

NI-FNET™

User's Manual

May 1986

p-cad®
PERSONAL CAD SYSTEMS INC.

000-0130-01

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

NI-FNET, PC-CARDS, PDIF, and PDIF-IN are trademarks of Personal CAD Systems, Inc. (P-CAD).

ABOUT THIS MANUAL

This manual describes NI-FNET, the P-CAD program that converts FutureNet schematic database files to PCB database files in the P-CAD Database Interchange Format (PDIF). The PDIF files can then be input to the P-CAD's PCB design system using the PDIF-IN program.

Chapter 1, INTRODUCTION, provides an overview of NI-FNET and installation instructions.

Chapter 2, PREPARING THE INPUT FILES, gives instructions for creating the files to be input into NI-FNET.

Chapter 3, USING NI-FNET, gives instructions for running NI-FNET.

Chapter 4, VIEWING THE OUTPUT FILES, tells how to view, print, and interpret the NI-FNET output files.

Appendix A, ERROR MESSAGES, explains error messages that may be produced during NI-FNET program operation.

Appendix B, SAMPLE FILES, explains the sample data files provided with NI-FNET and shows several files.



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

* 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

CONTENTS

CHAPTER 1. INTRODUCTION	1-1
Overview	1-1
System Requirements	1-4
Installation	1-4
CHAPTER 2. PREPARING THE INPUT FILES	2-1
Preparing the Schematic	2-1
Creating the Cross-Reference File	2-1
The CFG Section	2-5
The PARTS Section	2-7
The DEFAULTS Section	2-9
Example	2-10
Creating the Layer Structure File	2-12
CHAPTER 3. USING NI-FNET	3-1
Interactive Mode	3-2
Command Line Mode	3-5
Batch Mode	3-8
Creating the PCB Database	3-9
CHAPTER 4. VIEWING THE OUTPUT FILES	4-1
Viewing and Printing Files	4-1
The PDIF File	4-2
The Alias List File	4-4
Alias List File Format	4-5
Signal Aliasing Conventions	4-5
Signals Connected to Busses	4-5
Signals Not Connected to Busses	4-6
Signal Names With Special Characters	4-7

Contents (Continued)

APPENDIX A. ERROR MESSAGES A-1
 File Access and System Errors..... A-1
 Pin List Translation Errors A-2

APPENDIX B. SAMPLE FILES..... B-1
 Cross-Reference File: NIFNDEMO.XRF B-3
 PDF File: NIFNDEMO.PDF..... B-4
 Alias List File: NIFNDEMO.ALS..... B-7

FIGURES

1-1. NI-FNET Input and Output..... 1-3
2-1. Sample Cross-Reference File..... 2-10
3-1. NI-FNET Program Screen 3-2
3-2. Sample Program Screen 3-4
3-3. Sample Command Line Mode Screen 3-7
3-4. Sample Batch Control File..... 3-8

CHAPTER 1. INTRODUCTION

The NI-FNET interface program translates schematics created using the FutureNet DASH-2 and DASH-3C programs into a format compatible with P-CAD's PC-CARDS PCB design program.

This chapter gives an overview of NI-FNET and installation instructions.

OVERVIEW

NI-FNET, together with PDIF-IN, enables you to produce files in the format of the PC-CARDS PCB design program from a schematic created in the FutureNet system.

There are five stages in this process:

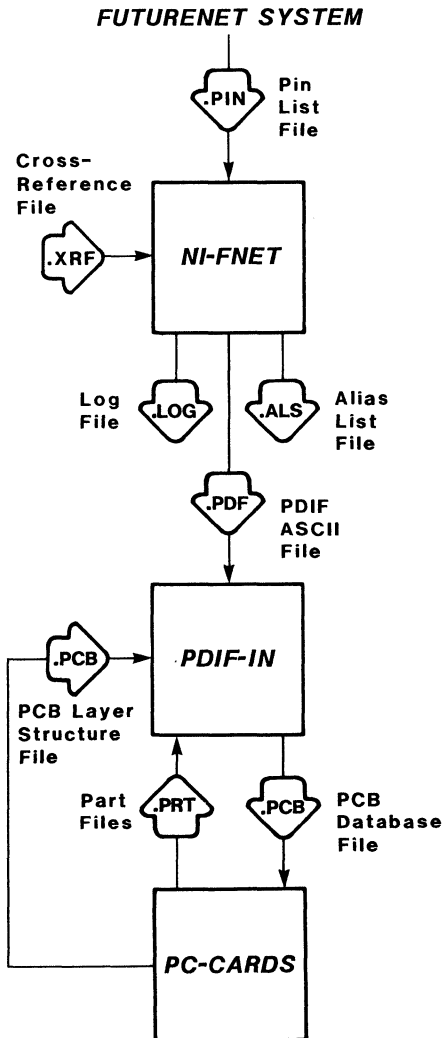
1. Using the FutureNet system, you prepare the schematic design (<filename>.DWG) and input it into the FutureNet Pin List Processor to extract a pin list file (<filename>.PIN).

NOTE: NI-FNET does not support hierarchical structure. You cannot input a pin list for a hierarchical design.

2. You use a text editing program to create the cross-reference file (<filename>.XRF), which maps FutureNet schematic symbols to the corresponding PC-CARDS PCB parts.
3. If necessary, you use PC-CARDS to create a layer structure file (<filename>.PCB) to provide basic environmental information for the PCB database.

