

UTILITIES

User's Manuals

August 1986

p-cad[®]
PERSONAL CAD SYSTEMS INC.

000-0070-03
048-0049-03

COPYRIGHT

Copyright (c) 1986 by Personal CAD Systems, Inc.
(P-CAD).

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior written permission of Personal CAD Systems, Inc.

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

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

TRADEMARKS

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

PREFACE

The P-CAD utilities are an interrelated set of programs that accompany P-CAD's electronic design automation programs. The utilities include:

PC-NODES

PC-LINK (used with CAE-1, CAE-2, PCB-2, and PCB-3 systems only)

PC-FORM

PREPACK

PC-PACK

PC-BACK

PC-PRINT

PC-PLOTS

PC-PHOTO

PC-DRC/NLC (used with PCB-1, PCB-2, and PCB-3 systems only)

The utility user's manuals in this package describe the input to, output from, and operation of each utility program.

The utility programs, combined with your main program(s), give you the capabilities for many applications. Your *System Overview* manual explains how you can use each utility program in the P-CAD system you have selected. We recommend that you read it before using the utilities.

1992-1993
1994-1995



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 extension .SYM in the current directory.

TESTFILE TESTFILE is a sample filename, which you must replace with the filename you intend to use. For example:

Database Filename :TESTFILE.SCH
Netlist Filename :TESTFILE.NLT



PC-NODES™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.



ABOUT THIS MANUAL

This manual describes PC-NODES, the P-CAD utility program that extracts binary format netlists from PC-CAPS schematic databases and from PC-CARDS printed circuit board (PCB) databases for input to other utility programs.

This manual provides an overview of PC-NODES, operating instructions, and an appendix describing common error messages.



CONTENTS

OVERVIEW.....	1-1
Input	1-2
Output	1-3
The Schematic Netlist.....	1-3
The Physical Netlist.....	1-4
USING PC-NODES	1-5
Interactive Mode.....	1-5
Command Line Mode	1-8
Batch Mode	1-10
APPENDIX A. ERROR MESSAGES	1-11

FIGURES

1-1. PC-NODES Input and Output	1-2
1-2. PC-NODES Program Screen.....	1-6
1-3. Sample PC-NODES Program Screen	1-7
1-4. Sample Batch File.....	1-10

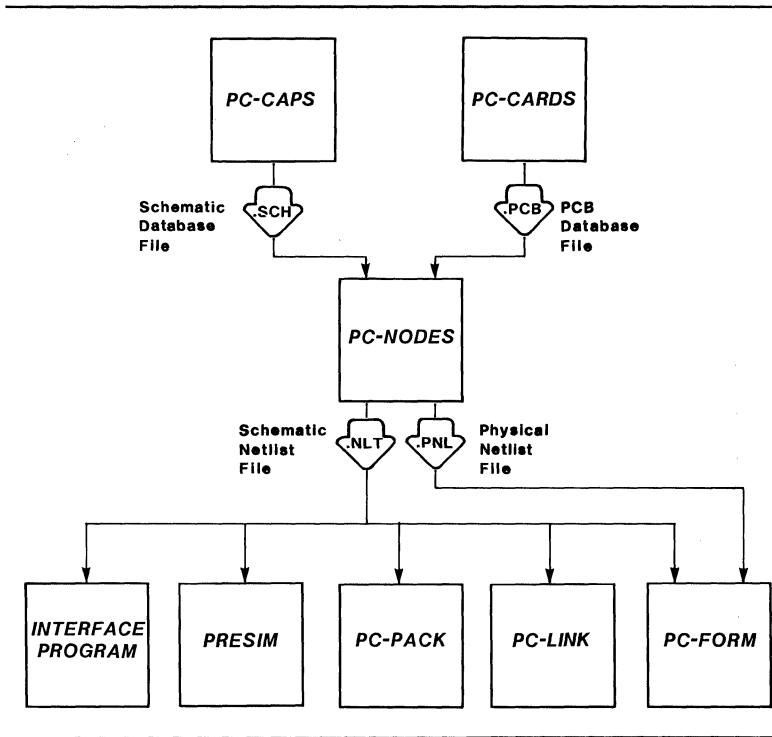


OVERVIEW

PC-NODES extracts a binary netlist from a PC-CAPS generated schematic database or from a PC-CARDS generated PCB database.

The output binary netlists can then be input to PC-LINK, PC-FORM, PC-PACK, PRESIM, and interface programs for operations specific to these programs. Refer to the appropriate user's manuals for more information about using these programs.

Figure 1-1 shows PC-NODES input and output.



86201

Figure 1-1. PC-NODES Input and Output

INPUT

PC-NODES input is one of two database types:

- A schematic database (<filename>.SCH) generated by PC-CAPS. The schematic database may be contained in one file (as with flat single-sheet schematics) or in several files (as with hierarchical or multisheet schematics). For designs using more than one file, each file must be input separately into PC-NODES.

- A PCB design database (<filename>.PCB) generated by PC-CARDS.

OUTPUT

The output of PC-NODES is a binary netlist extracted from either a schematic or a PCB database.

The Schematic Netlist

The schematic netlist (<filename>.NLT) can be input to the following P-CAD programs:

- PC-LINK, a utility that combines binary netlists extracted from several schematic database files that together form a single schematic design (for example, hierarchical and multisheet schematics) into a single binary netlist.
- PC-FORM, a report-generation utility that translates the binary netlist into ASCII format reports.

NOTE: If you do not name the components and/or nets in your PC-CAPS schematic or your PC-CARDS PCB, PC-NODES automatically names them for you. See Appendix B in the *PC-FORM User's Manual* for a description of the format PC-NODES uses to name components and nets.

- PC-PACK, a packaging utility that takes the binary netlist, packages the schematic components into physical parts, and then places the physical parts into a preliminary PCB database.

- **PRESIM**, a preprocessor for the **PC-LOGS** logic simulator. **PRESIM** translates the binary netlist into a format that can be used by **PC-LOGS**.
- **Interface programs**, which translate **PC-CAPS** schematic databases into the format required by several other **CAD** systems.

The Physical Netlist

The physical netlist (<filename>.PNL) can be input to **PC-FORM** to generate **ASCII** format reports.

USING PC-NODES

Before running PC-NODES, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-NODES program file (PCNODES.EXE).
- You have created the PC-CAPS schematic or the PC-CARDS PCB database file.

If you are using the P-CAD recommended directory structure, run PC-NODES from the appropriate project directory.

PC-NODES has three operating modes: interactive, command line, and batch. In interactive mode, PC-NODES prompts for input filenames on the program screen. In command line mode, you specify filenames following the program name when you start PC-NODES. In batch mode, you create a batch file containing command lines, and execute the file from DOS.

The following sections describe how to run PC-NODES in interactive mode, command line mode, and batch mode.

Interactive Mode

To run PC-NODES in interactive mode, follow the steps below.

1. Type:

PCNODES [Return]

When the PC-NODES Title Screen appears, press any key to continue.

The system prompts for the input database filename as shown in Figure 1-2.

PC-NODES

Database Filename : <Filename>.SCH

Enter the filename; Press [Return] or [Esc] to exit.

Figure 1-2. PC-NODES Program Screen

2. Type the database filename and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter the filename extension, PC-NODES enters the default .SCH extension. If the database file is a PCB layout, enter the filename with the extension .PCB.

The system prompts for the output netlist filename. The default is the input filename with a .NLT extension for a schematic or a .PNL extension for a PCB as shown in Figure 1-3.

PC-NODES

Database Filename : TESTFILE.SCH
Net-List Filename : TESTFILE.NLT

Enter the filename; Press [Return] to accept; [Esc] to cancel.

Figure 1-3. Sample PC-NODES Program Screen

3. Press [Return] to accept the default netlist filename, or type another filename and press [Return].

PC-NODES extracts the binary netlist. It displays progress reports and error messages, if any, at the bottom of the screen.

When processing is complete, the cursor returns to the **Database Filename** prompt. To end the program, press [Esc]. The system returns you to DOS.

Command Line Mode

Command line mode allows you to specify the input schematic or PCB database filename and, if you want, the output netlist filename when you start PC-NODES.

To use command line mode, at the DOS prompt, type PCNODES followed by the filenames, then press [Return].

The format of the command line is:

PCNODES <infile> <outfile>

where:

PCNODES initiates the program.

Infile is the name of the input file and is required. If you do not include the filename extension, PC-NODES assigns the .SCH extension. Therefore, if the database file is a PCB layout, enter the filename with the extension .PCB.

outfile is the name of the output file and is optional. If you specify a name with no extension, PC-NODES uses the .NLT extension if the input is a schematic database and the .PNL extension if the input is a PCB. If you do not specify a name, PC-NODES gives the output file the same name as the input file with a .NLT or .PNL extension, depending on the type of input database file.

Three examples of command lines are shown below.

```
PCNODES TESTFILE  
PCNODES TESTFILE.PCB  
PCNODES TESTFILE.SCH TEST1.NLT
```

The first example causes PC-NODES to extract a schematic netlist from the TESTFILE.SCH file. The output file will have the default filename TESTFILE.NLT

The second example causes PC-NODES to extract a physical netlist from the TESTFILE.PCB file. The output file will have the default filename TESTFILE.PNL.

The third example causes PC-NODES to extract a schematic netlist from the TESTFILE.SCH file and give the output file the filename TEST1.NLT.

After you enter the command line, PC-NODES runs automatically with no further input from you. It displays progress reports and error messages, if any. When processing is complete, the system returns you to DOS.

Batch Mode

Batch mode is a variation of command line mode where you can use a DOS batch file to run PC-NODES several times consecutively without user intervention.

You can use any text editor to create the batch file. Each line of the batch file has the same format as a command line. Each line contains the program name PCNODES and the name of the input netlist file and can include an output filename. You must use a .BAT filename extension when naming the batch file.

Figure 1-4 shows an example of a batch file.

```
PCNODES TESTFILE.SCH TESTFILE.NLT  
PCNODES TESTFILE.PCB TESTFILE.PNL
```

Figure 1-4. Sample Batch File

To run the batch control file, at the DOS prompt, type the filename, then press [Return].

PC-NODES is executed as many times as the number of command lines listed in the batch file. The system displays each netlist filename in turn. When processing is complete, the system returns you to DOS.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that are generated by PC-NODES.

Message: Disk is full...

Cause: Insufficient disk space to perform netlist extraction.

Action: Delete unnecessary files. Rerun PC-NODES.

Message: <filename> not found.

Cause: PC-NODES was not able to find the database file specified.

Action: Specify the appropriate drive or directory for the database file.

Message: <filename> format is incompatible.

Cause: The database filename specified is not an acceptable database format.

Action: Check the file to be sure you are processing the correct database. Be sure that you created the database on the current version of PC-CAPS or PC-CARDS.

**Message: Not enough memory space or...
CONFIG.SYS does not have FILES=15.**

Cause: Insufficient memory exists for program operation; or the CONFIG.SYS file does not exist or is insufficient.

Action: Make sure the CONFIG.SYS file in the root directory contains the text FILES=15. If the CONFIG.SYS file is correct, make sure you have 640K of memory available.

Message: !! Warning <filename> has no net.

Cause: The database does not have any nets or pins.

Action: Check the filename to be sure you are processing the correct database.

Message: <filename> is an empty database.

Cause: The database does not have any nets or components.

Action: Check the filename to be sure you are processing the correct database.



PC-LINK™
User's Manual

August 1986



p-cad®
PERSONAL CAD SYSTEMS INC.

000-0056-04



ABOUT THIS MANUAL

This manual describes PC-LINK, the P-CAD utility program that combines related binary netlists that are created by the PC-NODES utility into a single binary netlist that can be used for input to other utility programs.

NOTE: This program is used with the CAE-1, CAE-2, PCB-2, and PCB-3 systems only.

This manual provides an overview of PC-LINK, operating instructions, and an appendix describing common error messages.



CONTENTS

OVERVIEW.....	2-1
Input	2-2
Output	2-4
USING PC-LINK.....	2-7
Interactive Mode	2-8
Batch Mode	2-13
APPENDIX A. ERROR MESSAGES	2-17

FIGURES

2-1. PC-LINK Input and Output.....	2-2
2-2. PC-LINK Program Screen	2-8
2-3. Specifying Multisheet Input.....	2-10
2-4. Sample Netlist List File	2-11
2-5. Sample PC-LINK Program Screen.....	2-12
2-6. Sample Batch Control File.....	2-16

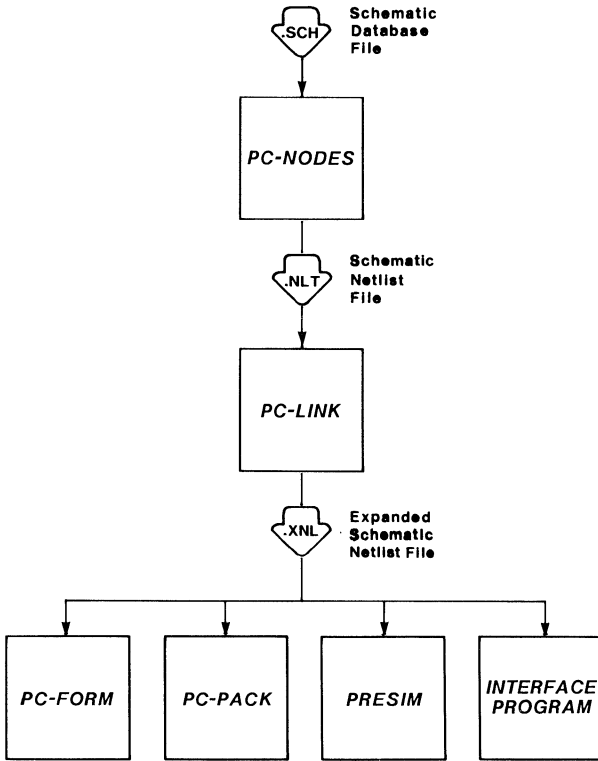


OVERVIEW

PC-LINK flattens and/or connects schematic netlists extracted by PC-NODES from hierarchical and multisheet schematic design databases.

These new "flattened" or "linked" binary netlists can then be input to PC-PACK, PC-FORM, PRESIM and interface programs for operations specific to these programs. Refer to the appropriate user's manuals for more information about using these programs.

Figure 2-1 shows PC-LINK input and output.



86202

Figure 2-1. PC-LINK Input and Output

INPUT

PC-LINK links two or more binary netlist files generated from a schematic database using the PC-NODES utility. All netlists to be linked must have

the same filename extension (usually .NLT). PC-LINK can link netlists from two types of schematics:

- **Hierarchical schematics.** PC-LINK "flattens" all the internal levels of hierarchy into a single "top-level" binary netlist. To flatten a hierarchical design, you must first use PC-NODES to extract netlists from all hierarchical components at all levels. You then input the top-level netlist into PC-LINK and, if necessary, specify the directory path to the netlists for the subcircuits.

NOTE: Each hierarchical subcircuit must be assigned a Component Type ID of 256 using the PC-CAPS SCMD/SCAT command in SYMB mode.

- **Multisheet schematics.** PC-LINK links several single-sheet schematics that together make up a multisheet design.

NOTE: Each of the schematic databases that make up a multisheet design must be assigned the SHEET attribute. To assign this attribute, use the PC-CAPS ATTR/ACOM command in SYMB mode and type the attribute in the format SHEET=<sheet id>, where **sheet id** can be up to three characters (generally digits) and is unique for each sheet (for example, SHEET=01 for sheet one and SHEET=02 for sheet two).

The PC-NODES program uses the sheet ID when naming unnamed nets and components. If sheet IDs are not assigned, the expanded netlist may contain several components with the same name.

OUTPUT

The output of PC-LINK is an expanded schematic netlist (<filename>.XNL) that represents the entire hierarchical or multisheet schematic design and can be used as the input binary netlist for PC-FORM, PC-PACK, PRESIM, and certain interface programs to other systems.

If PC-LINK encounters errors during program operation, it also outputs an error message file, PCLINK.MSG.

PC-LINK produces the expanded binary netlist file in the following manner:

- For a hierarchical design, PC-LINK produces an expanded netlist that includes all levels of hierarchy. For each schematic component with a Component Type ID of 256, PC-LINK finds the binary netlist for the detailed subcircuit it represents. PC-LINK adds this netlist to that of the higher-level circuit and connects the nets in the subcircuit with those in the higher level circuit.
- For a multisheet design that is non-hierarchical, PC-LINK produces an expanded netlist that includes all the components and nets on all the sheets. Nets that have the same name and are on different sheets are connected or merged into one net.
- For a multisheet design that includes hierarchy, PC-LINK produces an expanded netlist in a combination of the two manners described above.

However, if a hierarchical component is a multisheet design (that is, the multisheet design is "embedded" in the hierarchy), you must follow the steps below.

1. Use PC-LINK to link the sheets of the component. The first input filename you enter must be the filename of the sheet that contains the symbol that represents the component.
2. Change the filename extension of the file output in step 1 from .XNL to .NLT.
3. Use PC-LINK again to link the higher levels of the design.



USING PC-LINK

Before running PC-LINK, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-LINK program file (PCLINK.EXE).
- You have created all the input binary netlists to be linked and they all have the same filename extension (usually .NLT).

If you are using the P-CAD recommended directory structure, run PC-LINK from the appropriate project directory.

PC-LINK has two operating modes: interactive and batch. In interactive mode, PC-LINK prompts for each required input on the Program Screen. In batch mode, you create a batch control file containing the required inputs, and PC-LINK reads the inputs from the control file.

The following sections describe how to run PC-LINK in interactive mode and batch mode.

Interactive Mode

To run PC-LINK in interactive mode, follow the steps below.

1. Type:

PCLINK [Return]

When the PC-LINK Title Screen appears, press any key to continue.

The system prompts for the suffix (filename extension) of the input netlists as shown in Figure 2-2. The default is .NLT.

PC-LINK

Net-List Suffix : .NLT

Enter the suffix; Press [Return] to accept; [Esc] to exit.

Figure 2-2. PC-LINK Program Screen

2. Press [Return] to accept .NLT as the input netlist filename extension, or type in the appropriate extension and press [Return].

The system prompts for the work directory.

3. Type the name of the directory where the input netlist files are located and press [Return], or press [Return] to accept the default, "Current," if those files are in the current directory.

The system prompts for the library path.

4. Type the directory path to the netlists of the hierarchical components and press [Return], or press [Return] to accept the default, "Current," if those files are in the current directory or if your design is nonhierarchical.

The system prompts for the input netlist filename.

5. Type the input netlist filename and press [Return]. If you do not enter the filename extension, PC-LINK enters the extension you specified in step 2 for the netlist suffix.

For a hierarchical design, enter the filename of the schematic netlist for the top level of the design.

For a multisheet design, specify the names of the schematic netlists for the sheets, typing a plus sign after every filename except the last one. If necessary, you can use more than one line to enter the filenames, as shown in Figure 2-3. Press [Return] at the end of each line. The plus sign after the last filename on a line causes PC-LINK to display the prompt again so you can enter more filenames.

PC-LINK

Net-List Suffix : .NLT
Work Directory : Current

Library Pathname : Current

Net-List Filename : TSTFL1+TSTFL2+
Net-List Filename : TSTFL3+TSTFL4+
Net-List Filename : TSTFL5+TSTFL6

Enter the filename; Press [Return] or [Esc] to exit.

Figure 2-3. Specifying Multisheet Input

NOTE: You can use a text editing program to create a file containing a list of the netlist files in the multisheet designs before running PC-LINK. All filenames except the last one must be followed by a plus sign. The file can contain as many lines as necessary. A sample is shown in Figure 2-4.

```
SHEET1+SHEET2+  
SHEET3+  
SHEET4
```

Figure 2-4. Sample Netlist List File

To input this type of file into PC-LINK, at the **Net-List Filename** prompt, type @ followed by the filename. For example, if the file above is named NETLIST.LST, you would input it by typing @NETLIST.LST. PC-LINK reads the file as if each line were entered separately.

After you enter the input filename(s), the system prompts for the output expanded netlist filename. The default is the input filename with a .XNL extension as shown in Figure 2-5.

PC-LINK

Net-List Suffix : .NLT
Work Directory : Current

Library Pathname : Current

Net-List Filename : TESTFILE.NLT
Expanded Net-List : TESTFILE.XNL

Enter the filename; Press [Return] to accept; [Esc] to reject.

Figure 2-5. Sample PC-LINK Program Screen

4. Press [Return] to accept the default or enter another filename and press [Return]. If you do not enter the filename extension, PC-LINK enters the default .XNL.

PC-LINK expands the netlist(s). It displays progress reports at the bottom of the screen.

When processing is complete, the cursor returns to the **Net-List Filename** prompt. To end the program, press [Esc]. The system returns you to DOS.

If PC-LINK encounters any errors in expanding the netlists, it displays the following message:

Errors detected, Check PCLINK.MSG file.

To view the error message file, first exit from PC-LINK. Then, at the DOS prompt, type the following command:

```
TYPE PCLINK.MSG [Return]
```

Refer to Appendix A, "Error Messages," for a list of possible error messages, the cause of each error, and the appropriate action.

Batch Mode

Batch mode allows you to specify inputs to PC-LINK in a text file. The file is a batch control file (<filename>.BCF) containing several command lines to be processed consecutively without user intervention.

You can use any text editor to create this file. The default filename extension for the batch control file is .BCF

The batch control file consists of one or more entries. Each entry contains all the information for one expansion to be performed, in the format shown below:

```
SUFFIX <.ext>  
DIR <workdir>  
PATH <path>  
NAME <outfile>  
LINK <infiles>
```

where:

SUFFIX, DIR, PATH, NAME, AND LINK are keywords and refer to the parameters described below. Each keyword must be accompanied by the corresponding parameter value. **LINK** is required and must be the last keyword in the entry. The other keywords are optional and can be in any order. At least one blank space is required between a keyword and its parameter value.

.ext is the filename extension to be used for all the netlist filenames encountered during the linking process. If it is not present, PC-LINK uses the default **.NLT** extension. If an extension is specified in one entry, it is used for all subsequent entries until another **SUFFIX** keyword is used.

workdir specifies the working directory where PC-LINK will access the input files. If it is not present, PC-LINK looks for the files in the current directory. If a directory is specified in one entry, it is used for all subsequent entries until another **DIR** keyword is used.

path specifies the alternate directories where PC-LINK will look for hierarchical component netlist files. Directories must be separated by a plus sign (+). If a path is not present, PC-LINK

looks for the files in the current directory only. If a path is specified in one entry, it is used for all subsequent entries until another PATH keyword is used.

outfile specifies the name of the output file. If it is not present, PC-LINK takes the name of the first file specified with the LINK keyword and adds a .XNL extension.

infiles specifies the netlist files to link. For a hierarchical design, only the top level file must be specified. For a multisheet design, all the sheets must be included. Filenames must be separated by a plus sign (+). If a file is not in the working directory, include a drive designator or directory path.

Information for a keyword can take up one or more lines. A line can contain up to 80 characters.

You can add comments anywhere in the file. Comments begin with a percent sign (%). PC-LINK will ignore all text from the % to the end of the line.

Figure 2-6 shows a sample batch control file containing two entries.

% The following entry instructs PC-LINK to link a three
% sheet design into CAPS123.XNL. The input netlists
% can be in the current directory or in \PCAD\TEST.

```
DIR \PCAD\TEST
NAME CAPS123.XNL
LINK CAPS1 + CAPS2 + CAPS3
```

% In the following entry, TESTFILE.NLT can be in the
% current directory or \PCAD\TEST (as specified above).
% Hierarchical component netlists can be in the
% current directory, \PCAD\TEST, or \PCAD\NEWLIB.
% The output filename will default to TESTFILE.XNL.

```
PATH \PCAD\TEST + \PCAD\NEWLIB
LINK TESTFILE.NLT
```

Figure 2-6. Sample Batch Control File

To use batch mode, at the DOS prompt, type PCLINK followed by the name of the batch control file, then press [Return]. If you do not enter the filename extension, PC-LINK assumes the default .BCF extension.

PC-LINK expands the netlists according to the instructions in the batch control file. When processing is complete, the program returns you to DOS.

If PC-LINK encounters any errors in expanding the netlists, it displays the same message displayed in interactive mode. If errors occur, use the instructions in the section "Interactive Mode" to view the error message file.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that are generated by PC-LINK and are stored in the PCLINK.MSG file.

Message: <filename> not found.

Cause: The system could not find the named file.

Action: Make sure the file is on the current directory. Rerun the program, taking care to enter the filename correctly.

Message: <filename> format is incompatible.

Cause: The named input file was not a binary netlist.

Action: Input files for PC-LINK must be binary netlist files produced by PC-NODES.

Message: <filename> does not have any components.

Cause: PC-LINK cannot process a netlist that does not have components.

Action: Make sure that the input file you named contains the appropriate data, then rerun the program.

Message: Failed in loading <filename>.

Cause: The system does not have enough memory or hard disk space for PC-LINK to read the named file.

Action: Make sure you have 640K of memory available. If your hard disk is full, delete or move files to provide space.

Message: Failed linking definitions of <filename>.

Cause: PC-LINK could not add component definitions of the named file to the list of component definitions because the data in the file is damaged.

Action: Run PC-NODES to extract the netlist from the corresponding schematic database, then rerun PC-LINK.

Message: Failed linking nodes of <filename>.

Cause: PC-LINK could not add nodes of the named file to the list of nodes because the data in the file is damaged.

Action: Run PC-NODES to extract the netlist from the corresponding schematic database, then rerun PC-LINK.

Message: Net-List in <filename> is not complete.

Cause: The named netlist does not have interconnecting nets.

Action: Check to make sure the file should be linked. If so, use PC-CAPS to enter the necessary net names into the schematic database, then run PC-NODES to extract the corrected netlist and rerun PC-LINK.

Message: Failed expanding <filename>.

Cause: The program was not able to expand the hierarchical component mentioned due to a previously displayed error.

Action: Make sure the netlist for the component's schematic database file has been extracted, contains components, and is present in the current directory, then rerun PC-LINK. (If you do not require the component to be expanded, no action is necessary.)

Message: Fatal error in loading macro <filename>.

Cause: Data in the named netlist is damaged.

Action: Make sure the schematic database file is correct, then run PC-NODES to extract the netlist and rerun PC-LINK.

Message: Failure in heap allocation.

Cause: The system does not have 640K of memory available.

Action: Make sure you have 640K available.

Message: Macro <filename> does not have pins.

Cause: The program cannot process hierarchical components without pins.

Action: Use PC-CAPS to add pins to the SYMB view of the component and to reconstruct the schematic using the corrected component. Then use PC-NODES to extract the netlist and rerun PC-LINK.

Message: Failed linking sheets of Net-List.

Cause: PC-LINK could not link the netlist files because of an earlier error.

Action: Correct the previous error. (The message for the error was displayed when PC-LINK encountered the error.)

Message: Failed expanding & linking macros.

Cause: PC-LINK could not expand and link hierarchical components because of an earlier error.

Action: Correct the previous error. (The message for the error was displayed when PC-LINK encountered the error.)

Message: Failed saving expanded net-list.

Cause: The program could not save the output file because the disk is full.

Action: Check the space available on your disk and delete or move files, then rerun the program.

Message: Keyword : SUFFIX, DIR, PATH, NAME or LINK expected.

Cause: The format of the batch control file is incorrect.

Action: Correct the format of the batch control file.

Message: Not enough disk space.

Cause: There is not enough space on the disk to run the program.

Action: Delete or move files, then rerun the program.

**Message: Not enough memory space or ...
CONFIG.SYS file does not have FILES=15.**

Cause: The program requires 640K of memory and the appropriate text in the CONFIG.SYS file.

Action: Be sure you have 640K available. If necessary, use a text editor to correct the CONFIG.SYS file.





PC-FORM™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0057-04



ABOUT THIS MANUAL

This manual describes PC-FORM, the P-CAD utility program that generates ASCII format reports from the binary netlist files created by the PC-NODES, PC-LINK, and PC-PACK utility programs.

This manual provides an overview of PC-FORM, operating instructions, and appendixes describing common error messages and showing sample output reports.



CONTENTS

OVERVIEW.....	3-1
Input.....	3-3
The Schematic Netlist File.....	3-3
The Expanded Schematic Netlist File.....	3-3
The Physical Netlist File.....	3-3
The Back-Annotated Netlist File.....	3-3
Output.....	3-4
The Component List.....	3-4
The Node List.....	3-5
The Packaging List.....	3-5
The Wire List.....	3-6
The Material List.....	3-7
USING PC-FORM.....	3-9
Running PC-FORM.....	3-9
Interactive Mode.....	3-10
Command Line Mode.....	3-18
Batch Mode.....	3-19
Viewing and Printing the Output Files.....	3-20
APPENDIX A. ERROR MESSAGES.....	3-23
APPENDIX B. SAMPLE OUTPUT REPORTS ...	3-25
Example 1: Component List From Schematic Database With Pin Names.....	3-26
Example 2: Component List From Schematic Database With Pin Numbers.....	3-28
Example 3: Component List From PCB Database.....	3-30
Example 4: Node List From Schematic Database.....	3-32
Example 5: Node List From PCB Database....	3-34
Example 6: Packaging List.....	3-36

Contents (Continued)

Example 9: Material List From PCB Database	3-42
--	------

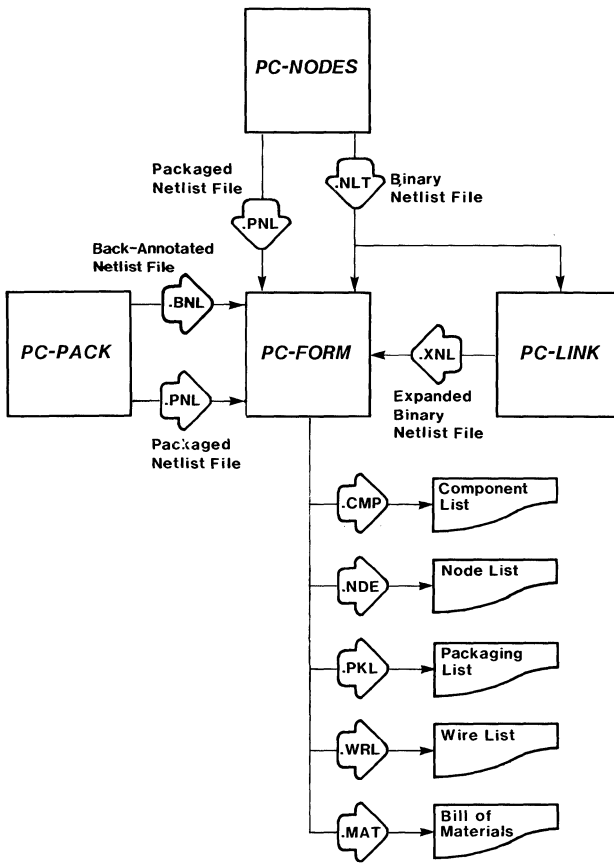
FIGURES

3-1. PC-FORM Input and Output	3-2
3-2. PC-FORM Program Screen	3-10
3-3. Entering Multiple Attributes	3-13
3-4. Sample Keyword List File	3-14
3-5. Sample PC-FORM Program Screen	3-17
3-6. Sample Batch File	3-20

OVERVIEW

PC-FORM converts binary format netlists generated by other P-CAD utility programs into ASCII reports for user reference. These reports can be organized by component, packaging, node, wire, or as a Bill of Materials.

Figure 3-1 shows PC-FORM input and output.



86203

Figure 3-1. PC-FORM Input and Output

INPUT

Four types of binary netlist files can be input into PC-FORM. The following sections describe these files.

The Schematic Netlist File

The schematic netlist file (<filename>.NLT) is extracted by PC-NODES from a schematic database.

The Expanded Schematic Netlist File

The expanded schematic netlist file (<filename>.XNL) is produced when PC-LINK combines a set of netlists that together describe a hierarchical or multisheet schematic design.

The Physical Netlist File

The physical netlist file (<filename>.PNL) is a binary netlist carrying PCB part information. It can be produced in either of two ways. PC-PACK generates a physical netlist file when it packages netlist or expanded netlist files. PC-NODES extracts a physical netlist file from a PCB database.

The Back-Annotated Netlist File

The back-annotated netlist file (<filename>.BNL) is a binary netlist in which all the components in the schematic are assigned PCB part reference designators and pin numbers. It is generated by PC-PACK.

OUTPUT

PC-FORM output is any combination of the ASCII reports described in this section. The reports are output as disk files, which you can view and print using the appropriate DOS commands. The form and content of each report are described briefly below. Appendix B, "Sample Output Reports," shows sample reports and gives detailed explanation of their contents.

The Component List

The component list (<filename>.CMP) is sorted by names of the symbol or part files of the components in the schematic or PCB database. It includes the following data:

- Component symbol or part name
- Instance name and/or reference designator
- Pin names or part/package pin numbers
- Associated net (signal) name
- User-selected attributes

The Node List

The node list (<filename>.NDE) is sorted by the net names assigned in the schematic or PCB database. It includes the following data:

- Net (signal) name
- Associated pin names and/or part/package pin numbers
- Symbol or part name
- Instance name and/or reference designator
- Hierarchical "path"

The Packaging List

The packaging list (<filename>.PKL) is sorted by physical reference designators. It includes the following data:

- Reference designator name
- Part/package pin numbers
- Associated net (signal) name
- User-selected attributes

To produce a packaging list from an input netlist extracted from a schematic database, reference designators and pin numbers must be preassigned to all components in the schematic database.

You can preassign this packaging information during schematic capture by using the PC-CAPS SCMD/PNLC and SCMD/PNUM commands, or after the schematic is complete by using the PC-PACK utility. For further information, refer to the *PC-CAPS* or *PC-PACK User's Manuals*.

The Wire List

The wire list (<filename>.WRL) is sorted by net names and respective reference designator/pin number pairs. It includes the following data:

- Net (signal) name
- Part/package reference designator
- Part/package pin numbers

Like the packaging list, the wire list requires preassignment of the schematic elements to package positions.

Because the wire list contains reference designator/pin number pairs, it is helpful in the construction of wire-wrap prototypes.

The Material List

The material list (<filename>.MAT) is a Bill of Materials. It includes the following data:

- Item number (assigned in sequence starting with 1)
- Quantity of item in database
- Symbol or part name
- Instance name
- User-selected attributes



USING PC-FORM

To produce ASCII reports, you must first run PC-FORM to produce the reports, then use DOS commands to view or print the reports. The following sections explain these operations.

RUNNING PC-FORM

Before running PC-FORM, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-FORM program file (PCFORM.EXE).
- You have prepared the input binary netlist.

If you are using the P-CAD recommended directory structure, run PC-FORM from the appropriate project directory.

PC-FORM has three operating modes: interactive, command line, and batch. In interactive mode, PC-FORM prompts for required inputs on the Program Screen. In command line mode, you specify the input netlist filename and the output report options. In batch mode, you create a batch file containing command lines, and execute the file from DOS.

Interactive Mode

To run PC-FORM in interactive mode, follow the steps below.

1. Type:

PCFORM [Return]

When the PC-FORM Title Screen appears, press any key to continue.

The system prompts for the name of the input netlist file as shown in Figure 3-2.

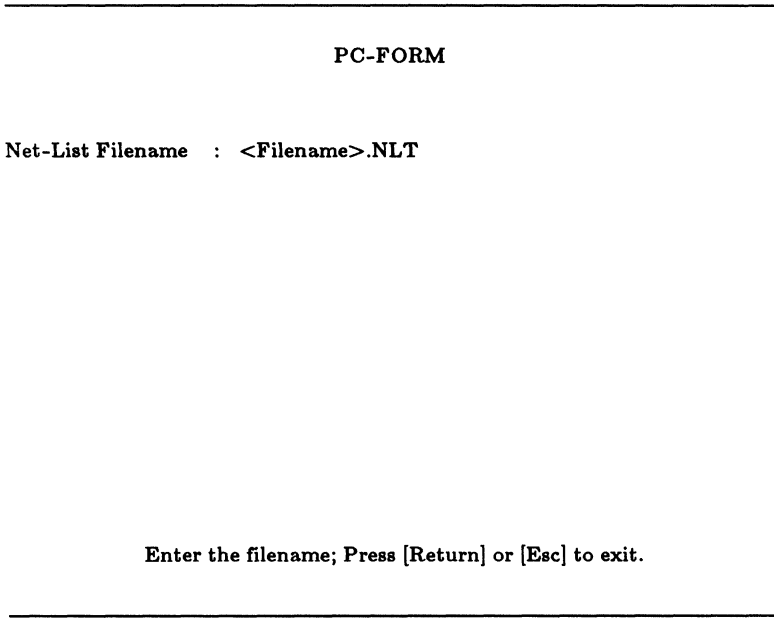


Figure 3-2. PC-FORM Program Screen

2. Type the filename of the input netlist file and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter the filename extension, PC-FORM enters the default .NLT extension.

