copy, or rename design directories. You can also add design directories that were previously created or copied using DOS commands. The system doesn't list existing designs in the design directory unless you add them as explained in this section.

To add a design directory

1. Cycle to Maintain Design Directories.
2. Select ADD.
3. Enter the design filename or path and filename in the Current Design data entry box.
4. Select EXIT to leave the Design Maintenance screen.

To delete a design directory

> **Caution**: When you delete a design directory, the system permanently deletes all schematic sheets, PCB layouts, and associated files contained in the directory, just as if you used the DOS *del *.** command. Note that any write-protected files that exist within the directory you're deleting are removed from the Design Manager configuration, but they still exist on disk.

1. Cycle to Maintain Design Directories.
2. Select the name of the design directory to delete.
3. Select DELETE. A decision box is displayed.
4. Select the appropriate delete option:

   Delete design *designname* (all files/subdirs)? deletes the entire design.

   Detach design *designname*? disassociates this design from the configuration file without any files actually being deleted from the hard disk.

   CANCEL cancels the operation and does not delete the design.

5. Select *EXIT* to leave the Design Maintenance screen.

To copy or rename a design

The system does not allow you to overwrite existing directories or files. For example, if you attempt to copy or rename one existing design directory by giving it the name of another existing directory, the system doesn't make a copy of the directory or any of the files contained in the directory.

1. Cycle to Maintain Design Directories.
2. Select the name of the design directory to copy or rename from the Design Directory list box.
3. Select *COPY* or *RENAME.*
4. Enter the new design filename in the Current Design data entry box. A decision box asks you to confirm the action.

5. Select *YES* to accept or *NO* to cancel the action.
6. Select *EXIT* to leave the Design Maintenance screen.

Maintaining Schematic Sheets

Schematic sheets contain logic symbols, wires, buses, and intelligent electrical information to be used in the PCB design. You can use the Design Maintenance tool to add, delete, copy, or rename schematic sheets.

To add a schematic sheet

1. Cycle to Maintain Schematic Sheets.
2. Select ADD.
3. Enter the schematic sheet filename or path and filename in the Current Schematic Sheet data entry box.

> **Note**: If you use a wildcard character in the data entry box (for example, *.sch or sp??.sch), all files that match this entry are added to the Schematic Sheets list box.

4. Select *EXIT* to leave the Design Maintenance screen.

To delete a schematic sheet

1. Cycle to *Maintain Schematic* Sheets.
2. Select the name of the schematic sheet you want to delete.
3. Select *DELETE*. A decision box pops up.
4. Select the appropriate delete option:

   Delete *filename*.sch? deletes only the schematic sheet file.

   Delete *filename*.*? deletes all files with this filename and any extension.

   Detach *filename*.sch? disassociates this schematic sheet from the current design so it doesn't appear in the Schematic Sheets list box. The schematic file isn't deleted from the directory. It is only disassociated with the design configuration so that when you are using a tool that processes all sheets of the current design, any sheets that you do not want processed are ignored by the tool, but are still retained in your directory.

   CANCEL cancels the operation and doesn't delete any schematic sheets.

5. Select *EXIT* to leave the Design Maintenance screen.

Once you detach a schematic from the design, you must add it again if you want to include it in the design configuration.

To copy or rename a schematic sheet

COPY and *RENAME* don't allow you to write over existing files. Make sure you choose a unique name for the new schematic sheet.

1. Cycle to Maintain Schematic Sheets.
2. Select the name of the schematic sheet to copy or rename from the Schematic Sheets list box.
3. Select COPY or RENAME.
4. Enter the new schematic sheet filename in the Current Schematic Sheet data entry box.

   When you rename a schematic sheet, a decision box pops up where you can select the appropriate option:

   Rename *filename*.sch? renames only the schematic sheet file.

   Rename *filename*.*? renames all files with this filename and any extension in the current design directory.

   CANCEL cancels the operation and doesn't rename any files.
5. Select *EXIT* to leave the Design Maintenance screen.

Maintaining PCB Layout Files

PCB layout files are files that eventually contain the completed PCB design. Many of the tools contained in the P-CAD modules expect the PCB filename to exist in the Design Manager module. You can use the *Design Maintenance* tool to add, delete, copy, or rename PCB layout files.

To add a PCB layout

When you add a new PCB layout, the system creates an empty PCB database file and places it in the current design directory. If you add an existing PCB layout that was created by using another tool, the existing layout is then accessible in the system from the PCB Layouts list box.

1. Cycle to Maintain PCB Layouts.
2. Select ADD.
3. Enter the PCB layout filename or path and layout filename in the Current PCB Layout data entry box.

Note: If you use a wildcard character in the data entry box (for example, *.pcb or sp??.pcb), all files that match this entry are added to the PCB Layouts list box.

4. Select EXIT to leave the Design Maintenance screen.

To delete a PCB layout

You can delete PCB layouts from the system by using *DELETE.* You can also detach a PCB layout from its design directory. The *Detach* option is useful if you don't want a particular PCB layout to be processed with other PCB layouts in the directory but you don't want to delete it. You can also save unattached layouts to keep track of revisions to the PCB layout. You can always reattach a PCB layout to your design by using the *ADD* button.

1. Cycle to *Maintain PCB Layouts.*
2. Select the name of the PCB layout to delete from the PCB Layouts list box.
3. Select *DELETE.* A decision box pops up.
4. Select the appropriate delete option:

   Delete *filename*.pcb? deletes only the PCB layout file.

   Delete *filename*.*? deletes all files with this filename and any extension from the current design directory.

   Detach *filename*.pcb? disassociates this PCB database from the design directory list so it doesn't appear in the PCB Layouts list box.

   CANCEL cancels the operation and doesn't delete any PCB layout.
5. Select *EXIT* to leave the Design Maintenance screen.

To copy or rename a PCB layout

COPY and *RENAME* don't allow you to write over existing files. Make sure you choose a unique name for the new PCB layout.

1. Cycle to *Maintain PCB Layouts.*
2. Select the name of the PCB layout to copy or rename.
3. Select *COPY* or *RENAME.*
4. Enter the new PCB layout filename in the Current PCB Sheet data entry box.

   When you rename a PCB layout, a decision box pops up where you can select the appropriate option:

   Rename *filename*.pcb? renames only the PCB layout file.

Rename *filename*.*? renames all files with this filename and any extension in the current design directory.

CANCEL cancels the operation and doesn't rename any files.

5. Select *EXIT* to leave the Design Maintenance screen.

Selecting a Cross-Reference File

A cross-reference file (*filename*.fil) maps logical symbols to physical parts. The cross-reference file must exist in the current directory or be found along the path specified by *Set Libs.* & *Search Paths* in the Library Manager module. You create or edit a cross-reference file by using the *Report Editor* tool or any text editor. See the section on packaging the schematic in the *Schematic Tools* manual for information about cross-reference files.

