NX-CALAY
User's Manual

October 1986

p·cad ®

PERSONAL CAD SYSTEMS INC.

ABOUT THIS MANUAL

This manual describes NX-CALAY, a program that converts P-CAD schematic database files to Calay-format netlists.

Chapter 1, INTRODUCTION, provides an overview of NX-CALAY and installation instructions.

Chapter 2, PREPARING THE SCHEMATIC DATABASE, gives instructions for using PC-CAPS to create a schematic database to be used with NX-CALAY.

Chapter 3, PREPARING THE INPUT FILES, gives instructions for creating the files to be input into NX-CALAY.

Chapter 4, USING NX-CALAY, gives instructions for running NX-CALAY.

Chapter 5, VIEWING THE OUTPUT FILES, tells how to view, print, and interpret the NX-CALAY output files.

Appendix A, ERROR MESSAGES, describes error messages that may be produced during NX-CALAY program operation.

iv

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 that you must replace with the filename you intend to use. For example:

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

CONTENTS

CONTENTS (Continued)

FIGURES

TABLES

CHAPTER 1. INTRODUCTION

The NX-CALAY interface program translates P-CAD circuit schematics to ASCII-format files that are compatible with the Calay printed circuit board (PCB) design system.

NX-CALAY packages P-CAD schematic symbols into Calay PCB parts and translates circuit connectivity information to the Calay format. It does not support transfer of graphics data.

This chapter gives an overview of NX-CALAY and installation instructions.

OVERVIEW

NX-CALAY, together with PC-NODES and (if needed) PC-LINK, enables you to produce files in the format of the Calay netlist file from a schematic created using PC-CAPS.

There are six stages in this process:

1. Using PC-CAPS, you prepare the schematic design. This design consists of one or more schematic files (<filename>.SCH), which are created by interconnecting basic schematic symbols (<filename>.SYM).

    NOTE: NX-CALAY does not support hierarchical structure. For further explanation, refer to Chapter 2, "Preparing the Schematic Database."

2. Using PC-NODES, you extract a netlist
    (<filename>.NLT) from each schematic file. The
    netlist file contains component and interconnection
    information for the schematic. If a design consists
    of a single schematic, you can input the
    corresponding netlist file directly to NX-CALAY.
    In this case, you do not need to perform step 3.

3. If your design consists of multiple sheets, after you
    extract the netlist for each separate sheet, you must
    use PC-LINK to link all the netlists into a single
    expanded netlist file (<filename>.XNL).

4. If necessary, you use a text editing program to
    create or modify the library file (<filename>.LBR)
    to specify reference designator prefixes for the
    PC-CAPS schematic symbols. The library file is
    described in Chapter 3, "Preparing the Input Files."

5. You then input the netlist or expanded netlist file
    to NX-CALAY. NX-CALAY uses this netlist with
    the library file to produce three output files, which
    are described below:

    •   The **Calay netlist file** (<filename>.NET) contains
        Calay PCB design information in an unsorted
        format. This file is described in detail in
        Chapter 5, "Viewing the Output Files."

    •   The **back-annotation command file**
        (<filename>.CMD) contains initial
        back-annotation information. You can input this
        file to PC-CAPS to update the original schematic.

    •   The **log file** (NXCALAY.LOG) contains all
        messages displayed during the execution of
        NX-CALAY. This file is useful for examining

any error messages that are reported by
NX-CALAY.

NOTE: Each time you run NX-CALAY, the
previous log file is overwritten. If you want to
save a log file, you must rename it or print it out.