The system prompts for the name of the output component list filename. The default is the input netlist filename with a .CMP extension.

3. If you do not want to output a component list, press [Esc]. The component list option is left blank and you can go on to step 5.

If you want to produce a component list, press [Return] to accept the default filename, or type a new filename and press [Return].

For a component list, you can specify whether or not you want a pin list for each component. The system displays the following prompt at the bottom of the screen:

Pin List Requested: YES

To omit pin lists, press the space bar to display the "NO" setting and press [Return].

To include pin lists in the output, press [Return] for YES. The system prompts you to specify whether the pins are to be identified by name or number:

Pins listed by: Name

Press [Return] to accept the default, "Name," or press the space bar to display the "Number" setting and press [Return].

The system prompts for the keywords of attributes to include in the component list. The default is "All."

4. To include all attributes, press [Return].

To omit all attributes from the component list, press [Esc]. The attribute keyword option is left blank.

To include one or more specific attributes, type the attribute keywords with a plus sign after each one, then press [Return]. You can enter as many keywords as you need. A maximum of 40 characters is allowed per line. Each time you end your entry with a plus sign, PC-FORM displays the prompt again for you to enter more keywords.

Figure 3-3 shows a sample screen that includes specification of several attributes on several lines.

PC-FORM

Net-List Filename : TESTFILE.NLT
Component List : TESTFILE.CMP
Attribute Keyword : PCL+PRT+
Attribute Keyword : RSY+
Attribute Keyword : MFR

Enter the keyword or filename; Press [Return] to accept; [Esc] to skip.

Figure 3-3. Entering Multiple Attributes

NOTE: You can use a text editing program to create a file containing the list of attributes before running PC-FORM. In this file, all keywords except the last one must be followed by a plus sign. A sample is shown in Figure 3-4.

PCL+123+
MDH+
KC+
SPC

Figure 3-4. Sample Keyword List File

To input this type of file into PC-FORM, at the attribute prompt, type @ followed by the filename. For example, if the file above is named KEYWORD.LST, you would input it by typing @KEYWORD.LST. PC-FORM reads the file as if each line were entered separately.

After you enter any attributes, the system prompts for the node list filename. The default is the name of the input netlist with a .NDE extension.

5. If you do not want to output a node list, press [Esc]. The node list option remains blank.

If you want to output a node list, press [Return] to accept the default filename, or type a new name and press [Return].

Node lists can be generated with either pin numbers or pin names. If you choose to create a node list, the system displays the following option at the bottom of the screen:

Pins Listed By: Name

Press [Return] to accept the default, "Name," or press the space bar to select "Number" and press [Return].

The system prompts for the packaging list filename. The default is the input netlist filename with a .PKL extension.

6. If you do not want to produce a packaging list, press [Esc]. The packaging list option is left blank.

If you want to produce a packaging list, press [Return] to accept the default filename or type a new name and press [Return].

NOTE: PC-FORM can produce a packaging list for a schematic database only if all components have been preassigned reference designators and package pin numbers.

If you choose to produce a packaging list, the system prompts for the keywords of attributes to include in the packaging list. The default is "All."

7. To include all attributes, press [Return].

To omit all attributes from the packaging list, press [Esc]. The attribute keyword option is left blank.

To specify one or more specific attributes, use the procedure given in step 4.

The system prompts for the output wire list filename. The default is the filename of the input netlist with a .WRL extension.

8. If you do not want to produce a wire list, press [Esc]. The wire list option is left blank.

If you want to produce a wire list, press [Return] to accept the default filename or type a new name and press [Return].

NOTE: PC-FORM can produce a wire list for a schematic database only if all components have been preassigned reference designators and package pin numbers.

The system prompts for the material list filename. The default is the filename of the input netlist with a .MAT extension.

9. If you do not want to produce a material list, press [Esc]. The material list option is left blank.

If you want to produce a material list, press [Return] to accept the default filename or type a new name and press [Return].

If you choose to generate the material list, the system prompts for the keywords of attributes to include in the material list. The default is "None."

10. To omit attributes from the material list, press [Return].

To include attributes, select them in the manner described in step 4.

Figure 3-5 shows a sample Program Screen with all default values selected.

PC-FORM

```
Net-List Filename : TESTFILE.NLT
Component List   : TESTFILE.CMP
Attribute Keyword : <All>
Node List        : TESTFILE.NDE
Packaging List   : TESTFILE.PKL
Attribute Keyword : <All>
Wire List        : TESTFILE.WRL
Material List     : TESTFILE.MAT
Attribute Keyword : <None>
```

Enter the keyword or filename; Press [Return] to accept; [Esc] to skip

Figure 3-5. Sample PC-FORM Program Screen

After you have selected the reports to produce, PC-FORM begins processing. It displays progress reports and error messages at the bottom of the screen.

When processing is complete, the system displays the **Net-List Filename** prompt again. To end the program, press [Esc]. The system returns you to DOS.

Command Line Mode

Command line mode allows you to specify the input netlist file and the report options when you start PC-FORM.

To use command line mode, at the DOS prompt, type PCFORM followed by the report options and the input netlist filename, then press [Return].

The format of the command line is:

PCFORM <options> <filename>

where:

PCFORM initiates the program.

options specify the output reports to be created. PC-FORM gives each output report the filename of the input database with the appropriate extension. The options and the output report filenames are listed below.

-C specifies a component list with pins listed by name and with no component attributes (<filename>.CMP).

-N specifies a node list with pins listed by name (<filename>.NDE).

-P specifies a packaging list (<filename>.PKL).

-W specifies a wire list (<filename>.WRL).

-M specifies a material list with no attributes (<filename>.MAT).

Options can be linked together in a chain. For example, -CMWNP is equivalent to -C -M -W -N -P. If you do not specify any options, PC-FORM uses -C as the default.

filename is the name of the input netlist file. If you do not specify a filename extension, PC-FORM uses the default .NLT extension.

An example of a command line is shown below.

```
PCFORM -CNPWM TESTFILE.NLT
```

After you enter the command line, the program starts execution with no further input from you. When processing is complete, the program returns you to DOS.

Batch Mode

Batch mode is a variation of command line mode where you can use a DOS batch file to run PC-FORM several times consecutively without user intervention.

You can use any text editor to create the batch file. Each line of the batch file is in the same format used for a command line.

You must use a .BAT filename extension when naming the batch file.

Figure 3-6 shows an example of a batch file.

```
PCFORM -CN TESTFILE.NLT  
PCFORM -CNW TEST2.PNL  
PCFORM -C TEST3.PNL
```

Figure 3-6. Sample Batch File

To run the batch file, at the DOS prompt, type the batch filename, then press [Return].

PC-FORM runs as many times as the number of command lines listed in the batch file, displaying each netlist filename in turn. When processing is complete, the program returns you to DOS.

VIEWING AND PRINTING THE OUTPUT FILES

To view or print an output file, first exit from PC-FORM. To view the file, at the DOS prompt, type the following command:

```
TYPE <filename> [Return]
```

To print the file, you can use the TYPE command with the DOS print function on:

```
TYPE <filename> [Ctrl]-[P] [Return]
```

When the file is printed, you can switch off the print function by pressing [Ctrl]-[P] again.

You can also use the DOS PRINT command to print a file.

Appendix B, "Sample Output Reports," describes all the reports generated by PC-FORM.



APPENDIX A. ERROR MESSAGES

This appendix lists and explains common error messages generated by PC-FORM.

**Message: Not enough memory space or ...
CONFIG.SYS does not have FILES=15.**

Cause: The program requires 640K of memory and the appropriate text in the CONFIG.SYS file.

Action: Be sure you have 640K available. If necessary, use a text editor to edit the CONFIG.SYS file.

Message: Failed producing <filename>.

Cause: PC-FORM has found a schematic symbol in the input netlist that was not pre-assigned a reference designator and pin numbers.

Action: To produce the named file, edit the original schematic database through PC-CAPS to assign reference designators and pin numbers, then rerun PC-NODES, PC-LINK, and PC-FORM.

Message: <filename> not found.

Cause: PC-FORM was unable to find the input binary netlist specified.

Action: Re-enter the correct filename, with the correct directory path or drive designator if necessary.

Message: <filename> format is not compatible.

Cause: The input file specified is not a binary netlist.

Action: Make sure that your input file is a schematic netlist, expanded schematic netlist, physical netlist, or back-annotated netlist.

APPENDIX S. SAMPLE OUTPUT REPORTS

This appendix contains samples of the output reports produced by PC-FORM. Only the first page of each report is shown. Included are three component lists, two node lists, a packaging list, a wire list, and two material lists.

Each output file begins with a header that identifies the file. The remaining information is explained with the examples. The PC-CAPS or PC-CARDS origin of each type of information is also given.

NOTE: PC-NODES assigns names to unnamed component instances and unnamed nets in the format, respectively, of UCsssnnn and UNsssnnn, where *sss* is the sheet ID (000 if the SHEET attribute is not assigned in the database file) and *nnn* is the number identifying the component or the net, assigned sequentially starting with 000.

EXAMPLE 1: COMPONENT LIST FROM SCHEMATIC DATABASE WITH PIN NAMES

This component list was produced from the netlist of a PC-CAPS database used for semicustom IC design. A pin list is included, identifying pins by name, and component attributes are given.

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

The **symbol name**, which is the name of the symbol file; for example, A02.

The **instance name** assigned using the NAME/COMP command in DETL mode or assigned by PC-NODES; for example, UC000000.

The **pin list**, which lists all of the symbol's pins. The entire pin list is enclosed in parentheses. Each entry in the pin list includes:

The **pin name** assigned to the symbol using the ENTR/PIN command in SYMB mode; for example, Y'.

The **net name** of the net connected to the pin, assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000014.

The **attribute list**, which gives all the attribute keywords and values specified using the ATTR/ACOM command in SYMB and DETL mode. This design uses only one attribute, FTYPE; for example, FTYPE="INP." The entire list is enclosed in square brackets ([]). When more than one attribute is listed, each attribute is on a separate line and attributes are separated with commas.

```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 03:46:15 PM
% File In : ICRSY.NLT
% File Out : ICRSY.CMP
% Format : P-CAD COMPONENT LIST
%
%*****

```

ICRSY()

```

BEGIN
A02:UC000000 ( Y' =UN000014, Y =,
              A =UN000000, B =CON1,
              C =UN000027 )
              [ FTYPE ="INP" ]
R01:UC000001 ( Y =INTN1, A =CON0,
              B =UN000001, C =CON0,
              E =CON1 )
              [ FTYPE ="OUT" ]
R01:UC000002 ( Y =INTN2, A =CON0,
              B =UN000002, C =CON0,
              E =CON1 )
              [ FTYPE ="OUT" ]
R01:UC000003 ( Y =INTN4, A =CON0,
              B =UN000026, C =CON0,
              E =CON1 )
              [ FTYPE ="OUT" ]
A01:UC000004 ( Y' =UN000018, Y =,
              A =UN000027, B =CON0 )
A01:UC000005 ( Y' =, Y =UN000009,
              A =UN000028, B =CON0 )
A01:UC000006 ( Y' =, Y =UN000011,
              A =UN000029, B =CON0 )
A01:UC000007 ( Y' =UN000012, Y =,
              A =UN000030, B =CON0 )
A01:UC000008 ( Y' =, Y =UN000016,
              A =UN000034, B =CON0 )
A01:UC000009 ( Y' =, Y =UN000017,
              A =UN000033, B =CON0 )
A01:UC000010 ( Y' =, Y =UN000005,
              A =UN000032, B =CON0 )
A01:UC000011 ( Y' =, Y =UN000003,
              A =UN000031, B =CON0 )
R01:UC000012 ( Y =INTN3, A =CON0,
              B =UN000024, C =CON0,
              E =CON1 )
              [ FTYPE ="OUT" ]
R03:UC000013 ( Y =INTN5, A =CON0,
              B =UN000025, C =UN000018,
              E =CON1 )
              [ FTYPE ="OUT" ]
H04:UC000014 ( Y' =UN000015, Y =,
              Z =UN000013, Z' =,
              B =UN000035, C =UN000014,

```

EXAMPLE 2: COMPONENT LIST FROM SCHEMATIC DATABASE WITH PIN NUMBERS

This component list was produced from the netlist of a PC-CAPS schematic database. A pin list is included, identifying pins by number. No component attributes are given.

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

The **symbol name**, which is the name of the symbol file; for example, 74107.

The **instance name** assigned using the NAME/COMP command in DETL mode or assigned by PC-NODES; for example, UC000000.

The **reference designator** assigned using the SCMD/PNUM command in DETL mode; for example, U1.

The **pin list**, which lists all of the symbol's pins. The entire pin list is enclosed in parentheses. Each entry in the pin list includes:

The **package pin number** assigned to the symbol using the SCMD/PNUM command in DETL mode; for example, 1.

The **net name** of the net connected to the pin, assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000014.

```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 02:15:15 PM
% File In : EZBNEWB.NLT
% File Out : EZBNEWB.CMP
% Format : P-CAD COMPONENT LIST
%
%*****
    
```

EZBNEW()

```

BEGIN
74107:UC000000:U1      ( 1      =UN000001, 12      =UN000026,
                       4          =UN000002, 13          =CLR,
                       2          =QA', 3              =QA )
74107:UC000001:U2      ( 1          =UN000005, 12          =UN000026,
                       4          =UN000006, 13          =CLR,
                       2          =QB', 3              =QB )
74107:UC000002:U1      ( 8          =UN000008, 9          =UN000026,
                       11         =UN000007, 10         =CLR,
                       6          =QC', 5              =QC )
74107:UC000003:U2      ( 8          =UN000016, 9          =UN000026,
                       11         =UN000015, 10         =CLR,
                       6          =QD', 5              =UN000022 )
7411:UC000004:U3       ( 1          =UN000021, 2          =QA,
                       13         =ENABLE/T, 12         =RCO )
7411:UC000005:U3       ( 3          =UN000028, 4          =QA,
                       5          =UN000021, 6          =UN000025 )
7411:UC000006:U4       ( 1          =UN000028, 2          =QA,
                       13         =QB, 12             =UN000020 )
7411:UC000007:U4       ( 3          =UN000028, 4          =QA,
                       5          =QD', 6              =UN000027 )
7404:UC000008:U5       ( 2          =LOAD, 1          =LOAD' )
7404:UC000009:U5       ( 4          =UN000026, 3          =CLOCK )
7404:UC000010:U5       ( 6          =CLR, 5          =CLEAR' )
7420:UC000011:U6       ( 1          =UN000028, 2          =QA,
                       4          =QB, 5              =QC,
                       6          =UN000024 )
7432:UC000012:U7       ( 1          =UN000024, 2          =UN000025,
                       3          =UN000023 )
7432:UC000013:U7       ( 4          =UN000023, 5          =LOAD,
                       6          =UN000017 )
7432:UC000014:U7       ( 9          =UN000020, 10         =LOAD,
                       8          =UN000014 )
7432:UC000015:U8       ( 1          =UN000028, 2          =LOAD,
                       3          =UN000004 )
7432:UC000016:U8       ( 4          =UN000027, 5          =LOAD,
                       6          =UN000011 )
7408:UC000017:U9       ( 1          =UN000003, 2          =UN000004,
                       3          =UN000001 )
7408:UC000018:U9       ( 4          =UN000004, 5          =UN000000,
                       6          =UN000002 )
7408:UC000019:U9       ( 10         =UN000009, 9          =UN000011,
                       8          =UN000005 )
    
```

EXAMPLE 3: COMPONENT LIST FROM PCB DATABASE

This component list was produced from the netlist of a PC-CARDS PCB database. A pin list is included, identifying the pins by number. No component attributes are given.

The component list has one record for each component (PCB part) in the design. Each record of the component list includes:

The **part name**, which is the name of the part file; for example, 74107.

The **instance name or reference designator**, assigned by one of the following: the PC-CARDS NAME/COMP command in DETL mode, the PC-CAPS SCMD/PNUM command in DETL mode, PC-PACK, or PC-NODES; for example, U1.

The **pin list**, which lists all of the part's pins. The entire pin list is enclosed in parentheses. Each entry on the pin list includes:

The pin number, which is determined by the sequence in which pins are entered on the part using the ENTR/PIN command in SYMB mode; for example, 1. (In a PCB database, this number is often the same as the pin name.)

The net name of the net connected to the pin, assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000001.


```

%*****
%
% Copyright (C) 1984,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 10:53:42 AM
% File In : EZBNEWC.PNL
% File Out : EZBNEWC.CMP
% Format : P-CAD COMPONENT LIST
%
%*****
    
```

EZBNEWA()

```

BEGIN
74107:U1 ( 1 =UN000001, 2 =QA',
          3 =QA, 4 =UN000002,
          5 =QC, 6 =QC',
          7 =, 8 =UN000008,
          9 =UN000026, 10 =CLR,
          11 =UN000007, 12 =UN000026,
          13 =CLR, 14 = )
74107:U2 ( 1 =UN000005, 2 =QB',
          3 =QB, 4 =UN000006,
          5 =UN000022, 6 =QD',
          7 =, 8 =UN000016,
          9 =UN000026, 10 =CLR,
          11 =UN000015, 12 =UN000026,
          13 =CLR, 14 = )
7411:U3 ( 1 =UN000021, 2 =QA,
          3 =UN000028, 4 =QA,
          5 =UN000021, 6 =UN000025,
          7 =, 8 =,
          9 =, 10 =,
          11 =, 12 =RCO,
          13 =ENABLE/T, 14 = )
7411:U4 ( 1 =UN000028, 2 =QA,
          3 =UN000028, 4 =QA,
          5 =QD', 6 =UN000027,
          7 =, 8 =,
          9 =, 10 =,
          11 =, 12 =UN000020,
          13 =QB, 14 = )
7404:U5 ( 1 =LOAD', 2 =LOAD,
          3 =CLOCK, 4 =UN000026,
          5 =CLEAR', 6 =CLR,
          7 =, 8 =,
          9 =, 10 =,
          11 =, 12 =,
          13 =, 14 = )
7420:U6 ( 1 =UN000028, 2 =QA,
          3 =, 4 =QB,
          5 =QC, 6 =UN000024,
          7 =, 8 =,
          9 =, 10 =,
          11 =, 12 =,
          13 =, 14 = )
7432:U7 ( 1 =UN000024, 2 =UN000025,
          3 =UN000023, 4 =UN000023,
    
```

EXAMPLE 4: NODE LIST FROM SCHEMATIC DATABASE

This node list was produced from the netlist of a PC-CAPS database used for semicustom IC design. Pins are identified by name.

The node list has one record for each node (net) in the design. Each record of the node list includes:

The **net name**, assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000000.

The **pin names** of all the pins connected to the net, assigned using the ENTR/PIN command in SYMB mode; for example, A.

The **symbol names**, which are the filenames of the symbols to which the pins belong; for example, A02.

The **instance names** of the symbols, assigned using the NAME/COMP command in DETL mode or assigned by PC-NODES; for example, UC000000.

The hierarchical design **path** to the net, which lists the the names of the component files at each hierarchical level from the component file in which the net is located to the top level of the design; for example, <ICRSY.

```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 03:46:17 PM
% File In : ICRSY.NLT
% File Out : ICRSY.NDE
% Format : P-CAD NODE LIST
%
%*****
    
```

```

%
% NODE      PINS      COMPONENT      PATHNAME
% ----      -
%
UN000000
  A          A02:UC000000
  OUT        PADIN:UC000026
          <ICRSY

UN000001
  B          R01:UC000001
  Q          H31:UC000018
          <ICRSY

UN000002
  B          R01:UC000002
  Q          H31:UC000021
          <ICRSY

UN000003
  Y          A01:UC000011
  B0        H40:UC000017
          <ICRSY

UN000004
  Z          H04:UC000015
  A1        H40:UC000017
          <ICRSY

UN000005
  Y          A01:UC000010
  A0        H40:UC000017
          <ICRSY

UN000006
  Y'        H04:UC000015
  B1        H40:UC000017
          <ICRSY

UN000007
  A          H04:UC000014
  Q'        H31:UC000021
  C         H59:UC000022
  F         H59:UC000022
          <ICRSY
    
```

EXAMPLE 5: NODE LIST FROM PCB DATABASE

This node list was produced from the netlist of a PC-CARDS PCB database. Pins are identified by number.

The node list has one record for each node (net) in the design. Each record of the node list includes:

The **net name**, assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000000.

The **pin numbers** of all the pins connected to the net; for example, 5. Pin numbers are determined by the sequence in which pins are entered on a part using the ENTR/PIN command in SYMB mode. (In a PCB database, this number is often the same as the pin name.)

The **part names**, which are the filenames of the parts to which the pins belong; for example, 7408.

The **instance names or reference designators** of the parts, assigned by one of the following: the PC-CARDS NAME/COMP command in DETL mode, the PC-CAPS SCMD/PNUM command in DETL mode, PC-PACK, or PC-NODES; for example, U1.

The hierarchical design **path** to the net, which lists the names of the component files at each hierarchical level from the component file in which the net is located to the top level of the design; for example, <ICRSY.

```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc. *
%
% Program : PC-FORM VERSION 1.31 *
% Date : Jul 11 1986 *
% Time : 10:53:45 AM *
% File In : EZBNEWC.PNL *
% File Out : EZBNEWC.NDE *
% Format : P-CAD NODE LIST *
%
%*****
    
```

```

%
% NODE      PINS      COMPONENT      PATHNAME
% -----
%
UN000000
      5      7408:U9      <EZBNEWA
      2      7400:U12
      6      7400:U12

UN000001
      1      74107:U1     <EZBNEWA
      3      7408:U9

UN000002
      4      74107:U1     <EZBNEWA
      6      7408:U9

UN000003
      1      7408:U9      <EZBNEWA
      3      7400:U12

LOAD
      2      7404:U5      <EZBNEWA
      5      7432:U7
      10     7432:U7
      2      7432:U8
      5      7432:U8
      1      7400:U12
      5      7400:U12
      9      7400:U12
      12     7400:U12
      2      7400:U13
      4      7400:U13
      10     7400:U13
      12     7400:U13

UN000004
      3      7432:U8      <EZBNEWA
      2      7408:U9
      4      7408:U9
    
```

EXAMPLE 6: PACKAGING LIST

This packaging list was produced from the netlist of a PC-CAPS schematic database that included packaging information.

The packaging list has one record for each schematic symbol. Symbols are sorted by reference designator. Each record of the packaging list includes:

The **reference designator** assigned using the SCMD/PNUM command in DETL mode; for example, U1.

The **pin list**, which lists all of the symbol's pins. The entire pin list is enclosed in parentheses. Each entry in the pin list includes:

The **package pin numbers**, assigned to the symbol using the SCMD/PNUM command in DETL mode; for example, 1.

The **net names** of the nets attached to the pins, assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000001.

The **attribute list**, which gives all the attribute keywords and values specified using the ATTR/ACOM command in SYMB and DETL mode. This design uses only one attribute, PCL; for example, PCL=(1,1,"d","d"). The entire list is enclosed in square brackets ([]). When more than one attribute is listed, each attribute is on a separate line and attributes are separated with commas.

```

*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 10:45:11 AM
% File In : EZBNEWB.NLT
% File Out : EZBNEWB.PKL
% Format : PACKAGING LIST
%
*****

```

```

U1 ( 1 =UN000001, 12 =UN000026, 4 =UN000002,
    13 =CLR, 2 =QA', 3 =QA )
U2 ( 1 =UN000005, 12 =UN000026, 4 =UN000006,
    13 =CLR, 2 =QB', 3 =QB )
U1 ( 8 =UN000008, 9 =UN000026, 11 =UN000007,
    10 =CLR, 6 =QC', 5 =QC )
U2 ( 8 =UN000016, 9 =UN000026, 11 =UN000015,
    10 =CLR, 6 =QD', 5 =UN000022 )
U3 ( 1 =UN000021, 2 =QA, 13 =ENABLE/T,
    12 =RCO )
    [ PCL =(1,1,"d","d") ]
U3 ( 3 =UN000028, 4 =QA, 5 =UN000021,
    6 =UN000025 )
    [ PCL =(1,1,"d","d") ]
U4 ( 1 =UN000028, 2 =QA, 13 =QB,
    12 =UN000020 )
    [ PCL =(1,1,"d","d") ]
U4 ( 3 =UN000028, 4 =QA, 5 =QD',
    6 =UN000027 )
    [ PCL =(1,1,"d","d") ]
U5 ( 2 =LOAD, 1 =LOAD' )
    [ PCL =(1,1,"D","D") ]
U5 ( 4 =UN000026, 3 =CLOCK )
    [ PCL =(1,1,"D","D") ]
U5 ( 6 =CLR, 5 =CLEAR' )
    [ PCL =(1,1,"D","D") ]
U6 ( 1 =UN000028, 2 =QA, 4 =QB,
    5 =QC, 6 =UN000024 )
    [ PCL =(1,1,"d","d") ]
U7 ( 1 =UN000024, 2 =UN000025, 3 =UN000023 )
    [ PCL =(1,1,"d","d") ]
U7 ( 4 =UN000023, 5 =LOAD, 6 =UN000017 )
    [ PCL =(1,1,"d","d") ]
U7 ( 9 =UN000020, 10 =LOAD, 8 =UN000014 )
    [ PCL =(1,1,"d","d") ]
U8 ( 1 =UN000028, 2 =LOAD, 3 =UN000004 )
    [ PCL =(1,1,"d","d") ]
U8 ( 4 =UN000027, 5 =LOAD, 6 =UN000011 )
    [ PCL =(1,1,"d","d") ]
U9 ( 1 =UN000003, 2 =UN000004, 3 =UN000001 )
    [ PCL =(1,1,"d","d") ]
U9 ( 4 =UN000004, 5 =UN000000, 6 =UN000002 )
    [ PCL =(1,1,"d","d") ]
U9 ( 10 =UN000009, 9 =UN000011, 8 =UN000005 )
    [ PCL =(1,1,"d","d") ]

```

EXAMPLE 7: WIRE LIST

This wire list was produced from the netlist of a PC-CAPS schematic database. The wire list has one record for each node, or net. Each record of the wire list includes:

The **net name** assigned using the NAME/NET command in DETL mode or assigned by PC-NODES; for example, UN000000.

The **pin list**, which includes, for each pin attached to the net:

The **reference designator** of the component, assigned using the SCMD/PNUM command in DETL mode.

The **package pin number**, assigned to the symbol using the SCMD/PNUM command in DETL mode; for example, 5.


```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 10:45:14 AM
% File In : EZBNEWB.NLT
% File Out : EZBNEWB.WRL
% Format : WIRE LIST
%
%*****
    
```

```

%
% NODE REF. DESIGNATOR-PIN
% ---- -
%
UN000000 U9-5 U12-2 U12-6
UN000001 U1-1 U9-3
UN000002 U1-4 U9-6
UN000003 U9-1 U12-3
LOAD U5-2 U7-5 U7-10
U8-2 U8-5 U12-1
U12-5 U12-9 U13-2
U13-4 U13-10 U12-12
U13-12
UN000004 U8-3 U9-2 U9-4
UN000005 U2-1 U9-8
UN000006 U2-4 U10-3
UN000007 U1-11 U10-8
UN000008 U1-8 U10-6
UN000009 U9-10 U12-8
UN000010 U10-4 U13-6
UN000011 U8-6 U9-9 U10-1
UN000012 U10-2 U12-10 U13-3
ENABLE/T U3-13 U11-9 P1-6
UN000013 U10-9 U13-5 U13-8
UN000014 U7-8 U10-5 U10-10
QA U1-3 U3-2 U3-4
U4-2 U4-4 U6-2
P1-1
UN000015 U2-11 U11-6
UN000016 U2-8 U11-3
UN000017 U7-6 U11-2 U11-4
UN000018 U11-1 U12-11
UN000019 U11-5 U12-13 U13-11
UN000020 U4-12 U7-9
CLEAR' U5-5 P1-2
UN000021 U3-1 U3-5
QB U2-3 U4-13 U6-4
P1-3
QD' U2-6 U4-5 P1-14
UN000022 U2-5 P1-7
UN000023 U7-3 U7-4
UN000024 U6-6 U7-1
UN000025 U3-6 U7-2
DATA/D P1-9 U13-13
DATA/A U12-4 P1-13
LOAD' U5-1 P1-11
DATA/B U13-1 P1-15
    
```

EXAMPLE 8: MATERIAL LIST FROM SCHEMATIC DATABASE

This material list, or Bill of Materials, was produced from the netlist of a PC-CAPS database used for semicustom IC design.

The material list has one record for each type of schematic symbol in the design. Each record of the material list includes:

The **item number**, which is assigned by PC-FORM in sequence starting with 1; for example, 1.

The **quantity**, or number of times the symbol is used in the design; for example, 1.

The **symbol name**, which is the filename of the symbol; for example, A02.

The **instance name** for each instance of the symbol, assigned using the NAME/COMP command in DETL mode or assigned by PC-NODES; for example, UC000000.

NOTE: The material list also has a column headed "DESCRIPTION." This space contains any attributes selected.

```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 03:46:20 PM
% File In : ICRSY.MLT
% File Out : ICRSY.MAT
% Format : P-CAD MATERIALS LIST
%
%*****
    
```

ITEM	QTY	COMP-NAME	INSTANCE-NAME	DESCRIPTION
----	---	-----	-----	-----
1	1	A02	UC000000	
2	4	R01	UC000001 UC000002 UC000003 UC000012	
3	8	A01	UC000004 UC000005 UC000006 UC000007 UC000008 UC000009 UC000010 UC000011	
4	1	R03	UC000013	
5	2	H04	UC000014 UC000015	
6	2	H40	UC000016 UC000017	
7	4	H31	UC000018 UC000019 UC000020 UC000021	
8	1	H59	UC000022	
9	9	PADIN	UC000023 UC000024 UC000025 UC000026 UC000027 UC000028 UC000029 UC000030 UC000031	
10	5	PADOUT	UC000032 UC000033 UC000034 UC000035 UC000036	

EXAMPLE 9: MATERIAL LIST FROM PCB DATABASE

This material list, or Bill of Materials, was produced from the netlist of a PC-CARDS PCB database.

The material list has one record for each type of PCB part in the design. Each record of the material list includes:

The **item number**, which is assigned by PC-FORM in sequence starting with 1; for example, 1.

The **quantity**, or number of times the part is used in the design; for example, 2.

The **part name**, which is the filename of the part; for example, 74107.

The **instance name or reference designator** for each instance of the part, assigned by one of the following: the PC-CARDS NAME/COMP command in DETL mode, the PC-CAPS SCMD/PNUM command in DETL mode, PC-PACK, or PC-NODES; for example, U1.

NOTE: The material list also has a column headed "DESCRIPTION." This space contains any attributes selected. For example, you might assign each part file the attribute PARTNO=nnn-nnnn-nn.

```

%*****
%
% Copyright (C) 1983,1986 - Personal CAD Systems, Inc.
%
% Program : PC-FORM VERSION 1.31
% Date : Jul 11 1986
% Time : 10:53:49 AM
% File In : EZBNEW.CPNL
% File Out : EZBNEW.CMAT
% Format : P-CAD MATERIALS LIST
%
%*****
    
```

ITEM	QTY	PART-NAME	REFERENCE-DESIGNATOR	DESCRIPTION
1	2	74107	U1, U2	
2	2	7411	U3, U4	
3	1	7404	U5	
4	1	7420	U6	
5	2	7432	U7, U8	
6	3	7408	U9, U10, U11	
7	1	CON20	P1	
8	2	7400	U12, U13	
9	1	RC07	R1	
10	1	CS13B	C1	
11	13	CK05	C2, C3, C4, C5, C6, C7, C8, C9, C10, C11, C12, C13, C14	





PREPACK™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.



ABOUT THIS MANUAL

This manual describes PREPACK, the P-CAD utility program that produces a binary component library cross-reference file for P-CAD's PC-PACK program. PC-PACK uses this file to package symbols in the schematic database into equivalent printed circuit board (PCB) parts for the PCB layout.

This manual provides an overview of PREPACK, instructions for creating the input file, operating procedures, and an appendix describing common error messages.



CONTENTS

OVERVIEW.....	4-1
Input	4-2
Output	4-2
CREATING OR EDITING THE ASCII	
INPUT FILE	4-5
Regular Format.....	4-5
Modular Format	4-8
USING PREPACK.....	4-15
APPENDIX A. ERROR MESSAGES	4-19

FIGURES

4-1. PREPACK Input and Output.....	4-1
4-2. Sample Input File	4-7
4-3. Sample Entries with Comments.....	4-8
4-4. Sample Modular Format File.....	4-12
4-5. PREPACK Program Screen	4-16
4-6. Sample PREPACK Program Screen.....	4-17

TABLES

4-1. Component Type IDs.....	4-10
------------------------------	------

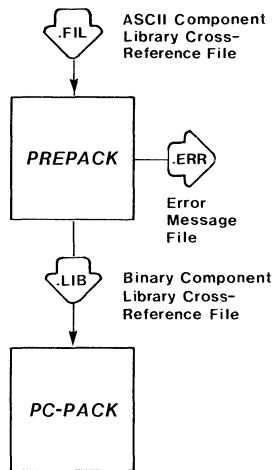


OVERVIEW

PREPACK translates a user-created ASCII component library cross-reference file, which maps schematic logic symbols to PCB parts, into binary format.

This binary file is used as input for PC-PACK.

Figure 4-1 shows PREPACK input and output.



86204

Figure 4-1. PREPACK Input and Output

INPUT

The input to PREPACK is an ASCII component library cross-reference file (<filename>.FIL) containing information for mapping PC-CAPS symbols to physical parts. If your database is to be packaged into a PCB database to be used with PC-CARDS, this input is in free-form tabular format. If you plan to use the packaging information only to update the schematic, the input file is in modular format.

Each P-CAD symbol and part library includes an ASCII component library cross-reference file that maps the components in the P-CAD symbol library to the parts in the corresponding part library. The P-CAD AUTOLOADER installs each ASCII cross-reference file in the appropriate PRT or SYM directory.

If your design includes parts from more than one part library or if you are using symbols and parts in addition to those supplied by P-CAD, you can combine and edit P-CAD ASCII cross-reference files as explained in the "Creating or Editing the ASCII Input File" section.

Modular format requires no part files; you specify detailed packaging information for each component symbol. You can create a modular format file using the procedures described in the "Creating or Editing the ASCII Input File" section.

OUTPUT

PREPACK outputs a binary component library cross-reference file (<filename>.LIB) that contains the same information as the input ASCII file. This binary file is a required input to the PC-PACK program.

Each P-CAD part and symbol library includes a binary cross-reference file produced by running the P-CAD supplied ASCII cross-reference file (<filename>.FIL) through PREPACK. The binary cross-reference file is located in the PRT and SYM directories. If your design includes only the P-CAD supplied symbols in one library, you can use this file as input to PC-PACK. You do not need to run PREPACK for the design.

If the ASCII file you input into PREPACK is in the modular format, the output binary file, when input into PC-PACK, will produce a packaged binary netlist and a command file for back annotation, but no PCB database.

If PREPACK encounters errors during program operation, it also outputs an error message file, PREPACK.ERR.



CREATING OR EDITING THE ASCII INPUT FILE

Before running PREPACK, you must either create an ASCII cross-reference file or edit a P-CAD supplied ASCII cross-reference file (<filename>.FIL). This file contains the schematic symbol to PCB part mapping information.

You can create or edit the ASCII input file for PREPACK using a text editor. The following sections provide guidelines and examples for creating and editing ASCII cross-reference files in both regular and modular formats.

REGULAR FORMAT

An entry in regular format consists of four items as shown below:

```
<gates> <part type> <part name> <sch symbol>
```

where:

gates is the number of schematic symbols or items (for instance, gates) contained in the PCB package device named in "part name."

part type is the manufacturer's part number or commentary data used to designate the PCB package named in "part name." This item is a maximum 8 alphanumeric characters long and must start and end on the same line. It is commentary only.

part name is the filename of the PCB package or part to which you want your schematic symbols mapped. This file must exist on your disk as a PC-CARDS part file (<filename>.PRT). Unless the part file is located in the directory from which you plan to run PC-PACK, the "part name" item must include a directory path. This item can be up to 40 characters long and must start and end on the same line.

sch symbol is the filename of the schematic symbol to be packaged into the PCB part specified in "part name." The "sch symbol" must be a valid name of a PC-CAPS created symbol (<filename>.SYM). It does not need the path to the directory where symbols reside. This item can be up to 40 characters long and must start and end on the same line.

An entry may be entirely on one line or spread over several lines. However, each item must be contained entirely on one line. Two adjacent items must be separated by at least one blank or comma.