To select a cross-reference file for the current design

1. Select the Part Cross-Reference data entry box on the Design Maintenance Screen. A file selector list box appears.
2. Enter the name of the cross-reference file in the file selector data entry box.

   or

   Select an existing cross-reference filename from the file selector list box.
3. Select *OK.*
4. Select *EXIT* to leave the Design Maintenance screen.

Selecting Hierarchical Symbols Associated With the Design

If you are using a hierarchical schematic design, you must list the hierarchical symbols used from within your current design directory by selecting them from among all the symbol files in the directory. Listing the hierarchical symbols adds them to the design configuration. Once they are listed, the program can identify the symbols and extract any netlist information it needs during the process of creating expanded netlists for the entire design.

A hierarchical design should be used only for logic simulation. Although you can package a hierarchical design into a PCB database, back- annotation is not supported. Any updates to the PCB file must be made manually to the schematic file. Refer to the section on using the Schematic Editor in the *Schematic Tools* manual for more information about creating a hierarchical design.

To list hierarchical symbols in your design

1. Select Netlist for Hierarchy.
2. Enter the symbol filename in the file selector data entry box.

   or

   Select an existing symbol filename from the file selector list box. To deselect a filename or to select more than one filename, press as you select the filename in the list box.
3. Select OK.
4. Select EXIT to leave the Design Maintenance screen.

Exiting the Design Maintenance Screen

To exit the Design Maintenance screen

Select *EXIT* in the Design Maintenance screen.

The P-CAD opening screen is displayed. The list boxes on the screen show the files you added. The highlighted design is your current working design directory. You can use other P-CAD features to begin working with the PCB design.

Configuring the System 3

This chapter applies only to P-CAD running on DOS-based platforms.

The *System Configuration* tool enables you to set the default text editor and the security device port.

This chapter describes the following tasks:

- opening the System Configuration screen
- specifying the text editor
- selecting the security device port
- selecting the printer port
- exiting the System Configuration screen

Opening the System Configuration Screen

To open the System Configuration screen from the P-CAD opening screen, select the *Design Manager* button. The button is highlighted and the *Design Manager* tool buttons are available for selection.

To open the System Configuration screen

- Select *System Configuration.* The System Configuration screen is displayed, as shown in Figure 3-1.

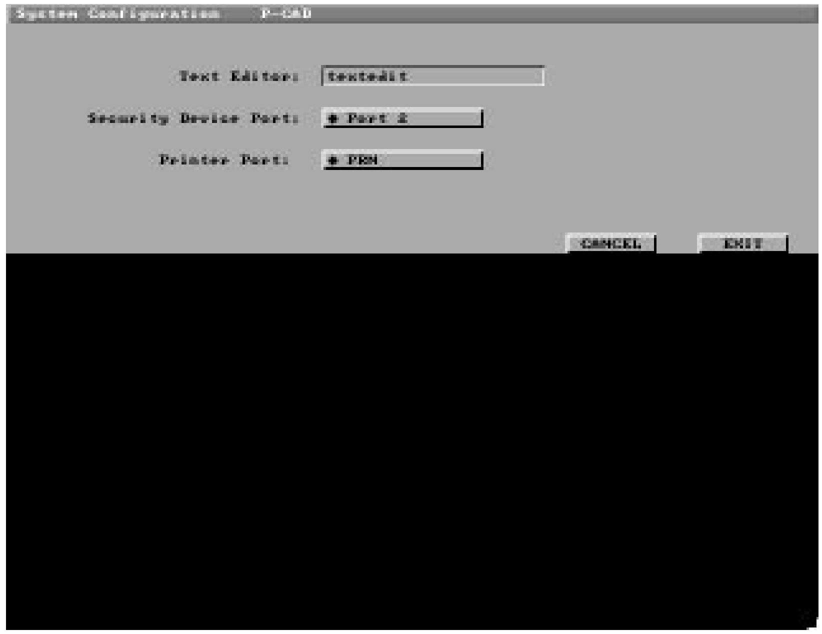

Figure 3-1 . System Configuration Screen

Specifying the Text Editor

The text editor you specify here is the default text editor the system uses when you edit an ASCII file in the *Report Editor*.

To specify the text editor

1. Select the Text Editor data entry box.
2. Enter the name of the text editor you want to use.

Selecting the Security Device Port

Before you can use your P-CAD software, the security device must be installed. You must specify which COM port you are using for the security device, so that the system recognizes that the device is installed.

To specify the security device port

- Cycle the Security Device Port to *Port 1* or *Port 2*.

> **Note**: The system checks port 1 first to see if the security device is installed. If port 1 fails, or there is no security device at port 1, meaning you have it installed at port2, the system automatically checks port 2.

Specifying the correct comport in the system configuration screen takes less time for the system to check than specifying the wrong port, or no port at all.

Selecting the Printer Port

The printer port you specify here is the printer port the system uses when you print from the Report Editor. The system default is *PRN*.

To specify the printer port

- Cycle the Printer Port to *PRN*, *LPT1*, or *LPT2*.

Exiting the System Configuration Screen

To exit the System Configuration Screen

- Select *EXIT* on the System Configuration screen.

  or

  Select *CANCEL* to exit without making any changes.

  The P-CAD opening screen is displayed. You can use other P-CAD features to continue work with the PCB design.

Monitoring Jobs 4

This chapter applies only to P–CAD running on UNIX–based platforms.

The Process Monitor shows your login access, all locations where you logged into the system, and the current P–CAD operations in your account.

This chapter describes the following:

- opening the Process Monitor window
- viewing the list of logged–in users
- viewing the list of locked files
- viewing the list of active jobs
- exiting the Process Monitor

Opening the Process Monitor Window

Open the Process Monitor window with the Design Manager icon selected.

To open the Process Monitor window

- Select *Monitor Jobs.* The system displays the Process Monitor window shown in Figure 4-1.