4. You then input the pin list file into NI-FNET. NI-FNET uses the pin list file with the cross-reference file to produce three files:
 - The ASCII netlist (<filename>.PDF) contains all the design information in P-CAD Database Interchange Format (PDIF) and contains the name of the layer structure file. The ASCII PDIF file is described in Chapter 4, "Viewing the Output Files."
 - The alias list (<filename>.ALS) contains signal alias information assigned by NI-FNET. It is created only if NI-FNET aliases any signal names. This file is described in Chapter 4, "Viewing the Output Files."
 - The log file (NIFNET.LOG) contains all messages generated during the execution of NI-FNET. This file is useful for examining any error messages that are reported by NI-FNET.
5. You input the PDIF ASCII file into PDIF-IN to produce a PCB database in the format of PC-CARDS.

Figure 1-1 illustrates the process described above.

**Figure 1-1. NI-FNET Input and Output**

SYSTEM REQUIREMENTS

Before you install NI-FNET, your computer system must have the following minimum configuration:

- IBM PC, PC/XT, PC/AT, TI PC, Tandy 2000, or equivalent
- 640K of RAM
- P-CAD supported graphics board and monitor
- DOS 2.0 or higher operating system
- The CONFIG.SYS file in the root directory, containing a BUFFERS value of at least 12 and a FILES value of at least 15
- The PCADDRV.SYS file (created automatically when you use the P-CAD INSTALL program) in the root directory and the appropriate loadable device driver files in the appropriate directory as specified in the PCADDRV.SYS file

INSTALLATION

Your NI-FNET package includes two diskettes, the NI-FNET diskette and the PDIF diskette.

The NI-FNET diskette contains the files listed below.

Program file:

NIFNET.EXE

Sample data files:

NIFNDEMO.XRF	8185.PRT
NIFNDEMO.PDF	8284A.PRT
NIFNDEMO.ALS	8755A-2.PRT
LAYS.PCB	CK06.PRT
2PIN.PRT	D07.PRT
74LS10.PRT	RC07.PRT
8088.PRT	XTAL.PRT

You can use the sample files to see how the program works. Appendix B discusses the sample data.

To install NI-FNET on your hard disk, insert the NI-FNET diskette in Drive A and use the following procedures. These procedures assume that you are using the P-CAD directory structure.

Install the program files in your \PCAD\EXE directory. To do this, first change to the \PCAD\EXE directory by typing:

```
CD \PCAD\EXE [Return]
```

Copy the files by typing:

```
COPY A:*.EXE [Return]
```

Copy the sample data files to your working project directory. For example, if you are using the PROJ0 directory, change to that directory by typing:

```
CD \PCAD\PROJ0 [Return]
```

Then copy the files by typing:

```
COPY A:NIFNDEMO.* [Return]  
COPY A:LAYS.PCB [Return]  
COPY A:*.PRT [Return]
```

Remove the diskette.

The PDIF diskette contains several files. The only PDIF file necessary for NI-FNET is the PDIF-IN program file, PDIFIN.EXE. To install this file, first change to the \PCAD\EXE directory by typing:

CD \PCAD\EXE [Return]

Then insert the diskette in Drive A. Copy the file by typing:

COPY A:PDIFIN.EXE [Return]

If you want to use the other files provided on the PDIF diskette, refer to the *PDIF User's Manual* for installation instructions.

CHAPTER 2. PREPARING THE INPUT FILES

The files you input into NI-FNET are the pin list file produced from the FutureNet schematic and the cross-reference file, which you create using a text editing program. You also input the name of the layer structure file produced by PC-CARDS. This chapter gives NI-FNET requirements for the schematic and explains how to create the cross-reference file and the layer structure file.

PREPARING THE SCHEMATIC

NI-FNET translates a pin list file extracted from any schematic created using the FutureNet DASH-2 or DASH-3C program, with the following qualifications:

- The schematic cannot be hierarchical.
- All schematic components must have preassigned circuit designators. If a part's **CIRCUIT DESIGNATOR** attribute is in the format **XNNN**, the part is not considered to be preassigned and NI-FNET cannot process it.

CREATING THE CROSS-REFERENCE FILE

The cross-reference file is an ASCII file that you can create using any text editing program. It generally has the **.XRF** filename extension but can have any extension.

The cross-reference file has three sections, **CFG**, **PARTS**, and **DEFAULTS**.

The CFG section sets several configuration options that affect how NI-FNET translates the pin list files.

The PARTS section contains part cross-referencing for specific FutureNet parts and specifies the pins that can be common to different gates in these parts.

The DEFAULTS section contains default part cross-referencing for FutureNet parts that are not listed in the PARTS section and whose FutureNet circuit designators have certain prefixes. It also specifies the pins that can be common to different gates in these parts.

None of the sections are required; for example, if you want to use all default configuration settings, you do not need to include the CFG section.

Each FutureNet part in a design must be listed in the cross-reference file if the P-CAD part name is different from the FutureNet name or has a filename extension other than .PRT or if the part has any common pins. If the P-CAD part has a filename that is the same as the FutureNet part name with the added .PRT extension and the part has no common pins, you do not need to enter it in the cross-reference file.

When you translate a pin list, NI-FNET checks the cross-reference file for each FutureNet part. It first looks in the PARTS section. If the part name is not found there, it then looks in the DEFAULTS section. If no reference is found for the part in either section, NI-FNET gives the part the FutureNet part name with the .PRT filename extension.