NOTE: For successful operation of PREPACK and PC-PACK, the part and symbol filenames listed in the ASCII cross-reference file can contain only alphanumeric characters (letters or numbers) and a period that separates the filename and extension. Nonalphanumeric marks, such as underscores, asterisks, percentage marks, etc., are not recognized by either program, except for the colon (:), backslash (\), and hyphen (-).

You can add comments anywhere in the file. Comments begin with a percent sign (%). All text from the % to the end of the line is ignored by the program.

Figure 4-2 shows a sample PREPACK input file with each entry specified entirely on one line.

```

% Component Library Cross-reference File
% Sample File
% Date: 4/1/86
%
% No.      Part          Component          Schematic
% Gates   Type           Part Name         Symbol Name
%-----
      4      7400      \PCAD\PR\TTL\7400.PRT      7400.SYM
      4      7402      \PCAD\PR\TTL\7402.PRT      7402.SYM
      6      7404      \PCAD\PR\TTL\7404.PRT      7404.SYM
      6      7407      \PCAD\PR\TTL\7407.PRT      7407.SYM
      4      7408      \PCAD\PR\TTL\7408.PRT      7408.SYM
      3      7410      \PCAD\PR\TTL\7410.PRT      7410.SYM
      3      7411      \PCAD\PR\TTL\7411.PRT      7411.SYM
      2      7420      \PCAD\PR\TTL\7420.PRT      7420.SYM
      2      74107     \PCAD\PR\TTL\74107.PRT     74107.SYM
% Discrete Components and Connectors
      1      RES      \PCAD\PR\DISCRETE\RC07.PRT  RC07.SYM
      1      CAP      \PCAD\PR\DISCRETE\CK05.PRT  CK05.SYM
      1      CAP      \PCAD\PR\DISCRETE\CS13B.PRT  CS13B.SYM
     20      CON      \PCAD\PR\DISCRETE\CON20.PRT  CON20.SYM

```

Figure 4-2. Sample Input File

Figure 4-3 shows two entries with items specified on more than one line and with comments.

4, 7400, \PCAD\PRT\TTL\7400.PRT	% PCB part name
7400.SYM	% schematic symbol name
1 RES, \PCAD\PRT\DISCRETE\RC07.PRT	% discrete case size
RC07.SYM	% symbol name

Figure 4-3. Sample Entries with Comments

You may want to combine two (or more) P-CAD supplied ASCII cross-reference files into one file containing all the parts in both libraries. The \PCAD\PRT directory is a good location for a combined file. Alternately, you can create this file in your working project directory.

To combine files, start in the appropriate directory, use the DOS COPY command as shown in the example below.

```
COPY \PCAD\PRT\TTL\7400.FIL+
\PCAD\PRT\DISCRETE\DISCRETE.FIL
\PCAD\PRT\CROSSREF.FIL
```

After you create the new file, you must edit it so that each part file entry includes the appropriate directory path. You can also delete any entries you do not need.

MODULAR FORMAT

If your system does not include the PC-CARDS program and you want to add packaging information to your

schematic, you must "manufacture" parts using a modular format.

Like regular format, modular format contains one entry for each component.

The file must begin with \$M, which is a flag that tells the program that the input file is written in modular rather than regular format. The structure of a modular format entry is shown below:

```
<package name> (<pin list>)
[
TYPE = <type>
GNAM = "<filename.sym>"
NGAT = <number of gates per package>
NPIN = <number of pins per gate>
PNAM = (<pin names>)
PREF = (<pin reference>)
GLOB = (<global pin reference>)
]
```

where:

package name is a designation for the PCB package device to be "manufactured"; for example, "package name" could be the manufacturer part code. It can be up to 14 alphanumeric characters long.

pin list is a list of pin names for the PCB package device, in the package device pin order. Each pin name can be up to 8 alphanumeric characters long. Pin names must be separated from each other by at least one blank or comma. The list can be continued on the next line; however, entire list must be enclosed in parentheses.

TYPE, GNAM, NGAT, NPIN, PNAM, PREF, and GLOB refer to the schematic component parameters described below. This entire group of parameters must be enclosed in square brackets.

type is the Component Type ID. PC-PACK uses this data for generating reference designator names and for part grouping. Table 4-1 shows valid "type" values, the component types they define, and the corresponding reference designator names.

Table 4-1. Component Type IDs

Type	Description	Reference Designator
10000	IC Packages	Un
11100	Resistors	Rn
11200	Capacitors	Cn
11300	Inductors	Ln
11400	Transistors	Qn
11000	Other Discretes	Xn
12000	Connectors	Un
13000	Miscellaneous	Xn

filename.sym is the name of the schematic symbol to map to the PCB package device. It must be enclosed in quotation marks.

number of gates is the number of schematic symbols contained in the PCB package device.

number of pins per gate is the number of pins per schematic symbol.

pin names is a list of pin names on the schematic symbol. Each pin name can be up to 8 alphanumeric characters long. Pin names must be separated from each other by at least one blank or comma. The entire list of pin names must be enclosed in parentheses.

pin reference is a listing of PCB package device pin names. The pins are listed in an order corresponding to the position of the schematic symbol pin names and schematic symbol logical position within the PCB package. The entire list of pin names must be enclosed in parentheses.

global pin reference is a list of PCB package device pinnames that are automatically connected to a net of the same name (for example, VDD, GND). The entire list of global pin names must be enclosed in parentheses.

An example of a modular format file with one entry is shown in Figure 4-4.

```
$M
7400PRT (1A, 1B, 1Y, 2A, 2B, 2Y GND
        3A, 3B, 3Y, 4A, 4B, 4Y, VCC)
[
TYPE = 10000
GNAM = "7400.SYM"
NGAT = 4
NPIN = 3
PNAM = (A, B, Y)
PREF = (1A, 1B, 1Y, 2A, 2B, 2Y, 3A, 3B, 3Y, 4A, 4B, 4Y)
GLOB = (VCC, GND)
]
```

Figure 4-4. Sample Modular Format File

In this example:

\$M is a modular list flag.

7400PRT is the PCB package device name.

(1A, . . . ,VCC) is a list of PCB package device pin names.

TYPE = 10000 states that the package device's Component Type ID is 10000, meaning that the component is an IC.

GNAM = "7400.SYM" states that the PCB package (7400PRT) contains gate(s) equivalent to schematic symbol logic gate named 7400.SYM in the component library.

NGAT = 4 means that the PCB package device contains four of the 7400.SYM components.

NPIN = 3 means that the 7400.SYM component contains 3 pins.

PNAM = (A, B, Y) means that the 7400.SYM pins are named A, B, and Y.

PREF = (1A, . . . , 4Y) means PCB packaged pins 1A, 1B, and 1Y correspond to pins A, B, Y of the first gate within this package; package pins 2A, 2B, 2Y correspond to pins A, B, Y of the second gate; and so on.

GLOB = (VCC, GND) means that these pins are connected to nets named VCC and GND. If nets VCC and GND do not exist in the original schematic, PC-PACK will create them for the packaged netlist.



USING PREPACK

Before running PREPACK, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview Manual* for a list of requirements.)
- You have installed the PREPACK program file (PREPACK.EXE).
- You have created the input ASCII file.

If you are using the P-CAD recommended directory structure and the input ASCII file is in regular format, run PREPACK from the appropriate PRT directory to ensure that all the PCB part files stored there can be accessed.

If your input ASCII file is in modular format, run PREPACK from the appropriate project directory.

To run PREPACK, follow the steps below.

1. Type:

PREPACK [Return]

When the PREPACK Title Screen appears, press any key to continue.

The system prompts you for the input ASCII component library cross-reference filename as shown in Figure 4-5.

PREPACK

Input Library : <Filename>

Enter the filename; Press [Return] or [Esc] to exit.

Figure 4-5. PREPACK Program Screen

2. Type the name and extension of the ASCII file and press [Return]. Include a directory path or drive designator if the file is not in the current directory.

The system prompts for the name of the output binary component library cross-reference file. The default output filename is the name of the input ASCII file with a .LIB extension as shown in Figure 4-6.

PREPACK

Input Library : TESTFILE.FIL
Output Filename : TESTFILE.LIB

Enter the filename; Press [Return] to accept; [Esc] to cancel.

Figure 4-6. Sample PREPACK Program Screen

3. Press [Return] to accept the default filename, or type another filename and press [Return].

After you enter the output filename, the system displays the message **Compiling...** at the bottom of the screen.

When processing is complete, the system displays the message **... Compilation completed** and moves the cursor to the **Input Library** prompt. To end the program, press [Esc]. The system returns you to DOS.

If PREPACK encounters any errors in compiling the binary file, it displays the following message at the bottom of the screen:

Errors Detected, check PREPACK.ERR file

To view the error message file, first exit from PREPACK. Then, at the DOS prompt, type the following command:

TYPE PREPACK.ERR [Return]

Refer to Appendix A, "Error Messages," for a list of possible error messages, the cause of the errors, and appropriate action.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that are generated by PREPACK and are stored in the PREPACK.ERR file. The messages contain the following information:

- The input line number of error
- The input line with error marked by a carat symbol
- The error message

Message: ***** Error on line : 8**
W 7474 7474.PRT 7474.SYM
 ^
**** : Valid number of gates expected.**
**** : Skipped to next line.**

Cause: A nonnumeric character has been used to indicate number of gates on line 8 of the input file. The program skipped to next line to resume reading.

Action: Edit the ASCII input file and rerun PREPACK.

Message: ** Error on line : 12
NPIN 4
 ^
**** : '=' Equal sign expected.**
**** Source skipped to:**
]
 ^

Cause: The equal sign has been omitted from the NPIN parameter on line 12 of the modular format input file. The program skipped to the end of the module to resume reading.

Action: Edit the ASCII input file and rerun PREPACK.

Message: ** Error on line : 23
2 #7480 7480.PRT 7480.SYM
 ^
**** : Valid Part-No expected.**
**** : Skipped to next line.**

Cause: An invalid (for example, nonalphanumeric) character has been used in the part number on line 23 of the input file. The program skipped to the next line to resume reading.

Action: Edit the ASCII input file and rerun PREPACK.

Message: ** Error on line: 19

```
PREF = ( A, B, Y )
          ^
```

**** : ')' Right parenthesis, String or
Alphanumeric expected.**

**** Source skipped to:**

```
PREF = ( A, B, Y )
          ^
```

Cause: In this case, the right parenthesis has actually been omitted from the pin name list on the preceding line (line 18); however, the program did not detect the error until line 19. The program skipped to the next right parenthesis to resume reading.

Action: Edit the ASCII input file and rerun PREPACK.





PC-PACK™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.



ABOUT THIS MANUAL

This manual describes PC-PACK, the P-CAD utility program that generates a PC-CARDS printed circuit board (PCB) database from a PC-CAPS schematic database.

This manual provides an overview of PC-PACK, operating instructions, and appendixes that describe common error messages and explain how to input the packaging command file into PC-CAPS.



CONTENTS

OVERVIEW.....	5-1
Input	5-3
The Schematic Netlist File	5-3
The Binary Component Library	
Cross-Reference File.....	5-3
The PCB Layer Database File.....	5-4
The PCB Part Files	5-6
Output	5-6
The Packaged PCB Database File.....	5-7
The Physical Netlist File.....	5-7
The Packaging Command File	5-8
The Back-Annotated Netlist File.....	5-8
The Error Message File.....	5-9
USING PC-PACK	5-11
APPENDIX A. ERROR MESSAGES	5-17
APPENDIX B. INPUTTING THE PACKAGING	
COMMAND FILE.....	5-21
FIGURES	
5-1. PC-PACK Input and Output.....	5-2
5-2. PC-PACK Program Screen	5-12
5-3. Sample PC-PACK Program Screen.....	5-15



OVERVIEW

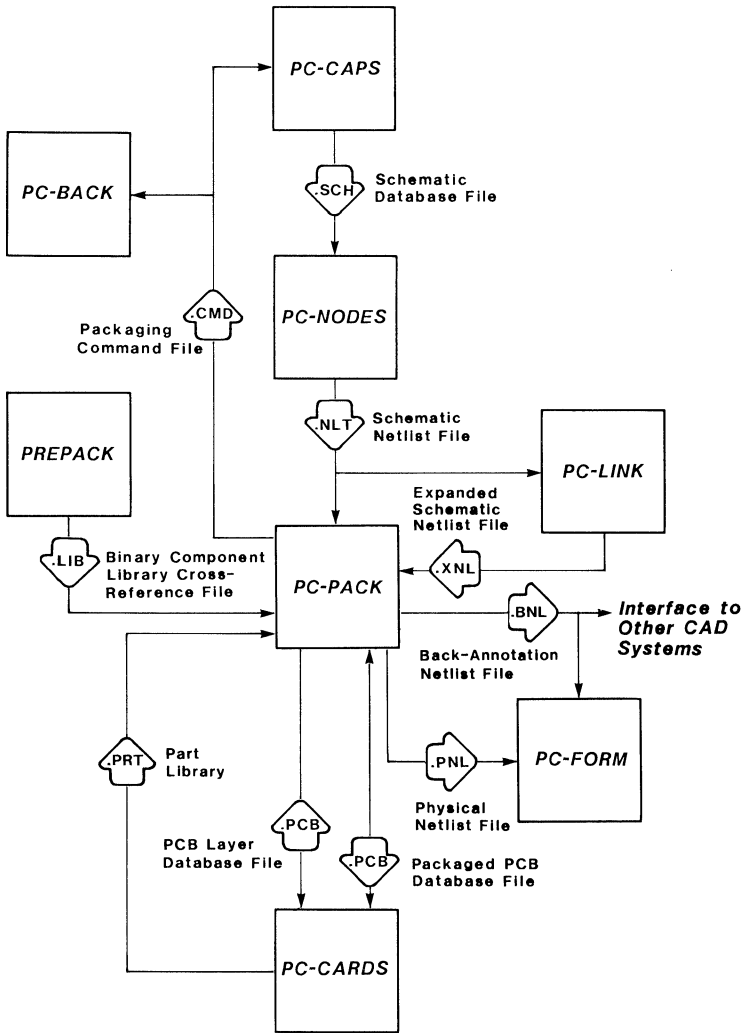
PC-PACK reads a schematic netlist or expanded schematic netlist from a single-sheet or multisheet schematic database, a binary component library cross-reference file, a PCB layer database file, and PCB part files.

It outputs:

- A PCB database with logic gates packaged in IC devices and connected in a preliminary arrangement of the board
- A physical netlist of the PCB database
- A packaging command file containing schematic symbol-to-physical-device packaging information for back annotation of the original schematic database
- A binary netlist of the schematic with package pin numbers and reference designators assigned to each component for input to interface programs

If you do not have the PC-CARDS program, you can use PC-PACK to add package pin numbers and reference designators to the components in a schematic. For this application, the input cross-reference file must be in modular format, the input PCB layer database is not required, and PC-PACK does not produce the PCB database.

Figure 5-1 shows PC-PACK input and output.



86205

Figure 5-1. PC-PACK Input and Output

INPUT

PC-PACK reads four types of input: a schematic netlist file, a binary component library cross-reference file, a PCB layer database file, and all the PCB part files (<filename>.PRT) referred to in the component library cross-reference file.

The Schematic Netlist File

The input schematic netlist file is produced by first running the PC-CAPS schematic database through PC-NODES to create a schematic binary netlist (<filename>.NLT).

If the schematic uses hierarchical or multisheet structure, the netlists for all subcircuits or sheets must then be run through PC-LINK to generate an expanded schematic netlist file (<filename>.XNL).

For more information, refer to the *PC-NODES User's Manual* and the *PC-LINK User's Manual*.

The Binary Component Library Cross-Reference File

The cross-reference file provides packaging information for the components used in the schematic. The binary cross-reference file (<filename>.LIB) is generated by using PREPACK to translate the ASCII cross-reference file (<filename>.FIL) into binary format.

The cross-reference file can be in regular or modular format. In either format, it must include an entry for each schematic component.

A regular format cross-reference file contains the information used by PC-PACK to map schematic symbols to PC-CARDS part files. The entry for each symbol includes the filename for the corresponding PCB part.

Each P-CAD part and symbol library includes a binary cross-reference file that contains information for mapping the PC-CAPS symbols in the symbol library to the PC-CARDS parts in the corresponding part library. This file is located in the PRT and SYM directories. If your design includes only P-CAD supplied symbols, you can use this cross-reference file as input to PC-PACK and you do not need to run PREPACK.

Modular format is useful only if you do not have PC-CARDS but want to add packaging information to your schematic. The packaging information in a modular format cross-reference file does not require PC-CARDS parts. For more information on modular format, refer to the *PREPACK User's Manual*.

The PCB Layer Database File

The PCB layer database file (<filename>.PCB) is generated by PC-CARDS. It provides basic environmental information for the new PCB database to be created.

Primarily used for layering information, this file also provides grid size, grid display, visible and enabled layers, and default wire width. A board outline can also be added to this file.

Each P-CAD part library includes the P-CAD standard PCB layer database file, named LAYS.PCB.

If your input .LIB cross-reference file is in modular format, you do not need to input the PCB layer database file into PC-PACK.

PC-PACK recognizes preplaced and fixed components within a board outline and placement parameters.

Therefore, before running PC-PACK, you can prepare a database for automatic placement in PC-PLACE and PC-CARDS if desired and use it as the input file for the PCB Layer Database File instead of LAYS.PCB.

To prepare the database, you can do one or more of the following:

1. Draw the board outline in PC-PLACE or PC-CARDS.
2. Preplace and fix components in PC-PLACE.

NOTE: Before you preplace and fix components in PC-PLACE, you must enter the components into the database in PC-CARDS. This must be done in PC-CARDS because the ENTR/COMP command is not available in PC-PLACE. You must also name the components. This can be done in PC-CARDS or PC-PLACE. The NAME/COMP command is available in both programs.

3. Specify placement lattice in PC-PLACE.
4. Draw placement barriers in PC-PLACE.
5. Specify components for placement in PC-PLACE.
6. Specify component clearance in PC-PLACE.

7. Specify component association in PC-PLACE.
8. Specify cutline placement in PC-PLACE.
9. Specify sorting parameters in PC-PLACE.

Refer to the *PC-PLACE User's Manual* for more information.

The PCB Part Files

PC-PACK automatically accesses all the PCB part files (<filename>.PRT) that are listed in the component library cross-reference file and are used in the schematic. These PCB part files must be located in the current directory or their directory path must be specified in the component library cross-reference file (refer to the *PREPACK User's Manual* for more information).

NOTE: When using part files other than those supplied by P-CAD, be sure to assign them the proper Component Type ID with the SCMD/SCAT command in PC-CARDS (refer to the SCMD/SCAT command in your *PC-CARDS User's Manual*).

If you are not using PC-PACK to produce a PCB database and your input cross-reference file is in modular format, PC-PACK does not use any part files.

OUTPUT

PC-PACK outputs four files that can be used as input to other programs: a packaged PCB database file to be edited by PC-CARDS, a physical netlist to be input to PC-FORM, a packaging command file to be input to PC-BACK or PC-CAPS, and a back-annotated netlist to

be input to PC-FORM or to interfaces to other CAD systems.

PC-PACK also outputs an error message file if it encounters any errors.

The Packaged PCB Database File

This database file (<filename>.PCB) contains the PCB components and their connecting nets arranged into a preliminary placement to be edited using PC-PLACE. All components are grouped into three categories and placed as follows.

- ICs are placed in even rows and columns, depending on the number used.
- Discretes are placed above the ICs.
- Connectors are placed below the ICs.

Connecting nets are represented in the form of ratsnest connections.

If your input cross-reference file is in modular format, PC-PACK does not produce the PCB database file.

The Physical Netlist File

The physical netlist (<filename>.PNL) is a binary netlist identical to a netlist that PC-NODES would produce from the packaged PCB database.

You can input the packaged netlist to PC-FORM to generate ASCII format reports.

The Packaging Command File

The packaging command file (<filename>.CMD) contains commands, readable by PC-CAPS, for assigning packaging information to components within the original schematic database.

You can input this file together with the swap report file (<filename>.RPT) produced by PC-CARDS to PC-BACK to produce a back-annotation command file, which you can then input to PC-CAPS.

You can also input the packaging command file directly to PC-CAPS to assign the reference designators and PCB part pin numbers generated by PC-PACK. Refer to Appendix B, "Inputting the Packaging Command File," for instructions.

The Back-Annotated Netlist File

The back-annotated netlist file (<filename>.BNL) is a binary netlist in which all the components in the schematic are assigned PCB part reference designators and pin numbers. This file is identical to a netlist that PC-NODES would produce from the schematic annotated with the packaging command file.

You can input the back-annotated netlist to PC-FORM to produce an ASCII report or to some interfaces to other CAD system. See your interface program manual for more information.

The Error Message File

The error message file is named PCPACK.ERR. You can use the DOS TYPE or PRINT commands to view the error messages. Refer to Appendix A, "Error Messages," for explanations of possible error messages.



USING PC-PACK

Before running PC-PACK, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-PACK program file (PCPACK.EXE).
- You have prepared the input binary netlist, the binary component library cross-reference file, and the PCB layer database.
- The part files used in the design are in the current directory or the path to each part file is in the cross-reference file.

If your input cross-reference file references parts from more than one P-CAD part library or if it is in modular format, run PC-PACK from the appropriate project directory.

If your input cross-reference file references parts from only one part library and does not specify a directory path to each part, run PC-PACK from the part directory that contains all the part files.

To run PC-PACK, follow the steps below.

1. Type:

PCPACK [Return]

When the PC-PACK Title Screen appears, press any key to continue.

The system prompts for the input netlist filename as shown in Figure 5-2.

PC-PACK

Schematic Net-List : <Filename>.NLT

Setting PC-Board database environment...

Enter the input filename; Press [Return] or [Esc] to exit.

Figure 5-2. PC-PACK Program Screen

2. Type the input netlist filename and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter the filename extension, PC-PACK enters the default .NLT extension.

The system prompts for the PREPACK-generated component library cross-reference filename.

3. Type the component library cross-reference filename and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter the filename extension, PC-PACK enters the default .LIB extension.

The program checks to see whether the file is in regular or modular format. If the file is in modular format, the program does not display the next two prompts, and you can continue to step 6 below.

The system prompts you for the input PCB layer database filename.

4. Type the name of the PC-CARDS generated PCB layer database file and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter the filename extension, PC-PACK enters the default .PCB extension.

If you do not have a layer database file that you created, press [Return] to display the next prompt. PC-PACK will take the layer information from one of the parts used in the PCB database to be created.

The system prompts for the output packaged PCB database filename. The default is the input netlist filename with a .PCB extension.

5. If you do not want a PCB database output, press [Esc] to display the next prompt.

If you want a PCB database output, press [Return] to accept the default filename or type another filename and press [Return].

The system prompts for the output physical netlist filename. The default filename is the input netlist filename with a .PNL extension.

6. If you do not want a physical netlist output, press [Esc] to display the next prompt.

If you want a physical netlist output, press [Return] to accept the default filename or type another filename and press [Return].

The system prompts for the output packaging command filename. The default is the input netlist filename with a .CMD extension.

7. If you do not want a packaging command file output, press [Esc] to display the next prompt.

If you want a packaging command file output, press [Return] to accept the default filename or type another filename and press [Return].

The last prompt asks for the back-annotated netlist filename. The default is the input netlist filename with a .BNL extension. Figure 5-3 shows a sample

screen with default filenames used for the output files.

PC-PACK

```
Schematic Net-List : \PCAD\PROJ\TESTFILE.NLT
Part Reference    : 7400.LIB
Layer Structure   : <filename>.PCB
PC-Board Database : \PCAD\PROJ\TESTFILE.PCB
PC-Board Net-List : \PCAD\PROJ\TESTFILE.PNL
Packaging Report  : \PCAD\PROJ\TESTFILE.CMD
Annotated Net-List : \PCAD\PROJ\TESTFILE.BNL
```

Enter the report filename; Press [Return] to accept; [Esc] to reject.

Figure 5-3. Sample PC-PACK Program Screen

8. If you do not want a back-annotated netlist, press [Esc] to begin processing.

If you want a back-annotated netlist, press [Return] to accept the default filename or type another filename and press [Return].

PC-PACK displays progress reports at the bottom of the screen.

When processing is complete, the cursor returns to the **Schematic Net-List** prompt. To end the program, press [Esc]. The system returns you to DOS.

If PC-PACK encounters any errors in producing the output files, it displays the following message:

Errors Detected, check PCPACK.ERR

To view the error message file, first exit from PC-PACK. Then at the DOS prompt, type the following command:

TYPE PCPACK.ERR [Return]

Refer to Appendix A, "Error Messages," for a list of possible error messages, the cause of the errors, and appropriate action.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains common error messages that are generated by PC-PACK and are stored in the PCPACK.ERR file.

Message: No Part listed for <filename>.SYM in the given library, this logic gate(s) left unpacked.

Cause: A symbol (<filename>.SYM) in the schematic netlist has not been mapped to a corresponding PCB part file (<filename>.PRT) in the input component library cross-reference file (<filename>.LIB).

Action: Edit your component library cross-reference file, run it through PREPACK, and rerun PC-PACK.

Message: <filename>.PRT, this part does not exist.
Logic gate to be packaged: <filename>.SYM
this logic gate(s) left unpacked.

Cause: The system cannot find the corresponding PCB part file for the specified schematic symbol file.

Action: Make sure the part file in question exists in the correct directory.

Message: <filename>.PRT, Part database format is incompatible.

Logic gate to be packaged: <filename>.SYM
this logic gate left unpacked.

Cause: The specified part file was created by an old version of PC-CARDS in a format incompatible with the current version of PC-PACK.

Action: Update your part library with current part files or generate a new part file with the current version of PC-CARDS.

Message: Failed entering part : <filename>.PRT.
Database may be full.

Cause: The program is incapable of supporting the size PCB database to be created.

Action: Refer to PC-CAPS/PC-CARDS documentation for system limitations.

Message: <filename>.SYM is to be packaged to an ambiguous section of a package referenced as <filename>.PRT.

Cause: The symbol file has been assigned to a non-existing section of the part file.

Action: Either reassign the symbol through PC-CAPS or edit the part file packaging assignment parameters through the SCMD/SPKG command in PC-CARDS.

Message: Assignment parameter does not exist in part named : <filename>.PRT.
Create this parameter thru SCMD/SPKG command in PC-CARDS.

Cause: No packaging assignment parameters have been entered for the specified .PRT file.

Action: Enter assignment parameters through the SCMD/SPKG command in PC-CARDS.

Message: Part <filename>.PRT is not listed in the library. This part is referred by <filename>.SYM thru its PRT parameter.

Cause: A part file that is assigned to a symbol in PC-CAPS using the ATTR/ACOM command and the keyword PRT is not listed in the cross-reference file.

Action: Edit the ASCII library cross-reference file to add the part filename, then run it through PREPACK.

Message: Too many <filename>.SYM are to be packaged into a part referenced as <refdes>.

Cause: Too many symbols in the schematic have been assigned to the part referenced as <refdes>. This mistake may happen when drawing multisheet schematics.

Action: Reassign your symbols through PC-CAPS.



APPENDIX B. INPUTTING THE PACKAGING COMMAND FILE

If components within the original schematic have not been assigned reference designators and pin numbers, you can input the packaging command file directly to PC-CAPS to insert the reference designators and pin numbers assigned by PC-PACK.

To input the packaging commands to the schematic, follow the procedure below:

1. Start PC-CAPS and select **Edit Database** at the Opening Menu.

The PC-CAPS Command Menu appears.

2. Type:

/

The system prompts:

Menu command:

3. Type:

EXE [Return]

The system prompts:

Exec filename:

4. Type:

<filename>.CMD [Return]

The system loads the appropriate schematic and updates the schematic with each assigned reference designator and pin numbers. If the schematic is a multisheet schematic, each sheet is automatically updated and saved in turn.

PC-BACK™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0066-02



ABOUT THIS MANUAL

This manual describes PC-BACK, the P-CAD utility program that allows you to update, or back-annotate, a PC-CAPS schematic database with the packaging assignments and modifications made to the packaged PC-CARDS PCB database.

This manual provides an overview of PC-BACK, operating instructions, and appendixes that describe error messages and explain how to edit swap reports.



CONTENTS

OVERVIEW	6-1
Input	6-2
The Packaging Command File	6-2
The Swap Report File	6-2
Output	6-4
The Back-Annotation File	6-4
The Error Listing File	6-4
USING PC-BACK	6-5
Running PC-BACK	6-5
Viewing the Error Listing File	6-9
Inputting the Back-Annotation File	6-9
APPENDIX A. ERROR MESSAGES	6-11
APPENDIX B. EDITING THE SWAP REPORT ..	6-13
The Swap Pin Record	6-13
The Swap Gate Record	6-15
The Rename Component Record	6-16
The Swap CNAME Record	6-17

FIGURES

6-1.	PC-BACK Input and Output	6-1
6-2.	PC-BACK Program Screen	6-6
6-3.	Sample PC-BACK Program Screen	6-8



OVERVIEW

PC-BACK reads packaging commands from PC-PACK and a swap report from PC-CARDS and generates a back-annotation file, which can be input into PC-CAPS to update, or back annotate, the original schematic with the packaging assignments made in the PCB database.

Figure 6-1 shows PC-BACK input and output.

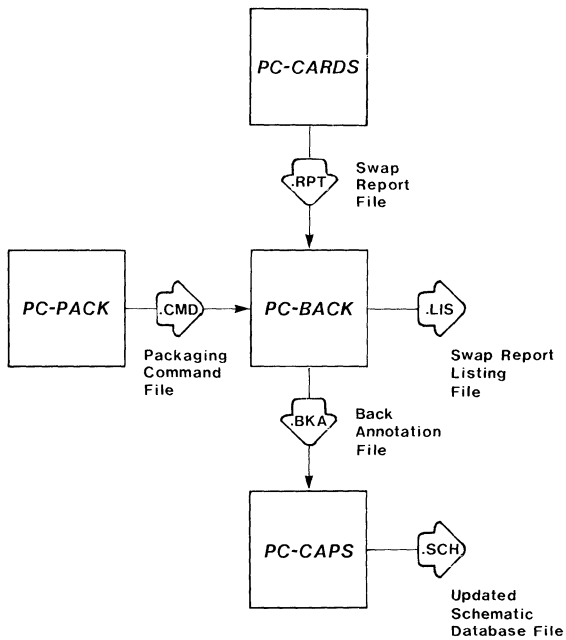


Figure 6-1. PC-BACK Input and Output

INPUT

Input to PC-BACK consists of a packaging command file and a swap report file. The following sections describe these files.

The Packaging Command File

The packaging command file (<filename>.CMD) is generated by PC-PACK, the P-CAD utility program that packages components from a schematic database into physical components in a PCB database.

The packaging command file contains records of these packaging assignments, including reference designators and pin numbers. This file is an ASCII file in executable format. (Refer to your *PC-CAPS* or *PC-CARDS User's Manual* for an explanation of executable files.)

The Swap Report File

After you use PC-PACK to produce a PCB database, you often improve upon the initial packaging assignments using the PC-CARDS SWAP command. The swap report file (<filename>.RPT) is produced automatically by PC-CARDS and contains records of changes made to the initial placement of PCB components.

Each time you swap gates or pins on the PCB design, PC-CARDS records the change in the swap report. The swap report is given the PCB database filename with a .RPT extension. This file contains swaps made during

one or several PC-CARDS editing sessions. For more information, refer to the *PC-CARDS User's Manual*.

NOTE: If you intend to back-annotate the schematic, use PC-BACK for each distinct swap report file generated. When you input a new swap report file into PC-BACK, an input packaging command file is still required. You can use PC-PACK to produce a new packaging command file, or you can use the previous back-annotation file as the command file.

The swap report file is an ASCII file which you can edit using a text editor. You can also add command records to rename components and to swap component names. The format of swap report records is described in Appendix B, "Editing the Swap Report."

NOTE: PC-CARDS produces the swap report only if the swapped gates and pins are assigned reference designators and pin numbers before PC-CARDS is used to edit the database.

If the components within the schematic were not assigned reference designators or pin numbers, you can use the following procedure to ensure that PC-CARDS will produce the swap report file.

1. As soon as you finish running PC-PACK, use the PC-CAPS /EXE command to input the packaging command file into the schematic database. The packaging command file inserts reference designators and pin numbers into the schematic.

2. Rerun the updated schematic through PC-NODES and PC-PACK. This process produces a PCB database with all reference designators and pin numbers assigned.

OUTPUT

PC-BACK outputs the back-annotation file and the error listing file. The following sections describe these files.

The Back-Annotation File

PC-BACK generates the back-annotation file (<filename>.BKA) from information in the packaging command file and the swap report. The format of this file is the same as that of the packaging command file. This file is input to PC-CAPS to update the schematic.

The Error Listing File

The error listing file (<filename>.LIS) is an ASCII file that contains the commands in the swap report file and notes any errors found in the file.

You must check this file for errors to ensure that the final back-annotation file is correct.

USING PC-BACK

To back-annotate the schematic database using PC-BACK, you must carry out three operations:

1. Run PC-BACK to compile a back-annotation file.
2. View the error listing file and correct errors if necessary.
3. Input the back-annotation file to PC-CAPS.

The following sections explain these operations.

RUNNING PC-BACK

Before running PC-BACK, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-BACK program file (PCBACK.EXE).
- You have prepared the schematic database and used PC-PACK to produce the PCB database.
- You have produced the packaging command file and the swap report file, and you have edited the swap report file if necessary (see Appendix B, "Editing the Swap Report").

If you are using the P-CAD recommended directory structure, run PC-BACK from the appropriate project directory.

To run PC-BACK, follow the steps below.

1. Type:

PCBACK [Return]

When the PC-BACK Title Screen appears, press any key to continue.

The system prompts for the input swap report filename as shown in Figure 6-2.

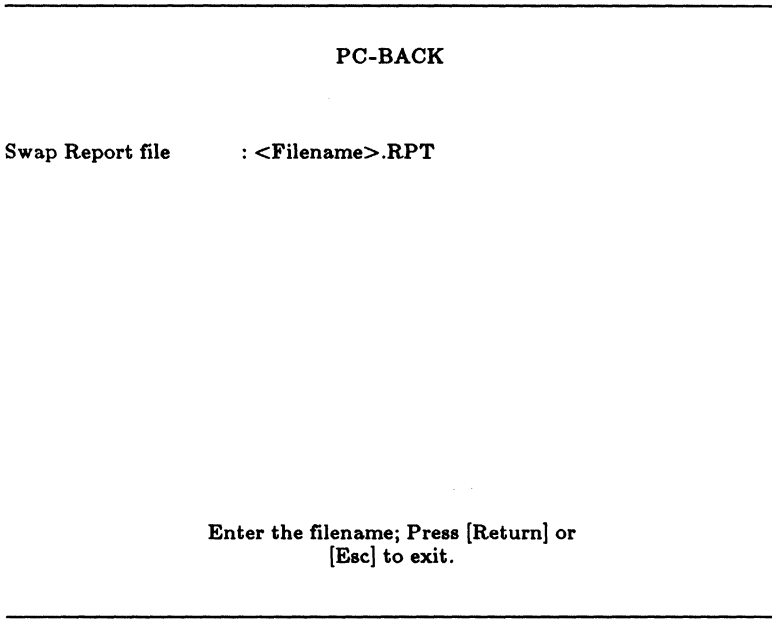


Figure 6-2. PC-BACK Program Screen

2. Type the swap report filename and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter the filename extension, PC-BACK enters the default .RPT extension.

The system prompts for the input packaging command filename. The default is the swap report filename with a .CMD extension.

3. Press [Return] to accept the default, or type the correct filename and press [Return].

The system prompts for the output "command" (back-annotation) filename. The default is the swap report filename with a .BKA extension.

4. Press [Return] to accept the default filename, or type another filename and press [Return].

The system prompts for the output error listing filename. The default is the output back-annotation filename with a .LIS extension. Figure 6-3 shows a sample Program Screen with default filenames used for the output files.

PC-BACK

Swap Report file : TESTFILE.RPT
Initial command file : TESTFILE.CMD
Final command file : TESTFILE.BKA
Error listing file : TESTFILE.LIS

Enter the filename; Press [Return] to
accept; [Esc] to reject.

Figure 6-3. Sample PC-BACK Program Screen

5. Press [Return] to accept the default filename, or type another filename and press [Return].

PC-BACK processes the files.

When processing is complete, the cursor returns to the **Swap report file** prompt. To end the program, press [Esc]. The system returns you to DOS.

VIEWING THE ERROR LISTING FILE

To view the error listing file, first exit from PC-BACK. Then, at the DOS prompt, type the following command:

```
TYPE <filename>.LIS [Return]
```

Read the error listing file. If it notes errors, correct the swap report using a text editor and rerun PC-BACK.

If no errors are noted, you can input the back-annotation file to PC-CAPS.