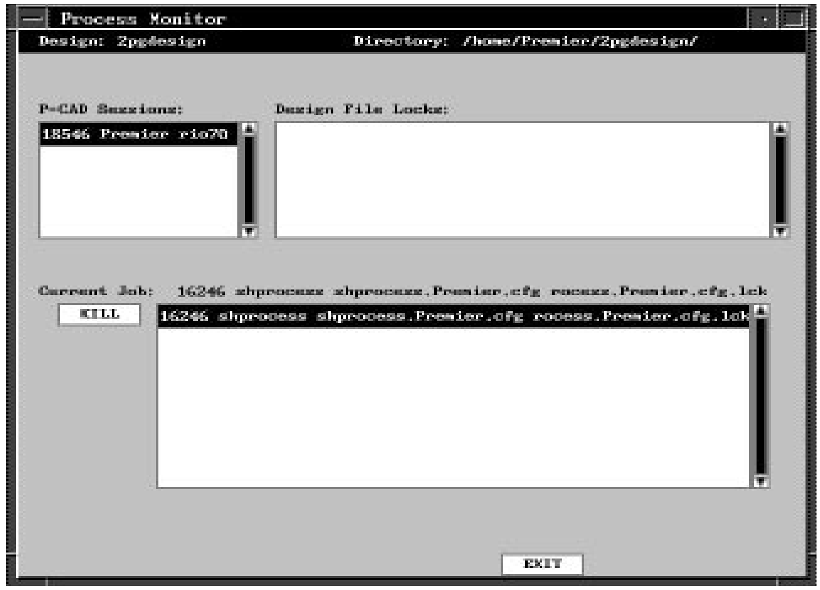

Figure 4-1. Process Monitor Window

Checking Logged–In Users

The P–CAD Sessions list box shows users who logged in with the same login account name as yours. If you logged in twice, the Process Monitor lists you twice. Each entry consists of the process ID number (the way UNIX identifies a job), the user name, and the name of the host system. The following is an example of the format:

18546 Premier rio70

Viewing the List of Locked Files

The Design File Locks list box shows the locked files.

P–CAD locks files to prevent simultaneous updates. The system locks files by creating a lock file, which is an empty file of the same name with a .@lk extension added on. For example, if you opened pcad.cfg, the system would create an empty file named pcad.cfg.@lk. the system automatically deletes the lock file when you close your original file.

If the process that locked the file dies before it can unlock it, you may see the following message when you try to re–open the locked file:

shcfg: Advisory lock on file *filename* broken on *date time*

If you don't have write permission in the directory where the system tries to open the lock file, you'll see the following error message:

> shcfg: Unable to remove advisory lock on file *filename* on *date time*

Neither of these error messages will affect P–CAD operation.

If you have problems with the file locking feature you can disable it entirely by setting the following environment variable:

```
setenv PCAD_NOLOCK 1
```

Viewing the List of Jobs in Process

The Current Job list box shows current P–CAD jobs. The Process Monitor automatically updates the jobs. If necessary, you can kill a job before it completes, as follows:

To kill a job in process

1. Select the job you want to kill in the Current Job list box.
2. Select KILL. The system displays a decision box for confirmation.
3. Select YES to confirm your decision.

Exiting the Process Monitor

To exit the Process Monitor

- Select *EXIT* in the Process Monitor window.

Generating Reports 5

The *Report Generator* tool in the Design Manager module produces report files that contain information about a specified design. With *Report Generator* you can create the following ASCII reports:

- Component List (*filename*.cmp) contains a list of component symbol names from a schematic design and the net names assigned to each pin on the component.
- Node List (*filename*.nde) contains a list of net names and net attributes in a schematic, and the pins and components to which they're connected.
- Packaging List (*filename*.pkl) contains one record for each symbol gate and is sorted by reference designator. Each record includes the reference designator, pin numbers, and net names.
- Wire List (*filename*.wrl) contains net names, net attributes, and respective reference designator and package pin numbers.
- Materials List (*filename*.mat) contains a list of component symbols used in the schematic.

This chapter presents information about

- using *Report Generator* to create and run reports
- using the command line mode to create and run reports

Running the *Report Generator* tool from within the Design Manager module allows you to see at a glance what settings and reports you use. A netlist is automatically generated for the schematic sheet selection you specify. You can run several reports at one time, but need only process one sheet or set of sheets to generate the reports.

The command line mode allows you to run the program by entering options on the command line at the DOS prompt. These options are the input netlist, the reports to produce, and the name of a keyword list files. After you enter the command line, the system runs automatically with no further input from you.

Running from the command line is useful if you want to process several different sets of schematic sheets from different designs in a batch process. No automatic netlist generation takes place when you use command line mode to generate reports.

Note: You must have at least one schematic database before you can generate reports.

Using *Report Generator* to Create and Run Reports

Report Generator extracts and links intelligent net information from schematic databases and creates ASCII report files. You can extract reports from a single schematic sheet or from an entire multisheet schematic design. When the tool produces reports from an entire schematic design, the system automatically links the extracted netlists of all sheets listed in the Schematic Sheet list box in the opening screen and creates an expanded netlist. The filename extensions for the reports are:

- *filename*.nlt: netlist for a single schematic sheet
- *filename*.xnl: expanded netlist for a multisheet design

The filename is the sheet name for single–sheet reports and the design name for reports from multisheet designs.

You can run one or more reports at a time or you can exit *Report Generator* without generating a report.

Opening the Report Generator Screen

You open the Report Generator screen from the P–CAD opening screen with the *Design Manager* button selected.

To open the Report Generator screen

- Select *Report Generator.* The Report Generator screen appears as shown in Figure 5-1.

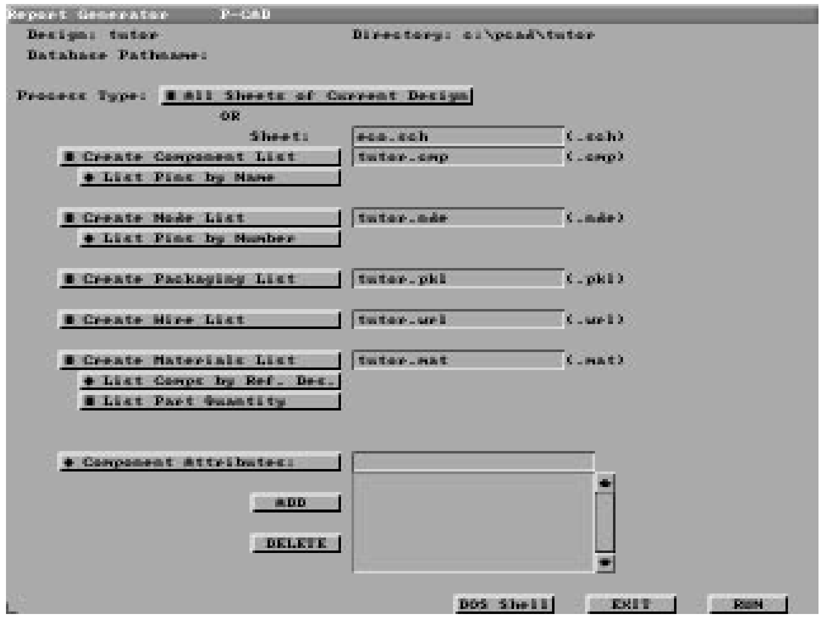

Figure 5-1. Report Generator Screen

Creating a Component List

A component list includes the following information:

- component (symbol) names
- instance names and reference designators
- pin names or part/pin package numbers
- associated net (signal) names
- user–selected attributes

To create a component list for all sheets

1. Fill the All Sheets of Current Design check box.
2. Fill the Create Component List check box.
3. Select the Create Component List data entry box to change the output filename.
4. Enter the output filename for the component list if you want it to be different from the input filename.
5. Cycle to *List Pins by Number, List Pins by Name,* or *Don’t List Pins.*
6. Add or delete attributes.