NI-FNET inserts the P-CAD part filenames in the appropriate positions in the output PDIF file. When

you run PDIF-IN, it uses the specified part files to create the PC-CARDS PCB database.

You can create as many cross-reference files as you want for use in different designs, or you can create one cross-reference file that contains all the components you use in all designs. When you run NI-FNET, the default cross-reference filename is NIFNET.XRF. If you want to be able to use the default choice, you must name your cross-reference file NIFNET.XRF.

The format of the cross-reference file is shown below, followed by explanations of the format of each of the three sections. Brackets ([]) are used to indicate optional information. The | character indicates that you can use either of the options shown.

CFG

[COMMIT|UNCOMMIT]

[GRID <x y>|NOGRID]

[CHARSUB <fnetchar> <pcadchar>]

ENDCFG

PARTS

<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]

<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]

.

CPIN <p1,p2, .. [P/G] .. pn>

<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]

<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]

.

ENDC

<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]

.

ENDP

DEFAULTS

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

.

CPIN <p1,p2, .. [P/G] .. pn>

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

.

ENDC

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

.

ENDD

The CFG Section

The CFG section contains the NI-FNET configuration information in the format:

```
CFG
[COMMIT|UNCOMMIT]
[GRID <x y>|NOGRID]
[CHARSUB <fnetchar> <pcadchar>]
ENDCFG
```

where:

CFG begins the section and is required.

COMMIT or **NONCOMMIT** signifies whether or not PWR and GND pins are to be committed (connected to the PWR or GND net). If you do not make an entry, NI-FNET commits these pins.

GRID *x y* specifies the grid spacing between parts in the P-CAD PCB database, allowing you to have the parts automatically arranged in rows rather than overlapping. The *x* and *y* values specify the horizontal and vertical distance between the origins of the parts, in PC-CARDS database units (DBUs). If you do not specify spacing and you do not use the **NOGRID** option, NI-FNET defaults to *x*=600 and *y*=1200, meaning that parts are 600 DBUs apart horizontally and 1200 DBUs apart vertically.

NOGRID specifies that the parts are to be clustered (overlapped) at the grid coordinates 0 0.

CHARSUB specifies the characters NI-FNET is to substitute for the five characters that may be in the pin list file but cannot be used in the P-CAD system. The CFG section can contain up to five

CHARSUB entries. Each entry must specify both the FutureNet character and the substitute character.

fnetchar is the FutureNet character and must be %, [,], {, or }.

pcadchar is the character to be substituted. It can be any character that can be used in the PDIF file. If you do not specify characters for substitution, NI-FNET substitutes ! for %, (for [and {, and) for] and }.

ENDC ends the section and is required.

The PARTS Section

The PARTS section contains explicit cross-referencing between the FutureNet part names and the P-CAD part files in the format:

PARTS

```
<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]
```

```
<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]
```

```
.
```

```
CPIN <p1,p2, .. [P/G] .. pn>
```

```
<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]
```

```
<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]
```

```
.
```

ENDC

```
<fnetpart> [<pcadpart> <p1,p2, .. [P/G] .. pn>]
```

```
.
```

ENDP

where:

PARTS begins the section and is required.

fnetpart is the FutureNet part name assigned with the PART attribute (attribute type 3) found in the pin list file. If a part does not have a PART attribute, fnetpart is the contents of the VAL attribute (attribute type 4).

pcadpart is the P-CAD part filename that corresponds with the FutureNet part name. If you enter a name without an extension, NI-FNET adds the .PRT extension. You do not need to enter the P-CAD filename if it is the same as the FutureNet name and has the .PRT extension. However, if you

do not specify the P-CAD filename and you do specify common pins for the part, you must enter an asterisk (*) in this position.

p1, p2, . . . pn is a list of the pins in the part that are common to two or more gates.

P/G indicates that the pins listed next are connected to power or ground.

CPIN indicates that the following list shows common pins that are global for all the parts listed on the following lines until an **ENDC** keyword or another **CPIN** statement is specified. The **PARTS** section can have as many **CPIN** statements as necessary, but the **CPIN** statement is not required.

ENDC marks the end of a series of parts with the common pins specified with **CPIN**.

ENDP ends the section and is required.

Each line can have a maximum of 80 characters. Each FutureNet part and its corresponding P-CAD part name and/or list of common pins must be on one line. Each **CPIN** statement must be on one line.

The DEFAULTS Section

The DEFAULTS section contains implicit cross-referencing information for parts that are not entered in the PARTS section in the format:

DEFAULTS

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

.

CPIN <p1,p2, .. [P/G] .. pn>

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

.

ENDC

<prefix> <pcadpart> <p1,p2, .. [P/G] .. pn>

.

ENDD

where:

DEFAULTS begins the section and is required.

prefix is the FutureNet prefix string used in the contents of the CIRCUIT DESIGNATOR attribute (attribute type 2).

pcadpart is the P-CAD part filename that will be used for all the FutureNet parts that have the preceding prefix and are not listed in the PARTS section.

ENDD ends the section and is required.

The remaining information is the same as in the PARTS section.