Appendix A, "Error Messages," describes the error listing file and explains common error messages.

INPUTTING THE BACK-ANNOTATION FILE

To back-annotate the schematic, use the following procedure.

1. Start PC-CAPS and select **Edit database** at the Opening Menu.

The PC-CAPS Command Menu appears.

2. Type:

/

The system prompts:

Menu command:

3. Type:

EXE [Return]

The system prompts:

Exec filename:

4. Type:

<filename>.BKA [Return]

The system loads the appropriate schematic and updates each symbol on the schematic to its physical configuration. If the schematic is a multisheet schematic, each sheet is automatically updated and saved in turn.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that are shown in the error listing file.

The error listing file (<filename>.LIS) gives the commands in the swap report file and notes errors found on any command lines.

Each line is preceded by a three-digit number and a colon. The number is the line number from the swap report file.

At each error, PC-BACK points to the item causing the error, inserts an error message, and then skips to the next command.

Examples of common error messages are given below.

```

Message: 001: SWAP PIN  U1 P1/P2 P3/P4
            002: SWAP GATE U1(1 2 3) U1(4 5 6)
            003: JUNK LINE
                ^
                *** Syntax error: Command expected.
            004:      1/2
            005: RENAME COMPONENT R8 R2
                ^ <-- Source skipped to this point.
            005: RENAME COMPONENT R8 R2
            006: SWAP CNAME U8 U1
  
```

Cause: An invalid command name has been used on line 3 of the swap report. The program skipped to the next command to resume reading.

Action: Edit the swap report and rerun PC-BACK.

Message: 005: RENAME COMPONENT R8 R2
***** R2 already exists. Use SWAP**
CNAME.
006: SWAP CNAME U8 U1
^ <-- Source skipped to this point.
006: SWAP CNAME U8 U1

Cause: The record on line 5 is invalid. The program skipped to the next command to resume reading.

Action: Edit the swap report and rerun PC-BACK.

Message: 001: SWAP PIN U1 P1/P2 P3/P4
***** No gate was packaged into U1.**
002: SWAP GATE U1(1 2 3) U1(4 5 6)
^ <-- Source skipped to this point.
002: SWAP GATE U1(1 2 3) U1(4 5 6)

Cause: The record on line 1 is invalid. The program skipped to the next command.

Action: Edit the swap report and rerun PC-BACK.

Message: 001: SWAP PIN U1 P1/P2 P3/P4
***** Pins P1 and P2 are not of the same**
gate.
002: SWAP GATE U1(1 2 3) U1(4 5 6)

Cause: An invalid pair of pin numbers has been specified on line 1. The program skipped to the next pin pair to resume reading. Pins P3 and P4 were successfully swapped.

Action: Edit the swap report and rerun PC-BACK.

APPENDIX B. EDITING THE SWAP REPORT

Four types of swap records can be entered in the swap report file: Swap Pin records, Swap Gate records, Rename Component records, and Swap CNAME records. (Swap Component commands are not recorded in the swap report.)

Swap Pin records and Swap Gate records are entered into the swap report file automatically by PC-CARDS but can be edited through a text editor.

Rename Component records and Swap CNAME records are not entered automatically; you can enter them into the swap report using a text editor.

The format and function of each type of record are described below. In all records, items are separated by one or more blank spaces and can be entered using uppercase or lowercase letters. The maximum line length is 80 characters. A record can be as long as needed, but an item within a record must be on the same line.

The Swap Pin Record

A Swap Pin record shows pairs of pins whose connections were swapped. The pins must belong to the same component and must be logically equivalent; for example, two input pins on an AND gate can be swapped. Logic equivalence is set using the PC-CARDS ENTER/PIN command.

The format of a Swap Pin record is shown below.

SWAP PIN <component-name> <pin#/pin#>

where:

SWAP PIN is the command.

component-name is the reference designator of the component within which pin connections are swapped.

pin# and **pin#** are the two pin numbers involved in the swap. They are separated by a slash. Any number of pin pairs can be specified for the component. Different pin pairs are separated by one or more blank spaces.

SWAP PIN and the component name must be on the same line. Pin pairs can be on separate lines. However, a pair must not be separated.

A Swap Pin record example is shown below.

```
SWAP PIN U27 12/13
           14/15 4/5
```

In this record, pin number 12 was swapped with pin number 13, pin 14 with 15, and pin 4 with 5, all within a PCB component with the reference designator U27.

The Swap Gate Record

A Swap Gate record shows entire sections, or gates, swapped within or between components. All pin connections of these gates were swapped and must therefore be listed. Each record shows one Swap Gate operation. The format of a Swap Gate record is shown below.

```
SWAP GATE <comp1> (<pin#, . . . , pin#>) <comp2>
(<pin#, . . . , pin#>)
```

where:

SWAP GATE is the command.

comp1 and **comp2** are the reference designators in which the gates or sections occur. **Comp1** may or may not be the same as **comp2**.

pin#, . . . , pin# is the list of pin numbers associated with the gate or section swapped. Both lists must have the same number of entries. The pin numbers must be enclosed in parentheses and separated by blanks or commas.

The reference designator and its associated pin list can be on a separate line from the SWAP/GATE command. For each component, the reference designator and pin list must be on the same line, but the information for the two components can be on two separate lines.

Two Swap Gate record examples are shown below.

```
SWAP GATE U1 (1, 2, 3) U1 (4, 5, 6)
```

In this record, two gates within PCB component U1 were swapped. Specifically, the gate connected to pins 1, 2, and 3 was swapped with the gate connected to pins 4, 5, and 6.

```
SWAP GATE
      U2 ( 1 2 3 )
      U3 ( 1 2 3 )
```

In this record, a gate in PCB component U2 was swapped with a gate in PCB component U3.

The Rename Component Record

A Rename Component record shows the renaming of a PCB component. The new name (reference designator) must be different from any other component name within the PCB database to eliminate the possibility of duplicate component names. Each record renames one component. The format of a rename component record and an example are shown below.

```
RENAME COMPONENT <oldname> <newname>
```

Example:

```
RENAME COMPONENT U8 U1
RENAME COMPONENT R7 R2
```

In this record, PCB component U8 was renamed to U1 and PCB component R7 to R2.

The Swap CNAME Record

A Swap CNAME record shows the swapping of names of two PCB components (reference designators). Both names must refer to existing components. Each record performs one name swap. The format of a Swap CNAME record and an example are shown below.

```
SWAP CNAME <name1> <name2>
```

Example:

```
SWAP CNAME R7 R2
```

In this record, the reference designator of the PCB component R7 is changed to R2 and the component that was R2 is changed to R7.





PC-PRINT™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0050-05



ABOUT THIS MANUAL

This manual describes PC-PRINT, the P-CAD utility program that enables dot matrix printers to produce plots from PC-CAPS, PC-CARDS, or PC-PLACE plot files.

This manual provides an overview of PC-PRINT, operating instructions, and appendixes that show error messages and required switch settings for certain printers.



CONTENTS

OVERVIEW.....	7-1
Input	7-2
Output	7-2
Printer Requirements.....	7-3
USING PC-PRINT.....	7-7
Configuring PC-PRINT.....	7-9
Plotting a File.....	7-14
Outputting to a Printer From Disk.....	7-21
APPENDIX A. ERROR MESSAGES	7-23
APPENDIX B. PRINTER SWITCH SETTINGS ..	7-25

FIGURES

7-1. PC-PRINT Input and Output	7-1
7-2. PC-PRINT Opening Menu.....	7-8
7-3. PC-PRINT Configuration Screen.....	7-10
7-4. Original and Default Text Dimensions.....	7-13
7-5. PC-PRINT Program Screen.....	7-15
7-6. Maximum and Minimum Plot Window Coordinates.....	7-17
7-7. Normal and Rotated Plots.....	7-18
7-8. Sample PC-PRINT Program Screen	7-20
7-9. C.Itoh Printer Switch Settings.....	7-25
7-10. Texas Instruments Printer Switch Settings ..	7-26

TABLES

7-1. PC-PRINT Output Filename Extensions. . . .	7-3
7-2. PC-PRINT Supported Printers	7-4



OVERVIEW

PC-PRINT reads data from a PC-CAPS, PC-CARDS, or PC-PLACE plot file and converts it to printer instructions. Figure 7-1 shows PC-PRINT input and output.

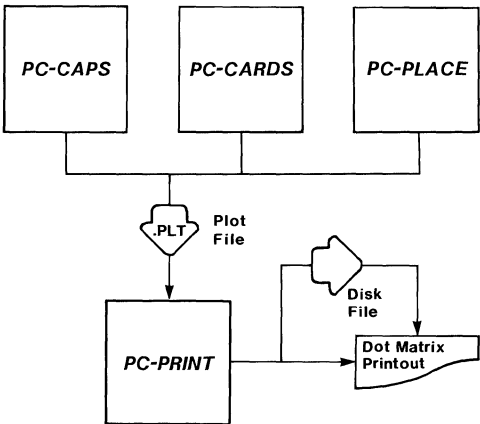


Figure 7-1. PC-PRINT Input and Output

INPUT

Input to PC-PRINT is a plot file generated by PC-CAPS, PC-CARDS, or PC-PLACE.

The plot file is generated from the schematic or PCB database using the SYS/PLOT command in PC-CAPS, PC-CARDS, or PC-PLACE and is usually given the filename extension .PLT. It contains graphic information readable by PC-PRINT.

NOTE: P-CAD provides a sample plot file (NEWTEST.PLT) that you can input to PC-PRINT to verify that the program, your computer, and the printer are working correctly. If you used the P-CAD Autoloader, the NEWTEST.PLT file is located in the PROJ0 directory.

OUTPUT

PC-PRINT outputs printer instructions, which can be sent directly to a printer to produce a dot matrix plot or stored in a disk file for later use.

If instructions are stored in a disk file, PC-PRINT gives this file the same filename as the input plot file and a filename extension based on the make of the plotter you specify. For example, a file for an Epson printer receives a .EPS extension; a file for a Texas Instruments printer receives a .TI extension.

Table 7-1 lists filename extensions and corresponding printers.

Table 7-1. PC-PRINT Output Filename Extensions

Disk File Extension	Printer Type
.CTH	C.Itoh
.EPS	Epson
.IBM	IBM
.OKI	Okidata
.TI	Texas Instruments

PRINTER REQUIREMENTS

PC-PRINT drives 16 printer models. Table 7-2 lists printer models, paper size, and graphics density.

Table 7-2. PC-PRINT Supported Printers

Printer	Paper Size	Graphics Density
C.Itoh 1550*	13"	single/double
C.Itoh 8510A*	8"	single/double
Epson MX-80	8"	single/double
Epson FX-80	8"	single/double
Epson FX-100	13"	single/double
Epson LQ-1500	13"	single/double
Epson FX-185	13"	single/double
Epson FX-85	8"	single/double
IBM 5553	13"	-----
IBM Proprinter	8"	single/double
Okidata ML92**	8"	single/double
Okidata ML93**	13"	single/double
Texas Instruments 865*	13"	single/double
Texas Instruments 850*	8"	single/double
Texas Instruments 855*	8"	single/double
Texas Instruments 860*	13"	single/double
Texas Instruments 865*	13"	single/double

* Requires special switch settings. These settings are shown in Appendix B, "Printer Switch Settings."

** Requires an IBM plug and play module.

Before running PC-PRINT, make sure all hardware is connected and configured according to the instructions for your particular printer.

Make sure there is paper in your print feeder.

For centered plots on Epson printers, align the top of the printer paper with the print head rather than the carriage ruler.



USING PC-PRINT

Before running PC-PRINT, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-PRINT program file (PCPRINT.EXE).
- You have produced the PC-CAPS, PC-CARDS, or PC-PLACE generated plot file.

If you are using the P-CAD recommended directory structure, run PC-PRINT from the appropriate project directory.

To run PC-PRINT, type:

```
PCPRINT [Return]
```

When the PC-PRINT Title Screen appears, press any key to continue. The PC-PRINT Opening Menu is displayed as shown in Figure 7-2.

PC-PRINT

Options:

Configure PC-PRINT

>> Plot a file <<

Exit PC-PRINT

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

Figure 7-2. PC-PRINT Opening Menu

This menu provides the following options:

Configure PC-PRINT - Allows you to examine or change PC-PRINT configuration options.

Plot a file - Allows you to run PC-PRINT. This option is the default when the Opening Menu is displayed.

Exit PC-PRINT - Returns you to DOS.

If you want to change the configuration, use the procedure in the next section, "Configuring PC-PRINT." If you do not want to change the configuration, continue to the section "Plotting a File."

CONFIGURING PC-PRINT

Configure PC-PRINT allows you to examine or change settings for several standard options.

To configure PC-PRINT, follow the steps below.

1. At the PC-PRINT Opening Menu, press the space bar or use the cursor keys to select **Configure PC-PRINT**, then press [Return]. The system displays the PC-PRINT Configuration Screen with the current settings, as shown in Figure 7-3.

You can now accept or change the settings for the configuration options.

PC-PRINT Configuration

Printer port..... LPT1
Output device..... printer

Default printer..... Epson MX-80
Default paper size..... A
Graphics density..... single

Text dimensions (% of text size) :
Text character width (including space)..... 70%
Text character height..... 80%
Space below text line..... 10%
Space above text line..... 10%
Space from top of text to bar..... 20%

Press: [SPACE] for the next port; [RETURN] to accept

Figure 7-3. PC-PRINT Configuration Screen

2. The first option, **Printer port**, allows you to specify the communications port to which the printer is connected. The choices are "LPT1," "LPT2," "LPT3," and "PRN."

Press the space bar to select the port to which the printer is connected and press [Return]. If no printer is connected, just press [Return].

3. The next option, **Output device**, allows you to specify the device to which the program outputs plotting instructions. The choices are "printer" and "disk."

The "printer" setting is the default. It sends plotting instructions directly to the printer.

The "disk" setting causes PC-PRINT to store plotting instructions in a disk file which is later input directly to the printer. This file has the same name as the input plot file and a filename extension based on the printer model. Table 7-1 lists the filename extensions and corresponding printers.

You can use the "disk" setting if no printer is connected to your computer or if you wish to print a file at a later time.

Press [Return] to accept the default setting, or press the space bar to select "disk" and press [Return].

4. The next option, **Default printer**, allows you to select one of the printers listed in Table 7-2, "PC-PRINT Supported Printers."

Press [Return] to accept the default setting or press the space bar to select a different model and press [Return].

5. The next option, **Default paper size**, allows you to specify the size of the paper surface on which you intend to print.

The surface size choices range from 10.5" x 8.0" to 43.5" x 32.0" on 8" printers and from 10.5" x 13" to 43.5" x 52" on 13" printers.

Surface sizes for 8" printers are labeled (from smallest to largest) A, B, C, D, or E; surface sizes for 13" printers are labeled AX, BX, CX, DX, and EX.

If you select a surface size larger than A or AX, the plot will be printed on more than one sheet of paper. The surface sizes, their letter designations, and the number of standard sheets they require for printing are listed below:

For 8" printers:

A = 10.5" x 8.0" Prints on 1 sheet
B = 16.0" x 10.5" Prints on 2 sheets
C = 21.5" x 16.0" Prints on 4 sheets
D = 32.0" x 21.5" Prints on 8 sheets
E = 43.5" x 32.0" Prints on 16 sheets

For 13" printers:

AX= 10.5" x 13.0" Prints on 1 sheet
BX= 13.0" x 21.5" Prints on 2 continuous sheets
CX= 21.5" x 26.0" Prints on 4 sheets
DX= 52.0" x 21.5" Prints on 8 sheets
EX= 43.5" x 52.0" Prints on 16 sheets

Press [Return] to accept the default setting or press the space bar to select another paper size and press [Return].

6. The next option, **Graphics density**, allows you to specify the density used by the printer. The choices are "single" and "double."

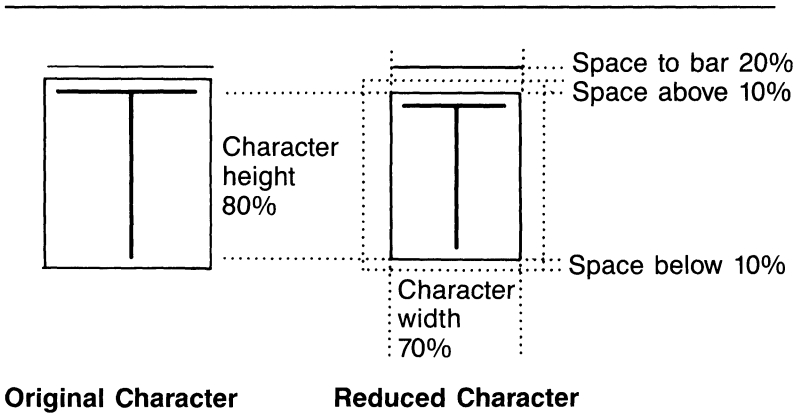
The higher the density, the more dots per inch on a printed page. "Single" (normal) density varies for each printer; "double" is twice normal density.

Press [Return] to accept the default setting or press the space bar to select the other setting and press [Return].

NOTE: The IBM 5553 printer uses only one density.

- The remaining options, which are displayed under the heading **Text dimensions (% of text size)**, allow you to specify the dimensions at which text will be plotted. Choices range from 0 to 1000% of the text size specified in PC-CAPS or PC-CARDS. Default values are provided.

Figure 7-4 shows how the text will be plotted using the default specifications.



86211

Figure 7-4. Original and Default Text Dimensions

For each option, press [Return] to accept the default proportion, or type a new proportion and press [Return].

8. After you change the last option, if you have changed any options, PC-PRINT displays the following prompt:

Save this configuration? No

To save the configuration, type Y for YES and press [Return]. The system saves the modified configuration as a file named PCPRINT.CFG.

To use the configuration for the current session only, press [Return] for NO. Changes to the configuration will be used for this PC-PRINT program session but will not be saved after you exit PC-PRINT.

NOTE: If you modify the P-CAD supplied configuration and later wish to restore it, from DOS, type the following command:

DEL PCPRINT.CFG [Return]

The original configuration will be restored.

When configuration is complete, the system returns you to the Opening Menu.

PLOTTING A FILE

Plot a file allows you to generate the printer instructions and transmit them either directly to a printer or to a print plot disk file.

To plot a file, follow the steps below.

1. At the Opening Menu, select **Plot a file** and press [Return]. The system displays the PC-PRINT Program Screen and prompts for the input filename as shown in Figure 7-5.

PC-PRINT

Plot filename: filename.plt

Enter the filename; Press: [RETURN] to accept

Figure 7-5. PC-PRINT Program Screen

2. Type the filename of the input plot file and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter a filename extension, PC-PRINT enters the default .PLT extension.

The system prompts for the printer model. The default is the model specified in the current configuration.

NOTE: At any of the following prompts you can cancel your entry and return to the previous prompt by pressing [Esc].

3. Press [Return] to accept the default printer, or press the space bar until the correct printer name is displayed, then press [Return].

The system prompts for the paper size. This option specifies the size of the paper surface on which you will be printing.

The choice of surface sizes ranges from 10.5" x 8.0" to 43.5" x 32.0" on 8" printers and from 10.5" x 13" to 43.5" x 52" on 13" printers.

Surface sizes for 8" printers are labeled (from smallest to largest) A, B, C, D, or E; surface sizes for 13" printers are labeled AX, BX, CX, DX, and EX.

If you select a surface size larger than A or AX, the plot will be printed on more than one sheet of paper. The surface sizes, their letter designations, and the number of standard sheets they require for printing are listed below:

For 8" printers:

A = 10.5" x 8.0" Prints on 1 sheet
B = 16.0" x 10.5" Prints on 2 sheets
C = 21.5" x 16.0" Prints on 4 sheets
D = 32.0" x 21.5" Prints on 8 sheets
E = 43.5" x 32.0" Prints on 16 sheets

For 13" printers:

AX= 10.5" x 13.0" Prints on 1 sheet

BX= 13.0" x 21.5" Prints on 2 continuous sheets

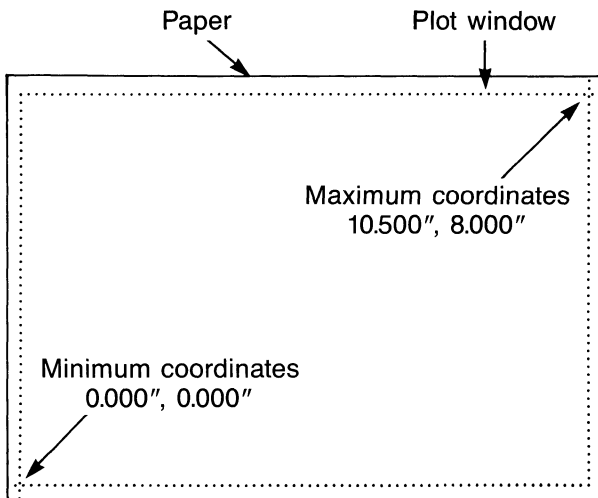
CX= 21.5" x 26.0" Prints on 4 sheets

DX= 52.0" x 21.5" Prints on 8 sheets

EX= 43.5" x 52.0" Prints on 16 sheets

4. Press the space bar to select the paper size you intend to use and press [Return].

The system prompts for the plot window coordinates, which specify the locations of the lower left corner and the upper right corner of the plot area on the paper, as shown in Figure 7-6.



86212

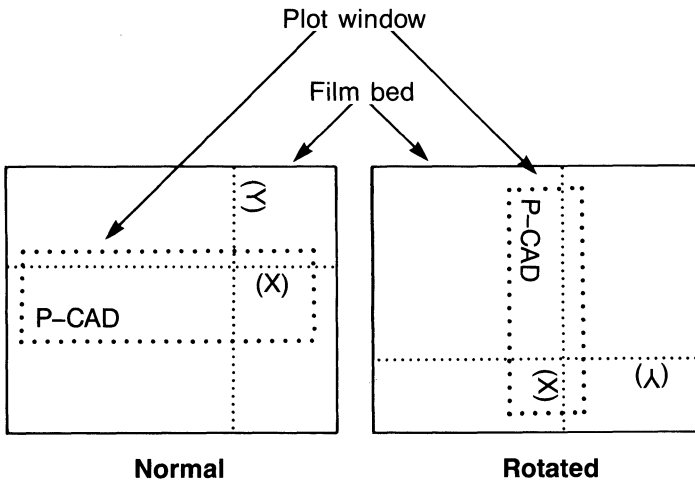
Figure 7-6. Maximum and Minimum Plot Window Coordinates

The default coordinates show the maximum plot area for the paper size and printer selected. The plot will be centered in this area unless you enter new coordinates to specify a smaller plotting area.

5. At the prompt for each coordinate, press [Return] to accept the default, or type a new coordinate and press [Return].

The system prompts for the plot orientation. The settings are "normal" and "rotated."

A normally oriented plot is printed with its horizontal (x) axis parallel to the long side of the printer paper. A rotated plot is printed with its vertical (y) axis parallel to the long side of the paper. Figure 7-7 shows normal and rotated plots.



86213

Figure 7-7. Normal and Rotated Plots

For each plot, the default setting is the orientation that allows the design to be plotted at the largest scale. Thus, a plot window with a longer x than y dimension has a default of "normal," and a plot window with a longer y dimension has a default of "rotated."

6. Press [Return] to accept the default orientation, or press the space bar to select the other setting and press [Return].

The system prompts for the scale factor, which specifies the enlargement or reduction of the design for the plot.

PC-PRINT uses the plot dimensions, plot window size, and orientation to compute the largest scale for the plot and displays this value as the default.

If you want a 1 to 1 plot of your PC-CAPS, PC-CARDS, or PC-PLACE plot file, you should set the scale factor to 1.

NOTE: A DBU is 0.01 inches in PC-CAPS and 0.001 inches in PC-CARDS. Therefore, if you set the scale factor to 1, a line that is 100 DBUs long in PC-CAPS will plot out at 1 inch and a line that is 1000 DBUs long in PC-CARDS will plot out at 1 inch.

NOTE: Any scale other than 1 for a PC-CAPS, PC-CARDS, or PC-PLACE plot file increases or decreases line widths on the plot by the specified amount.

7. Press [Return] to accept the default scale, or type a smaller scale factor and press [Return].

Figure 7-8 shows a sample Program Screen with all values entered.

PC-PRINT

Plot filename: newtest.plt

Plotter: Epson FX-80

Paper size: A

Plot window:

 Minimum x,y: 0.257", 0.257"

 Maximum x,y: 10.743", 8.243"

Plot orientation: Normal

Scale factor: 1.664

Enter the scale factor; Press: [Return] to accept

Figure 7-8. Sample PC-PRINT Program Screen

After you specify the scale factor, PC-PRINT generates print instructions and outputs them to the printer or to disk.

PC-PRINT rasterizes the plot file (generates a page of data). This may take several minutes, depending on the amount of data.

When rasterizing is complete, PC-PRINT sends instructions to the printer or writes instructions to disk.

PC-PRINT displays progress reports at the bottom of the screen.

NOTE: If you want to abort a plot, press [Esc]. The system prompts:

Abort this Plot? NO

Type Y for Yes and press [Return] to abort the plot, or press [Return] to continue plotting.

When PC-PRINT has finished plotting or writing to disk, it returns you to the Opening Menu.

OUTPUTTING TO A PRINTER FROM DISK

If you save your print instructions to a disk file and wish to print from this file, follow the steps below.

1. Connect your printer to LPT1, LPT2, or LPT3 of your computer.
2. Change to the directory where the file is located.
3. Use the DOS PRINT command to print the file.
For example:

PRINT TESTFILE.EPS [Return]

The system prompts:

Name of List Device? PRN

4. Type LPT1, LPT2, or LPT3, depending on which port the printer is connected to. Press [Return].

The printer plots the file.

NOTE: Subsequent PRINT commands will send data to the device designated here until the system is rebooted.

APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that may appear on the screen when you run PC-PRINT. These errors must be corrected before PC-PRINT can generate printing instructions.

Message: ** PCADDRV.SYS file not found **

Cause: Your P-CAD software is not installed correctly on your computer system.

Action: Use the P-CAD INSTALL program, which creates the required file.

Message: Error writing file <filename>

Error writing file PCPRINT.CFG

Failed to fire pins

Failed to enter graphic mode

Failed to write to the CMD file

Cause: Your disk is full.

Action: Delete files from your disk to create space.



APPENDIX B. PRINTER SWITCH SETTINGS

Figures 7-9 and 7-10 show the switch settings required by C.Itoh and Texas Instruments printers to correctly process printing instructions from PC-PRINT.

	8	7	6	5	4	3	2	1
SW1	C						C	
	O		O	O	O	O		O

O = OPEN = OFF
C = CLOSE = ON

SWITCHES	FUNCTION
1,2,3,4	National character set (US)
5	Processing of DC1 and DC3
6	No line feed when DATA BUFFER is full
7	CR, LF, VT, FF, US
8	Line feed by CR command code (No)

	8	7	6	5	4	3	2	1
SW2	C							
	O		O	O	O	O	O	O

O = OPEN = OFF
C = CLOSE = ON

SWITCHES	FUNCTION
1	Printing of numeral 0(0)
2	Select buffer option. No buffer
3	Length between (11 inches)
4	Line spacing (1/16 inch)
5	Power on character pitch (10 CPI)
6	Select of 7 or 8 data (8-bit)
7	Power on select/dselect (select)
8	Uni- or bi-direction print (bi-direct)

Figure 7-9. C.Itoh Printer Switch Settings

		1	SWITCHES	FUNCTION
	C		1,2	8 data bits, no parity
O			3	No automatic line feed on [Return]
	C		4	Can be either ON or OFF
	C		5	Will be set to DATA processing within the program
	C		6	
	C		7	Parallel
	C		8	

O = OPEN = OFF
C = CLOSE = ON

86215

Figure 7-10. Texas Instruments Printer Switch Settings

PC-PLOTS™

User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.



ABOUT THIS MANUAL

This manual describes PC-PLOTS, the P-CAD utility program that enables plotters to produce pen plots from PC-CAPS, PC-CARDS, or PC-PLACE plot files.

This manual provides an overview of PC-PLOTS, operating instructions, and appendixes that show error messages and plotter specifications.



CONTENTS

OVERVIEW.....	8-1
Input	8-2
Output.....	8-2
Plotter Requirements.....	8-3
USING PC-PLOTS.....	8-5
Configuring PC-PLOTS.....	8-7
Plotting a File.....	8-15
Outputting to a Plotter From Disk.....	8-23
APPENDIX A. ERROR MESSAGES.....	8-27
APPENDIX B. COMPUTER TO PLOTTER INTERFACE SPECIFICATIONS.....	8-29
CalComp Plotters.....	8-29
Hewlett-Packard and IBM 7371, 7372, 7374, and 7375 Plotters.....	8-34
Seiko Instruments Plotter.....	8-39
Houston Instruments Plotters.....	8-40
IBM XY/749 Plotter.....	8-42
Nicolet Zeta Plotters.....	8-44
Mutoh Plotters.....	8-47
Graphtec Plotters.....	8-49
APPENDIX C. PLOTTER PEN VELOCITY, ACCELERATION, AND FORCE CAPACITIES.....	8-51
APPENDIX D. PLOT WINDOW DIMENSIONS.....	8-53
APPENDIX E. HOUSTON INSTRUMENTS DMP PLOTTER TEXT LIMITATIONS.....	8-59

Contents (Continued)

APPENDIX F. SETTING PEN VELOCITY/ ACCELERATION ON DMP-51, 52, 56, 51MP, AND 52MP PLOTTERS	8-61
---	------

FIGURES

8-1. PC-PLOTS Input and Output	8-1
8-2. PC-PLOTS Opening Menu	8-6
8-3. PC-PLOTS Configuration Screen	8-8
8-4. Original and Default Text Dimensions. . .	8-14
8-5. PC-PLOTS Program Screen	8-16
8-6. Minimum and Maximum Plot Window Coordinates	8-18
8-7. Normal and Rotated Plots	8-20
8-8. Sample PC-PLOTS Program Screen	8-22
8-9. Hewlett-Packard 7470A & 7475A Switch Settings	8-36
8-10. Hewlett-Packard 7580B & 7585B Switch Settings	8-37
8-11. Hewlett-Packard 7580A Switch Settings. .	8-38
8-12. DMP-29 Switch Settings	8-41
8-13. IBM XY/749 Switch Settings	8-43
8-14. Nicolet Zeta 8 & 822 Switch Settings . . .	8-45
8-15. Nicolet Zeta 824 & 836 Switch Settings . .	8-46
8-16. Mutoh Switch Settings	8-48
8-17. Graphtec MP-2000 Switch Settings	8-50
8-18. Sample Configuration Sheet for the DMP-51, 52, 51MP, and 52MP	8-65
8-19. Sample Configuration Sheet for DMP-56	8-66

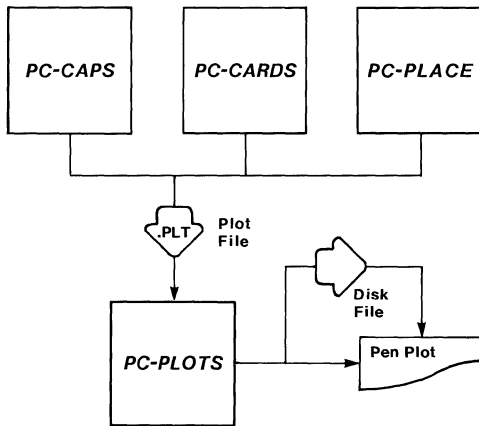
TABLES

8-1.	PC-PLOTS Output Filename Extensions. . .	8-3
8-2.	CalComp Cable Configuration.	8-30
8-3.	Control Panel Settings for CalComp 1043	8-31
8-4.	Hewlett-Packard Cable Configuration . . .	8-35
8-5.	DSCAN GP-5400 Cable Configuration. . .	8-39
8-6.	Houston Instruments Cable Configuration.	8-40
8-7.	IBM Cable Configuration.	8-42
8-8.	Nicolet Zeta Cable Configuration.	8-44
8-9.	Mutoh Cable Configuration.	8-47
8-10.	Graphtec MP-2000 Cable Configuration. . .	8-49
8-11.	Pen Velocity, Acceleration, and Force . . .	3-51
8-12.	Plot Window Dimensions (U.S. Paper Sizes)	8-53
8-13.	Plot Window Dimensions (Metric Paper Sizes)	8-56
8-14.	Recommended DMP-51, 52, 51MP, & 52MP Pen Velocity Settings.	8-62
8-15.	Recommended DMP-56 Pen Velocity Settings	8-63



OVERVIEW

PC-PLOTS reads data from a PC-CAPS, PC-CARDS, or PC-PLACE plot file and converts it to plotter instructions. Figure 8-1 shows PC-PLOTS input and output.



86208

Figure 8-1. PC-PLOTS Input and Output

INPUT

PC-PLOTS input is a plot file generated by PC-CAPS, PC-CARDS, or PC-PLACE.

The plot file is generated from the schematic or PCB database using the SYS/PLOT command in PC-CAPS, PC-CARDS, or PC-PLACE and is usually given the filename extension .PLT. It contains graphic information readable by PC-PLOTS.

NOTE: P-CAD provides a sample plot file (NEWTEST.PLT) that you can input to PC-PLOTS to verify that the program, your computer, and the pen plotter are all working correctly. If you used the P-CAD Autoloader, the NEWTEST.PLT file is located in the PROJ0 directory.

OUTPUT

PC-PLOTS outputs plotter instructions, which can be output directly to a plotter to produce a pen plot or stored in a disk file for later use.

If instructions are stored in a disk file, PC-PLOTS gives this file the same filename as the input plot file and a filename extension based on the make of the plotter you specify. For example, a disk file for a CalComp plotter receives a .CC extension; a file for a Hewlett-Packard plotter receives a .HP extension.

Table 8-1 lists filename extensions and corresponding plotters.

Table 8-1. PC-PLOTS Output Filename Extensions

Disk File Extension	Plotter Type
.CC	CalComp
.HP	Hewlett-Packard
.DMP	Houston Instruments DMP
.IBM	IBM
.NZ	Nicolet Zeta
.DSN	Seiko Instruments DSCAN
.MTH	Mutoh
.GTK	Graphtec

PLOTTER REQUIREMENTS

PC-PLOTS drives the following pen plotters:

- CalComp 945, 960, 965, 1043, 1073, 1075, 1077, and Calcomp PCI Controller
- Hewlett-Packard 7470, 7475, 7580, and 7585
- Houston Instruments DMP-29, DMP-40, DMP-41, DMP-42, DMP-51, DMP-52, DMP-56, DMP-51MP, and DMP-52MP
- IBM XY/749, 7371, 7372, 7374, and 7375

- Nicolet Zeta 8, 822, 824, and 836
- Seiko Instruments DSCAN GP-5400
- Mutoh IP-100 A2 and A3
- Graphtec MP-1000 and MP-2000

Before running PC-PLOTS, make sure all hardware is connected and configured according to instructions for your particular plotter. Cable and switch specifications are included in Appendix B, "Computer to Plotter Interface Specifications." In addition, make sure paper and pens are properly inserted in holders.

USING PC-PLOTS

Before running PC-PLOTS, make sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements. Note that PC-PLOTS requires only 256K of RAM.)
- You have installed the PC-PLOTS program file (PCPLOTS.EXE).
- You have produced the PC-CAPS, PC-CARDS, or PC-PLACE generated plot file.

If you are using the P-CAD recommended directory structure, run PC-PLOTS from the appropriate project directory.

To run PC-PLOTS, type:

```
PCPLOTS [Return]
```

When the PC-PLOTS Title Screen appears, press any key to continue. The Opening Menu is displayed as shown in Figure 8-2.

PC-PLOTS

Options:

Configure PC-PLOTS

>> **Plot a file** <<

Exit PC-PLOTS

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

Figure 8-2. PC-PLOTS Opening Menu

This menu provides the following options:

Configure PC-PLOTS - Allows you to examine or change PC-PLOTS configuration options.

Plot a file - Allows you to run PC-PLOTS. This option is the default when the Opening Menu is displayed.

Exit PC-PLOTS - Returns you to DOS.

If you want to change the configuration, use the procedure in the next section, "Configuring PC-PLOTS." If you do not want to change the configuration, continue to the section "Plotting a File."

CONFIGURING PC-PLOTS

Configure PC-PLOTS allows you to examine or change settings for several standard options.

To configure PC-PLOTS, follow the steps below.

1. At the PC-PLOTS Opening Menu, press the space bar or use the cursor keys to select **Configure PC-PLOTS**, then press [Return]. The system displays the PC-PLOTS Configuration Screen with the current settings, as shown in Figure 8-3.

You can now accept or change the settings for the configuration options.