To add attributes

[1] Cycle to Component Attributes next to the Attributes list box.

[2] Select ADD.

[3] Enter the attribute keyword in the data entry box. If you enter **all**, the system carries all component attributes.

To delete attributes

[1] Cycle to *Component Attributes* next to the Attributes list box.

[2] Select the attribute keyword in the list box.

[3] Select *DELETE.*

To create a component list for one sheet

1. Empty the All Sheets of Current Design check box.
2. Select the Sheet data entry box.
3. Enter the schematic sheet filename in the data entry box in the file selector window.

   or

   Select the schematic sheet filename from the file selector list box.
4. Select *OK* to enter the sheet name in the Sheet data entry box.
5. Fill the Create Component List check box.
6. Select the Create Component List data entry box to change the output filename.
7. Enter the output filename for the component list if you want it to be different from the input filename.
8. Cycle to *List Pins by Number, List Pins by Name,* or *Don't List Pins.*
9. Add or delete attributes for one sheet the same way you would if you were processing all sheets of the design.

Component List Examples

This section shows samples of component lists generated using the Design Manager. Only the first page of each report is shown.

Component List (Pins Listed by Name)

The component list in Figure 5-2 was produced from a netlist extracted from a schematic database. The pins are identified by name, and the attribute, FTYPE, is listed for each occurrence.

The component list has one record for each component (schematic symbol) in the design. Each record of the component list includes the following:

- *symbol name* is the base filename of the component symbol assigned using *File*!*Save* in the Symbol Editor; for example, 8250.
- *instance name* is the component or device name you assign using *Name*!*Component* in the Schematic Editor, or the system default name. UC00000001 is an example of a system–assigned net name.
- *pin list* lists all the symbol's pins names (and net assignments, if applicable). The entire pin list is enclosed in parentheses. Each entry in the pin list includes the following:
  - *pin name* is the name of the pin you assign using *Enter*!*Pin* in the Symbol Editor.
  - *net name* is the name of the net connected to the pin and is assigned by the system when you enter a wire in Schematic Editor or by you with *Name*!*Net*. If you use *Enter*!*Wire*, the system generates a net name. UN00010001 is an example of a system–assigned net name.
- *attribute list* lists all the specified attribute keywords and values. The design for which the report in Figure 5-2 was generated uses only one attribute, FTYPE; for example, FTYPE="INP". The entire list is enclosed in square brackets. When two or more attributes are listed, each attribute is on a separate line and attributes are separated by commas. You enter attributes using *Enter*!*Attribute* in the Schematic Editor or the Symbol Editor.

```
%*********************************************************************
%                                                                    *
%                                                                    *
%     Program  :   PC*FORM VERSION 8.5                               *
%     Date     :   Aug 15 1995                                       *
%     Time     :   10:16:30 AM                                       *
%     File In  :   aca.nlt                                           *
%     File Out :   aca.cmp                                           *
%     Format   :   P-CAD COMPONENT LIST                              *
%                                                                    *
%*********************************************************************
ACA(  )
BEGIN
  8250:UC000000    ( MR      =RESET,    D0      =DATA0,
                     D1      =DATA1,    D2      =DATA2,
                     D3      =DATA3,    D4      =DATA4,
                     D5      =DATA5,    D6      =DATA6,
                     D7      =DATA7,    ADS'    =GND,
                     DISTR   =GND,      DOSTR   =GND,
                     DISTR'  =IO-READ', DOSTR'  =IOW',
                     A0      =A0,       A1      =A1,
                     A2      =A2,       CS1     =UN000009,
                     CS0     =UN000009, CS2'    =ENABLE',
                     XTAL2   =,         XTAL1   =CLK,
                     RLSD'   =RSLD',    DSR'    =DSR',
                     CTS'    =CTS',     RI'     =RI',
                     SIN     =RX_DATA,  RCLK    =SIO_CLK,
                     OUT1'   =UN000007, OUT2'   =UN000005,
                     INTRPT  =UN000000, CSOUT   =,
                     DDIS    =,         N/C     =,
                     DTR'    =DTR',     RTS     =RTS,
                     SOUT    =TX_DTA1,  BAUDOUT'=SIO_CLK )
                   [ FTYPE   ="INP" ]
  74ls245:UC000001 ( B1      =DATA0,    B2      =DATA1,
                     B3      =DATA2,    B4      =DATA3,
                     B5      =DATA4,    B6      =DATA5,
                     B7      =DATA6,    B8      =DATA7,
                     G'      =ENABLE',  DIR     =IO-READ',
                     A1      =D0,       A2      =D1,
                     A3      =D2,       A4      =D3,
                     A5      =D4,       A6      =D5,
                     A7      =D6,       A8      =D7 )
  osc:UC000002     ( OSC     =OSC )
  75150:UC000003   ( INA     =DTR',     INB     =UN000004,
                     INC     =RTS,      OUTY1   =EIA_DTR,
                     OUTY2   =EIA_RTS )
  75150:UC000004   ( INA     =TX_DTA2,  INB     =UN000001,
                     INC     =,         OUTY1   =TX_DTA6',
                     OUTY2   = )
  75154:UC000005   ( OUT     =RSLD',    EN      =+5V,
                     IN      =EIA_CDET )
  75154:UC000006   ( OUT     =DSR',     EN      =+5V,
                     IN      =EIA_DSR )
  75154:UC000007   ( OUT     =CTS',     EN      =+5V,
                     IN      =EIA_CTS )
  75154:UC000008   ( OUT     =RI',      EN      =+5V,
                     IN      =EIA_RI )
  75154:UC000009   ( OUT     =RX_DTA2,  EN      =+5V,
                     IN      =EIA_RXD )
```

Figure 5-2. Component List With Pins Listed by Name

Component List (Pins Listed by Number)

The component list in Figure 5-3 was produced from a netlist extracted from a schematic database. The pins are identified by number. No attributes are listed.