Example

Figure 2-1 shows a sample cross-reference file.

```
CFG
GRID 600 1000
CHARSUB % @
ENDCFG

PARTS
7400 * 1,14
400mF CK05
ENDP

DEFAULTS
C      CAP
ENDD
```

Figure 2-1. Sample Cross-Reference File

The part information from three sample pin list file entries is shown below.

```
(SYM
DATA,2,U1
DATA,3,7400
)
(SYM
DATA,2,C10
DATA,4,100mF
)
(SYM
DATA,2,C11
DATA,4,400mF
)
```


If NI-FNET uses the cross-reference file in Figure 2-1 with the pin list entries above, it will use the following P-CAD part names for the FutureNet parts:

- For the first part, 7400, NI-FNET will use the default P-CAD part name, 7400.PRT. In this part, NI-FNET will allow pins 1 and 14 to be common to more than one gate. (If the part was not listed in the cross-reference file, it would be given the same default P-CAD part name but could not have common pins.)
- For the second part, 100mF, because no entry exists in the PARTS section, NI-FNET will check the DEFAULTS section for an entry for the circuit designator prefix, C, and use the CAP.PRT P-CAD part name.
- For the third part, 400mF, NI-FNET will use the CK05.PRT P-CAD part name listed in the PARTS section. (The 400mF part also has a circuit designator prefix of C, so if there was no entry for it in the PARTS section, it would receive the CAP.PRT P-CAD part name.)

The CFG section of the cross-reference file in this example will cause NI-FNET to specify component locations that are 600 DBUs apart horizontally and 1000 DBUs apart vertically and to substitute the @ character for the % character.

The default substitution character (will be used for [and { , and the default character) will be used for] and } . NONCOMMIT was not specified, so PWR and GND pins will be connected to the power and ground nets.

CREATING THE LAYER STRUCTURE FILE

The PCB layer structure file is generated by PC-CARDS. When you run NI-FNET, you specify a layer structure filename. NI-FNET inserts this filename in the output PDIF file. PDIF-IN uses the layer structure file to set basic environmental information for the PCB database to be created.

This information in the layer structure file includes the layers used in the database and sets grid size, grid display, visible and enabled layers, and default wire width. You can also add a board outline to this file if you want.

P-CAD recommends a standard layer structure to be used for PCB databases. This layer structure is supplied on the NI-FNET diskette as the LAYS.PCB file. The parts in all the P-CAD part libraries are created using the standard layers, so you must use these layers to use these parts. LAYS.PCB is the default layer structure file used by NI-FNET.

To make changes to the LAYS.PCB file (for example, to add a board outline), load the file into PC-CARDS, make the changes, then save the changed layer structure file with a different filename.

Refer to the *PC-CARDS User's Manual* for more information on layer structure and the standard layers.

CHAPTER 3. USING NI-FNET

This chapter describes the required conditions and procedures for configuring and running NI-FNET.

Before running NI-FNET, be sure that:

- Your system is correctly configured.
- You have installed the NI-FNET program file (NIFNET.EXE).
- You have transferred the input pin list file to your system.
- You have prepared the cross-reference file and, if necessary, the layer structure file.

NI-FNET has three operating modes: interactive, command line, and batch. In interactive mode, NI-FNET displays a series of screens and you select options and specify filenames. In command line mode, you specify filenames on the command input line when you start the program, and NI-FNET translates the pin list automatically. In batch mode, when you start the program, you specify the name of a batch file that instructs NI-FNET to process several pin lists automatically.

The following sections describe how to use interactive, command line, and batch modes and explain how to translate the output PDIF file into a PC-CARDS PCB database.

INTERACTIVE MODE

To run NI-FNET, start from the appropriate project directory and follow the steps below.

1. Type:

NIFNET [Return]

When the NI-FNET Title Screen appears, press any key to continue.

The system displays the NI-FNET Program Screen and prompts for the name of the input pin list file as shown in Figure 3-1.

NI-FNET

Pin List Filename : <filename>.PIN

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

Figure 3-1. NI-FNET Program Screen

2. Type the filename of the pin list to be read by NI-FNET and press [Return]. If you do not enter a filename extension, NI-FNET adds the .PIN extension.

NOTE: At any prompt or during file processing, if you decide not to proceed with the program, you can press [Esc] to quit.

The system prompts for the name of the input cross-reference file. The default is NIFNET.XRF.

3. Press [Return] to accept the default, or type another cross-reference filename and press [Return]. If you do not enter the filename extension, NI-FNET adds the .XRF extension.

The system prompts for the name of the input layer structure file. The default is LAYS.PCB.

4. Press [Return] to accept the default, or type another layer structure filename and press [Return]. If you do not enter the filename extension, NI-FNET adds the .PCB extension.

The system prompts for the name of the output file. The default is the input pin list filename with the .PDF extension as shown in Figure 3-2.

NI-FNET

Pin List Filename : TESTFILE.PIN
Cross-Ref Filename : NIFNET.XRF
Layer Filename : LAYS.PCB
Output Filename : TESTFILE.PDF

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

Figure 3-2. Sample Program Screen

5. Press [Return] to accept the default, or type another filename and press [Return]. If you do not enter the filename extension, NI-FNET adds the .PDF extension.

After you enter the output filename, NI-FNET begins file processing. It displays progress reports and error messages, if any, at the bottom of the screen.

When processing is complete, the cursor returns to the Pin List Filename prompt. You can translate another pin list file, or press [Esc] to exit to DOS.

COMMAND LINE MODE

Command line mode allows you to specify input and output filenames when you start NI-FNET.

To use command line mode, at the DOS prompt, type the command line and press [Return].