PC-PLOTS Configuration

Plotter port	PORT1
Output device	plotter
Output record length	80
Default plotter	CalComp 945
Paper Size Setting	US
Default paper size	A
Pen width	0.010 "
Pen velocity	---%
Pen acceleration	---%
Pen force	---%
Text dimensions (% of text size):	
Text character width (including space)	70 %
Text character height	80 %
Space below text line	10 %
Space above text line	10 %
Space from top of text to bar	20 %

Press: [SPACE] for the next port; [RETURN] to accept

Figure 8-3. PC-PLOTS Configuration Screen

2. The first option, **Plotter port**, allows you to specify the communications port to which the plotter is connected. The choices are "PORT1" and "PORT2." The default is "PORT1."

To select "PORT1," press [Return]. To select "PORT2," press the space bar to display the "PORT2" setting, then press [Return]. If no plotter is connected, just press [Return].

3. The next option, **Output device**, allows you to specify the device to which the program outputs plotting instructions. The choices are "plotter" and "disk."

The "plotter" setting is the default. It sends plotting instructions directly to the pen plotter.

The "disk" setting causes PC-PLOTS to store plotting instructions in a disk file which is later input directly to the plotter. This file has the same name as the input plot file and a filename extension based on the plotter model. Table 8-1 lists the filename extensions and corresponding plotters.

You can use the "disk" setting if no plotter is connected to your computer or if you wish to plot a file at a later time.

Press [Return] to accept the default setting or press the space bar to select "disk" and press [Return].

4. The next option, **Output record length**, allows you to specify the record length of the output disk file. The default record length of the output disk file is 80. Do not change it unless your plotter instructions specifically tell you to do so.

This option applies only if you selected "disk" as your output device. If you are outputting directly to a pen plotter, press [Return] to ignore this option.

Press [Return] to accept the default setting, or type a new record length and press [Return].

5. The next option, **Default plotter**, allows you to select one of the plotters listed in the "Plotter Requirements" section. The default plotter is "Calcomp 945."

Press [Return] to accept the default setting, or press the space bar to select a different plotter and press [Return].

6. The next option, **Paper Size Setting**, allows you to specify that the maximum and minimum plot coordinates be given in inches or millimeters. The choices are "US" or "METric."

The "US" option is the default. If you select "US," the plot window coordinates will be given in inches and the paper sizes will be A to E.

If you select "METric," the plot window coordinates will be given in millimeters and the paper sizes will be A4 to A0.

NOTE: Both options will work only if the plotter you are using supports both options. Otherwise, you will get the plot window coordinates in millimeters or inches according to the plotter type.

Press [Return] to accept the default option or press the space bar to select "METric" and press [Return].

7. The next option, **Default paper size**, allows you to specify the size of the paper on which you intend to plot.

Plotters use five paper sizes labeled A, B, C, D, and E. These letters correspond to the dimensions listed below.

A = 11" x 8.5"	D = 34" x 22"
B = 17" x 11"	E = 44" x 34"
C = 22" x 17"	

Press [Return] to accept the default or press the space bar to select a paper size supportable by your plotter and press [Return].

8. The next option, **Pen width**, allows you to specify the pen width to be used on the plotter.

All plot file lines are plotted at least as wide as the selected pen regardless of the line width specified in PC-CAPS or PC-CARDS.

Press [Return] to accept the default or type the pen width that matches the width of the pens currently in the plotter and press [Return].

9. The next options, **Pen velocity, acceleration, and force**, allow you to specify percentages of the maximum velocity, acceleration, and speed for your pens to plot at.

Maximum and minimum pen velocity, acceleration, and force vary from one plotter to another. Consult Appendix C, "Plotter Pen Velocity, Acceleration, and Force Capacities," for specifications applying to your plotter.

To use the plotter default, press [Return] to set pen velocity, force, or acceleration to 0. Three dashes (---) are displayed and the plotter defaults will be used.

To set the pen velocity, acceleration, or force to a percentage of the maximum, type the desired percentage (up to 100) and press [Return].

If you type a value less than the minimum, the system uses the minimum. Refer to your plotter manual for defaults and ranges of these parameters.

NOTE: Pen velocity and acceleration for the DSCAN GP-5400 plotter; the CalComp 960 and 1043 plotters; and the DMP 51, 52, 56, 51MP, and 52MP plotters must be set on the plotters. When configuring PC-PLOTS, use the zero settings only. CalComp plotters adjust pen velocity according to the type of pen inserted. The DSCAN GP-5400 and DMP plotters adjust pen velocity according to input from their plotter control panel. Refer to DSCAN documentation for instructions on setting DSCAN pen velocity. Refer to Appendix F, "Setting Pen Velocity/ Acceleration on DMP 51, 52, 56, 51MP, and 52MP Plotters," for instructions on setting DMP pen velocity.

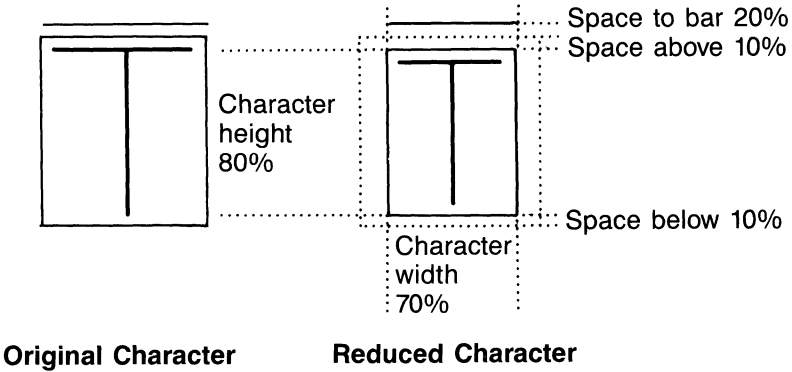
If you selected "Calcomp PCI Controller" for the default plotter option, the "Pen velocity" and "Pen acceleration" options are replaced by "RADIX" and "CHECKSUMMING" options.

The RADIX option allows you to specify the radix value. Radix is the maximum value of the character that can be transmitted by the Calcomp PCI Controller. The default is "64." Press [Return] to accept the default or type a different setting and press [Return].

The CHECKSUMMING option allows you to specify whether or not an error message is displayed if the plotter does not correctly receive the data from your plot file. The options are "YES" or "NO." If you select "YES" and the plotter does not correctly receive the data from your plot file, the system will display an error message that indicates there is a checksumming error. If you select "NO" for this option and the plotter does not correctly receive the data, no error message will be displayed. The default is "NO." Press [Return] to accept "NO" or press the space bar to select "YES" and press [Return].

10. The remaining options which are displayed under the heading **Text dimensions (% of text size)**, allow you to specify the dimensions at which text will be plotted. Choices range from 0 to 1000% of the text size specified in PC-CAPS or PC-CARDS. Default values are provided.

Figure 8-4 shows how the text will be plotted using the default specifications.



86211

Figure 8-4. Original and Default Text Dimensions

For each option, press [Return] to accept the default proportion, or type a new proportion and press [Return].

NOTE: If you are plotting with a Houston Instruments DMP plotter, refer to Appendix E, "Houston Instruments DMP Plotter Text Limitations," for a list of restrictions.

11. After you change the last option, if you have changed any options, PC-PLOTS displays the following prompt:

Save this configuration? No

To save the configuration, type Y for YES and press [Return]. The system saves the modified configuration as a file named PCPLOTS.CFG.

To use the configuration for the current session only, press [Return] for NO. Changes to the configuration will be used for this PC-PLOTS program session but will not be saved after you exit PC-PLOTS.

NOTE: If you modify the original P-CAD supplied configuration and later wish to restore it, from DOS, type the following command:

DEL PCPLOTS.CFG [Return]

The original configuration will be restored.

When the configuration is complete, the system returns you to the Opening Menu.

PLOTTING A FILE

Plot a file allows you to generate the pen plotter instructions and transmit them directly to a pen plotter or to a pen plot disk file.

To plot a file, follow the steps below.

1. At the Opening Menu, select **Plot a file** and press [Return]. The system displays the PC-PLOTS Program Screen, and prompts for the input filename as shown in Figure 8-5.

PC-PLOTS

Plot filename: filename.plt

Enter the filename; Press: [RETURN] to accept

Figure 8-5. PC-PLOTS Program Screen

2. Type the filename of the input plot file and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter a filename extension, PC-PLOTS enters the default .PLT extension.

The system prompts for the plotter name. The default is the model specified in the current configuration.

NOTE: At any of the following prompts, you can cancel your entry and return to the previous prompt by pressing [Esc].

3. Press [Return] to accept the default plotter or press the space bar until the correct plotter name is displayed, then press [Return].

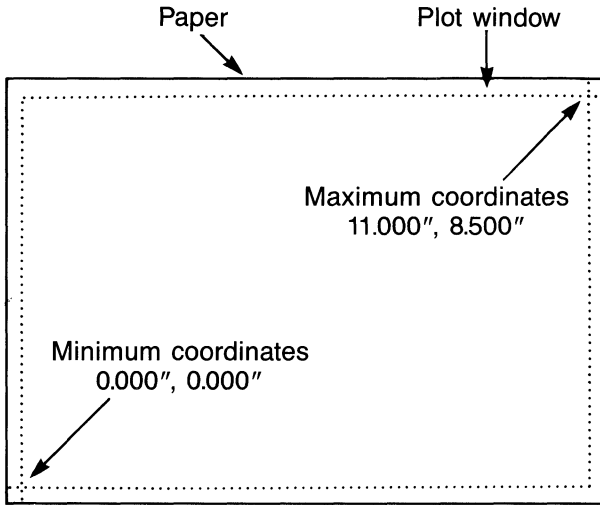
The system prompts for the paper size.

Plotters use five U.S. standard sizes of paper labeled A, B, C, D, and E. These letters correspond to the dimensions listed below:

A = 11" x 8.5"	D = 34" x 22"
B = 17" x 11"	E = 44" x 34"
C = 22" x 17"	

4. Press the space bar to select a paper size supported by your plotter and press [Return].

The system prompts for the plot window coordinates, which specify the locations of the lower left corner and upper right corner of the plot window on the paper, as shown in Figure 8-6.



86216

Figure 8-6. Minimum and Maximum Plot Window Coordinates

The default coordinates show the maximum plot window for the paper size and plotter selected. The plot will be centered in this area unless you enter new coordinates to specify a smaller plotting area.

If you are using cut sheet paper with the CalComp plotters, you must specify the following values for the plot window coordinates.

Paper Size	Window Coordinates
A	Minimum x,y = 0.000",0.000" Maximum x,y = 9.500",7.000"
B	Minimum x,y = 0.000",0.000" Maximum x,y = 15.500",9.500"
C	Minimum x,y = 0.000",0.000" Maximum x,y = 20.500",15.500"
D	Minimum x,y = 0.000",0.000" Maximum x,y = 32.500",20.500"
E	Minimum x,y = 0.000",0.000"

- At the prompt for each coordinate, press [Return] to accept the default, or type a new coordinate and press [Return].

The system prompts for the plot orientation. The settings are "normal" and "rotated."

A normally oriented plot is printed with its horizontal (x) axis parallel to the long side of the plotter paper. A rotated plot is printed with its vertical (y) axis parallel to the long side of the paper. Figure 8-7 shows normal and rotated plots.

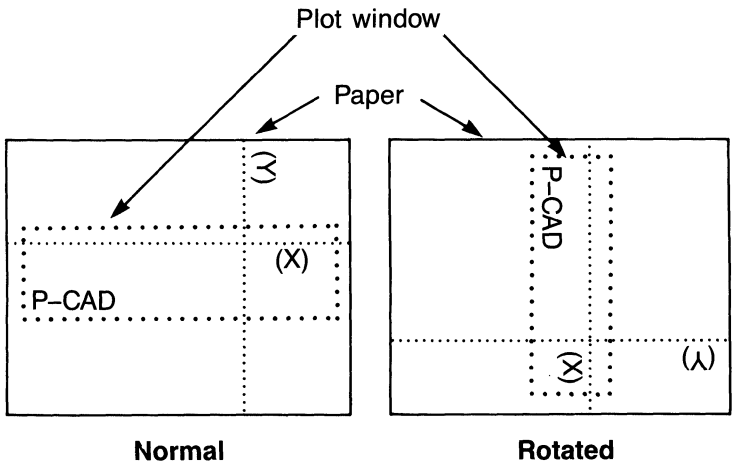


Figure 8-7. Normal and Rotated Plots

86213

For each plot, the default setting is the orientation that allows the design to be plotted at the largest scale. Thus, a plot window with a longer x than y dimension has a default of "normal" and a plot window with a longer y dimension has a default of "rotated."

6. Press [Return] to accept the default orientation, or press the space bar to select the other setting and press [Return].

The system prompts for the scale factor, which specifies the enlargement or reduction of the design for the plot.

PC-PLOTS uses the plot dimensions, plot window size, and plot orientation to compute the largest scale for the plot and displays this value as the default.

If you want a 1 to 1 plot of your PC-CAPS, PC-CARDS, or PC-PLACE plot file, you should set the scale factor to 1.

NOTE: A DBU is 0.01 inches in PC-CAPS and 0.001 inches in PC-CARDS. Therefore, if you set the scale factor to 1, a line that is 100 DBUs long in PC-CAPS will plot out at 1 inch and a line that is 1000 DBUs long in PC-CARDS will plot out at 1 inch.

NOTE: Any scale other than 1 for a PC-CAPS, PC-CARDS, or PC-PLACE plot file increases or decreases line widths and text size on the pen plot by the specified amount.

7. Press [Return] to accept the default scale, or type a smaller scale factor and press [Return].

If you use a smaller scale, the plot will be within the plot window specified.

Figure 8-8 shows a sample Program Screen with all values entered.

PC-PLOTS

Plot filename: newtest.plt

Plotter: Calcomp 945

Paper size: A

Plot window:

Minimum x,y: 0.000", 0.000"

Maximum x,y: 11.000", 8.500"

Plot orientation: Normal

Scale factor: 2.316

Enter the scale factor; Press: [Return] to accept

Figure 8-8. Sample PC-PLOTS Program Screen

After you specify the scale factor, PC-PLOTS generates pen plot instructions and outputs them to the plotter or to disk.

PC-PLOTS displays progress reports and error messages, if any, at the bottom of the screen.

The plotter may pause before or during plotting and prompt you for a pen change, for example:

Put pen 1 & 2 in pen holder slot 1 & 2.

Press: [SPACE] when ready

If the plotter pauses, put the plotter pens in their respective slots. Press the space bar to continue.

NOTE: If you want to abort a plot, press [Esc]. The system prompts:

Abort This Plot? No

Type Y for Yes and press [Return] to abort the plot, or press [Return] to continue plotting.

When PC-PLOTS has finished plotting or writing to disk, it returns you to the Opening Menu.

OUTPUTTING TO A PLOTTER FROM DISK

Outputting to a plotter from disk allows you to continue to work at your computer while plotting is taking place.

If you save your plot instructions to a disk file and wish to plot from this file, follow the steps below.

1. Connect your plotter to Communications Port 1 or Communications Port 2 of your computer.
2. Change to the directory where the file is located.
3. At the DOS prompt, type:

```
MODE COM<n>:9600,N,8,2,P
```

where:

<n> is the number of the RS-232 comport connected to the plotter. If your computer is connected to the plotter through PORT1, enter 1 for n. If your computer is connected to the

plotter through PORT2, enter 2 for n.
9600 is the baud rate.

N is no parity.

8 is the databits.

2 is the stopbits. (If you are using a CalComp
960 plotter, enter 1 instead of 2.)

P indicates that the asynchronous adapter is
being used for a serial interface printer. If you
enter the P, time-out errors are continuously
retried. You can stop the retry loop by pressing
[Ctrl]-[Z]. To stop the time-out errors from being
continuously retried when you have entered P,
you must reinitialize the asynchronous adapter
without entering the P.

4. Use the DOS PRINT command to plot the file. For
example:

```
PRINT TESTFILE.HP [Return]
```

The system prompts:

```
Name of List Device? PRN
```

5. Type COM1 if the plotter is connected to
Communications Port 1, or type COM2 if the plotter
is connected to Communications Port 2. Press
[Return].

The plotter begins to plot.

NOTE: Subsequent PRINT commands will send data to the device designated here until the system is rebooted.

NOTE: If the plotter fails to plot the file, start again at Step 3 above and change the baud rate from 9600 to 4800 or 2400, then try plotting again.



APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that may appear on the screen when you run PC-PLOTS. These errors must be corrected before PC-PLOTS can generate plotting instructions.

Message: ** PCADDRV.SYS file not found **

Cause: Your P-CAD software is not installed correctly on your computer system.

Action: Use the P-CAD INSTALL program, which creates the required file.

Message: Plotter is offline.

Cause: The plotter is not connected to the port you specified on the Configuration Screen.

Action: Make sure the plotter is connected to the appropriate port.

Message: Port<n> not found.

Cause: The port you specified on the Configuration Screen does not exist in your system.

Action: Select another port on the Configuration Screen.



APPENDIX B. COMPUTER TO PLOTTER INTERFACE SPECIFICATIONS

PC-PLOTS and the IBM PC communications port interface require the following signals for plotter communication:

- Pin 1: AC Ground
- Pin 2: Transmit
- Pin 3: Receive
- Pin 5: Clear To Send
- Pin 6: Data Set Ready
- Pin 7: Signal Ground

PC-PLOTS checks pin 5, Clear to Send (CTS), for buffer status (High = buffer not full, Low = buffer full).

All communication between PC-PLOTS and the plotter is done through hardware handshaking. The transmitted data format is 8 data bits, 2 stop bits, no parity. For the CalComp 960, the transmitted data format is 8 data bits, 1 stop bit, no parity.

CALCOMP PLOTTERS

CalComp plotters should be installed only with the advice of your CalComp dealer or service representative.

In CalComp 1073, 1075, and 1077 models, the "plotter control board" must be wired "host" to "9600 baud."

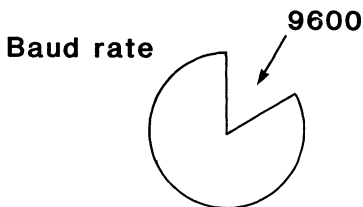
The 1073, 1075, and 1077 models must be installed to communicate directly through the "plotter control board," not through a "PCI interface" board.

Check the configuration of the interface lines on the plotter communication interface board.

If the I/O signals are installed so that the communication is direct control CTS (Local/Host), use the straight through cable configuration for Calcomp plotters shown in Table 8-2.

Table 8-2. CalComp Cable Configuration

Computer			Plotter	
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	2	Trans
Recv	3	<-----	3	Recv
CTS	5	<-----	5	CTS
DSR	6	<-----	6	DSR
DCGnd	7	-----	7	DCGnd
DTR	20	<-----	20	DTR



If the I/O signals are installed so that the communication is direct control DTR (Modem/Remote), use the cable configuration shown in Table 8-4 for the Calcomp plotters as well as the Hewlett-Packard plotters.

Table 8-3 shows required settings input from the control panel of the CalComp 1043 plotter prior to the operation of PC-PLOTS.

Table 8-3. Control Panel Settings for CalComp 1043

Key	Parameter	Setting
0	PARITY	NO
	BITS/CHAR	8
1	STOP BITS	2
	CLOCK	INT
2	INTERFACE	SERIAL ASYNC
	LISTEN ADDRESS	--
3	HOST BAUD RATE	9600
	MODE	960
4	TERMINAL MUTING	YES OR NO
	CHECKSUM ENABLE	NO
5	ISOCRONOUS	NO
	END OF MESS CHAR	--
6	DIRECT CONTROL	YES
	XON/XOFF	NO
7	TERMINAL BAUD RATE	9600
	DUPLEX	FULL
8	SYNC CODES	1
	SYNC CODE VALUE	--

The following list describes the required settings input from the control panel of the CalComp PCI Controller.

- The plotter mode should be set 907 (906) or PCI mode.
- The sync characters should be set to 02 (hex).
- The number of sync characters should be set to 2.
- The end of message character should be set to 03 (hex).
- The bias character should be set to 20 (hex).
- The radix value should be set on the PC-PLOTS Configuration Screen.
- The checksumming option should be set on the control panel and on the PC-PLOTS Configuration Screen.

When you start PC-PLOTS, header information is sent to the CalComp PCI Controller plotter at the beginning of the plot. This information sets the plotter to:

- Two sync chars with values = HEX 02.
- Bias char with the value = HEX 20.
- The RADIX value you selected. The default is 64.
- "GOOD EOT" escape sequence = ASCII '0.'
- "BAD EOT" escape sequence = ASCII '1.'
- Trigger = [Return] and linefeed (OD OA).

- Enable double buffering.
- Turnaround time = 0 milliseconds.
- Select pen number 1.
- Checksum char (if enabled). If checksumming is disabled, then this char will not be sent.
- Pen up.
- EOT (end of message char) = HEX 03.

HEWLETT-PACKARD AND IBM 7371, 7372, 7374, AND 7375 PLOTTERS

This section shows specifications for Hewlett-Packard plotters.

The cable configuration for the IBM 7371, 7372, 7374, and 7375 is the same as the Hewlett-Packard cable configuration.

The switch settings for the IBM 7371 and 7372 are the same as the switch settings for the Hewlett-Packard 7470A and 7475A.

The switch settings for the IBM 7374 and 7375 are the same as the switch settings for the Hewlett-Packard 7580B and 7585B.

Table 8-4 shows the cable configuration and baud rate for Hewlett-Packard plotters.

Plug the cable into the COMPUTER/MODEM port on HP 7580B and 7585B plotters.

Table 8-4. Hewlett-Packard Cable Configuration

Computer			Plotter	
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

Baud rate: 9600

Figures 8-9, 8-10, and 8-11 show the switch settings for HP 7470A & 7475A; HP 7580B & 7585B; and HP7580A plotters.

	0	1	
Parity	X		S2
	X		S1
D	X		Y
Met		X	US
A4		X	A3
Baud		X	B4
	X		B3
		X	B2
	X		B1

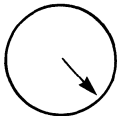
86217

Figure 8-9. Hewlett-Packard 7470A & 7475A Switch Settings. Switches are located in rear of plotter; X's represent switch positions.

Interface mode					
		Expand	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Normal
		Emulate	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Normal
HP-1B	<input type="checkbox"/>	Stand alone	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Eavesdrop
RS-232-C	<input checked="" type="checkbox"/>	Monitor node	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Normal
		Local	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Normal

HP-1B switches: Ignore

RS-232-C switches:

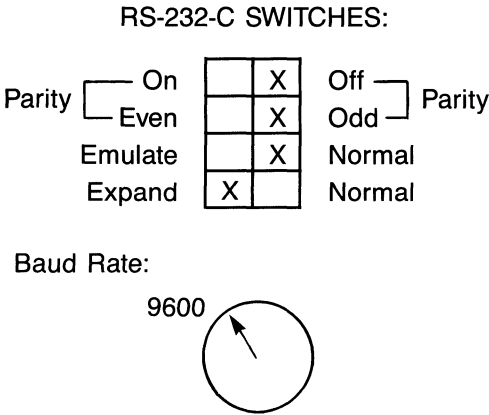
Parity	<input type="checkbox"/>	On	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Off	 9600
	<input type="checkbox"/>	Even	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Odd	
	<input type="checkbox"/>	Duplex-half	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Full	
	<input type="checkbox"/>	Hard-wire	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Modem	
	<input type="checkbox"/>	DTR-bypass	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Normal	

86218

Figure 8-10. Hewlett-Packard 7580B & 7585B Switch Settings. Switches are located in rear of plotter; X's represent switch positions; ignore the HP-1B port switches.

You can expand the plotting window for the 7580B and 7585B models by setting the expand/normal switch to the expand position.

In this configuration, the pen is allowed to plot to within 29 mm of the upper paper edge and to within 5 mm of the other three edges.



86219

Figure 8-11. Hewlett-Packard 7580A Switch Settings. Switches are located in rear of plotter; X's represent switch positions.

SEIKO INSTRUMENTS PLOTTER

Table 8-5 shows the cable configuration for the DSCAN GP-5400 plotter.

Table 8-5. DSCAN GP-5400 Cable Configuration

Computer		Plotter		
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

To be supported by PC-PLOTS, the DSCAN GP-5400 requires that a CalComp 960 emulator be installed.

Prior to operation of PC-PLOTS, the plotter requires CalComp 960 format and the following settings input from its plotter control panel:

Baud rate: 4800
 Bits/char: 8
 Stop bits: 2

HOUSTON INSTRUMENTS PLOTTERS

Table 8-6 shows the cable configuration and baud rate for Houston Instruments DMP plotters. Figure 8-12 shows the switch settings for the DMP-29.

Table 8-6. Houston Instruments Cable Configuration

Computer		Plotter		
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

Baud rate: 2400 (default) - DMP-29 and DMP-40 series
 9600 (default) - DMP-50 series

	1	2	3	4	5	6	7	8
OFF	X	X			X	X	X	X
ON			X					

SWITCHES	FUNCTION
1,2,3	Baud rate 001
4	No parity
5	IF = 0
6	SW6
7	NET/ENG
8	SW8

86220

Figure 8-12. DMP-29 Switch Settings. Switches are located in rear of plotter; X's represent switch positions.

IBM XY/749 PLOTTER

Table 8-7 shows the cable configuration and baud rate for the IBM XY/749 plotter.

Table 8-7. IBM Cable Configuration

Computer		Plotter		
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

Baud rate: 9600

Figure 8-13 shows the switch settings for the IBM XY/749 plotter.

ON	X	X	X	X	X	X	X	X	X	X
OFF										
	1	2	3	4	5	6	7	8	9	10

SWITCHES	FUNCTION
1,2,3	Baud rate 9600
4	Parity
5	Even
6	7 bit
7	1 stop
8	ADDR
9	DTR
10	RS-232

86221

Figure 8-13. IBM XY/749 Switch Settings. Switches are located in rear of plotter; X's represent switch positions.

NICOLET ZETA PLOTTERS

Table 8-8 shows the cable configuration and baud rate for Nicolet Zeta plotters.

Table 8-8. Nicolet Zeta Cable Configuration

Computer		Plotter		
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

Baud rate: 9600

Figures 8-14 and 8-15 show the switch settings for the NZ 8, 822, 824, and 836 plotters.

SW1

OFF	X	X	X	X
ON				
	1	2	3	4

SWITCHES	FUNCTION
1	Plot speed slow/fast
2,3	Reserved
4	Print on/off

SW2

OFF	X	X	X		X	X	X	X
ON				X				
	1	2	3	4	5	6	7	8

SWITCHES	FUNCTION
1	Off
2	Off
3	Reserved
4	Online/offline
5	Parity: enabled/disabled
6	Parity: even/odd
7	Modem/hardwired
8	Reserved

SW3

OFF	X	X	X		X	X	X	X
ON				X				
	1	2	3	4	5	6	7	8

SWITCHES	FUNCTION
1	Mode select
2	Mode select GML-RS-232-C
3	Mode select
4	Halt on limit/limit select
5	Virtual/hard limit
6	Single/multiple pen
7	Page size: metric/ASA
8	Pen pressure: nylon/liquid roller or auto-sense

Switch 4A: 7 = 4800 baud
 Switch 4B: 5 = pen pressure (0=minimum, 9=maximum)

Figure 8-14. Nicolet Zeta 8 and 822 Switch Settings. Switches are located on right of plotter; X's represent switch positions.

GML-RS-232-C

SW1

X	X	X	X

All four switches are forward (down) which sets the plotter to mode ZERO (GML mode).

SW2

UP	X	X	X	X	X		X	X
DOWN						X		
	1	2	3	4	5	6	7	8

SWITCHES	FUNCTION
1,2,3	Set the resolution to 0.0025 inch
4	Sets it to non-error correct mode
5	Disable the line turnaround delay
6	Not used
7	Clipping (virtual)
8	Continuous sheet

SW3

UP	X			X	X	X		
DOWN		X	X				X	X
	1	2	3	4	5	6	7	8

SWITCHES	FUNCTION
1,2,3	Set the baud rate to 9600
4,5	Set the parity to none
6,7,8	Set the positioning mode to either no repositioning or free-form repositioning
	If 6, 7, 8 are all down, it will set it to FREE-FORM REPOS
	If 6 is up and 7, 8 are down, it will set it to NO REPOS

86223

Figure 8-15. Nicolet Zeta 824 & 836 Switch Settings. Switches are located on right of plotter; X's represent switch positions.

MUTOH PLOTTERS

Table 8-9 shows the cable configuration for Mutoh IP-100 plotters.

Table 8-9. Mutoh Cable Configuration

Computer			Plotter	
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

Baud rate: 9600

Note that the baud rate should be 9600 when PC-PLOTS is outputting directly to the plotter. However, if you use PC-PLOTS to create a disk file, you must change the baud rate to 4800 to plot the file. Also make sure that the baud rate in your system is set to 4800.

Figure 8-16 shows the switch settings for the Mutoh IP-100 plotters.

	1	2	3	4	5	6	7	8
ON	X	X			X	X	X	X
OFF			X	X				

SWITCHES	FUNCTION
1,2	Stop bits
3	Parity
4	Parity check enable/disable
5	Data length 8 bits
6,7,8	Baud rate 9600

86224

Figure 8-16. Mutoh Switch Settings

Pen velocity and pen force are set by the user using the switches on the front of the plotter.

To use the Mutoh with PC-PLOTS, you must turn on the REMOTE switch on the front of the plotter.

GRAPHTEC PLOTTERS

The cable configuration for the Graphtec MP-1000 is the same as the Hewlett-Packard cable configuration.

Table 8-10 shows the cable configuration for the Graphtec MP-2000 plotter.

Table 8-10. Graphtec MP-2000 Cable Configuration

Computer			Plotter	
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	----->	3	Recv
Recv	3	<-----	2	Trans
CTS	5	<-----	20	DTR
DSR	6	<-----	20	DTR
DCGnd	7	-----	7	DCGnd

Baud rate: 9600

Figure 8-17 shows the switch settings for the Graphtec MP-2000 plotter.

SW1

ON		
OFF	X	X

SWITCHES	FUNCTION
1,2	Command mode: HP-GL

SW2

ON	X	X		X				X		
OFF			X		X	X	X		X	X

SWITCHES	FUNCTION
1	RS-232
2	Handshake: X ON/OFF
3	RS control: out only
4	8 bit
5	No parity
6	Parity: Even/odd
7	Stop bit: 1 bit
8,9,10	Baud rate 9600

86233

Figure 8-17. Graphtec MP-2000 Switch Settings (RS-232-C interface, HP-GL emulation commands). Switches are located on the base of the plotter; X's represent switch positions.

APPENDIX C. PLOTTER PEN VELOCITY, ACCELERATION, AND FORCE CAPACITIES

Table 8-11 lists plotter specifications for pen velocity, acceleration, and force capacity.

Table 8-11. Pen Velocity, Acceleration, and Force

Plotter	Command	Velocity	Accel	Force
CC945	0101	2-43ips	1-4g	-
CC960				
CC965	0101	2-43ips	1-4g	-
CC1073	0101	2-21ips	1g	-
CC1075	0101	2-35ips	1-2g	-
CC1077	0101	2-52ips	1-4g	-
HP7470/75	VS		-	-
HP7580/85		23ips	1-4g	10-66gr
DMP29	V	1,2,4, 8,16 ips	-	-
DMP40/41/42	V	1,2,3 ips	-	-
IBM749		16,32 ips	-	-



APPENDIX D. PLOT WINDOW DIMENSIONS

Tables 8-12 and 8-13 show the minimum and maximum plot windows and the corresponding plot sizes for plotters supported by PC-PLOTS.

**Table 8-12. Plot Window Dimensions
(U.S. Paper Sizes)**

Plotter Paper		Minimum (")		Maximum (")		Plot Size (")
		X	Y	X	Y	
CC945, CC965, CC1043, CC107X	A	0.000	0.000	11.000	8.500	11.000 x 8.500
	B	0.000	0.000	17.000	11.000	17.000 x 11.000
	C	0.000	0.000	22.000	17.000	22.000 x 17.000
	D	0.000	0.000	34.000	22.000	34.000 x 22.000
	E	0.000	0.000	44.000	34.000	44.000 x 34.000
CC960	A	0.000	0.000	11.000	8.500	11.000 x 8.500
	B	0.000	0.000	17.000	11.000	17.000 x 11.000
	C	0.000	0.000	22.000	17.000	22.000 x 17.000
	D	0.000	0.000	34.000	22.000	34.000 x 22.000
	E	0.000	0.000	44.000	32.000	44.000 x 32.000
DMP29	A	0.500	0.750	10.500	7.750	10.000 x 7.000
	B	0.750	0.875	15.750	10.125	15.000 x 9.250
DMP40	A	1.000	0.750	10.000	7.750	9.000 x 7.000
	B	0.750	0.875	15.750	10.125	15.000 x 9.250
DMP42	C	0.945	1.020	22.445	15.520	21.500 x 14.500
	D	1.040	0.945	32.040	22.445	31.000 x 21.500

Table 8-12 Continued

Plotter Paper		Minimum (")		Maximum (")		Plot Size (")
		X	Y	X	Y	
DMP52	C	0.945	1.020	22.445	15.520	21.500 x 14.500
	D	1.040	0.945	32.040	22.445	31.000 x 21.500
DMP41, DMP51, DMP51MP	C	1.000	1.000	21.000	16.000	20.000 x 15.000
	D	1.000	1.000	33.000	21.000	32.000 x 20.000
DMP52MP	C	0.945	1.020	22.445	15.520	21.500 x 14.500
	D	1.040	0.945	32.040	22.445	31.000 x 21.500
DMP56	A	1.000	1.000	10.800	8.300	9.800 x 7.300
	B	1.000	1.000	16.800	10.800	15.800 x 9.800
	C	1.000	1.000	21.800	16.800	20.800 x 15.800
	D	1.000	1.000	33.800	21.800	32.800 x 20.800
	E	1.000	1.000	43.800	33.800	42.800 x 32.800
DSCAN GP5400	A	0.000	0.000	11.000	8.500	11.000 x 8.500
	B	0.000	0.000	17.000	11.000	17.000 x 11.000
	C	0.000	0.000	22.000	17.000	22.000 x 17.000
	D	0.000	0.000	34.000	22.000	34.000 x 22.000
	E	0.000	0.000	44.000	32.000	44.000 x 32.000
Graphtec MP1000	A	0.399	0.332	10.601	8.168	10.202 x 7.836
	B	0.311	0.399	16.689	10.601	16.378 x 10.202
Graphtec MP2000	B	0.000	0.000	15.500	11.000	15.500 x 11.000
HP7470	A	0.431	0.485	10.569	8.015	10.138 x 7.530
HP7475	A	0.399	0.332	10.601	8.168	10.202 x 7.836
	B	0.311	0.399	16.689	10.601	16.378 x 10.202
HP7580	B	1.142	0.197	16.803	10.803	15.661 x 10.606
	C	1.142	0.197	21.803	16.803	20.661 x 16.606
	D	1.142	0.197	33.803	21.803	32.661 x 21.606
HP7585	B	1.142	0.197	16.803	10.803	15.661 x 10.606
	C	1.142	0.197	16.803	21.803	20.661 x 16.606
	D	1.142	0.197	33.803	21.803	32.661 x 21.606
	E	1.142	0.197	43.803	33.803	42.661 x 33.606

Table 8-12 Continued

Plotter Paper		Minimum (")		Maximum (")		Plot Size (")
		X	Y	X	Y	
IBM749	A	0.205	0.195	10.795	8.305	10.590 x 8.110
IBM7371		0.431	0.485	10.569	8.015	10.138 x 7.530
IBM7372	A	0.399	0.332	10.601	8.168	10.202 x 7.836
	B	0.311	0.399	16.689	10.601	16.378 x 10.202
IBM7374	B	1.142	0.197	16.803	10.803	15.661 x 10.606
	C	1.142	0.197	21.803	16.803	20.661 x 16.606
	D	1.142	0.197	33.803	21.803	32.661 x 21.606
IBM7375	B	1.142	0.197	16.803	10.803	15.661 x 10.606
	C	1.142	0.197	16.803	21.803	15.661 x 21.606
	D	1.142	0.197	33.803	21.803	32.661 x 21.606
	E	1.142	0.197	43.803	33.803	42.661 x 33.606
NZ8	A	0.000	0.000	11.000	8.500	11.000 x 8.500
	B	0.000	0.000	17.000	11.000	17.000 x 11.000
NZ822/ 824	A	0.000	0.000	11.000	8.500	11.000 x 8.500
	B	0.000	0.000	17.000	11.000	17.000 x 11.000
	C	0.000	0.000	22.000	17.000	22.000 x 17.000
	D	0.000	0.000	34.000	22.000	34.000 x 22.000
NZ836	A	0.000	0.000	11.000	8.500	11.000 x 8.500
	B	0.000	0.000	17.000	11.000	17.000 x 11.000
	C	0.000	0.000	22.000	17.000	22.000 x 17.000
	D	0.000	0.000	34.000	22.000	34.000 x 22.000
	E	0.000	0.000	44.000	34.000	44.000 x 34.000

**Table 8-13. Plot Window Dimensions
(Metric Paper Sizes)**

Plotter Paper	Minimum (")	Maximum (")		Plot Size (")
		X	Y	
DMP42	A2	24.00, 25.91	570.10, 394.21	546.10 x 368.30
	A1	26.42, 24.00	813.82, 570.10	787.40 x 546.10
DMP52	A2	24.00, 25.91	570.10, 394.21	546.10 x 368.30
	A1	26.42, 24.00	813.82, 570.10	787.40 x 546.10
DMP52MP	A2	24.00, 25.91	570.10, 394.21	546.10 x 368.30
	A1	26.42, 24.00	813.82, 570.10	787.40 x 546.10
DMP56	A4	25.40, 25.40	291.85, 204.72	266.45 x 179.32
	A3	25.40, 25.40	414.78, 291.85	389.38 x 266.45
	A2	25.40, 25.40	588.77, 414.78	563.37 x 389.38
	A1	25.40, 25.40	835.91, 588.77	810.51 x 563.37
	A0	25.40, 25.40	1183.89, 835.91	1158.49 x 810.51
DSCAN GP5400	A4	0.00, 0.00	279.40, 215.90	279.40 x 215.90
	A3	0.00, 0.00	431.80, 279.40	431.80 x 279.40
	A2	0.00, 0.00	558.80, 431.80	558.80 x 431.80
	A1	0.00, 0.00	863.60, 558.80	863.60 x 558.80
	A0	0.00, 0.00	1117.60, 812.80	1117.60 x 812.80
Graphtec MP1000	A4	10.46, 8.39	286.46, 201.41	276.00 x 193.02
	A3	7.58, 10.46	411.52, 286.46	403.94 x 276.00
Graphtec MP2000	A3	0.00, 0.00	393.70, 279.40	393.70 x 279.40
HP7470	A4	12.21, 9.28	284.71, 200.53	272.50 x 191.25
HP7475	A4	10.46, 8.39	286.46, 201.41	276.00 x 193.02
	A3	7.58, 10.46	411.52, 286.46	403.94 x 276.00
HP7580	A3	29.00, 5.00	414.95, 291.95	385.95 x 286.95
	A2	29.00, 5.00	585.15, 414.95	556.15 x 409.95
	A1	29.00, 5.00	835.95, 586.40	806.95 x 581.40

Table 8-13 Continued

Plotter Paper		Minimum (")		Maximum (")		Plot Size (")
		X	Y	X	Y	
HP7585	A3	29.00	5.00	414.95	291.95	385.95, 286.95
	A2	29.00	5.00	414.95	585.15	385.95 x 580.15
	A1	29.00	5.00	835.95	586.40	806.95 x 581.40
	A0	29.00	5.00	1183.95	835.95	1154.95 x 830.95
IBM7371	A4	12.21	9.28	284.71	200.53	272.50 x 191.25
IBM7372	A4	10.46	8.39	286.46	201.41	276.00 x 193.02
	A3	7.58	10.46	411.52	286.46	403.94 x 276.00
IBM7374	A3	29.00	5.00	414.95	291.95	385.95 x 286.95
	A2	29.00	5.00	585.15	414.95	556.15 x 409.95
	A1	29.00	5.00	835.95	586.40	806.95 x 581.40
IBM7375	A3	29.00	5.00	414.95	291.95	385.95 x 286.95
	A2	29.00	5.00	414.95	585.15	385.95 x 580.15
	A1	29.00	5.00	835.95	586.40	806.95 x 581.40
	A0	29.00	5.00	1183.95	835.95	1154.95 x 830.95
Mutoh	A3	9.55	8.55	409.55	283.55	400.00 x 275.00
IP100	A2	10.85	9.55	573.35	409.55	562.50 x 400.00



APPENDIX E. HOUSTON INSTRUMENTS DMP PLOTTER TEXT LIMITATIONS

The following restrictions apply to Houston Instrument DMP-29, 40, 41, & 42 plotters.