The component list has one record for each component (schematic symbol) in the design. Each record of the component list includes the following:

- *symbol name* is the base filename of the component symbol; for example, 74ls245.
- *instance name* is the component or device name you assign using *Name*!*Component* in the Schematic Editor, or the default name automatically assigned by the system. UC00000001 is an example of a system–assigned net name.
- *reference designator* is assigned using *Enter*!*Ref. Des. & Section* in the Schematic Editor; for example, U1.
- *pin list* lists all the symbol's pins by package pin number and their net assignment. The entire pin list is enclosed in parentheses. Each entry in the pin list includes the following:
  - — *package pin number* is assigned using *Enter*!*Ref. Des. & Section* in the Schematic Editor; for example, 35.
  - — *net name* is the name of the net connected to the pin and is assigned by the system when you enter a wire in Schematic Editor or by you with *Name*!*Net*. If you use *Enter*!*Wire*, the system generates a net name. UN00010001 is an example of a system–assigned net name.
- attribute list lists all the specified attribute keywords and values. The design for which the report in Figure 5-3 was generated uses only one attribute, FTYPE; for example, FTYPE="INP". The entire list is enclosed in square brackets. When two or more attributes are listed, each attribute is on a separate line and attributes are separated by commas. You enter attributes using Enter!Attribute in the Schematic Editor or the Symbol Editor.

```
%*********************************************************************
%     Program  :   PC*FORM VERSION 8.5                               *
%     Date     :   Aug 15 1995                                       *
%     Time     :   10:28:16 AM                                       *
%     File In  :   aca.nlt                                           *
%     File Out :   aca.cm1                                           *
%     Format   :   P-CAD COMPONENT LIST                              *
%                                                                    *
%*********************************************************************
ACA(  )
BEGIN
  8250:UC000000:U1          ( 35      =RESET,    1       =DATA0,
                              2       =DATA1,    3       =DATA2,
                              4       =DATA3,    5       =DATA4,
                              6       =DATA5,    7       =DATA6,
                              8       =DATA7,    25      =GND,
                              22      =GND,      19      =GND,
                              21      =IO-READ', 18      =IOW',
                              28      =A0,       27      =A1,
                              26      =A2,       13      =UN000009,
                              12      =UN000009, 14      =ENABLE',
                              17      =,         16      =CLK,
                              38      =RSLD',    37      =DSR',
                              36      =CTS',     39      =RI',
                              10      =RX_DATA,  9       =SIO_CLK,
                              34      =UN000007, 31      =UN000005,
                              30      =UN000000, 24      =,
                              23      =,         29      =,
                              33      =DTR',     32      =RTS,
                              11      =TX_DTA1,  15      =SIO_CLK )
  74ls245:UC000001:U2       ( 18      =DATA0,    17      =DATA1,
                              16      =DATA2,    15      =DATA3,
                              14      =DATA4,    13      =DATA5,
                              12      =DATA6,    11      =DATA7,
                              19      =ENABLE',  1       =IO-READ',
                              2       =D0,       3       =D1,
                              4       =D2,       5       =D3,
                              6       =D4,       7       =D5,
                              8       =D6,       9       =D7 )
  osc:UC000002:U3           (8        =OSC )
  75150:UC000003:U4         ( 2       =DTR',     1       =UN000004,
                              3       =RTS,      6       =EIA_DTR,
                              7       =EIA_RTS )
  75150:UC000004:U5         ( 2       =TX_DTA2,  1       =UN000001,
                              3       =,         6       =TX_DTA6',
                              7       = )
  75154:UC000005:U6         ( 3       =RSLD',    1       =+5V,
                              2       =EIA_CDET )
  75154:UC000006:U6         ( 6       =DSR',     4       =+5V,
                              5       =EIA_DSR )
  75154:UC000007:U6         ( 8       =CTS',     10      =+5V,
                              9       =EIA_CTS )
  75154:UC000008:U7         ( 3       =RI',      1       =+5V,
                              2       =EIA_RI )
  75154:UC000009:U7         ( 8       =RX_DTA2,  10      =+5V,
                              9       =EIA_RXD )
```

Figure 5-3. Component List With Pins Listed by Pin Number

Creating a Node List

A node list is the ASCII version of a netlist. A node list (*filename*.nde) shows the pin–to–pin connections for each net in a design and includes the following information:

- net (signal) name
- associated pin names or package pin numbers
- net attributes
- symbol name
- instance name and reference designator
- hierarchical path

To create a node list for all sheets

1. Fill the All Sheets of Current Design check box.
2. Fill the Create Node List check box.
3. Select the Create Node List data entry box to change the output filename.
4. Enter the output filename of the node list if you want it to be different from the input filename.
5. Cycle to *List Pins by Number* or *List Pins by Name.*
6. Add or delete attributes.

   To add attributes

   [1]Cycle to Node Attributes next to the Attributes list box.

   [2] Select ADD.

   [3]Enter the attribute keyword in the data entry box. If you ent

   To delete attributes

   [1]Cycle to *Node Attributes* next to the Attributes list box.

   [2]Select the attribute keyword in the list box.

   [3] Select *DELETE.*

To create a node list for one sheet

1. Empty the All Sheets of Current Design check box.
2. Select the Sheet data entry box.
3. Enter the schematic sheet filename in the data entry box in the file selector window.

   or

Select the schematic sheet filename from the file selector list box.

4. Select *OK* to accept the name.
5. Fill the Create Node List check box.
6. Select the Create Node List data entry box to change the output filename.
7. Enter the output filename of the node list if you want it to be different from the input filename.
8. Cycle to *List Pins by Number or List Pins by Name.*

Node List Example

The node list in Figure 5-4 was produced from a netlist extracted from a schematic database. The node identifies the pins by name.

The node list has one record for each node (net) in the schematic database. Each record in the list includes the following:

- net name *is the name of the net connected to the pin and is assigned by the system when you enter a wire in Schematic Editor or by you with* Name!Net*. If you use* Enter!Wire*, the system generates a net name. UN00010001 is an example of a system–assigned net name.*
- pin names *are the names of all the pins committed to the net. You assign pin names using* Enter!Pin *in the Symbol Editor. D0 is an example of a pin name. The pin names column also lists any net attributes in the node.*
- symbol names *are the base filenames of the symbols to which the pins belong; for example, 8250.*
- instance name *is the component or device name you assign using* Name!Component *in the Schematic Editor, or the system default name. UC00000001 is an example of a system–assigned net name. If a reference designator was assigned, its name is listed following the instance name; for example, 7240:UC00000001:U1.*
- hierarchical design path *to the net lists the names of the components at each level in which the net is located relative to the top level of the hierarchical sheet.*

```
%********************************************************************
%                                                                  *
%     Program  :   PC*FORM VERSION 8.5                             *
%     Date     :   Aug 15 1995                                     *
%     Time     :   10:16:33 AM                                     *
%     File In  :   aca.nlt                                         *
%     File Out :   aca.nde                                         *
%     Format   :   P-CAD NODE LIST                                 *
%                                                                  *
%********************************************************************


%
% NODE      PINS            COMPONENT                    PATHNAME
% ----      ----            ---------                    --------
%


  DATA0                                             <ACA
            D0              8250:UC000000
            B1              74ls245:UC000001
           [WIDTH=12  ]

  DATA1                                             <ACA
            D1              8250:UC000000
            B2              74ls245:UC000001
           [WIDTH=12  ]

  DATA2                                             <ACA
            D2              8250:UC000000
            B3              74ls245:UC000001
           [WIDTH=12  ]

  DATA3                                             <ACA
            D3              8250:UC000000
            B4              74ls245:UC000001
           [WIDTH=12  ]

  DATA4                                             <ACA
            D4              8250:UC000000
            B5              74ls245:UC000001
           [WIDTH=12  ]

  DATA5                                             <ACA
            D5              8250:UC000000
            B6              74ls245:UC000001
           [WIDTH=12  ]

  DATA6                                             <ACA
            D6              8250:UC000000
            B7              74ls245:UC000001
           [WIDTH=12  ]

  DATA7                                             <ACA
            D7              8250:UC000000
            B8              74ls245:UC000001
           [WIDTH=12  ]

  D0                                                <ACA
            A1              74ls245:UC000001
            IO              CON1:UC000051
           [WIDTH=12  ]
```