The format of the command line is:

```
NI-FNET <pinlist> <xrf> <layer> <outfile>
```

where:

NI-FNET initiates the program.

pinlist is the name of the input pin list file. This name is required. If you do not enter the filename extension, NI-FNET adds the .PIN extension.

xrf is the name of the input cross-reference file. This name is optional. If you do not specify a cross-reference filename, NI-FNET uses the default NIFNET.XRF. If you enter a name with no extension, NI-FNET uses the .XRF extension.

If you do not specify a cross-reference filename and you do specify a layer structure filename or an output filename, you must include an asterisk (*) in

the command line in place of the cross-reference filename.

layer is the name of the input layer structure file. This name is optional. If you do not specify a layer structure filename, NI-FNET uses the default, LAYS.PCB. If you specify a name with no extension, NI-FNET uses the .PCB extension.

If you do not specify a layer structure filename and you do specify an output filename, you must include an asterisk (*) in the command line in place of the layer structure filename.

outfile is the name of the PDIF file to be output. This name is optional. If you do not specify a name, NI-FNET uses the input pin list filename with the .PDF extension. If you specify a name with no extension, NI-FNET uses the .PDF extension.

Three examples of command lines are shown below.

```
NIFNET TESTFILE  
NIFNET TESTFILE PARTS  
NIFNET TESTFILE * OUTLINE TEST
```

The first example causes NI-FNET to create a PDIF file from the TESTFILE.PIN file using the default NIFNET.XRF cross-reference file and the default layer structure filename LAYS.PCB. The output file will have the default filename TESTFILE.PDF.

The second example causes NI-FNET to create a PDIF file from the TESTFILE.PIN file using the PARTS.XRF cross-reference file and the default layer structure filename LAYS.PCB. The output file will have the default filename TESTFILE.PDF.

The third example causes NI-FNET to create a PDIF file from the TESTFILE.PIN file using the default NIFNET.XRF cross-reference file and the OUTLINE.PCB layer structure filename, and to give the output file the filename TEST.PDF.

After you enter the command line, the program runs automatically with no further input from you. First, the NI-FNET Title Screen appears, followed by the Command Line Mode Screen. Figure 3-3 shows a sample Command Line Mode Screen.

NI-FNET

Pin List Filename : TESTFILE.PIN

Cross-Ref Filename : NIFNET.XRF

Layer Filename : LAYS.PCB

Output Filename : TESTFILE.PDF

Processing TESTFILE.PIN

Press ESC to abort

Figure 3-3. Sample Command Line Mode Screen

NI-FNET displays progress reports and error messages, if any, as it processes the files. When processing is complete, the system returns you to DOS.

BATCH MODE

Batch mode is a variation of command line mode where the input file is a batch control file (<filename>.BCF) containing several command lines to be run consecutively without user intervention.

Use a text editor to create the batch control file. You must use the .BCF filename extension when naming the batch control file.

Each line of the batch control file has the same format as a command line without the NIFNET command. Each line contains the name of an input pin list file and can include a cross-reference filename, a layer structure filename, and/or an output filename.

Figure 3-4 shows a sample batch control file.

```
TEST1 * OUTLINE
TEST2
TEST3 TEST3.XRF * NEW.PDF
TEST4 * * TEST4.ASC
```

Figure 3-4. Sample Batch Control File

To run the batch control file, at the DOS prompt, type:

NIFNET @<filename> [Return]

You do not need to enter the extension; NI-FNET assumes it is .BCF. You can leave a space between the @ and the filename, but the space is not required.

NI-FNET displays the Title Screen, then the Command Line Mode Screen. NI-FNET processes the files sequentially, displaying each set of filenames in turn, and displays progress reports and error messages, if any. When processing is complete, the system returns you to DOS.

Batch mode produces all the output files specified in the batch control file.

CREATING THE PCB DATABASE

After you translate a pin list into PDIF format, you must input the PDIF file into PDIF-IN to translate it into a PC-CARDS PCB database. Use the procedure given in the "Running PDIF-IN" section of the *PDIF User's Manual*.

The new PCB database will have the components arranged on a grid as specified with the GRID <x y> or NOGRID option in the cross-reference file. It will contain network connectivity ("ratsnest" connections) but no physical traces. Use PC-CARDS or PC-PLACE to examine and edit the design.



CHAPTER 4. VIEWING THE OUTPUT FILES

NI-FNET has three types of output files, the PDIF file (<filename>.PDF), the alias list file (<filename>.ALS), and the log file (NIFNET.LOG), which contains error and information messages displayed during the generation of the other output files.

This chapter describes how to view and print the output files and describes the output PDIF file and alias list file.

Error messages that might be in the log file are listed and explained in Appendix A.

VIEWING AND PRINTING FILES

To display or print an output file, use the DOS TYPE and [Ctrl]-[P] commands. For example, to display a file, type:

```
TYPE TESTFILE.LOG [Return]
```

Or, to print a file, type:

```
TYPE TESTFILE.LOG [Ctrl]-[P] [Return]
```

When the file is printed, press [Ctrl]-[P] again to turn off the print function.

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

THE PDIF FILE

The PDIF file output by NI-FNET is an ASCII file in a format consistent with P-CAD's standard PDIF format. This section explains the structure of the NI-FNET output PDIF file and tells where NI-FNET finds this information in the pin list. For further information about PDIF file structure and contents, refer to the PDIF User's Manual.