- Text height is reduced to the next lower multiple of
7 * resolution
= 7 * 0.001" = 0.007"
- Text width is reduced to the next lower multiple of
6 * resolution
= 6 * 0.001" = 0.006"
- Maximum text height
= 255 * 7 * resolution
= 255 * 7 * 0.001" = 1.785"
- Maximum text width
= 255 * 6 * resolution
= 255 * 6 * 0.001" = 1.530"
- Text which is larger than the above maximums and text which is mirrored will be plotted using the PC-PLOTS character font.



APPENDIX F. SETTING PEN VELOCITY/ ACCELERATION ON DMP 51, 52, 51MP, AND 52MB PLOTTERS

Set pen velocity/acceleration in manual mode on the Houston Instruments DMP-51, 52, 56, 51MP, and 52MP plotters.

Before running PC-PLOTS, turn on your plotter.

Insert the paper and pens in holders.

Locate the plotter control buttons on the right front panel.

Press the plotter [Enter] and [Scale UR] buttons to start the configuration routine.

After you press [Scale UR], the plotter writes a line of text specifying possible Pen Down Velocity settings:

PEN DOWN VELOCITY (IPS):

2 3 4 5 6 7 8 9 10 11 12 13 14 15 16a

The plotter positions the pen by the current setting.

P-CAD recommends the plotter settings listed in Table 8-14 for the DMP-51, 52, 51MP, and the 52MP plotters. P-CAD recommends the plotter settings listed in Table 8-15 for the DMP-56 plotter.

Table 8-14. Recommended DMP-51, 52, 51MP, & 52MP Pen Velocity Settings

Parameter	Setting
Pen Down Velocity	16a (ball point pen) 8 (liquid ink pen)
Pen Up Velocity	16a (ball point pen) 8 (liquid ink pen)
Pen Down Acceleration	4G
Pen Up Acceleration	4G
Pen Down Delay	35
Pen Up Delay	35
Paper Size	LARGE
Pen Change Action	IGNORE
Constant Velocity	ON
Addressing	.001IN (.1MM for Metric)
Character Set	G0
Text Font*	F0
Baud Rate	9600
UART Parity	BITS=0
Handshake RTS/DTR	TOGGLE
Pass-thru Port Option	ALWAYS ON
Machine Type*	14 PEN

*These settings apply only to DMP-51MP and 52MP models.

Table 8-15. Recommended DMP-56 Pen Velocity Settings

Parameter	Setting
Pen Down Velocity	12
Pen Up Velocity	12
Pen Down Acceleration	1
Pen Up Acceleration	1
Pen Down Delay	45
Pen Up Delay	40
Paper Size	LARGE
Pen Change Action	IGNORE
Constant Velocity	ON
Addressing	.001IN (.1MM for Metric)
Character Set	G0
Text Font	F0
Baud Rate	9600
UART Parity	BIT8=0
Handshake RTS/DTR	TOGGLE
Pass-thru Port Option	ALWAYS ON

Press the arrow buttons on the plotter control panel to move the pen to the recommended setting and press the plotter [Enter] button.

The plotter underlines your selection and writes the next line of text indicating Pen Down Velocity selections.



Continue to enter the recommended settings in the same manner as above. As you enter your setting on each option, the next option is written beneath it.

After you have set all the parameters, the configuration sheet resembles Figure 8-18 for the DMP-51, 52, 51MP, and the 52MP plotters and Figure 8-19 for the DMP-56.



PEN DOWN VELOCITY (IPS):
2 3 4 5 6 7 8 9 10 11 12 13 14 15 16a

PEN UP VELOCITY (IPS):
2 3 4 5 6 7 8 9 10 11 12 13 14 15 16a

PEN DOWN ACCELERATION:
0.5G 1G 2G 3G 4G

PEN DOWN DELAY (MS)
0 5 10 15 20 25 30 35 40 45 50 55 60 65 70

PEN UP DELAY (MS)
0 5 10 15 20 25 30 35 40 45 50 55 60 65 70

PAPER SIZE:
SMALL LARGE

PEN CHANGE ACTION:
IGNORE PAUSE

CONSTANT VELOCITY OPTION:
ON OFF

ADDRESSING:
.001IN .005IN .1MM .125MM

CHARACTER SET:
G0 G1 G2 G3 G4 G5 G6 G7

BAUD RATE:
9600 4800 2400 1200 600 300

UART PARITY:
BITS=1 BITS=0 EVEN ODD

HANDSHAKE RTS/DTR:
TOGGLE ALWAYS HIGH

PASS-THRU PORT OPTION:
TOGGLE ALWAYS ON

**Figure 8-18. Sample Configuration Sheet for the
DMP-51, 52, 51MP, and 52MP**

PEN DOWN VELOCITY (PS):
2 3 4 5 6 7 8 9 10 11 12

PEN UP VELOCITY (PS):
2 3 4 5 6 7 8 9 10 11 12

PEN DOWN ACCELERATION (G):
.5 1

PEN UP ACCELERATION (G):
.5 1

PEN DOWN DELAY (MS):
40 45 50 55 60 65 70

PEN UP DELAY (MS):
20 25 30 35 40 45 50 55 60 65 70

PAPER SIZE:
SMALL LARGE

PEN CHANGE ACTION:
IGNORE PAUSE

CONSTANT VELOCITY:
ON OFF

ADDRESSING
.0011N .0051N .1MM .025MM

CHARACTER SET:
G8 G1 G2 G3 G4 G5 G6 G7

TEXT FONT:
F0 F1

ZERO CHARACTER:
SLASH NO SLASH

BAUD RATE:
9600 4800 2400 1200 600 300

UART PARITY:
BIT8=1 BIT8=0 EVEN ODD

HANDSHAKE RTS/DTR:
TOGGLE ALWAYS HIGH

PASS-THRU PORT:
TOGGLE ALWAYS ON

Figure 8-19. Sample Configuration Sheet for the DMP-56

Press [Scal UR] to exit the configuration routine.

Once the configuration is set, it remains in plotter memory until changed through the same routine.

You can now run PC-PLOTS.

DO NOT attempt to change the default pen velocity and acceleration settings (0%) on the PC-PLOTS Configuration Screen.

Refer to the DMP-51/52 or DMP 56 plotter manual for more information.





PC-PHOTO™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0064-04



ABOUT THIS MANUAL

This manual describes PC-PHOTO, the P-CAD utility program that produces instructions for Gerber, GTCO, and Patek Philippe Flashscan photoplotters to make film photoplots of PC-CARDS generated printed circuit board (PCB) designs.

This manual provides an overview of PC-PHOTO, operating instructions, and appendixes that describe error messages and photoplotter features.



CONTENTS

OVERVIEW.....	9-1
Input	9-2
Output	9-3
Photoplot Instructions.....	9-3
The Error Message File.....	9-4
Photoplotter Requirements	9-4
USING PC-PHOTO.....	9-5
Configuring PC-PHOTO	9-7
Editing the Aperture List	9-14
The Gerber Aperture List.....	9-16
The GTCO FP1-1622 Aperture List.....	9-19
The Flashscan Aperture List.....	9-22
Plotting a File.....	9-24
Outputting to Magnetic Tape From Disk.....	9-31
Outputting to a Photoplotter From Disk	9-32
APPENDIX A. ERROR MESSAGES	9-35
System Errors	9-35
Plot File Errors.....	9-36
The Error Message File	9-37
The "Messages" Section	9-39
The "Error Messages" Section	9-41
APPENDIX B. PHOTOPLOTTERS, APERTURES, FLASHES, AND LINES	9-43
Gerber Photoplotters	9-43
The GTCO FP1-1622 Photoplotter	9-45
The Flashscan Photoplotter.....	9-46
APPENDIX C. COMPUTER TO GTCO PLOTTER INTERFACE SPECIFICATIONS.....	9-49

CONTENTS (Continued)

APPENDIX D. GTCO FIRMWARE SYMBOL NUMBERS AND SIZES	9-51
APPENDIX E. GERBER FLASH APERTURE SYMBOL NUMBERS	9-55
APPENDIX F. COMPUTER TO FLASHSCAN PHOTOPLOTTER INTERFACE SPECIFICATIONS	9-57

FIGURES

9-1. PC-PHOTO Input and Output	9-1
9-2. PC-PHOTO Opening Menu	9-6
9-3. PC-PHOTO Configuration Screen	9-8
9-4. Original and Default Text Dimensions	9-13
9-5. Gerber Aperture List	9-16
9-6. GTCO Aperture List	9-20
9-7. Flashscan Aperture List	9-23
9-8. PC-PHOTO Program Screen	9-24
9-9. Minimum and Maximum Plot Window Coordinates	9-27
9-10. Normal and Rotated Plots	9-28
9-11. Sample PC-PHOTO Program Screen	9-30
9-12. Sample PC-PHOTO.ERR File	9-38

TABLES

C-1. GTCO Cable Configuration	9-49
F-1. Flashscan Cable Configuration	9-57

OVERVIEW

PC-PHOTO reads data from a plot file and converts it to photoplotter instructions. These instructions are input directly or indirectly to a photoplotter, which reads the plotting, aperture size, and flash commands and plots the design.

Figure 9-1 shows PC-PHOTO input and output.

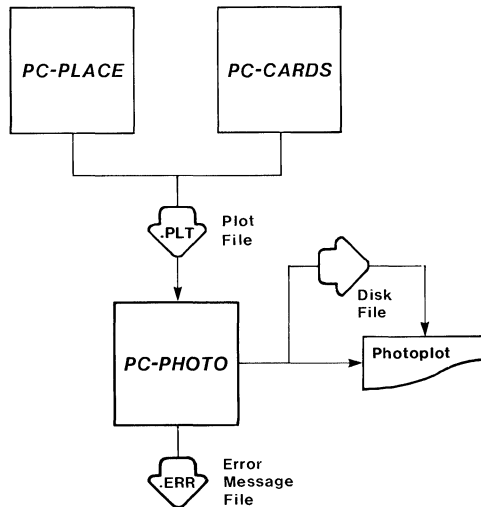


Figure 9-1. PC-PHOTO Input and Output

INPUT

Input to PC-PHOTO is a PCB plot file generated using PC-CARDS or PC-PLACE.

The PCB plot file is created from the PCB database using the SYS/PLOT command in PC-CARDS or PC-PLACE and is usually given the filename extension .PLT. It contains graphic information readable by PC-PHOTO. Refer to the "Commands" chapter in the *PC-CARDS User's Manual* for instructions for using the SYS/PLOT command to generate the PCB plot file.

In addition to the regular graphic information, the PCB database used to produce the plot file may contain "Flash" information which can be entered using the PC-CARDS DRAW/FLSH command. This command instructs the photoplotter to draw flash symbols that are unique to that plotter. Refer to the "Commands" chapter in the *PC-CARDS User's Manual* for further explanation of the DRAW/FLSH command.

NOTE: Before generating the plot file from the PCB database, be sure of the following:

- Aperture numbers specified for flash symbols in the PCB database match the aperture numbers or flash symbols on the plotter you will be using.

Refer to Appendix D, "GTCO Firmware Symbol Numbers and Sizes," and Appendix E, "Gerber Flash Aperture Symbol Numbers," for listings.

- Line widths specified in the PCB database are no smaller than the specified minimum aperture diameter for the photoplotter you will be using.
- PCB layers not to be photoplotted are turned off with the VLYR command.

OUTPUT

PC-PHOTO outputs photoplot instructions in the format of the specified photoplotter and, when applicable, an error message file.

Photoplot Instructions

Photoplot instructions can be output in two ways:

- They can be output directly from the computer to a photoplotter to produce a film photoplot.

NOTE: Currently, photoplot instructions can be output directly from the computer to only the GTCO FP1-1622 and the Patek Philippe Flashscan photoplotters to produce a film photoplot.

- They can be written to disk in the form of a photoplot file. This file can later be input directly to a photoplotter or transmitted to magnetic tape for input to a photoplotter.

If instructions are store in a disk file, PC-PHOTO gives this file the same filename as the input plot file and the filename extension .GBR for the Gerber photoplotter, .GTC for the GTCO photoplotter, or .FLS for the Flashscan photoplotter.

The Error Message File

The error message file (PCPHOTO.ERR) contains messages telling why the attempted photoplot did not run correctly or why certain elements in the plot file did not photoplot.

PHOTOPLOTTER REQUIREMENTS

PC-PHOTO supports the following photoplotters: Gerber photoplotter models 32, 33, and 41; the GTCO FP1-1622 photoplotter; and the Patek Philippe Flashscan.

Gerber, GTCO, and Flashscan photoplotters operate with different aperture arrangements and drawing patterns. They interface with PC-PHOTO in different ways. Refer to Appendix B, "Photoplotters, Apertures, Flashes, and Lines," for more information.

USING PC-PHOTO

Before running PC-PHOTO, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements. Note that PC-PHOTO requires only 256K of RAM.)
- You have installed the PC-PHOTO program file (PCPHOTO.EXE).
- You have produced the PC-CARDS or PC-PLACE generated plot file.

If you are using the P-CAD recommended directory structure, run PC-PHOTO from the appropriate project directory.

To run PC-PHOTO, type:

PCPHOTO [Return]

When the PC-PHOTO Title Screen appears, press any key to continue. The PC-PHOTO Opening Menu is displayed as shown in Figure 9-2.

PC-PHOTO

Options:

Configure PC-PHOTO

Edit aperture list

>> **Plot a file** <<

Exit PC-PHOTO

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

Figure 9-2. PC-PHOTO Opening Menu

This menu provides the following options.

Configure PC-PHOTO - Allows you to examine or change PC-PHOTO configuration options.

Edit aperture list - Allows you to edit the list of aperture types, shapes, and sizes for the photoplotter you are using.

Plot a file - Allows you to run PC-PHOTO. This option is the default when the Opening Menu is displayed.

Exit PC-PHOTO - Returns you to DOS.

CONFIGURING PC-PHOTO

Configure PC-PHOTO allows you to examine or change settings for several standard options.

To configure PC-PHOTO, follow the steps below.

1. At the PC-PHOTO Opening Menu, press the space bar or use the cursor keys to select **Configure PC-PHOTO**, then press [Return]. The system displays the PC-PHOTO Configuration Screen with the current settings, as shown in Figure 9-3.

You can now accept or change the settings for configuration options.

PC-PHOTO Configuration

Plotter port.....	PORT1
Output device	disk
Output record length	512
Default plotter.....	Gerber Model 32
System Setting (Type).....	US
Text aperture width.....	0.010"
Zero-width line diameter.....	0.000"
Pen velocity (GTCO).....	---%
Film speed (GTCO).....	3
Text dimensions (% of text size) :	
Text character width (including space).....	70%
Text character height.....	80%
Space below text line.....	10%
Space above text line.....	10%
Space from top of text to bar.....	20%

Press: [SPACE] for the next port; [RETURN] to accept

Figure 9-3. PC-PHOTO Configuration Screen

2. The first option, **Plotter port**, allows you to specify the communications port to which the plotter is connected. The choices are "PORT1" and "PORT2." The default is "PORT1."

To select "PORT1," press [Return]. To select "PORT2," press the space bar to display the "PORT2" option, then press [Return].

3. The next option, **Output Device**, allows you to specify the device to which the program outputs plotting instructions. The choices are "disk" and "plotter."

The "disk" setting is the default. It causes PC-PHOTO to store photoplotting instructions in a disk file which is later input to the photoplotter either directly or through transfer to magnetic tape. The output file has the same name as the input file and a .GBR extension for the Gerber photoplotter, a .GTC extension for the GTCO photoplotter, or a .FLS for the Flashscan photoplotter.

The "plotter" setting sends plotting instructions directly to a photoplotter. Currently, PC-PHOTO supports direct input to the GTCO FP1-1622 and Patek Philippe Flashscan photoplotters only.

Press [Return] to accept the default setting or press the space bar to select "plotter" and press [Return].

4. The next option, **Output record length**, allows you to specify the record length of the output disk file.

The default record length is 512. Do not change it unless you are specifically instructed to do so by your photoplotter operator.

This option applies only if you selected "disk" as your output device. If you are outputting directly to a photoplotter, press [Return] to ignore this option.

Press [Return] to accept the default setting, or type a new record length and press [Return].

5. The next option, **Default plotter**, allows you to select "Gerber Model 32," "Gerber Model 33," "Gerber Model 41," "GTCO FP1-1622," or "Flashscan Model 4PG2-AAA." The default plotter is "Gerber Model 32."

Press [Return] to accept the default setting or press the space bar to select a different photoplotter and press [Return].

6. The next option, **System Setting (Type)**, allows you to specify that the maximum and minimum plot coordinates be given in inches or millimeters. The choices are "US" or "METric."

The "US" setting is the default. If you select "US," the plot window coordinates will be given in inches.

If you select "METric," the plot window coordinates will be given in millimeters.

NOTE: Both settings will work only if the photoplotter you are using supports both options. Otherwise, you will get the plot window coordinates in metric or inches according to the photoplotter's type.

Press [Return] to accept the default setting or press the space bar to select "METric" and press [Return].

7. The next option, **Text aperture width**, allows you to specify in mils the aperture width that PC-PHOTO assigns for plotting text. The default width is "0.010."

Press [Return] to accept the default setting or type a new text width and press [Return].

If the text aperture width is less than the smallest size in the aperture list and the file contains text, an error message will be displayed when the file is plotted.

8. The next option, **Zero-width line diameter**, allows you to assign a photoplot line width to lines or wires drawn with a line width of zero in PC-CARDS.

PC-PHOTO will not plot lines or wires drawn with zero width in PC-CARDS unless you assign an aperture width with this option.

If you do not require a zero-width value, press [Return] to accept the 0 default setting. Otherwise, type a line diameter no smaller than the smallest line size in your PC-PHOTO aperture list and press [Return].

9. The next option, **Pen velocity (GTCO)**, allows you to specify the pen velocity for the GTCO photoplotter. This option applies only to the GTCO photoplotter.

If you are not using a GTCO photoplotter, press [Return] to ignore the pen velocity option.

If you are using a GTCO photoplotter and are intending to pen plot, you can set the speed at which your pens move.

The maximum velocity for the GTCO FP1-1622 is about 6 centimeters, or 1.6 inches/second. The default velocity is 33% (about 2 centimeters or .8 inches) of maximum.

To select the GTCO photoplotter default velocity, press [Return]. Three dashes (---) are displayed.

To set the pen velocity to a larger percentage of the maximum, type the desired percentage (34 to 100) and press [Return]. If you specify a value less than 33, the default velocity will be used.

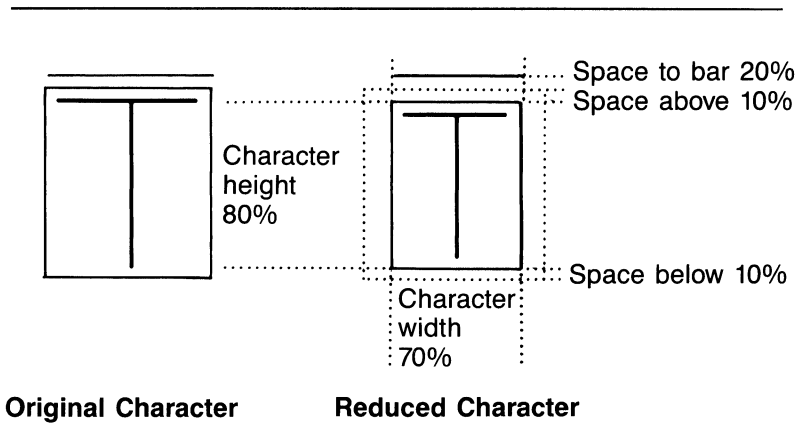
10. The next option, **Film speed (GTCO)**, allows you to specify the film speed for the GTCO photoplotter. The GTCO photoplotter uses four speeds of film: 1, 2, 3, and 4. Changing the film speed setting from 1 to 4 doubles the intensity of the GTCO photoplotting light.

If you are not using the GTCO photoplotter, press [Return] to ignore the film speed option.

If you are using the GTCO photoplotter, press [Return] to accept the default setting, or type the new film speed and press [Return].

11. The remaining options, which are displayed under the heading **Text dimensions (% of text size)**, allow you to specify the dimensions at which text will be plotted. Choices range from 0 to 1000% of the text size specified in PC-CARDS. Default values are provided.

Figure 9-4 shows how the text will be plotted using the default specifications.



86211

Figure 9-4. Original and Default Text Dimensions

For each option, press [Return] to accept the default value or type a new value and press [Return].

12. After you set the last option, PC-PHOTO displays the following prompt:

Save this configuration? No

To save the configuration, type Y for YES and press [Return]. The system saves the modified configuration as a file named PCPHOTO.CFG.

To use the configuration for the current session only, press [Return] for NO. Changes to the configuration will be used for this PC-PHOTO program session, but will not be saved after you exit PC-PHOTO.

NOTE: If you modify the P-CAD supplied configuration and later wish to restore it, from DOS, type the following command:

DEL PCPHOTO.CFG [Return]

The original configuration will be restored.

When configuration is complete, the system returns you to the Opening Menu.

EDITING THE APERTURE LIST

Edit Aperture List allows you to change the specifications for photoplotter apertures. Traces, pads, and other shapes entered in PC-CARDS are photoplotted by exposing film to a controlled beam of light filtered through apertures of various shapes and sizes. The apertures are attached to the photohead.

The aperture list displays the current settings for the apertures on the model of photoplotter selected on the Configuration Screen.

PC-PHOTO reads the line widths and flash commands in the PCB plot file, assigns them aperture settings from this list, and includes these aperture settings in the photoplot output instructions.

Lines are plotted with an aperture equal to or slightly larger than their width. Flashes are plotted according to an aperture or "flash" number specified with the DRAW/FLSH command in PC-CARDS. Refer to Appendix B, "Photoplotters, Apertures, Flashes, and Lines," for background information.

You can modify the aperture list to accommodate apertures of different sizes and shapes. We recommend that you compare this aperture list to the one from your photoplotter to make sure they match.

If you modify the Gerber aperture list, you must communicate the changes to the Gerber photoplot operator, who must then make changes in the photoplotting aperture wheel.

Changes made to the GTCO aperture list will be read directly by the photoplotter and incorporated into the photoplot instructions.

The Flashscan aperture list cannot be modified.

To edit the Gerber or GTCO aperture list, at the Opening Menu, use the space bar or the cursor keys to select **Edit Aperture List**, then press [Return].

The aperture list that displays settings for the default plotter appears on the screen.

The following sections give instructions for editing the aperture lists for Gerber and GTCO photoplotters, and show the aperture list for the Flashscan photoplotter.

The Gerber Aperture List

If you selected a Gerber model for the default photoplotter, the system displays the aperture list shown in Figure 9-5.

PC-PHOTO Gerber Apertures							
<u>#</u>	<u>size</u>	<u>shape</u>	<u>type</u>	<u>#</u>	<u>size</u>	<u>shape</u>	<u>type</u>
1	5	round	line	13	75	round	flash
2	10	round	line	14	75	square	flash
3	12	round	line	15	50	round	flash
4	25	round	line	16	55	round	flash
5	50	round	line	17	80	round	flash
6	75	round	line	18	85	round	flash
7	100	round	line	19	90	round	flash
8	0	#1	line	20	100	round	flash
9	60	round	flash	21	125	round	flash
10	60	square	flash	22	0	#4	flash
11	70	round	flash	23	50	#5	flash
12	70	square	flash	24	100	#6	flash

Special shape descriptions:

#1: aperture not used	#4: aperture not used
#2:	#5: drill crosshair, 5 mil
#3:	#6: bowtie target, Q2 & 4

Enter the size; Press: [Return] to accept

Figure 9-5. Gerber Aperture List

The Gerber aperture list shows the number, size, shape, and type for each of the 24 positions on the Gerber aperture wheel.

The aperture list settings have no effect on the operation of the photoplotter. The purpose of the list is to allow you to specify physical changes for the photoplotter operator to make on the aperture wheel. Before modifying any settings, consult your Gerber photoplotter operator to make sure the wheel can accommodate your changes. The operator must note your changes and change the apertures accordingly.

When the aperture list appears, the cursor is in the size column of aperture 1. To change a setting, use the following procedure:

1. Use the cursor keys to move the cursor to the aperture number you want to change.
2. The first setting is size, which gives the aperture size in mils.

To accept the current size, press [Return]. To change the size, type another size and press [Return].

The cursor moves to the **shape** column.

3. The **shape** setting gives the shape of the aperture. The shape can be "round," "square," or one of the six special shapes, #1 through #6.

To accept the current shape, press [Return]. To change the shape, press the space bar until the setting you want is displayed, then press [Return].

If you select one of the special shape numbers, the cursor moves to the **Special shape descriptions** section on the lower part of the screen. A default description, "Aperture not used," is displayed. Press [Return] to accept the default, or type a description of the aperture and press [Return].

NOTE: Each special shape number can be used for only one aperture. If you select a number that is already in use, an error message appears and you must select an unused number.

4. The **type** setting specifies whether the aperture is a "flash" or a "line."

To accept the current type, press [Return]. To change the type, press the space bar to display the other setting and press [Return].

5. When you finish making all your changes, print a screen copy of the revised aperture list to send to your photoplotter operator by pressing [Shift]-[PrtSc].
6. When you finish making all your changes, press [Esc] to exit the aperture list. The following prompt appears:

Save this aperture list? No

To save the aperture list, type Y for YES and press [Return]. The system saves the modified list as a file named PCPHOTO.APR and returns you to the Opening Menu.

To use the aperture list for the current session only, press [Return] for NO. Changes to the list will not be saved after you exit PC-PHOTO.

NOTE: If you modify the P-CAD supplied aperture list and later wish to restore it, from DOS, type the following command:

```
DEL PCPHOTO.APR [Return]
```

The original aperture list will be restored.

At the Opening Menu, use the space bar or cursor keys to select another option and press [Return].

The GTCO FP1-1622 Aperture List

If you selected a GTCO FP1-1622 for the default photoplotter, the system displays the aperture list shown in Figure 9-6.

PC-PHOTO GTCO Apertures

Physical Aperture Diameter (mils):

1: 12 2: 50 3: 60 4: 8

Centered Firmware Symbol Map:

#	Hex	#	Hex	#	Hex	#	Hex	#	Hex
1	20	14	2D	27	40	40	52	53	60
2	21	15	2E	28	41	41	53	54	61
3	22	16	2F	29	42	42	54	55	62
4	23	17	30	30	43	43	55	56	63
5	24	18	31	31	44	44	56	57	64
6	25	19	32	32	45	45	57	58	65
7	26	20	33	33	46	46	58	59	6F
8	27	21	34	34	47	47	59	60	--
9	28	22	35	35	48	48	5B	61	--
10	29	23	36	36	49	49	5C	62	--
11	2A	24	37	37	4A	50	5D	63	--
12	2B	25	38	38	50	51	5E	64	--
13	2C	26	39	39	51	52	5F	65	--

Enter the size; Press: [Return] to accept

Figure 9-6. GTCO Aperture List

The GTCO aperture list shows the physical aperture diameter settings for the four apertures on the GTCO photohead. This list matches the "flash" aperture numbers assigned in PC-CARDS with the corresponding GTCO setting.

When the aperture list appears, the cursor is on the diameter setting for aperture 1.

To edit the aperture list, use the following procedure:

1. The first section of the aperture list is **Physical Aperture Diameter (mils)**. It shows sizes in mils for each of the four apertures (1, 2, 3, and 4) on the GTCO photohead.

Aperture sizes vary for different GTCO photoheads. Measure the apertures on the photohead you are using. The settings in this section of the aperture list should match the sizes of the apertures on your photohead.

For each aperture in turn, press [Return] to accept the current size, or type another size and press [Return].

After you enter all four settings, press the cursor down key to move the cursor to the next section, **Centered Firmware Symbol Map**.

2. The firmware map matches the PC-CARDS "flash" numbers 1 through 65 to hexadecimal numbers representing the symbol plotting instructions programmed into the GTCO firmware.

Refer to Appendix D, "GTCO Firmware Symbol Numbers and Sizes," for samples of firmware symbols.

To change a setting, use the cursor keys to move the cursor to the number you want to change. Then type the new hexadecimal value and press [Return].

NOTE: To move from this section of the aperture list back to the **Physical Aperture Diameter** section, use the cursor keys.

3. When you finish making all your changes, press [Esc] to exit the aperture list. The following prompt appears:

Save this aperture list? No

To save the aperture list, type Y for YES and press [Return]. The system saves the modified list as a file named PCPHOTO.APR and returns to the Opening Menu.

To use the aperture list for the current session only, press [Return] for NO. Changes will not be saved after you exit PC-PHOTO.

NOTE: If you modify the P-CAD supplied aperture list and later wish to restore it, from DOS, type the following command:

DEL PCPHOTO.APR [Return]

The original aperture list will be restored.

At the Opening Menu, use the space bar or cursor keys to select another option and press [Return].

The Flashscan Aperture List

If you selected the Patek Philippe Flashscan for the default photoplotter, the system does not display an aperture list.

The aperture settings for each of the 68 positions on the Flashscan aperture wheel cannot be modified.

Figure 9-7 shows the Flashscan aperture list.

Flashscan Apertures							
<u>#</u>	<u>size</u>	<u>#</u>	<u>size</u>	<u>#</u>	<u>size</u>	<u>#</u>	<u>size</u>
1	5	18	27	35	52	52	78
2	6	19	28	36	54	53	80
3	7	20	30	37	55	54	81
4	8	21	31	38	57	55	82
5	9	22	33	39	58	56	84
6	11	23	34	40	60	57	85
7	12	24	35	41	61	58	87
8	13	25	37	42	63	59	88
9	15	26	39	43	65	60	89
10	16	27	40	44	66	61	91
11	17	28	41	45	68	62	92
12	19	29	43	46	69	63	91
13	20	30	44	47	70	64	95
14	21	31	45	48	72	65	96
15	22	32	48	49	74	66	97
16	24	33	49	50	75	67	99
17	26	34	51	51	76	68	100

Figure 9-7. Flashscan Aperture List

The Flashscan aperture list shows the number and size in mils for each of the 68 positions on the Flashscan aperture wheel. The shape of each aperture is hexagonal.

PLOTTING A FILE

Plot a file allows you to generate the photoplot instructions and transmit them either directly to a photoplotter or to a photoplot disk file.

To plot a file, follow the steps below.

1. At the Opening Menu, select **Plot a file** and press [Return]. The system displays the PC-PHOTO Program Screen and prompts for the input filename as shown in Figure 9-8.

PC-PHOTO

Plot filename: filename.plt

Enter the filename; Press: [RETURN] to accept

Figure 9-8. PC-PHOTO Program Screen

2. Type the filename of the input plot file and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter a filename extension, PC-PHOTO enters the default .PLT extension.

NOTE: At any of the following prompts you can cancel your entry and return to the previous prompt by pressing [Esc].

The system prompts for the plotter model. The default is the model specified in the current configuration.

3. Press [Return] to accept the default plotter or press the space bar until the correct plotter name is displayed, then press [Return].

The system prompts for the coordinate mode. This option specifies whether the coordinates in the plot file will be interpreted as "Absolute" or "Incremental" during plotting.

NOTE: This prompt appears for Gerber photoplotters only. If you selected either the GTCO or the Flashscan photoplotter, you can go on to step 6.

In absolute mode, all coordinates are relative to the origin (in the lower left corner) at coordinate 0,0. In incremental mode, all coordinates are relative to the previous coordinate.

In PC-CARDS, all coordinates are expressed in absolute terms, so you will normally use the "Absolute" setting.

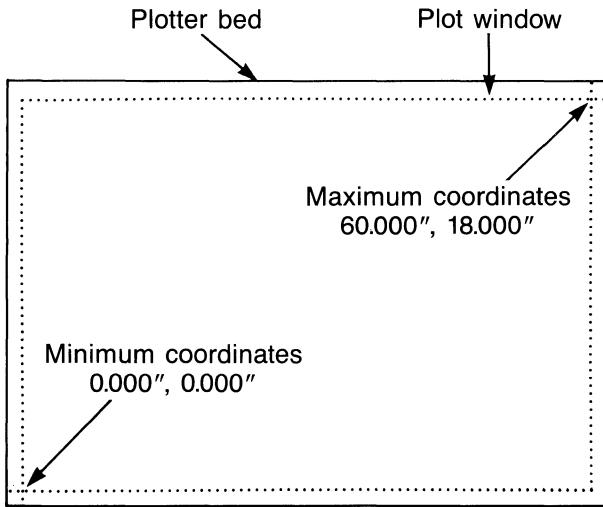
4. Press [Return] to accept the default coordinate mode, "Absolute," or press the space bar to display the "Incremental" setting, then press [Return].