Figure 5-4. Node List

Creating a Packaging List

A packaging list contains one record for each symbol gate and is sorted by reference designator. You must have already assigned reference designators and packaging information to the symbols to create a packaging list. Use *Enter*!*Ref. Des. & Section* in the

Schematic Editor to assign the reference designator and package section to each symbol.

A packaging list contains the following information:

- reference designator
- part or package pin numbers
- associated net (signal) name
- user–selected attributes

To create a packaging list for all sheets

1. Fill the All Sheets of Current Design check box.
2. Fill the Create Packaging List check box.
3. Select the Create Packaging List data entry box to change the output filename.
4. Enter the output filename of the packaging list if you want it to be different from the input filename.
5. Add or delete attributes.

   To add attributes

   [1] Cycle to *Packaging Attributes* next to the Attributes list box.

   [2] Select *ADD.*

   [3]Enter the attribute keyword in the data entry box. If you enter **all**, the system carries all packaging attributes.

   To delete attributes

   [1] Cycle to *Packaging Attributes* next to the Attributes list box.

   [2]Select the attribute keyword in the list box.

   [3] Select *DELETE.*

To create a packaging list for one sheet

1. Empty the All Sheets of Current Design check box.
2. Select the Sheet data entry box.
3. Enter the schematic sheet filename in the data entry box in the file selector window.

   or

   Select the schematic sheet filename from the file selector list box.
4. Select *OK* to enter the sheet name in the Sheet data entry box.

5. Fill the Create Packaging List check box.
6. Select the Create Packaging List data entry box to change the output filename.
7. Enter the output filename of the packaging list if you want it to be different from the input filename.
8. Add or delete attributes for one sheet the same way you would for all sheets.

Packaging List Example

The packaging list in Figure 5-5 was produced from a netlist extracted from a schematic database that was assigned reference designators.

The packaging list has one record for each schematic symbol. Symbols are sorted by reference designators or instance numbers. Each record of the packaging list includes the following:

- reference designator *is assigned* using Enter!Ref. Des. & Section *in the Schematic Editor (you can't generate a packaging list unless reference designators are assigned); for example, U1.*
- pin list *lists all the symbol's pins by package number and their net assignment, if applicable. The entire pin list is enclosed in parentheses. Each entry in the pin list includes the following:*
  - — *package pin number* is assigned to the symbol using *Enter*!*Ref. Des. & Section* in the Schematic Editor; for example, 36.
  - — *net name* is the name of the net connected to the pin and is assigned by the system when you enter a wire in Schematic Editor or by you with *Name*!*Net*. If you use *Enter*!*Wire*, the system generates a net name. UN00010001 is an example of a system–assigned net name.
- attribute list *lists all the specified attribute keywords and values; for example FTYPE="INP". The entire list is enclosed in square brackets. When two or more attributes are listed, each is on a separate line and the attributes are separated by commas. You assign attributes to a symbol using* Enter!Attribute *in the Schematic or Symbol Editor.*

```
%********************************************************************
%                                                                    *
%     Program  :   PC*FORM VERSION 8.5                               *
%     Date     :   Aug 15 1995                                       *
%     Time     :   10:16:44 AM                                       *
%     File In  :   aca.nlt                                           *
%     File Out :   aca.pkl                                           *
%     Format   :   PACKAGING LIST                                    *
%                                                                    *
%********************************************************************

U1      ( 35      =RESET,    1       =DATA0,    2       =DATA1,
          3       =DATA2,    4       =DATA3,    5       =DATA4,
          6       =DATA5,    7       =DATA6,    8       =DATA7,
          25      =GND,      22      =GND,      19      =GND,
          21      =IO-READ', 18      =IOW',     28      =A0,
          27      =A1,       26      =A2,       13      =UN000009,
          12      =UN000009, 14      =ENABLE',  17      =,
          16      =CLK,      38      =RSLD',    37      =DSR',
          36      =CTS',     39      =RI',      10      =RX_DATA,
          9       =SIO_CLK,  34      =UN000007, 31      =UN000005,
          30      =UN000000, 24      =,         23      =,
          29      =,         33      =DTR',     32      =RTS,
          11      =TX_DTA1,  15      =SIO_CLK )
        [ FTYPE   ="INP" ]
U2      ( 18      =DATA0,    17      =DATA1,    16      =DATA2,
          15      =DATA3,    14      =DATA4,    13      =DATA5,
          12      =DATA6,    11      =DATA7,    19      =ENABLE',
          1       =IO-READ', 2       =D0,       3       =D1,
          4       =D2,       5       =D3,       6       =D4,
          7       =D5,       8       =D6,       9       =D7 )
U3      ( 8       =OSC )
U4      ( 2       =DTR',     1       =UN000004, 3       =RTS,
          6       =EIA_DTR,  7       =EIA_RTS )
U5      ( 2       =TX_DTA2,  1       =UN000001, 3       =,
          6       =TX_DTA6', 7       = )
U6      ( 3       =RSLD',    1       =+5V,      2       =EIA_CDET )
U6      ( 6       =DSR',     4       =+5V,      5       =EIA_DSR )
U6      ( 8       =CTS',     10      =+5V,      9       =EIA_CTS )
U7      ( 3       =RI',      1       =+5V,      2       =EIA_RI )
U7      ( 8       =RX_DTA2,  10      =+5V,      9       =EIA_RXD )
U8      ( 3       =RXCLRET,  2       =RXCLDATA, 8       =+5V,
          6       =RX_DTA1', 5       =GND,      7       = )
U9      ( 1       =UN000006, 2       =TX_DTA4', 3       =RX_DTA1,
          4       =,         5       =,         6       =,
          7       =TX_DTA6', 8       =RX_DTA2,  9       =RX_DATA,
          10      =EIA_TXDT, 11      =,         12      =,
          13      =,         14      =RX_DATA,  15      =XMITCLDT,
          16      =XMITCLRT )
U10     ( 1       =IRQ,      2       =A8',      3       =IRQ,
          4       =A8,       5       =UN000008, 6       =IRQ4,
          7       =UN000008, 8       =IRQ3 )
J1      ( 20      =EIA_DTR )
J1      ( 4       =EIA_RTS )
J1      ( 9       =XMITCLRT )
J1      ( 11      =XMITCLDT )
J1      ( 2       =EIA_TXDT )
J1      ( 8       =EIA_CDET )
J1      ( 6       =EIA_DSR )
J1      ( 5       =EIA_CTS )
```