The PDIF file output by NI-FNET consists of the P-CAD header, which identifies the file, a two-line heading, and a series of records that define the parts in the design.

The format of the heading is shown below.

```
{COMPONENT <dbf>  
  {ENVIRONMENT <brd> {DETAIL {SUBCOMP
```

where:

dbf is the P-CAD PCB database filename. For this filename, NI-FNET uses the output PDIF filename you specified on the NI-FNET Program Screen with the .PCB extension.

brd is the layer structure or board outline filename you specified on the NI-FNET Program Screen when translating the pin list.

After the heading, the PDIF file contains a record for each part in the design in the format shown below.

```
{I <part> <refdes>
  {CN <pinname> <net> <pinname> <net> . . . }
  {ATR {IN {P1 <x> <y>} } }
}
```

where:

part is the P-CAD part name, taken from the cross-reference file. If the name is not in the cross-reference file, NI-FNET uses the FutureNet part name with the .PRT extension.

refdes is the reference designator of this instance of the part, taken from the CIRCUIT DESIGNATOR attribute in the pin list file. If this attribute value ends with an alphabetic section name (for example, U1A), NI-FNET omits the letter. For example, if the attribute is U1A, the reference designator is U1.

pinname is the pin name taken from the PIN entry in the pin list file. The pin name and corresponding net name are listed for each pin in the part. If a pin name in the pin list file is in the format **An**, where **A** is an alphabetical prefix and **n** is a number, NI-FNET omits the letter. For example, if the pin name in the pin list is A2, the pin name in the PDIF file is 2. (In the pin list file, this pin name format can be used only for a part that has only one gate per package.)

net is the net to which the pin is connected, taken from the signal name field of the PIN entry in the pin list file or assigned by NI-FNET as an alias. If a signal is unnamed in the pin list file and has a

value of *****XXXYYY**, the net value is ****XXXYYY**. If the signal is connected to a bus, the net value is **<bus_name\$signal_name>**. If the net value would be longer than eight characters, the net value is an alias assigned by NI-FNET (refer to the next section, "The Alias List File").

x and y are the horizontal (x) and vertical (y) coordinates of the part in the PC-CARDS database. These values are present only if the NOGRID option was not specified in the cross-reference file. If the NOGRID option was specified, the entire **{ATR {IN {PI <x> <y> } } }** section is not present.

After the last part entry, the PDIF file ends with three ending braces:

```
}}}
```

THE ALIAS LIST FILE

The alias list file is an ASCII file that cross-references pin list file signal names with the aliases assigned by NI-FNET. You can use this file when comparing the input pin list file with the output PDIF file.

An alias is assigned to a signal name only if the signal name is longer than the P-CAD limit of eight characters or contains any of the FutureNet special characters.

NOTE: For a signal connected to a bus, NI-FNET assigns a signal name in the format **<bus_name>\$<signal_name>**. If this concatenated name is longer than eight characters, NI-FNET assigns an alias.

The following sections describe the format of the alias list file and the conventions NI-FNET uses for aliasing.

Alias List File Format

The alias list file contains one record for each net name that was aliased in the format:

BUSS=<bus> SIGNAL=<signal> ALIAS=<alias>

where:

BUSS=bus is present only if the signal is connected to a bus. The **bus** value is the name of the bus in the pin list file.

signal is the signal name in the pin list file.

alias is the new net name assigned by NI-FNET.

Signal Aliasing Conventions

NI-FNET uses the following conventions when assigning aliases.

Signals Connected to Busses

When aliasing a signal that is connected to a bus, NI-FNET uses one of two conventions, depending on the length of the FutureNet signal name, as described below.

Signal Name Three Characters or Less. NI-FNET assigns a net name in the format:

BB<busnum>\${<signame>

where:

busnum is a two-digit number identifying the bus, assigned sequentially starting with 00.

signame is the signal name. If this name is shorter than three characters, it is padded to the left with zeroes.

Examples of this type of alias are BB00\$006 and BB01\$0AA.

Signal Name Longer Than Three Characters. NI-FNET assigns a net name in the format:

BB<busnum><signum>

where:

busnum is a two-digit number identifying the bus, assigned sequentially starting with 00.

signum is a four-digit number identifying the signal, assigned sequentially starting with 0000.

Examples of this type of alias are BB000006 and BB010100.

Signals Not Connected to Busses

When aliasing a signal that is not connected to a bus, NI-FNET uses the format:

SS<signum>

where:

signum is a six-digit number identifying the signal, assigned sequentially starting with 000000.

Examples of this type of alias are SS000006 and SS000100.

Signal Names With Special Characters

If NI-FNET finds any of the FutureNet special characters (%, [, { ,], and }) in a signal or bus name, it replaces the special character with the substitute character specified in the CFG section of the cross-reference file or the default substitute character.



APPENDIX A. ERROR MESSAGES

NI-FNET displays error messages on the screen and stores them in the log file (NIFNET.LOG), which you can view or print.

This appendix lists and explains NI-FNET error messages. Errors are grouped into two types:

- File access and system errors
- Pin list translation errors

Within each group, error messages are listed in alphabetical order.

FILE ACCESS AND SYSTEM ERRORS

Errors in this group are due to file access problems. You cannot run the program until the error is corrected.

Message: Batch file does not exist

Cause: NI-FNET is unable to find the batch control file you specified.