The system prompts for the coordinate m.n format. This option sets the number of digits used in expressing photoplot coordinates. It is required for the Gerber photoplotters, which assumes the location of the decimal point. The "m" setting specifies the number of digits to the left of the decimal point, and the "n" setting specifies the number of digits to the right of the decimal point.

The default setting is 2 for m and 3 for n, mm.nnn. This setting accommodates coordinates with two digits to the left of the decimal and three digits to the right of the decimal (for example, 65.220 or 03.534).

5. Press [Return] to accept the default settings, or type new settings and press [Return].

The system prompts for the plot window coordinates, which specify the locations of the lower left corner and the upper right corner of the plot area on the plotter film bed, as shown in Figure 9-9.



86224

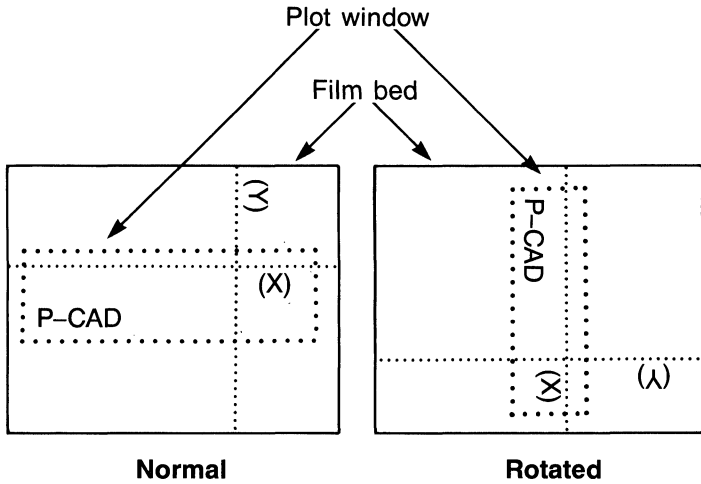
Figure 9-9. Minimum and Maximum Plot Window Coordinates

The default coordinates show the maximum plot area for the selected photoplotter. The plot will be centered in this area unless you enter new coordinates to specify a smaller plotting area.

6. At the prompt for each coordinate, press [Return] to accept the default, or type a new coordinate and press [Return].

The system prompts for the plot orientation. The settings are "normal" and "rotated."

A normally oriented plot is printed with its horizontal (x) axis parallel to the long side of the plotter bed. A rotated plot is printed with its vertical (y) axis parallel to the long side of the plotter. Figure 9-10 shows normal and rotated plots.



86225

Figure 9-10. Normal and Rotated Plots

For each plot, the default setting is the orientation that allows the design to be plotted at the largest scale. Thus, a plot window with a longer x than y dimension has a default of "normal" and a plot window with a longer y dimension has a default of "rotated."

WARNING: Apertures do not rotate. If you use oval or other nonequilateral apertures in a PCB database, specifying a rotated orientation will result in an erroneous photoplot.

7. Press [Return] to accept the default orientation or press the space bar to select the other setting and press [Return].

The system prompts for the scale factor, which specifies the enlargement or reduction of the design for the photoplot.

PC-PHOTO uses the plot dimensions, plot window size, and orientation to compute the largest scale for the plot and displays this value as the default.

NOTE: Any scale other than 1 for a PCB plot file increases or decreases line widths by the specified amount. However, the scale factor does not change flash aperture sizes.

8. Press [Return] to accept the default scale, or type a smaller scale factor and press [Return].

Figure 9-11 shows a sample Program Screen with all values entered.

PC-PHOTO

Plot filename: newtest.plt
Plotter: Gerber model 32
Coordinate mode: Absolute
Coordinate m.n format:
 Integer digits: 2 Fractional digits: 3
Plot window:
 Minimum x,y: 0.000", 0.000"
 Maximum x,y: 60.000", 48.000"
Plot orientation: Normal
Scale factor: 12.632

Enter the scale factor; Press: [RETURN] to accept

Figure 9-11. Sample PC-PHOTO Program Screen

After you specify the scale factor, PC-PHOTO generates photoplotting instructions and outputs them to the photoplotter or to disk.

PC-PHOTO displays progress reports and error messages, if any, at the bottom of the screen. If necessary, it creates an error message file, PCPHOTO.ERR.

To view the error message file, first exit from PC-PHOTO. Then, at the DOS prompt, type the following command:

TYPE PCPHOTO.ERR [Return]

Refer to Appendix A, "Error Messages," for a list of possible error messages, the cause of the errors, and appropriate action.

NOTE: If you want to abort a plot, press [Esc]. The system prompts:

Abort This Plot? No

Type Y for Yes and press [Return] to abort the plot, or press [Return] to continue plotting.

When PC-PHOTO has finished plotting or writing to disk, it returns you to the Opening Menu.

OUTPUTTING TO MAGNETIC TAPE FROM DISK

Gerber photoplotters read photoplot instructions from magnetic tape.

Consult your photoplotter operator for directions on transferring disk file data to magnetic tape.

The following information and an aperture list must be furnished with the magnetic tape to the photoplot service for each output file to be plotted on a Gerber photoplotter:

CODE: ASCII or EBCDIC
UNITS: English
ZERO SUPPRESSION: None
INTEGER DIGITS: 2 DECIMAL DIGITS: 3
(or other, if so formatted)
COORDINATE MODE: Absolute or
Incremental

OUTPUTTING TO A PHOTOPLOTTER FROM DISK

The GTCO photoplotter can read photoplot instructions directly from disk.

If you save your photoplot instructions to a disk file and wish to plot from this file, follow the steps below.

1. Connect the GTCO photoplotter to Communications Port 1 or Communications Port 2 of your computer.
2. Change to the directory where the file is located.
3. Use the DOS PRINT command to plot the file. For example:

```
PRINT TESTFILE.GTC [Return]
```

The system prompts:

```
Name of List Device? PRN
```


4. Type COM1 if the plotter is connected to Communications Port 1, or type COM2 if the plotter is connected to Communications Port 2. Press [Return].

The system plots the file.

NOTE: Subsequent PRINT Commands will send data to the device designated here until the system is rebooted.



APPENDIX A. ERROR MESSAGES

This appendix lists and explains error messages that may appear on the screen when you run PC-PHOTO and describes the error message file.

SYSTEM ERRORS

The following errors must be corrected before PC-PLOTS can generate plotting instructions.

Message: ** PCADDRV.SYS file not found **

Cause: Your P-CAD software is not installed correctly on your computer system.

Action: Use the P-CAD INSTALL program, which creates the required file.

Message: Plotter is offline.

Cause: The plotter is not connected to the port you specified on the Configuration Screen.

Action: Make sure the plotter is connected to the appropriate port.

Message: Port <n> not found.

Cause: The port you specified on the Configuration Screen does not exist in your system.

Action: Select another port on the Configuration Screen.

PLOT FILE ERRORS

Before generating photoplot instructions, PC-PHOTO scans the input plot file. If PC-PHOTO finds errors, it displays one or both of the messages described below. If applicable, it also creates the error message file, PCPHOTO.ERR, which is described in the next section, "The Error Message File."

Message: n Zero-width lines are ignored

Cause: Zero-width lines are present in your plot file and database. PC-PHOTO will not plot zero-width lines.

Action: If you want to plot the zero-width lines, press [Esc] to abort the plot. At the **Abort this plot? No** prompt, type Y [Return] to display the Opening Menu. Select **Configure PC-PHOTO** and enter a "zero-width line diameter" no smaller than the smallest aperture in the aperture list. Then, rerun **Plot a file**.

**Message: n Error(s) exist -> please check
PCPHOTO.ERR.**

Cause: Errors are present in the plot file.

Action: Check the error message file to read the error messages.

To access the error message file, press [Esc] to abort the plot. At the **Abort this plot? No** prompt, type Y [Return] to display the Opening Menu. At the opening menu, select "Exit PC-PHOTO" to exit to DOS. Enter the command **TYPE PCPHOTO.ERR** [Return]. Press [CTRL]-[S] to start and stop automatic scrolling.

THE ERROR MESSAGE FILE

The error message file, PCPHOTO.ERR, has two main sections, "messages" and "error messages." Figure 9-12 shows a sample error message file for a Gerber photoplot. The file for a GTCO and the Flashscan photoplot is similar.

Gerber messages:

```
>>> WIDTH - Rounded scale width.          <<<
>>> Scale factor = 1.00.                  <<<
>>> Smallest aperture diameter = 100 mils. <<<
>>> Text aperture WIDTH = 10 mils.       <<<
>>> Zero-width line diameter = 0.00 mils. <<<
>>> Zero-width lines and Filled rectangles
      WIDTH = n mils.                      <<<
```

Gerber error messages:

ERROR CODES:

1. Aperture is not a Gerber flash aperture.
2. Scaled line width is less than the smallest aperture diameter.

<u>ERROR CODE</u>	<u>TYPE</u>	<u>WIDTH(mils)</u>	<u>COORDINATES</u>
2	text	10	4000, 3100
2	text	10	4000, 3250
2	line	25	3700, 1400

Figure 9-12. Sample PCPHOTO.ERR File

The "Messages" Section

The "messages" section of the error message file shows several settings for this plot. The first five messages in Figure 9-11 are always present. They are explained below.

"WIDTH" is the specified width of a given segment times the scale factor (rounded to the nearest integer) specified in the "Plot a file" routine.

"Scale factor" is the factor specified in **Plot a file**.

"Smallest aperture diameter" refers to the smallest aperture in the aperture list.

"Text aperture WIDTH" is the text aperture width specified in **Configure PC-PHOTO** times the scale factor specified in **Plot a File**.

"Zero-width line diameter" refers to the diameter specified in **Configure PC-PHOTO**.

The "messages" section also contains either the last line in the Figure 9-11 example or one of two other sets of messages. Each of the possible messages is explained below.

Message: >>> **Zero-width lines are ignored.** <<<
>>> **Filled rectangles are ignored.** <<<

Cause: The width of a line is equal to zero, and therefore cannot be photoplotted. PC-PHOTO does not photoplot filled rectangles.

Action: Return to **Configure PC-PHOTO** and enter a "zero-width line diameter" that, when multiplied by the scale factor, will produce a width no smaller than the smallest width in the aperture list.

Zero-width lines will be plotted at the new width. Filled rectangles are plotted as hollow rectangles. To fill them, you can manually paint in the hollow rectangles on the finished photoplot with opaque solution.

Message: >>> **Zero-width lines and Filled rectangles**
WIDTH = n mils.

Cause: The width of a line times the scale factor is greater than zero but less than the smallest aperture size, and therefore cannot be photoplotted.

Action: Return to **Configure PC-PHOTO** and follow the steps specified for the previous message.

Message: >>> Zero-width lines will be drawn with the
WIDTH = n.n mils.

>>> Filled rectangles will be drawn as an
outline with the WIDTH = n.n mils.

Cause: The "zero-width line diameter" times the scale
factor is greater than or equal to the smallest
aperture diameter.

Zero-width lines will be plotted at the width
specified. Filled rectangles will be plotted as
hollow rectangles at the width specified.

Action: None.

The "Error Messages" Section

The "error messages" section of the error message file
lists the error codes, then a list including the code for
each error, the type of segment containing the error, its
scaled width, and its coordinate location in the database
file.

The error message codes are discussed below.

Message: 1. Aperture is not a Gerber flash aperture.

Cause: This message applies to the Gerber photoplotter only. A flash aperture in the PCB database is not specified as a flash aperture in the Gerber aperture list. The "flash" signal will be omitted from the output photoplot data.

Action: If the aperture number is correct, use **Edit aperture list** and change the type setting to "flash."

If the aperture number is incorrect, edit the PCB database through PC-CARDS at the location shown, regenerate the plot file, and rerun PC-PHOTO.

Message: 2. Scaled line width is less than the smallest aperture diameter.

Cause: The width of a line, rectangle, circle, or arc in the PCB plot file and database is less than the smallest aperture diameter specified in the aperture list file.

That line will be omitted from the output photoplot data.

Action: Use PC-CARDS to change the line width at the specified coordinates in the PCB database to at least equal the smallest aperture in the aperture list. Then regenerate the plot file and rerun PC-PHOTO.

APPENDIX B. PHOTOPLOTTERS, APERTURES, FLASHES, AND LINES

Photoplotters are the machines which create the finished tooling film from the PCB plot file data. The PCB plot is photoplotted directly on film, using what are referred to as "lines" and "flashes."

All photoplotters consist of a flat bed, on which the film is secured, and a photohead, consisting of an overhead light source and aperture housing.

The aperture housing contains apertures of various sizes and shapes. Apertures are the openings through which the light is focused on the film. The shapes and sizes of the apertures determine the shapes and sizes of the lines and flashes photoplotted on the film.

GERBER PHOTOPLOTTERS

Most Gerber photoplotters consist of a movable flat bed supported by motor drives underneath a stationary photohead.

Unexposed film is secured in the plotter bed and passed underneath the photohead. Some Gerber models move the photohead over a stationary film bed.

The Gerber aperture housing is a rotating wheel (called a "Gerber wheel") on which the necessary apertures are set to accommodate rapid switching.

Gerber photoplotters plot both "flash" and "line" images.

"Flash" images are photoplotted by turning the light source on and off while the plotter bed remains stationary. A flash, or image, of the aperture is exposed on the film.

"Line" images are plotted by turning the light source on, moving the plotter bed in an x and/or y direction, and turning the light source off. This procedure produces a line the width of the aperture diameter between the coordinates where the light is turned on and off.

Both flash and line images are plotted in response to PC-CARDS commands. Flash images are specified by the DRAW/FLSH or SCMD/GSSF commands in PC-CARDS. The flash number in PC-CARDS specifies which numbered aperture on the Gerber wheel is used to plot the flash image. Line images are used to plot all other graphical data. The apertures used are determined by the line widths assigned to the graphic data in PC-CARDS.

Gerber bed movement and aperture settings are controlled by the PC-PHOTO photoplot instructions, which are transferred from a disk file to 1/2 inch, 9-track computer tape, formatted at 800 BPI, and input to the photoplotter through a tape controller.

THE GTCO FP1-1622 PHOTOPLOTTER

The GTCO FP1-1622 photoplotter consists of a movable photohead over a stationary film bed.

Unexposed film is secured in the film bed and the photohead is passed over it.

The GTCO aperture housing holds four round apertures which generate all lines and shapes plotted.

The GTCO FP1-1622 plots "lines" and "firmware symbols."

"Lines" are photo plotted by the photohead moving across the film. Line width is determined by the diameter of the aperture used.

"Firmware symbols" are images of predetermined size and shape drawn by the photohead from instructions programmed into the GTCO firmware.

Like Gerber lines, GTCO lines are used to plot nonflash graphical data.

Like Gerber flash images, GTCO firmware symbols are specified by the PC-CARDS DRAW/FLSH command. The aperture number in PC-CARDS specifies which numbered GTCO firmware symbol will be used.

GTCO photohead movement and aperture settings are also controlled by the PC-PHOTO photoplot instructions. These instructions are either input to the GTCO photoplotter directly from the computer, or stored on disk to be input later.

THE FLASHSCAN PHOTOPLOTTER

The Flashscan photoplotter operation can be viewed as a system of four simultaneous processes:

1. Longitudinal movement of the horizontal drawing table (X axis).
2. Perpendicular movement of the photoprojection head (Y axis).
3. Exposure by means of a pulse rate modulated flash lamp.
4. Variation of the light source aperture.

The variation aperture lens projects an octagonal equiangular image onto the film.

Aperture dimensions are microprocessor controlled and range from 0.12 to 2.54 mm (5 mils to 100 mils) in 68 steps. This enables the Flashscan to produce an equal number of track widths and octagonal pad diameters.

When drawing lines, the microprocessor controls the flash interval so that octagons are projected with just enough overlap to create a line along the X and Y axes as well as in a 45 degree angle.

Like the other two photoplotter flash images, Flashscan flashes are specified by the PC-CARDS DRAW/FLASH command. The aperture number in PC-CARDS specifies which aperture should be selected in the Flashscan photoplotter. The instructions from PC-PHOTO can be either input to the Flashscan photoplotter directly or stored on disk and input later.



APPENDIX C. COMPUTER TO GTCO PLOTTER INTERFACE SPECIFICATIONS

When using PC-PHOTO directly with the GTCO FP1-1622 photoplotter, use the following specifications:

Baud rate = 9600

Data bits = 8

Parity = None

The required cable configuration is shown in Table C-1.

Table C-1. GTCO Cable Configuration

Computer		Plotter	
Signal	Pin	Pin	Signal
ACGnd	1	1	ACGnd
Trans	2	3	Recv
Recv	3	2	Trans
CTS	5	20	DTR
DSR	6	20	DTR
DCGnd	7	7	DCGnd



APPENDIX D. GTCO FIRMWARE SYMBOL NUMBERS AND SIZES

GTCO firmware symbols are shown on the following pages, with corresponding symbol numbers, sizes, and hexadecimal codes.

<u>Flash Number</u>	<u>Size (mils)</u>	<u>Hex Code</u>	<u>Symbol</u>	<u>Flash Number</u>	<u>Size (mils)</u>	<u>Hex Code</u>	<u>Symbol</u>
1	12	20	·	16	122	2F	●
2	32	21	•	17	150	30	●
3	39	22	•	18	169	31	●
4	51	23	•	19	177	32	●
5	55	24	•	20	39	33	•
6	59	25	•	21	47	34	▪
7	59	26	•	22	51	35	▪
8	59	27	•	23	59	36	▪
9	65	28	•	24	71	37	▪
10	71	29	•	25	98	38	■
11	71	2A	•	26	126	39	■
12	79	28	•	27	44	40	•
13	85	2C	•	28	51	41	•
14	91	2D	•	29	59	42	•
15	98	2E	•	30	59	43	•



<u>Flash Number</u>	<u>Size (mils)</u>	<u>Hex Code</u>	<u>Symbol</u>	<u>Flash Number</u>	<u>Size (mils)</u>	<u>Hex Code</u>	<u>Symbol</u>
31	71	44	●	46	118x59	58	—
32	98	45	●	47	413x59	59	—
33	124	46	●	48	398x98	5B	—
34	150	47	●	49		5C	—
35	171	48	●	50		5D	⊕
36	362x12	49	—	51		5E	+
37	402x73	4A	—	52		5F	#
38	205x20	50	—	53		60	+
39	591x20	51	—	54		61	-
40	402x47	52	—	55		62	×
41	79x55	53	●	56		63	□
42	102x55	54	●	57		64	○
43	362x55	55	—	58		65	△
44	398x55	56	—	59		6F	⊕
45	669x59	57	—				



APPENDIX E. GERBER FLASH APERTURE SYMBOL NUMBERS

Gerber flash aperture symbols are shown below, with corresponding flash aperture numbers, sizes, and "D-Codes."

<u>Flash Number</u>	<u>Size (mils)</u>	<u>D-Code</u>	<u>Symbol</u>	<u>Flash Number</u>	<u>Size (mils)</u>	<u>D-Code</u>	<u>Symbol</u>
1	5	D10	—	13	75	D20	●
2	10	D11	—	14	75	D21	■
3	12	D12	—	15	50	D22	●
4	25	D13	—	16	55	D23	●
5	50	D14	—	17	80	D24	●
6	75	D15	—	18	85	D25	●
7	100	D16	—	19	90	D26	●
8	60	D17	○	20	100	D27	●
9	60	D18	●	21	125	D28	●
10	60	D19	■	22	60	D29	○
11	70	D70	●	23	50	D72	+
12	70	D71	■	24	100	D73	●



APPENDIX F. COMPUTER TO FLASHSCAN PLOTTER INTERFACE SPECIFICATIONS

When using PC-PHOTO directly with the Flashscan photoplotter, use the following specifications:

- Baud rate = 9600
- Data bits = 8
- Parity = None
- Stop bits = 2
- Photoplotter resolution = 0.1mm

The required cable configuration is shown in Table F-1.

Table F-1. Flashscan Cable Configuration

Computer		Plotter		
Signal	Pin		Pin	Signal
ACGnd	1	-----	1	ACGnd
Trans	2	-----	3	Recv
Recv	3	-----	2	Trans
CTS	5	-----	4	RTS

The input format is Houston Instruments.



PC-DRC/NLC™
User's Manual

August 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0077-02



ABOUT THIS MANUAL

This manual introduces you to PC-DRC/NLC, the P-CAD utility program that checks a PC-CARDS designed printed circuit board (PCB) layout for violations of physical continuity and design rules.

NOTE: This program is used with the PCB-1, PCB-2, and PCB-3 systems only.

This manual provides an overview of PC-DRC/NLC, operating instructions, and appendixes that describe error messages and show sample output reports.



CONTENTS

OVERVIEW.....	10-1
Input.....	10-2
The PCB Database File.....	10-2
User-Specified Design Rules.....	10-3
Output.....	10-4
The Highlighted PCB Database File.....	10-4
The Error Report File.....	10-4
Continuity Errors.....	10-4
Design Errors.....	10-7
USING PC-DRC/NLC.....	10-9
Editing the Design Rules.....	10-11
Creating or Modifying a Rules Set.....	10-12
Renaming a Rules Set.....	10-16
Deleting a Rules Set.....	10-16
Saving Design Rules Changes.....	10-17
Editing the Check Passes.....	10-17
Creating or Modifying a Check Pass.....	10-19
Renaming a Check Pass.....	10-21
Deleting a Check Pass.....	10-21
Saving Check Pass Changes.....	10-22
Running PC-DRC/NLC.....	10-23
Viewing and Printing the Error Report.....	10-27
The Continuity Error Report.....	10-28
The Design Rules Error Report.....	10-28
Viewing the Highlighted PCB Database.....	10-30
APPENDIX A. ERROR MESSAGES.....	10-37
APPENDIX B. SAMPLE CONTINUITY ERROR REPORT.....	10-43
APPENDIX C. SAMPLE DESIGN RULES ERROR REPORT.....	10-45

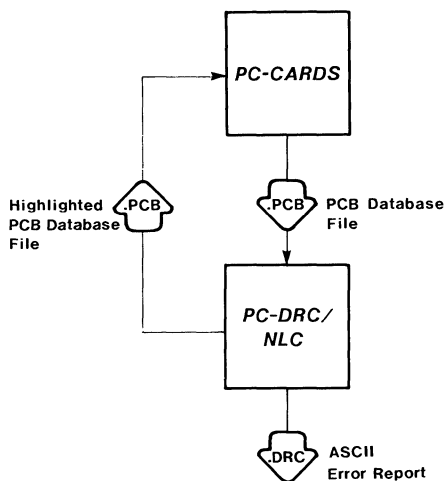
FIGURES

10-1. PC-DRC/NLC Input and Output.	10-1
10-2. Ratsnest and Trace Lines.	10-5
10-3. Continuity Error.	10-6
10-4. PC-DRC/NLC Opening Menu	10-10
10-5. Design Rules Set Menu.	10-12
10-6. Design Rules Set Screen.	10-14
10-7. Check Pass Menu.	10-18
10-8. Check Pass Screen	10-20
10-9. PC-DRC/NLC Program Screen	10-24
10-10. Sample PC-DRC/NLC Program Screen.	10-26
10-11. Line Showing Unwired Net	10-31
10-12. Highlighted Trace Size Error.	10-32
10-13. Highlighted Pad Size Error	10-33
10-14. Highlighted Trace Spacing Error.	10-34
10-15. Outlined Pads and Vias	10-35

OVERVIEW

PC-DRC/NLC reads a PC-CARDS generated printed circuit board (PCB) database and checks for violations of continuity rules and user-specified design rules. It generates additional layers in the PCB database on which the continuity and design errors are highlighted. It also generates an ASCII report file that lists the errors.

Figure 10-1 shows PC-DRC/NLC input and output.



86210

Figure 10-1. PC-DRC/NLC Input and Output

INPUT

PC-DRC/NLC input is the PC-CARDS generated PCB database and user-specified design rules.

The PCB Database File

The PCB database file (<filename>.PCB) contains the graphic representation of the physical PCB plus all relevant packaging and logical netlist information.

To check for continuity, PC-DRC/NLC requires the following conditions in the PCB database:

- Logical netlist connectivity (represented by ratsnest lines) should exist. Netlist connectivity is established when the PCB database is generated by PC-PACK from a schematic netlist. Netlist connectivity may be modified in PC-CARDS by SWAP and ENTR commands.
- Physical trace connectivity must be established. Physical traces or printed circuits, connecting one end of the ratsnest line with the other, should be entered on the board with the PC-CARDS ENTR/WIRE command.

Traces drawn with the DRAW/LINE command are not recognized by PC-DRC/NLC continuity check.

- Traces moved and connected to other traces with the EDIT commands must be merged and renamed with the PC-CARDS ENTR/WIRE command.

To check for adherence to design rules, PC-DRC/NLC requires the following conditions in the PCB database:

- Padstack symbols must be attached to pin and via locations on the PCB layout with the SCMD/GSSF command in PC-CARDS.
- The layer structure of the PCB database must be consistent with the layer groupings specified in PC-DRC/NLC, and items placed in the database must be placed on the appropriate layers. All solder side traces, for instance, should be entered on the SOLDER layer.

P-CAD provides a default layer structure for you to use when preparing your PCB database.

PC-DRC/NLC will read up to 48 layers on a database.

- The PCB outline must be drawn on a layer named BRDOUT. The system does not recognize shapes on any other layer as board outlines.

User-Specified Design Rules

Before you run PC-DRC/NLC, you input design rules specifications for the PCB. The **Edit Design Rules** option allows you to specify minimum size and spacing requirements for items in the PCB database. The **Edit Check Pass** option allows you to specify which layers of the PCB database to include in the design rules check.

OUTPUT

When you run PC-DRC/NLC, it reads the design rules specifications and the specified layers of the PCB database. It then outputs a highlighted PCB database, which displays connectivity and design rule errors, and an ASCII file that lists these errors in report form. The following sections describe these files.

The Highlighted PCB Database File

The marked PCB database (<filename>.PCB) is the same as the input database, except for the addition of two layers, \$CONT and \$DRC, on which errors are highlighted.

The \$CONT layer displays continuity errors; the \$DRC layer displays component size errors and spacing errors.

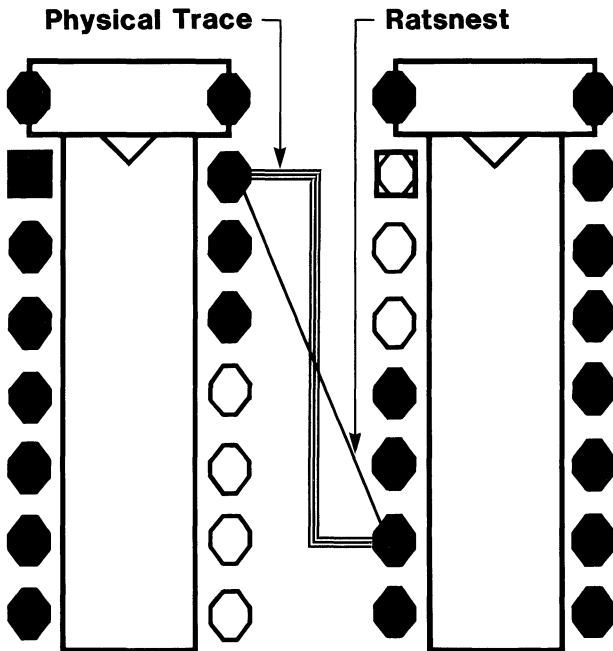
The Error Report File

PC-DRC/NLC also outputs an ASCII file (<filename>.DRC) listing continuity and design rule errors in report form.

Continuity Errors

Continuity errors are caused by discrepancies between the logical netlist connectivity and the physical printed circuit connectivity.

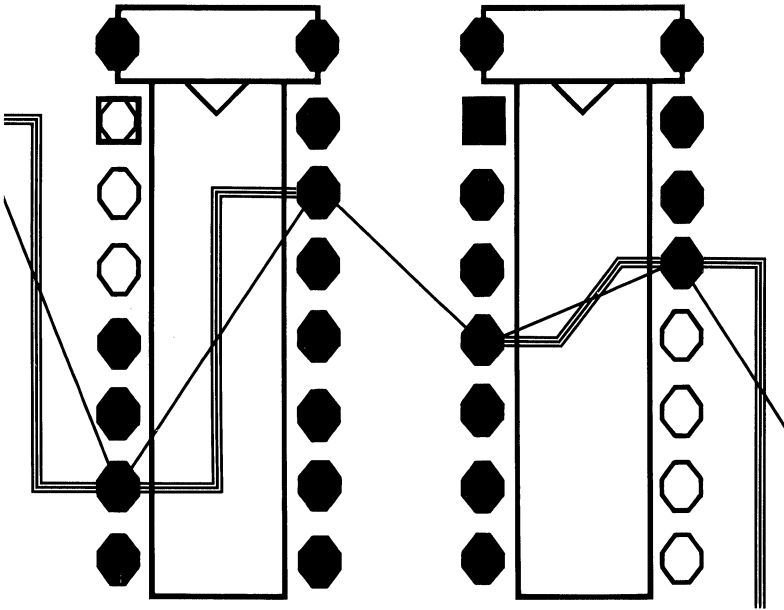
Logical netlist connectivity is indicated in the PCB database by ratsnest lines running between logically connected pins. Physical connectivity is established by traces drawn with the ENTR/WIRE command in the PCB database as shown in Figure 10-2.



86226

Figure 10-2. Ratsnest and Trace Lines

If pins connected by ratsnest lines are not also connected by corresponding physical traces, a continuity error exists as shown in Figure 10-3.



86227

Figure 10-3. Continuity Error

Design Errors

Design errors are violations of user-specified PCB design rules. PC-DRC/NLC checks for size and spacing design errors.

Size Errors. Size errors in PCB design occur when elements of the PCB are designed at smaller than user-specified dimensions.

PC-DRC/NLC checks for size errors with the following items: pads, vias, and traces (width).

Spacing Errors. Spacing, or "airgap," refers to the distance between one specified entity and another.

Spacing errors in PCB design occur when items are placed in the PCB layout closer than a user-specified minimum distance to one another.

PC-DRC/NLC checks for spacing errors in the following spaces: pad to pad, pad to wire, trace to wire, pad to board edge, and trace to board edge.

PC-DRC/NLC also checks spacing between the above items and non-electrical items (text or random shapes).

The items checked may be on the same or different layers.

NOTE: Lines drawn as traces can touch other items (traces, pads, vias and other lines) if the line endpoint is the point of contact. Lines drawn too close to other items without touching are flagged as spacing errors regardless of endpoint.



USING PC-DRC/NLC

To check design rules and continuity, you must carry out the following operations:

1. Edit or specify the design rules (size and spacing requirements).
2. Edit or specify the check passes (which PCB layers to check).
3. Run PC-DRC/NLC.
4. Read the ASCII report file.
5. View the highlighted PCB database.

Before running PC-DRC, be sure that:

- Your system is correctly configured. (See the "System Configuration" section in your *System Overview* manual for a list of requirements.)
- You have installed the PC-CARDS and PC-DRC/NLC program files (PCCARDS.EXE and PCDRC.EXE).
- You have prepared the PCB database to be checked.

If you are using the P-CAD recommended directory structure, run PC-DRC/NLC from the appropriate project directory.

To run PC-DRC/NLC, type:

PCDRC [Return]

When the PC-DRC/NLC Title Screen appears, press any key to continue. The PC-DRC/NLC Opening Menu is displayed as shown in Figure 10-4.

Options:

 Edit Design Rules
 Edit Check Pass
>> Run DRC/NLC <<
 Exit PC-DRC/NLC

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

Figure 10-4. PC-DRC/NLC Opening Menu

This menu provides the following options.

Edit Design Rules - Allows you to specify minimum size and space requirements for various items or pairs of items on the PCB layout.

Edit Check Pass - Allows you to specify the layers of the PCB layout to be checked on each pass of PC-DRC.

Run DRC/NLC - Allows you to run PC-DRC/NLC. This option is the default when the Opening Menu is displayed.

Exit PC-DRC/NLC - Returns you to DOS.

If you want to add to, create, modify, or review design rules or check pass specifications, use the procedures in the next sections, "Editing the Design Rules" and "Editing the Check Passes."

If you already have design rules or check pass specifications for your program and you do not wish to add to or modify them, continue to the section "Running PC-DRC/NLC."

EDITING THE DESIGN RULES

Edit Design Rules allows you to set or change minimum size and spacing specifications. These specifications are contained in design rules sets. You can create several design rules sets and use different sets as needed when you run PC-DRC/NLC.

To edit the design rules, at the PC-DRC/NLC Opening Menu, press the space bar or use the cursor keys to select **Edit Design Rules** and press [Return]. The system displays the Design Rules Set Menu, which shows the names of all your rules sets.

The first time you edit design rules, the only set is "pcad," which contains rules preset by P-CAD. Figure 10-5 shows the Design Rules Set Menu.

PC-DRC Design Rules

Edit Design Rules Set:

pcad

Enter the design rules set; Press: [RETURN] to accept

Figure 10-5. Design Rules Set Menu

You can now examine or edit an existing design rules set or create a new one. You can also rename or delete a rules set. The next sections show how to perform these operations.

Creating or Modifying a Rules Set

To create or modify a rules set, start at the Design Rules Set Menu and follow the procedure below.

1. To edit an existing rules set, use the cursor keys to move the cursor to the name of the set and press [Return].

To create a new rules set, use the cursor key to move the cursor to the space below the last set listed, then type the name of the new set and press [Return] twice.

The system displays the Design Rules Set Screen, which lists all the rules and allows you to enter settings for each one.

The rules specify minimum sizes and spacings. Settings are expressed in inches and to a thousandth of an inch (a mil). A setting of 1.000 means one inch; 0.001 means one mil.

For an existing rules set, the screen shows the current settings. For a new rules set, dashes are shown instead of settings.

Figure 10-6 shows the Design Rules Set Screen for the "pcad" rules set.

PC-DRC Design Rules

Edit Design Rules Set: pcad

Round pad size	0.060
Non-round pad size	0.060
Via size	0.050
Trace width	0.012
Pad to pad spacing	0.010
Pad to trace spacing	0.010
Trace to trace spacing	0.010
Pad to board edge spacing	0.125
Trace to board edge spacing	0.125

Enter the minimum value; Press [RETURN] to accept

Figure 10-6. Design Rules Set Screen

2. The first rule, **Round pad size**, sets the minimum size for pads. Press [Return] to accept the current setting or type a new setting and press [Return].

NOTE: You can use the cursor keys to move up and down between the rules.

3. The next rule, **Non-round pad size**, sets the minimum size for pads that are not round. Press [Return] to accept the current setting or type a new setting and press [Return].

4. The next rule, **Via size**, sets the minimum size for vias. Press [Return] to accept the current setting or type a new setting and press [Return].
5. The next rule, **Trace width**, sets the minimum width for traces. Press [Return] to accept the current setting or type a new setting and press [Return].
6. The next rule, **Pad to pad spacing**, sets the minimum distance between pads. Press [Return] to accept the current setting or type a new setting and press [Return].
7. The next rule, **Pad to trace spacing**, sets the minimum distance from a pad to a trace. Press [Return] to accept the current setting or type a new setting and press [Return].
8. The next rule, **Trace to trace spacing**, sets the minimum distance between traces. Press [Return] to accept the current setting or type a new setting and press [Return].
9. The next rule, **Pad to board edge spacing**, sets the minimum distance from a pad to the edge to the board, which is the board outline entered on the BRDOUT layer. Press [Return] to accept the current setting or type a new setting and press [Return].
10. The next rule, **Trace to board edge spacing**, sets the minimum distance from a trace to the edge of the board (the board outline). Press [Return] to accept the current setting or type a new setting and press [Return].
11. When all the settings are correct, press [Esc]. The system returns you to the Design Rules Set Menu.

The section "Saving Design Rules Changes" gives instructions for saving the new design rules settings.

Renaming a Rules Set

To rename a rules set, start at the Design Rules Set Menu. Use the cursor key to move the cursor to the name of the set, then type the new name and press [Return].

The section "Saving Design Rules Changes" gives instructions for saving the change.

Deleting a Rules Set

To delete a rules set, start at the Design Rules Set Menu and follow the steps below.

1. Use the cursor key to move the cursor to the name of the rules set.
2. Press the space bar and press [Return].

The system prompts:

Delete Rules Set "<name>"? No

3. Type Y for Yes and press [Return]. The system deletes the rules set.

The next section gives instructions for saving the change.

Saving Design Rules Changes

When you have created, edited, renamed, and deleted rules sets as required, at the Design Rules Set Menu, press [Esc]. The system prompts:

Save these rules? No

To save the rules sets, type Y for Yes and press [Return]. The system saves the modified rules sets as a file named PCDRC.RUL and returns you to the Opening Menu.

To use the rules sets for the current session, press [Return] for No. Changes to the rules set will be used for this PC-DRC/NLC program session but will not be saved after you exit PC-DRC/NLC.