Figure 5-5. Packaging List

Creating a Wire List

A wire list contains the following information:

- net (signal) name
- net attribute
- part or package reference designator
- part or package pin numbers

To create a wire list for all sheets

1. Fill the All Sheets of Current Design check box.
2. Fill the Create Wire List check box.
3. Select the Create Wire List data entry box to change the output filename.
4. Enter the output filename of the wire list if you want it to be different from the input filename.
5. Add or delete attributes.

   To add attributes

   [1] Cycle to *Wire Attributes* next to the Attributes list box.

   [2] Select *ADD.*

   [3] Enter the attribute keyword in the data entry box. If you enter **all**, the system carries all wire attributes.

   To delete attributes

   [1] Cycle to *Wire Attributes* next to the Attributes list box.

   [2] Select the attribute keyword in the list box.

   [3] Select *DELETE.*

To create a wire list for one sheet

1. Empty the All Sheets of Current Design check box.
2. Select the Sheet data entry box.
3. Enter the schematic sheet filename in the data entry box in the file selector window.

   or

   Select the schematic sheet filename from the file selector list box.
4. Select *OK* to enter the name in the Sheet data entry box.
5. Fill the Create Wire List check box.

6. Select the Create Wire List data entry box to change the output filename.
7. Enter the output filename of the wire list if you want it to be different from the input filename.

Wire List Example

The wire list in Figure 5-6 was produced from a netlist extracted from a schematic database.

The wire list has one record for each node, or net. Each record of the wire list includes the following:

- net name *is the name of the net connected to the pin and is assigned by the system when you enter a wire in Schematic Editor or by you with* Name!Net*. If you use* Enter!Wire*, the system generates a net name. UN00010001 is an example of a system–assigned net name. The net name column also contains any net attributes.*
- pin list *lists all pins that are attached to the net. Use* Enter!*Wire in the Schematic Editor to attach wires to the pins. Each entry in the pin list includes the following:*
  - — *reference designator* (of the component) is assigned using *Enter*!*Ref. Des. & Section* in the Schematic Editor (or it can be back–annotated during packaging); for example, U1 or U2.
  - — *package pin number* is assigned to the symbol using *Enter*!*Pin Numbers* in the Symbol Editor; for example, 36.

```
%********************************************************************
%                                                                   *
%     Program  :   PC*FORM VERSION 8.5                              *
%     Date     :   Aug 15 1995                                      *
%     Time     :   10:16:46 AM                                      *
%     File In  :   aca.nlt                                          *
%     File Out :   aca.wrl                                          *
%     Format   :   WIRE LIST                                        *
%                                                                   *
%********************************************************************


%
% NODE        REF. DESIGNATOR-PIN
% ----        -------------------
%
  DATA0           U1-1                U2-18
  [ WIDTH=12  ]

  DATA1           U1-2                U2-17
  [ WIDTH=12  ]

  DATA2           U1-3                U2-16
  [ WIDTH=12
    LENGTH=40
    LAYER=COMP  ]

  D0              U2-2                P1-9
  [ WIDTH=12
    NETCLASS=TTL  ]

  D1              U2-3                P1-8
  [ WIDTH=12
    LAYER=COMP
    LENGTH=60
    NETCLASS=ECL  ]

  IOW'            U1-18               P1-44
  [ NETATR1=value1  ]

  A2              U1-26               P1-29
  [ WIDTH=10
    NETCLASS=CLK
    LAYER=COMP  ]

  A5              U13-4               P1-26
  [ WIDTH=12
    NETCLASS=ECL
    LENGTH=45  ]

  A8              P1-23               U11-8
  [ WIDTH=15
    LENGTH=50  ]

  CLK\            U14-3               U14-5
  [ WIDTH=10  ]

  OSC             U11-11              R1-2
  [ WIDTH=12  ]
```

Figure 5-6. Wire List

Creating a Materials List

A materials list includes the following information:

- item number (assigned in sequence, starting with 1)
- number of symbols found in the schematic netlist
- number of parts that can be created from the symbols in the netlist
- symbol name
- reference designator or instance name
- user–selected attributes

Listing the Number of Parts

You can specify whether to list part quantities in a materials list. The program lists the number of parts that can be created from the symbol packaging information found in the schematic netlist. When the program finds symbols without packaging information, a series of question marks (???) is listed in the part quantity field for that component.

Grouping by Attribute

You can specify which user–assigned attributes you want included in the materials list. Components with the same symbol filename and the same group of attribute values and keywords are listed in a single entry. Each component is counted once, and the report provides a count for each group.

For example, in a schematic where each discrete resistor symbol is called res.sym, you assign the resistor value using the attribute val and the power rating using the attribute pwr. Any occurrences of res.sym in the design that have both the same resistor value and power rating are considered the same part number.

Specifying the attribute keywords pwr and val when you are generating a materials list shows the value and power rating for any symbols having those attributes assigned to them in the schematic.

Excluding Symbols With a Component ID Type of 0

If you create nonelectrical symbols such as title blocks or notes in your schematic designs, you can assign a component ID type of 0 to each nonelectrical symbol. The system ignores any component with an ID type of 0 when it extracts a netlist from the schematic sheets. Because the system ignores these symbols, they aren’t included in any of the reports. Component ID types are assigned to symbols in the Symbol Editor using *Enter!Component Type.*

Including Symbols With a Component ID Type of –1

You can assign a component ID type of –1 to your symbols such as power, ground, and page connector in schematic designs. These components are extracted and included in the netlist and show up in the reports, but they aren't packaged to the PCB. Assigning a component ID type of –1 is usually reserved for symbols with electrical intelligence, but no corresponding physical package.

To create a materials list for all sheets

1. Fill the All Sheets of Current Design check box.
2. Fill the Create Materials List check box.
3. Select the Create Materials List data entry box to change the output filename.
4. Enter the output filename of the materials list if you want it to be different from the input filename.
5. Add or delete attributes.

   To add attributes

   [1] Cycle to *Material Attributes* next to the Attributes list box.

   [2] Select *ADD.*

   [3] Enter the attribute keyword in the data entry box. If you enter **all**, the system carries all materials attributes.