Action: Make sure you enter the correct filename, using a drive designator or directory path if necessary. Make sure the file has the .BCF extension.

Message: Cannot open <filename>

Cause: NI-FNET was unable to find the specified file.

Action: Make sure you enter the correct filename, using a drive designator or directory path if necessary.

**Message: Memory deallocation error <xxx:nnn>
Program aborted**

Cause: The system may have insufficient memory. NI-FNET requires 640K.

Action: Be sure you have 640K. Use the DOS CHKDSK command to check the system if necessary. Reboot your system.

**Message: Memory allocation error
Program aborted**

Cause: The system may have insufficient memory. NI-FNET requires 640K.

Action: Be sure you have 640K. Use the DOS CHKDSK command to check the system if necessary. Reboot your system.

PIN LIST TRANSLATION ERRORS

The following error messages are produced when NI-FNET encounters incorrect data in the input file.

**Message: Hierarchical design detected <filename>
Processing terminated**

Cause: The design you are inputting to NI-FNET is hierarchical.

Action: Input only flat designs.

**Message: Incompatible pin list format <filename>
Processing terminated**

Cause: The input pin list is in a format that is not compatible with FutureNet's DASH-2 or DASH-3C pin list file.

Action: Be sure you input the correct file.

Message: Invalid common pin <n> for part <partname>

Cause: NI-FNET found a common pin in the specified part that was not specified as allowable in the cross-reference file. NI-FNET processes the part as if the common pin were allowable.

Action: Check to see if the components are meant to have the common pin. If so, change the cross-reference file; if not, correct your FutureNet design and rerun NI-FNET.

**Message: Maximum # of busses (100) exceeded
<busname> bus cannot be processed**

**Cause: The design contains more than 100 busses.
NI-FNET processes only the first 100 busses.**

**Action: Limit the number of busses to 100 in the
schematic database, then rerun NI-FNET.**

APPENDIX B. SAMPLE FILES

This appendix explains and shows the sample files supplied with NI-FNET. The sample files are listed below.

NIFNDEMO.XRF	Cross-reference file
NIFNDEMO.PDF	PDIF file
NIFNDEMO.ALS	Alias list file
LAYS.PCB	Layer structure file
2PIN.PRT	Part file
74LS10.PRT	Part file
8088.PRT	Part file
8185.PRT	Part file
8284A.PRT	Part file
8755A-2.PRT	Part file
CK06.PRT	Part file
D07.PRT	Part file
RC07.PRT	Part file
XTAL.PRT	Part file

The NIFNDEMO PDIF file and alias list file were produced by NI-FNET from a pin list produced using the FutureNet system. The NIFNDEMO cross-reference file and the LAYS.PCB layer structure filename were also used to produce these files. The part files are all the P-CAD parts used in the design.

You can examine the NIFNDEMO cross-reference file, PDIF file, and alias list file as an aid to understanding the structure of these files.

You can also use PDIF-IN to translate the PDIF file into a PC-CARDS PCB database, then use PC-CARDS to examine the new PCB database. (The sample layer structure file and the sample part files are required to input the PDIF file to PDIF-IN.)

The next sections show the NIFNDEMO sample files.

CROSS-REFERENCE FILE: NIFNDEMO.XRF

CFG
UNCOMMIT
CHARSUB [&
CHARSUB] #
ENDCFG

PARTS
8088 * A2, A3, A4
8284A *
74LS10 * P/G 7,14
1UF CK06
1N914 D07
8085 P/G 20,40
ENDP

DEFAULTS
R RC07
SW 2PIN
Y XTAL
ENDD

PDF FILE: NIFNDEMO.PDF

```
*****
%
%                               NI-FNET ASCII NETLIST
%
% NI-FNET Version 1.30
% Copyright (C) 1986 - Personal CAD Systems, Inc.
%
% Date       : MAR 13 1986
% Time       : 02:06 PM
% File In    : NIFNDEMO.PIN
% File Out   : NIFNDEMO.PDF
% Log File   : NIFNET.LOG
%
%*****
```