NOTE: If you modify the original P-CAD supplied rules set and later want to restore it, from DOS, type the following command.

DEL PCDRC.RUL [Return]

The original rules set will be restored; however, all additional rules sets will be deleted.

EDITING THE CHECK PASSES

Edit Check Pass allows you to specify the passes PC-DRC/NLC will make when you run the design rules check. For each check pass, you specify layers to be checked and the design rules set to be used for those layers. You can have as many check passes as you want, but before running the design rules check, you must remove from the list any pass that you do not want to run.

To edit the check passes, at the PC-DRC/NLC Opening Menu, press the space bar or use the cursor keys to select **Edit Check Pass** and press [Return]. The system displays the Check Pass Menu, which shows the names of all the current check passes. Figure 10-7 shows the Check Pass Menu.

PC-DRC Check Passes

Edit Check Pass:

PASS_1
PASS_2

Enter the name; Press: [RETURN] to accept

Figure 10-7. Check Pass Menu

P-CAD supplies a default set of two check passes, **PASS_1** and **PASS_2**, for a standard PCB with two physical layers. These check passes are set up for use with P-CAD standard layer structure.

You can now examine, edit, delete, or rename the current check passes or create new passes. The next sections show how to perform these operations.

Creating or Modifying a Check Pass

To create or modify a check pass, start at the Check Pass Menu and follow the procedure below.

1. To edit an existing check pass, use the cursor keys to move the cursor to the name of the pass and press [Return].

To create a new check pass, use the cursor key to move the cursor to the space below the last set listed, then type the name of the new pass and press [Return] twice.

The system displays the Check Pass Screen, which shows the name of the design rules set and the layers for which it is to be used. The cursor is on the current rules set.

Figure 10-8 shows the Check Pass Screen for the PASS_1 check pass.

PC-DRC Check Passes

Edit Check Pass: PASS__1

Rules set: pcad

layers:

COMP
PADCOM

Press: [Space] for the next Rules Set; [Return] to accept

Figure 10-8. Check Pass Screen

2. Press [Return] to accept the current design rules set or press the space bar until the rules set you want is displayed, then press [Return].
3. Next, specify all the layers to be checked in this pass.

To add a layer, use the cursor down key to move the cursor to the line below the last layer displayed. Then type in the layer name and press [Return].

To delete a layer, use the cursor keys to move the cursor to the layer, then press the space bar and press [Return].

To change a layer name, use the cursor keys to move the cursor to the layer, then type the new name and press [Return].

4. When the the correct design rules set and layers are entered, press [Esc]. The system returns you to the Check Pass Menu.

The section "Saving Check Pass Changes" gives instructions for saving the new check pass specifications.

Renaming a Check Pass

To rename a check pass, start at the Check Pass Menu. Use the cursor key to move the cursor to the name of the pass, then type the new name and press [Return].

The section "Saving Check Pass Changes" gives instructions for saving the change.

Deleting a Check Pass

To delete a check pass, start at the Check Pass Menu and follow the steps below.

1. Use the cursor key to move the cursor to the name of the check pass.

2. Press the space bar and press [Return].

The system prompts:

Delete Check Pass "<name>"? No

3. Type Y for Yes and press [Return]. The system deletes the check pass.

The next section gives instructions for saving the change.

Saving Check Pass Changes

When you have created, edited, renamed, and deleted check passes as required, at the Check Pass Menu, press [Esc]. The system prompts:

Save this configuration? No

To save the check passes, type Y for Yes and press [Return]. The system saves the modified check passes as a file named PCDR.CFG and returns you to the Opening Menu.

To use the check passes for the current session, press [Return] for No. Changes to the check passes will be used for this PC-DRC/NLC program session but will not be saved after you exit PC-DRC/NLC.

NOTE: To temporarily omit certain check passes from a design rules check session, you can delete the passes you want to omit, and then not save the configuration. The check passes are not used for the current session, but are restored after you exit to DOS and rerun PC-DRC/NLC.

NOTE: If you modify the P-CAD supplied check passes and later want to restore them, from DOS, type the following command:

DEL PCDRC.CFG [Return]

The original check passes will be restored; however, all additional passes will be deleted.

RUNNING PC-DRC/NLC

To run PC-DRC, follow the steps below.

1. At the Opening Menu, select **Run DRC/NLC** and press [Return]. The system displays the PC-DRC/NLC Program Screen and prompts for the input filename as shown in Figure 10-9.

PC-DRC/NLC

PCB Filename: : <Filename>.PCB

Enter the filename; Press [Return] or [Esc] to exit.

Figure 10-9. PC-DRC/NLC Program Screen

2. Type the filename of the PCB database to check and press [Return]. Include a directory path or drive designator if the file is not in the current directory. If you do not enter a filename extension, the program adds a .PCB extension.

The system prompts for the output ASCII report filename. The default is the PCB filename with a .DRC extension.

3. Press [Return] to accept the default filename, or type another filename and press [Return].

The system prompts:

Perform Continuity Check? Yes

4. Press [Return] for Yes to run the continuity check or press the space bar to select No and press [Return] to omit the continuity check.

The system then prompts:

Perform Design Rule Check? Yes

5. Press [Return] for Yes to run the design rule check or press the space bar to select No and press [Return] to omit the Design Rule check.

Figure 10-10 shows a sample Program Screen.

PC-DRC/NLC



PCB Filename : TESTFILE.PCB
Report Filename : TESTFILE.DRC

Perform Design Rule Check? Yes



Figure 10-10. Sample PC-DRC/NLC Program Screen

PC-DRC/NLC performs one or both checks, as you specified. It displays progress reports at the bottom of the screen.

If no design rule errors are found on a check pass, the system displays the following message:

No DRC errors were found on this pass.



If too many design rule errors are found for a particular specification, the system stops checking the file and displays the following message:

Too many errors (see report for messages)

If this message appears, check the design rules error report, edit your PCB database accordingly, and run PC-DRC/NLC again.

PC-DRC/NLC lists error messages in the message section of the design rules check report. (Refer to Appendix A, "Error Messages," for a listing of common error messages.)

When processing is complete, PC-DRC/NLC returns to the Opening Menu. Use the space bar or the cursor keys to select another option and press [Return].

VIEWING AND PRINTING THE ERROR REPORT

To view or print the ASCII error report file, first exit from PC-DRC/NLC. To view the file, at the DOS prompt, type the following command:

```
TYPE <filename>.DRC
```

To print the file, you can use the TYPE command with the DOS print function on:

```
TYPE <filename> [Ctrl]-[P] [Return]
```

When the file is printed, you can switch off the print function by pressing [Ctrl]-[P] again.

You can also use the DOS PRINT command to print the file.

The two reports in the error report file are described below.

The Continuity Error Report

If you ran the continuity check, the first part of the report is the continuity error report.

The continuity error report lists the unwired or disjoint nets and lists coordinates of the closest pins of the unconnected net.

An example of a continuity error entry is shown below:

```
Net: QD  
(1850,2100) (3200,7500)
```

This entry indicates a net named QD lacks a physical connecting trace between coordinates 1850,2100 and 3200,7500 on the PCB layout.

Refer to Appendix B, "Sample Continuity Report," for a complete report.

The Design Rules Error Report

If you ran a design rules check, the error report file includes a design rules error report for each check pass made by PC-DRC/NLC.

Each design rules error report lists: the layers checked, all the settings of the design rules set used, a list of errors flagged, messages, and a statistical record of the number of items checked and errors found.

Each entry in the error list lists the type of error (spacing or size), the value or dimensions of the item or airgap flagged, the types of items involved, and the coordinates at which or between which the items are located.

Two examples of error list entries are shown below:

<u>TYPE</u>	<u>VALUE</u>	<u>ITEM</u>	<u>X1</u>	<u>Y1</u>	<u>X2</u>	<u>Y2</u>
SIZE	5	TRACE	4250	2600	4250	2500

This entry indicates an undersized trace with end coordinates 4250,2600 and 4250,2500 drawn at 5 mils wide (less than minimum specified width).

<u>TYPE</u>	<u>VALUE</u>	<u>ITEM</u>	<u>X1</u>	<u>Y1</u>	<u>X2</u>	<u>Y2</u>
SPACE	0	TRACE	2050	1700	2050	600
		PAD	2100	1150		

This entry indicates an airgap of 0 mils (a short) between a pad, centered at coordinates 2100,1150, and a trace running between coordinates 2050,1700 and 2050,600.

Refer to Appendix C, "Sample Design Rules Error Report," for a complete sample report.

Refer to Appendix A, "Error Messages," for a listing of errors that appear in the message section of the report.

NOTE: For any given pin type, only the first instance of a pad or via size error is reported; this entry indicates the same error exists for all instances of that pin type. Pad or via errors can be corrected by editing the special symbol file of the database.

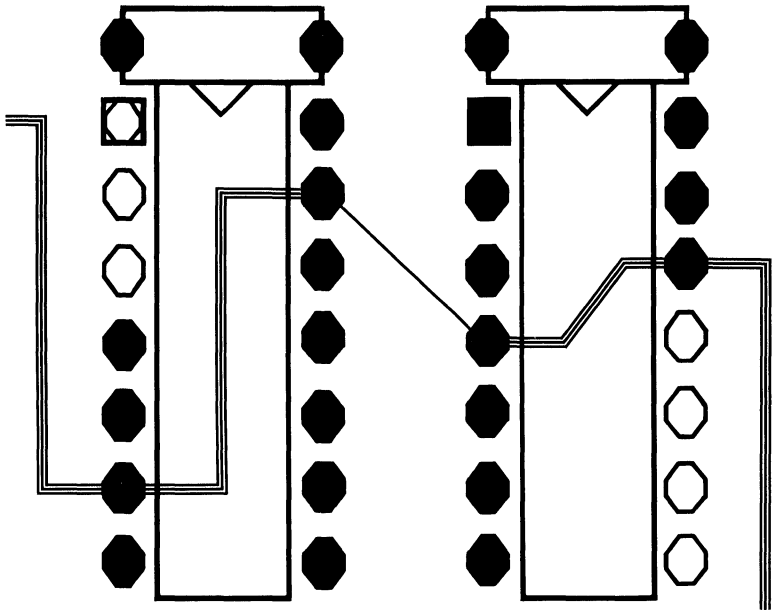
VIEWING THE HIGHLIGHTED PCB DATABASE

To view the graphic representation of continuity and design rule errors flagged by PC-DRC, access your original PCB database using the PC-CARDS FILE/LOAD command.

With the original database on the screen, select the VLYR command to view the layer structure. Note on the layer screen the addition of one or two layers, \$DRC and \$CONT. The \$DRC layer displays highlighted design rule errors. The \$CONT layer displays connectivity errors.

To view the errors, toggle the \$DRC and/or \$CONT layers ON; select a color number for each layer; exit from the layer menu; and select the REDR command to redraw the screen.

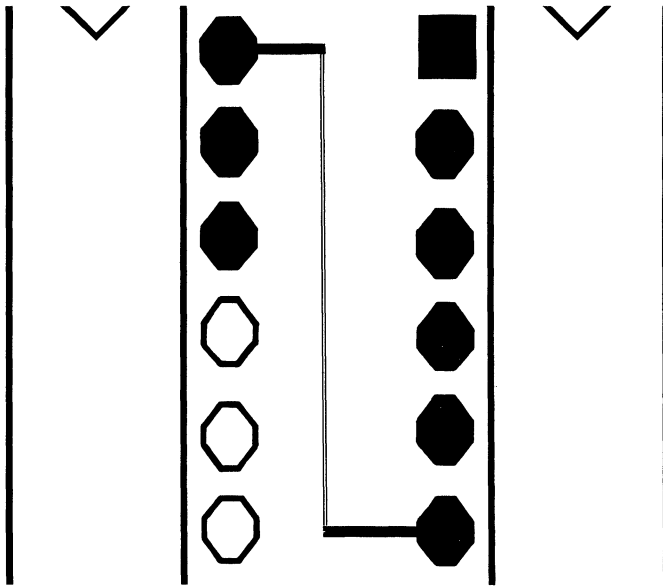
Continuity errors are displayed on the \$CONT layer as point-to-point lines (similar to ratsnest lines) from one unwired point on the net to the other. Figure 10-11 shows an example of a continuity error.



86228

Figure 10-11. Line Showing Unwired Net

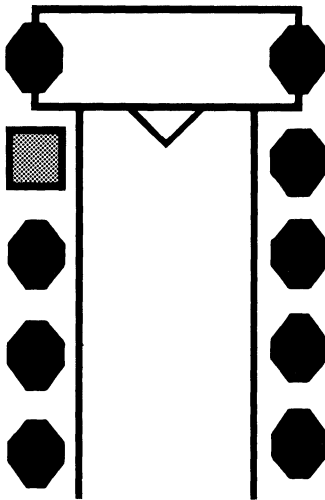
Trace size errors are displayed on the \$DRC layer as solid lines covering the error segments. Figure 10-12 shows an example of a trace size error.



86229

Figure 10-12. Highlighted Trace Size Error

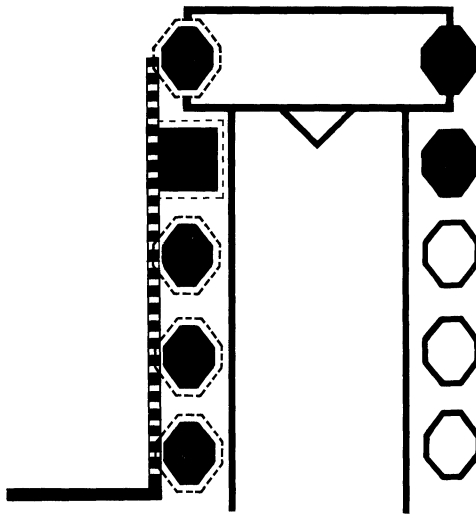
Nontrace size errors are displayed with the item in question highlighted in the color of the \$DRC layer. Figure 10-13 shows an example of a pad size error.



86230

Figure 10-13. Highlighted Pad Size Error

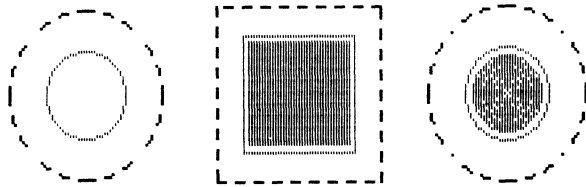
Trace spacing errors are displayed as dashed lines highlighting incorrectly spaced segments in the color of the \$DRC layer. Figure 10-14 shows an example of a trace space error.



86231

Figure 10-14. Highlighted Trace Spacing Error

Nontrace spacing errors are displayed with the items outlined in the color of the \$DRC layer. Figure 10-15 shows an example of this type of error.



86232

Figure 10-15. Outlined Pads and Vias



APPENDIX A. ERROR MESSAGES

Operational error messages appear in the message section(s) of the ASCII error report (<filename>.DRC).

Examples of common error messages are given below:

Message: No names specified in the layer group.

Cause: No layer names were specified for this check pass.

Action: Edit the check pass to include layer names.

Message: Layers not found:

**BOARD
PAD**

Cause: PC-DRC/NLC was unable to find layer names (in this case BOARD and PAD) specified for this check pass set in the PCB database.

Action: Check the PCB database layer names against the names shown and edit the check pass if necessary.

Message: FLASH not implemented.

Cause: PC-DRC/NLC does not include photoplot FLASH specifications in its design rules check.

Action: Make sure the dimensions of the shape specified in your flash commands do not violate your design rules.

Message: No special symbol found - (uncommitted pin)
Location: 2390,2400 Type: 24

Cause: PC-DRC/NLC was unable to find a padstack symbol for pin type 24. This message applies to all instances of pin type 24, but PC-DRC/NLC reports only the first instance of this error for each pin type.

Action: Check the special symbol file used in the PCB database to confirm that a padstack symbol is specified for the pin in question. If not, edit the special symbol file to include a padstack symbol for pin type 24 and use the PC-CARDS SCMD/GSSF command to include the padstack symbol in the database.

Message: No special symbol for vias found.

Cause: PC-DRC/NLC was unable to find a via symbol.

Action: Check the special symbol file used in the PCB database to confirm that a via symbol is specified. If not, edit the special symbol file to include a via symbol and use the PC-CARDS SCMD/GSSF command to include the via symbol in the database.

Message: No board edge defined.

Cause: The board outline has not been drawn on the BRDOUT layer. PC-DRC/NLC will not recognize lines drawn on any other layer as a board outline.

Action: Draw the board outline on the BRDOUT layer.

Message: Use only lines or rectangle to define board edge.

Cause: The board edge has been outlined with more than one type of DRAW command. Such combinations are not recognized by PC-DRC/NLC.

Action: Redraw the board outline using only the PC-CARDS DRAW/LINE command or only the PC-CARDS DRAW/REC command.

Message: Pad undefined on layer group - (uncommitted pin)

Location: 2650, 2300 Type: 5

Cause: The pad is undefined on layers specified in check pass set. This message applies to all instances of pin type 5, but PC-DRC/NLC reports only the first instance of this error for each pin type.

Action: Check the padstack symbol and the PCB database. Make sure layer names match. If names do not match, change the layer names to match and use the PC-CARDS SCMD/GSSF command to include the padstack symbol in the database.

Check your check pass set. Make sure the layer names match those of padstack symbol. If the names do not match, add the necessary layer names to the check pass set, or edit your padstack symbol to change the padstack layers and use the PC-CARDS SCMD/GSSF command to include the revised padstack symbol in the database.

Message: Too many trace width errors
Maximum allowed: 50 (Question TRACE
WIDTH rule).

Cause: Trace width errors have exceeded the maximum allowable number. Additional width errors are unlisted for this check pass. The design rules may contain an incorrect width specification.

Action: Check the design rules set for this check pass and edit it if necessary. If the design rules set is correct, redraw all affected wires in the database in the correct width.



APPENDIX B. SAMPLE CONTINUITY ERROR REPORT

This appendix shows a sample continuity error report. The report includes the heading and a listing of disjoint nets.

```
*****
*
* Copyright (C) 1985, 1986 - Personal CAD Systems *
*
* Program : DRC/NLC V1.30 *
* Date : Jul 11 1986 *
* Time : 04:44PM *
* Database : TESTFILE.PCB *
* Output : TESTFILE.DRC *
*
*****
```

Disjoint Nets:

```
Net: UN000015
    (1500,1050) (2400,1150)

Net: UN000019
    (2400,1250) (1500,1650)

Net: QD'
    (3300,1250) (1800,300)

Net: QA'
    (600,1250) (1700,300)
```

Net: +5V
(2400,1450) (2400,400)

Net: CLR
(1500,1250) (1500,950)

Net: GND
(2400,300) (2100,1450)
(3900,400) (3900,1450)
(2700,2350) (2700,1450)
(3300,1450) (3900,1450)
(3300,2350) (3900,2350)
(2100,1450) (2700,1450)
(300,1450) (900,1450)
(300,2350) (900,2350)
(900,2350) (1500,2350)
(2100,2350) (2700,2350)
(900,1450) (1500,1450)
(2700,1450) (3300,1450)
(2100,2350) (1500,2350)
(2700,2350) (3300,2350)
(1500,1450) (2100,1450)



APPENDIX C. SAMPLE DESIGN RULES ERROR REPORT

This appendix shows a sample design rules error report for PASS_1. The report includes the heading, the check pass layers, the design rules specifications, a listing of design errors, operational error messages, and a statistical record of items checked and errors found.

```
*****
*                                                                 *
* Copyright (C) 1984, 1986 - Personal CAD Systems *
*                                                                 *
* Program   :   DRC V1.30 *
* Date      :   Jul 11 1986  4:45:44 PM *
* Database  :   TESTFILE.PCB *
* Pass      :   PASS_1 *
*                                                                 *
*****
```

LAYERS GROUPED:

```
-----
COMP
PADCOM
PADVIA
```

DESIGN RULES SET:

```
-----
Pad size..... 60
Non-round pad size..... 60
Via size..... 50
Wire width..... 12
Pad to pad spacing..... 10
Pad to wire spacing..... 10
Wire to wire spacing..... 10
Pad to board edge spacing..... 125
Wire to board edge spacing..... 125
```

	TYPE	VALUE	ITEM	X1	Y1	X2	Y2
	----	-----	----	--	--	--	--
1	SIZE	7	WIRE	1000	3533	940	3593
2	SIZE	7	WIRE	940	3593	760	3593
3	SIZE	8	WIRE	760	3593	700	3533
4	SIZE	7	WIRE	700	3533	700	2833
5	SPACE	0	WIRE	680	993	660	1013
			PAD	670	1003	730	1063
6	SPACE	0	WIRE	680	993	660	1013
			PAD	670	1003	730	1063
7	SPACE	6	WIRE	660	1013	660	1133
			PAD	670	1003	730	1063
8	SPACE	6	WIRE	660	1013	660	1133
			PAD	670	1003	730	1063

MESSAGES:

1. Layers not found:
PADVIA
2. No special symbol found - (uncommitted pin).
Location: 4688, 2410 Type: 6
3. No special symbol found - (committed pin).
Location: 21872, 24527 Type: 20

STATISTICS:

Number of pads	851
Number of vias	4
Number of wires	50
Number of other shapes	2
Number of pad size errors	0
Number of via size errors	0
Number of wire width errors	4
Number of pad to pad errors	0
Number of pad to wire or shape errors	4
Number of wire to wire or shape errors	0
Number of pad to board edge errors	0
Number of wire or shape to board edge errors	0
Total number of entities	907
Total number of size violations	4
Total number of spacing violations	4
Total number of violations	8

Time finished: 4:48:03PM

INDEX

A

- Absolute mode, 9-25
- Aperture lists
 - editing in PC-PHOTO, 9-14
 - Flashscan, 9-22
 - Gerber, 9-15, 9-16
 - GTCO, 9-19, 9-20
- Aperture wheel, 9-16
- ASCII component library
 - cross-reference file
 - creating, 4-5, 4-6
 - modular format, 4-8
 - PREPACK, 4-2
 - regular format, 4-5, 4-6
- ASCII reports
 - generating, 1-4, 3-9
- Attributes
 - and PC-FORM component lists, 3-12, 3-26
 - and PC-FORM material lists, 3-16
 - and PC-FORM packaging lists, 3-15, 3-36

B

- Back-annotated netlist file
 - and PC-FORM, 3-3
 - and PC-PACK, 5-8
- Back-annotation, 6-1, 6-5
- Back-annotation file, 6-4
 - inputting, 6-9
- Batch control files
 - and PC-LINK, 2-14, 2-15
 - and PC-NODES, 1-10
- Batch mode
 - and PC-FORM, 3-19
 - and PC-LINK, 2-7, 2-16
 - and PC-NODES, 1-10
- Bill of Materials, 3-7
 - sample, 3-40, 3-42
- Binary component library
 - cross-reference file, 4-16
 - and PC-PACK, 5-3, 5-4
 - overview, 4-3
 - PREPACK, 4-2

- Binary netlists
 - extracting with PC-NODES, 1-8
 - linking with PC-LINK, 2-2, 2-4

C

- Calcomp plotters, 8-29
- Check passes
 - creating/editing, 10-17, 10-19
 - default sets, 10-18
 - deleting, 10-21
 - renaming, 10-21
 - restoring P-CAD supplied passes, 10-23
 - saving changes, 10-22
 - temporarily omitting, 10-22
- CHECKSUMMING option, 8-13
- Command line mode
 - and PC-FORM, 3-18
 - and PC-NODES, 1-8
- Comments
 - in PC-LINK batch file, 2-15
 - in PREPACK ASCII
 - cross-reference file, 4-6
- Component lists, 3-4, 3-11
 - samples, 3-26, 3-28, 3-30
- Configuration file
 - PC-PHOTO, 9-13
 - PC-PLOTS, 8-15
 - PC-PRINT, 7-14
- Connectivity, 10-2, 10-5
- \$CONT layer, 10-30
- Continuity
 - checking, 10-9
 - error report, 10-28
 - errors, 10-4, 10-30
 - sample error report, 10-43
- Coordinate mode, 9-25

D

- Design rules
 - checking, 10-9
 - editing, 10-11

- error report, 10-28
- errors, 10-7, 10-26, 10-27, 10-30
- sample error report, 10-45
- specifications, 10-3
- Design rules sets
 - creating/editing, 10-12
 - deleting, 10-16
 - renaming, 10-16
 - restoring P-CAD supplied set, 10-17
 - saving, 10-17
- Disk file
 - printing from, 7-21
- Dot matrix plot file
 - storing, 7-2
- \$DRC layer, 10-30

E

- Error listing file
 - PC-BACK, 6-4, 6-9
- Error message file
 - PC-PACK, 5-9
- Error messages
 - PC-BACK, 6-11
 - PC-DRC/NLC, 10-37
 - PC-FORM, 3-23
 - PC-LINK, 2-17
 - PC-NODES, 1-11
 - PC-PACK, 5-17
 - PC-PHOTO, 9-35
 - PC-PLOTS, 8-27
 - PC-PRINT, 7-23
 - PREPACK, 4-19
- Error report file
 - PC-DRC/NLC, 10-4
- Expanded schematic netlists
 - PC-FORM, 3-3
 - PC-LINK, 2-4, 2-5

F

- Files
 - plotting with PC-PHOTO, 9-24
 - plotting with PC-PLOTS, 8-15
 - plotting with PC-PRINT, 7-14

- Film speed
 - for GTCO photoplotter, 9-12
- Firmware symbols, 9-51
- Flash aperture symbols, 9-55
- Flash images, 9-44
- Flashscan aperture list, 9-22
- Flashscan photoplotter, 9-46, 9-57

G

- Gerber aperture list, 9-15, 9-16, 9-17
- Gerber photoplotter, 9-43
 - and magnetic tape, 9-31
 - flash aperture symbols, 9-55
- Graphics density option, 7-12
- GTCO FP1-1622 aperture list, 9-19
 - modifying, 9-21
- GTCO FP1-1622 photoplotter, 9-45, 9-49
 - firmware symbols, 9-51

H

- Hewlett-Packard plotters, 8-34
- Hierarchical schematics
 - flattening, 2-3
- Houston Instruments plotters, 8-40

I

- Interactive mode
 - and PC-FORM, 3-10
 - and PC-LINK, 2-7
 - and PC-NODES, 1-5

L

- LAYS.PCB file, 5-4
- Line images, 9-44

M

Magnetic tape
 outputting to, 9-31

Material lists, 3-7, 3-16
 samples, 3-40, 3-42

Modular format ASCII
 cross-reference files, 4-9
 and PC-PACK, 5-4

Multisheet input
 specifying for PC-LINK, 2-10

Multisheet schematics
 and PC-BACK, 6-10
 linking, 2-3

MUTOH plotters, 8-47

N

NEWTEST.PLT file, 7-2, 8-2

Nicolet Zeta plotters, 8-44

Node lists, 3-5, 3-14
 samples, 3-32, 3-34

Nontrace size errors, 10-33

Nontrace spacing errors, 10-35

O

Output record length option,
 9-9

Output report samples
 PC-FORM, 3-25

P

Package pin numbers
 adding to components, 5-1

Packaged PCB database file, 5-7

Packaging command file
 and PC-BACK, 6-2, 6-3
 and PC-PACK, 5-8
 inputting into schematic, 5-21

Packaging information
 adding to schematic without
 PC-CARDS, 4-8

Packaging lists, 3-5, 3-15
 samples, 3-36

Pads
 distance between, 10-15
 distance between pad

and trace, 10-15
 distance from pad to
 board edge, 10-15
 size for PC-DRC/NLC
 design rules, 10-14

Paper size

for 13" printers, 7-12, 7-17
 for 8" printers, 7-12, 7-16
 specifying for PC-PLOTS,
 8-11, 8-17, 8-18
 specifying for PC-PRINT,
 7-11, 7-16

PC-BACK

back-annotation file, 6-4
 error listing file, 6-4, 6-9
 error messages, 6-11
 input, 6-2
 output, 6-4
 overview, 6-1
 running, 6-5

PC-DRC/NLC

check pass menu, 10-18
 check pass screen, 10-20
 check passes, 10-17
 continuity error report, 10-28
 continuity errors, 10-4
 design errors, 10-7
 design rules error report,
 10-28
 design rules set menu, 10-11
 error messages, 10-37
 error report file, 10-27
 input, 10-2
 opening menu, 10-10
 output, 10-4
 overview, 10-1
 "pcad" design rules set
 screen, 10-14
 running, 10-9, 10-23
 size errors, 10-7
 spacing errors, 10-7

PC-FORM

and PC-LINK, 2-1
 and PC-NODES, 1-1
 error messages, 3-23
 input, 3-3
 operating modes, 3-9
 output, 3-4
 overview, 3-1
 running in batch mode, 3-19
 running in command line
 mode, 3-18

- running in interactive mode, 3-10
- viewing and printing output, 3-20
- PC-LINK
 - and hierarchical designs, 2-4
 - and multisheet/hierarchical designs, 2-4, 2-5
 - and multisheet/non-hierarchical designs, 2-4
 - and PC-NODES, 1-1
 - batch control file, 2-14
 - batch mode, 2-13
 - error message file, 2-4, 2-13
 - error messages, 2-17
 - input, 2-2
 - operating modes, 2-7
 - output, 2-4
 - overview, 2-1
 - running in interactive mode, 2-8, 2-9
- PC-NODES
 - batch control file, 1-10
 - input, 1-2, 1-3
 - operating modes, 1-5
 - output, 1-3
 - overview, 1-1
 - running in batch mode, 1-10
 - running in command line mode, 1-8
 - running in interactive mode, 1-5, 1-6, 1-7
- PC-PACK
 - and PC-LINK, 2-1
 - and PC-NODES, 1-1
 - error message file, 5-16
 - error messages, 5-17
 - input, 5-3
 - output, 5-6
 - overview, 5-1
 - running, 5-11
 - using without PC-CARDS, 5-1
 - viewing error message file, 5-9
- PC-PHOTO
 - configuring, 9-7, 9-8
 - error message file, 9-4, 9-35, 9-37
 - input, 9-2
 - opening menu, 9-6
 - output, 9-3
 - output filename extensions, 9-3
 - overview, 9-1
 - running, 9-5
 - supported photoplotters, 9-4
- PC-PHOTO plot files
 - errors in, 9-36
- PC-PLOTS
 - configuration file, 8-15
 - configuring, 8-7
 - error messages, 8-27
 - input, 8-2
 - output, 8-2
 - output filename extensions, 8-2
 - overview, 8-1
 - plot orientation settings, 8-19
 - program file, 8-5
 - running, 8-5
 - sample plot file, 8-2
 - supported pen plotters, 8-3
- PC-PRINT
 - configuring, 7-9
 - error messages, 7-23
 - input, 7-2
 - output, 7-2
 - output filename extensions, 7-2
 - overview, 7-1
 - plot orientation settings, 7-18
 - running, 7-7
 - sample plot file, 7-2
 - supported printers, 7-4
- PCB database file, 1-3
 - PC-DRC/NLC, 10-2, 10-24
- PCB highlighted database file
 - PC-DRC/NLC output, 10-4
 - viewing, 10-30
- PCB layer database file, 5-3, 5-4
- PCB part files
 - and PC-PACK, 5-3, 5-6
- PCBACK.EXE program file, 6-5
- PCCARDS.EXE program file, 10-9
- PCDRC.EXE program file, 10-9
- PCLINK.MSG file, 2-4, 2-13, 2-17
- PCPACK.ERR file, 5-9, 5-16

- PCPACK.EXE program file, 5-11
- PCPHOTO.APR file, 9-18, 9-22
- PCPHOTO.CFG file, 9-13
- PCPHOTO.ERR file, 9-4, 9-30, 9-31, 9-36
- PCPHOTO.EXE program file, 9-5
- PCPLOTS.CFG file, 8-15
- PCPLOTS.EXE program file, 8-5
- PCPRINT.CFG file, 7-14
- PCPRINT.EXE program file, 7-7
- Pen acceleration
 - plotter specifications for, 8-51
 - specifying for PC-PLOTS, 8-11
- Pen force
 - plotter specifications for, 8-51
 - specifying for PC-PLOTS, 8-11
- Pen plot file, 8-2
- Pen plotters
 - outputting from disk, 8-23
 - supported by PC-PLOTS, 8-3
- Pen velocity
 - plotter specifications for, 8-51
 - specifying for GTCO photoplotter, 9-11
 - specifying for PC-PLOTS, 8-11
- Pen width
 - specifying, 8-11
- Photoplot files, 9-3
- Photoplot instructions, 9-3
- Photoplot line width, 9-11
- Photoplotters
 - changing aperture specs, 9-14
 - Flashscan, 9-46
 - Gerber, 9-43
 - GTCO, 9-45
 - outputting to, 9-32
 - overview, 9-43
 - requirements, 9-4
 - supported by PC-PHOTO, 9-4
- Physical netlist file
 - and PC-FORM, 1-4, 3-3
 - and PC-PACK, 5-7
 - extracting with PC-NODES, 1-9
- Pin numbers
 - inserting, 5-21
- Plot a file option
 - PC-PHOTO, 9-24
 - PC-PLOTS, 8-15
 - PC-PRINT, 7-14
- Plot files
 - PC-PHOTO, 9-2, 9-24
 - PC-PLOTS, 8-2
 - PC-PRINT, 7-2
 - rasterizing, 7-20, 7-21
- Plot orientation
 - for PC-PHOTO, 9-27
 - for PC-PLOTS, 8-19
 - for PC-PRINT, 7-18
- Plot window
 - dimensions, 8-53
 - for PC-PHOTO, 9-27
 - for PC-PLOTS, 8-10, 8-17
 - for PC-PRINT, 7-17
- Plots
 - aborting, 7-21, 8-23, 9-31
 - normal, 7-18, 8-19, 9-28
 - rotated, 7-18, 8-19, 9-28
- Plotting files
 - PC-PHOTO, 9-24
 - PC-PLOTS, 8-15
 - PC-PRINT, 7-14
- PREPACK
 - error messages, 4-19
 - input, 4-2
 - output, 4-2
 - overview, 4-1
 - running, 4-15, 4-16, 4-17, 4-18
- PREPACK.ERR file, 4-3, 4-18
- PRESIM
 - and PC-LINK, 2-1
 - and PC-NODES, 1-1
- Printer density
 - for PC-PRINT, 7-12
- Printers
 - outputting to from disk, 7-21
 - supported by PC-PRINT, 7-4

R

- RADIX option, 8-13
- Radix value for PC-PLOTS, 8-13
- Rasterizing plot files, 7-20
- Reference designators, 5-1, 5-21
- Regular format ASCII cross-reference files, 4-5 and PC-PACK, 5-4
- Rename Component record, 6-16

S

- Scale factor
 - for PC-PHOTO, 9-29
 - for PC-PLOTS, 8-21
 - for PC-PRINT, 7-19
- Schematic database file, 1-2
- Schematic netlist files
 - and interface programs, 1-4
 - and PC-FORM, 1-3, 3-3
 - and PC-LINK, 1-3
 - and PC-PACK, 1-3, 5-3
 - and PRESIM, 1-4
 - extracting with PC-NODES, 1-9
 - flattening with PC-LINK, 2-1
- Schematics
 - back-annotating, 6-9
- Seiko Instruments plotter, 8-39
- Sheet attribute, 2-3
- Sheet IDs
 - and PC-NODES, 2-3
- Size errors, 10-7
- Spacing errors, 10-7
- Swap CNAME record, 6-17
- Swap Gate record, 6-15
- Swap Pin record, 6-13, 6-14
- Swap records
 - overview, 6-13
- Swap report file
 - and PC-BACK, 6-2, 6-3
 - editing with PC-BACK, 6-13
 - records, 6-3

Switch settings

- C.Itoh printer, 7-25
 - Hewlett Packard 7470A & 7475A printers, 8-36
 - Hewlett Packard 7580A printer, 8-38
 - Hewlett Packard 7580B & 7585B printers, 8-37
 - Houston Instruments DMP 29 printer, 8-41
 - IBM XY/749 printer, 8-43
 - MUTOH printer, 8-48
 - Nicolet Zeta 8 & 822 printers, 8-45
 - Nicolet Zeta 824 & 836 printers, 8-46
 - Texas Instruments printer, 7-26
- System Setting (Type) option, 9-10

T

- Text dimensions
 - for PC-PHOTO, 9-12
 - for PC-PLOTS, 8-13
 - for PC-PRINT, 7-13
- Trace size errors, 10-32
- Trace spacing errors, 10-34
- Traces
 - distance between, 10-15
 - distance from trace to board edge, 10-15
 - size for PC-DRC/NLC design rules, 10-14, 10-15

V

- Vias
 - size for PC-DRC/NLC design rules, 10-15

W

- Wire lists, 3-6, 3-16
- samples, 3-38