   To delete attributes

   [1] Cycle to *Material Attributes* next to the Attributes list box.

   [2]Select the attribute keyword in the list box.

   [3] Select *DELETE.*

6. Cycle to *List Comps by Ref. Des.* or *List Comps by Name*.

To create a materials list for one sheet

1. Empty the All Sheets of Current Design check box.
2. Select the Sheet data entry box.
3. Enter the schematic sheet filename in the data entry box in the file selector window.

   or

Select the schematic sheet filename from the file selector list box.

4. Select *OK* to enter the sheet name in the Sheet data entry box.
5. Fill the Create Materials List check box.
6. Select the Create Materials List data entry box to change the output filename.
7. Enter the output filename of the materials list if you want it to be different from the input filename.
8. Cycle to *List Comps by Ref. Des.* or *List Comps by Name*.
9. Fill or empty the List Part Quantity check box.
10. Add or delete attributes for one sheet the same way you would for all sheets.

Materials List Example

The materials list in Figure 5-7 was produced from a netlist extracted from a schematic database with the List Part Quantities option enabled. The components are listed by reference designator.

A materials list has one record for each type of schematic symbol in the design. Each record in this version of the materials list includes the following fields:

- *item number* is assigned in sequence in the report, starting with 1.
- *symbol quantity (SQTY)* is the number of times the symbol occurs in the design.
- *part quantity (PQTY)* is the number of parts that can be created from the number of symbols found in the netlist.

> **Note:** The system uses the reference designator to determine the number of parts that can be created from the number of symbols found in the netlist. When no reference designator has been assigned, the program divides the number of symbols having the same filename by the number of gates per package to determine the number of parts shown in the list. This method is used whether you create reports listed by component name or by reference designator. It is left to the designer to determine the validity of partially occupied packages.

- *component name* is the base filename of the component symbol; for example, 74LS00.
- *reference designator* is assigned using *Enter*!*Ref. Des. & Section* in the Schematic Editor. You can also create a materials

list in which the components are listed by the instance name assigned by using *Name*!*Component* in the Schematic Editor or by the system generated name; for example, UC00000003. If you selected *List Comps by Name* this column would be headed *Instance Name*.

- *description* lists all the specified attribute keywords and values. When two or more attributes are listed, each attribute is on a separate line and attributes are separated by commas.

```
%**********************************************************************
%                                                                     *
%     Program  :   PC-FORM VERSION 8.5                                *
%     Date     :   Aug 15 1995                                        *
%     Time     :   11:58:57 AM                                        *
%     File In  :   test2.nlt                                          *
%     File Out :   test2.mat                                          *
%     Format   :   P-CAD MATERIALS LIST                               *
%                                                                     *
%**********************************************************************
ITEM  SQTY PQTY  COMP-NAME  REFERENCE-DESIGNATOR       DESCRIPTION
----  ---- ----  ---------  -------------              -----------
1      2    2    z84c42     U1 UC00000019
2      2    ???  empty      UC00000001 NO_PKG
3      3    3    resh       R1 UC00000004 R2
4      10   3    74ls00     U3 U3 UC00000007 U3
                            UC00000009 UC00000010
                            U2 U2 U2 U2
5      14   3    74ls04     U4 U4 U4 U4 U4 U4
                            UC00000017 UC00000018
                            UC00000024 UC00000025
                            UC00000026 U5 U5 U5
```

Figure 5-7. Materials List With List Part Quantities Enabled

Running the Reports

After you specify which report or reports you want, you can run *Report Generator* or exit without generating any reports.

To run *Report Generator*

- Select *RUN.*

Exiting the Report Generator Screen

You can exit *Report Generator* anytime without generating any reports.

To exit *Report Generator*

- Select *EXIT* on the Report Generator screen.

The P–CAD opening screen is then displayed.

Using the Command Line Format

You can run reports without using the P–CAD graphical interface anytime by entering options on the command line.

You must have an extracted netlist *(filename*.nlt) or an expanded netlist (*filename*.xnl) before you can generate reports from the command line.

For more information about creating these netlist files using command line format, see Chapter 9, "Packaging the Schematic," in the *Schematic Tools* manual.

- Use the following format for generating reports at the DOS command line prompt:

  pcform [*options*] *netlist*

  where

  pcform initiates the execution.

  options is one or more of the options listed below that correspond to the configuration options:

  -*h* prints the help message on the screen. The –h is optional.

  -*c* creates a component list with pins listed by name and with no component attributes (.cmp). This is the default option.

  -*p* creates a packaging list with no attributes (.pkl).

  -*a* creates a packaging list and component list with all attributes.

  -*n* creates a node list with pins listed by name (.nde).

  -*w* creates a wire list. (.wrl).

  -*m* creates a materials list (.mat) without part quantity.

  -*s* creates a materials list (.mat) with part quantity.

  @ *atrfile* lists component attributes to be used in the report. *atrfile* is the name of the ASCII component keyword list file. No extension is assumed. The @ sign is required. You can create this file with any text editor.

  % *atrfile* lists net attributes to be used in the report. *atrfile* is the name of the ASCII net keyword list file. No extension is assumed. The % sign is required. You can create this file with any text editor.

netlist is the name of the netlist file and is required. If you don't specify a filename extension, pcform uses the .nlt extension.

An example of a command line using *pcform* is shown below.

pcform -c -n -p -w -m @keyword.lst testfile.nlt

In this example, the system generates all the available output reports for an input netlist named testfile.nlt and includes the attributes listed in the keyword.lst file.

After you enter the command line, *Report Generator* runs automatically with no further input from you. When you generate reports from the command line, *Report Generator* writes any progress and error messages to the pcform.log file. When processing is complete, the system returns to the DOS prompt.

Creating a Batch File

A batch file can contain the command line instructions for running pcform several times consecutively. Use a text editor to create the ASCII batch file. Each line of the batch file is in the same format used for a command line.

You must use .bat as the filename extension when naming the batch file. For example, a batch file named report.bat might contain the following:

```
pcform -c -n \pcad\proj0\testfile.nlt
pcform -c -n -w \pcad\proj0\test2.pnl
pcform -c \pcad\proj0\test3.pnl
```

If you want to run the batch file from various directories, include the pathnames to the files used by pcform.

To run the batch file, enter the filename at the DOS or UNIX prompt. You don't need to enter the extension. For example, if you wanted to run the batch file report.bat and it is in the current directory, you enter

report

To run a batch file that isn't in the current directory, you enter the pathname. For example, if you wanted to run report.bat and it is in the \pcad\proj0 directory, you enter

\pcad\proj0\report

The program runs as many times as the number of pcform command lines listed in the batch file. When processing is complete, the system returns to the DOS or UNIX prompt.

Index