{COMPONENT NIFNDEMO.PCB
{ENVIRONMENT LAYS.PCB} {DETAIL {SUBCOMP

```
{I XTAL.PRT Y1
  {CN 1 **024022 2 **051028 }
  {ATR {IN {PL 0 0} } }
}
{I RC07.PRT R4
  {CN 1 +5V 2 **057049 }
  {ATR {IN {PL 600 0} } }
}
{I RC07.PRT R3
  {CN 1 +5V 2 **040053 }
  {ATR {IN {PL 1200 0} } }
}
{I RC07.PRT R2
  {CN 1 **051028 2 GND }
  {ATR {IN {PL 1800 0} } }
}
{I RC07.PRT R1
  {CN 1 **024022 2 GND }
  {ATR {IN {PL 2400 0} } }
}
{I 8755A-2.PRT U3
  {CN 1 **140166 2 BB01$019 3 **092025 4 **092055
    5 +5V 6 **057049 7 10/M 8 +5V 9 RD-
    10 WR- 11 ALE 12 BB00$000 13 BB00$001 14 BB00$002
    15 BB00$003 16 BB00$004 17 BB00$005 18 BB00$006 19 BB00$007
    20 GND 21 BB01$008 22 BB01$009 23 BB01$010 24 I/OA$0
    25 I/OA$1 26 I/OA$2 27 I/OA$3 28 I/OA$4 29 I/OA$5
    30 I/OA$6 31 I/OA$7 32 I/OB$0 33 I/OB$1 34 I/OB$2
    35 I/OB$3 36 I/OB$4 37 I/OB$5 38 I/OB$6 39 I/OB$7
    40 +5V }
  {ATR {IN {PL 3000 0} } }
}
```

```

(I 8185.PRT U5
(CN 1 BB00$000 2 BB00$001 3 BB00$002 4 BB00$003
  5 BB00$004 6 BB00$005 7 BB00$006 8 BB00$007 9 GND
  10 BB01$008 11 BB01$009 12 BB01$012 13 BB01$011 14 IO/M
  15 ALE 16 WR- 17 RD- 18 +5V )
(ATR (IN (PL 3600 0) ) )
)
(I 8185.PRT U4
(CN 1 BB00$000 2 BB00$001 3 BB00$002 4 BB00$003
  5 BB00$004 6 BB00$005 7 BB00$006 8 BB00$007 9 GND
  10 BB01$008 11 BB01$009 12 BB01$011 13 BB01$012 14 IO/M
  15 ALE 16 WR- 17 RD- 18 +5V )
(ATR (IN (PL 4200 0) ) )
)
(I CK06.PRT C1
(CN 1 **040053 2 GND )
(ATR (IN (PL 0 1200) ) )
)
(I 8088.PRT U2
(CN 1 GND 2 BB01$014 3 BB01$013 4 BB01$012
  5 BB01$011 6 BB01$010 7 BB01$009 8 BB01$008 9 BB00$007
  10 BB00$006 11 BB00$005 12 BB00$004 13 BB00$003 14 BB00$002
  15 BB00$001 16 BB00$000 17 GND 18 GND 19 **092025
  20 GND 21 **092055 22 **092052 23 GND 24 ?
  25 ALE 26 ? 27 ? 28 IO/M 29 WR-
  30 ? 31 GND 32 RD- 33 +5V 34 ?
  35 BB01$019 36 BB01$018 37 BB01$017 38 BB01$016 39 BB01$015
  40 +5V )
(ATR (IN (PL 600 1200) ) )
)
(I D07.PRT CR1
(CN 1 **040053 2 +5V )
(ATR (IN (PL 1200 1200) ) )
)
(I 8284A.PRT U1
(CN 1 GND 2 ? 3 GND 4 **057049
  5 **092052 6 ? 7 GND 8 **092025 9 GND
  10 **092055 11 **040053 12 ? 13 GND 14 ?
  15 ? 16 **051028 17 **024022 18 +5V )
(ATR (IN (PL 1800 1200) ) )
)
(I 2PIN.PRT SW1
(CN 1 **040053 2 GND )
(ATR (IN (PL 2400 1200) ) )
)
(I 74LS10.PRT U6
(CN 1 BB01$018 2 BB01$019 3 ? 4 ?
  5 ? 6 ? 7 ? 8 ? 9 ?
  10 ? 11 ? 12 **140166 13 BB01$017 14 ? )
(ATR (IN (PL 3000 1200) ) )
)
)))

```

ALIAS LIST FILE: NIFNDEMO.ALS

```
** *****  
** *  
** *                               NI-FNET ALIAS LIST                               *  
** *  
** * NI-FNET Version 1.30 *  
** * Copyright (C) 1986 - Personal CAD Systems, Inc. *  
** *  
** * Date      : MAR 13 1986 *  
** * Time      : 02:06 PM *  
** * File In   : NIFNDEMO.PIN *  
** * File Out  : NIFNDEMO.PDF *  
** * Log File  : NIFNET.LOG *  
** *  
** *****  
BUSS = ADR/DATA  SIGNAL = 0  ALIAS = BB00$000  
BUSS = ADR/DATA  SIGNAL = 1  ALIAS = BB00$001  
BUSS = ADR/DATA  SIGNAL = 2  ALIAS = BB00$002  
BUSS = ADR/DATA  SIGNAL = 3  ALIAS = BB00$003  
BUSS = ADR/DATA  SIGNAL = 4  ALIAS = BB00$004  
BUSS = ADR/DATA  SIGNAL = 5  ALIAS = BB00$005  
BUSS = ADR/DATA  SIGNAL = 6  ALIAS = BB00$006  
BUSS = ADR/DATA  SIGNAL = 7  ALIAS = BB00$007  
BUSS = ADDRESS   SIGNAL = 8  ALIAS = BB01$008  
BUSS = ADDRESS   SIGNAL = 9  ALIAS = BB01$009  
BUSS = ADDRESS   SIGNAL = 11  ALIAS = BB01$011  
BUSS = ADDRESS   SIGNAL = 12  ALIAS = BB01$012  
BUSS = ADDRESS   SIGNAL = 14  ALIAS = BB01$014  
BUSS = ADDRESS   SIGNAL = 13  ALIAS = BB01$013  
BUSS = ADDRESS   SIGNAL = 10  ALIAS = BB01$010  
BUSS = ADDRESS   SIGNAL = 19  ALIAS = BB01$019  
BUSS = ADDRESS   SIGNAL = 18  ALIAS = BB01$018  
BUSS = ADDRESS   SIGNAL = 17  ALIAS = BB01$017  
BUSS = ADDRESS   SIGNAL = 16  ALIAS = BB01$016  
BUSS = ADDRESS   SIGNAL = 15  ALIAS = BB01$015
```