6. To sort the Calay netlist file into its format for
input to the Calay system, you use the DOS SORT
command.

Figure 1-1 shows the process described above.
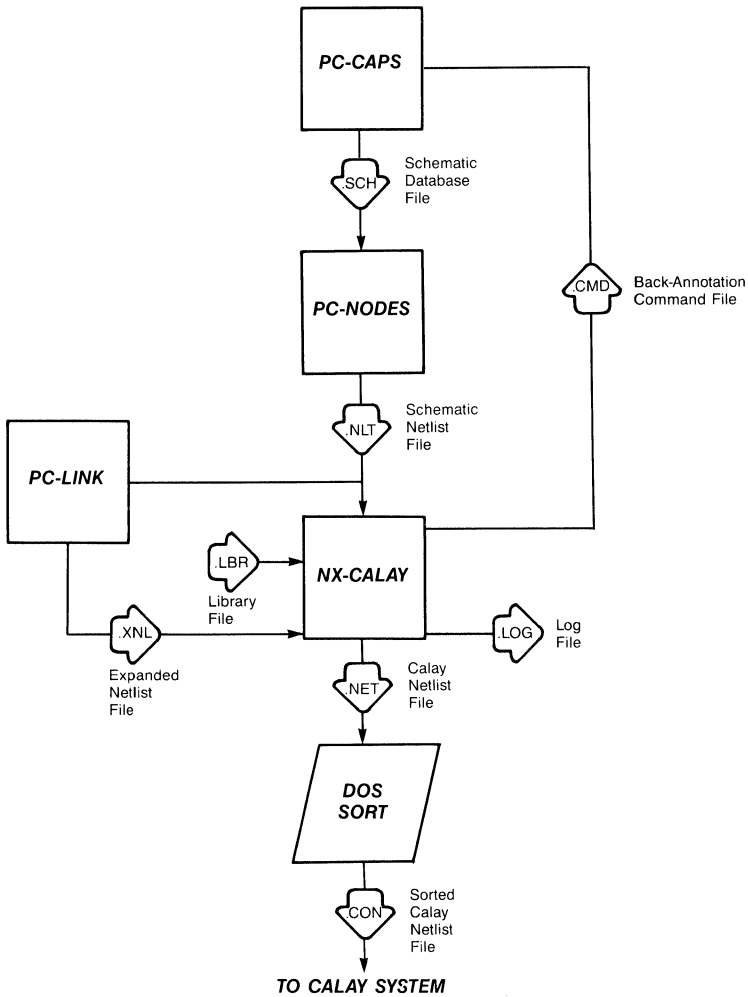
Figure 1-1. NX-CALAY Input and Output

SYSTEM REQUIREMENTS

Before you install NX-CALAY, your computer system must have the following minimum configuration:

- IBM PC/AT, PC/XT, HP Vector, TI Business Pro, TI Professional, 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

INSTALLATION

Your NX-CALAY diskette contains the following files:

Program file:

NXCALAY.EXE

Sample data files:

CSHEET1.SCH
CSHEET1.NLT
CSHEET2.SCH
CSHEET2.NLT
CALAY.XNL
CALAY.NET
CALAY.CMD

You can use the sample data to see how the program works. "Sample Calay Netlist File" in Chapter 5 provides more information about the sample data.

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

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

CD \PCAD\EXE [Return]

Copy the file 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:*.SCH [Return]
COPY A:*.NLT [Return]
COPY A:CALAY.* [Return]

CHAPTER 2. PREPARING THE SCHEMATIC DATABASE

Before using NX-CALAY, you must use PC-CAPS to create a schematic database. This chapter describes PC-CAPS schematic structure and gives guidelines for entering the special information required to construct a Calay-compatible schematic. You should read this chapter before creating your schematic to ensure that your database is compatible with the Calay system.

SCHEMATIC DATABASE STRUCTURE

A PC-CAPS schematic consists of an interconnected group of predefined symbols. These symbols can be user-defined or can be part of a P-CAD supplied library. Each symbol is a logical and pictorial representation of a component. For example, a 7400 symbol is a single 2-input NAND gate.

Symbols are created in PC-CAPS SYMB mode as files with the .SYM filename extension. You assemble symbols into a schematic in DETL mode, producing a schematic database file with the .SCH filename extension.

NX-CALAY does not support hierarchical structure. It requires "flat" design structure, which consists of symbols with no DETL data placed directly on the final schematic. Because of this, you should not use hierarchical design in a schematic if you intend to input it to NX-CALAY.

NX-CALAY does support multisheet structure, which you can use for a schematic that requires more than one file.

A multisheet schematic is created by producing a number of single-sheet schematics. Connections between sheets are made by giving connected wires the same net name on each sheet. When you use PC-LINK, it merges together wires on different sheets that have the same name.

To indicate that a given schematic is one sheet of a multisheet schematic, assign the SHEET attribute to the sheet. In PC-CAPS SYMB mode, use the ATTR/ACOM command and enter, anywhere in the design, the following attribute:

SHEET=<sheet id>

where:

sheet id is one to three characters and is typically a number, with the first sheet being sheet 1 and so on. Sheet id's in a multisheet schematic must be unique.

SYMBOLS

You can use any PC-CAPS symbol in a schematic to be processed by NX-CALAY. If the Calay device name of the part into which the component is to be packaged is different from the P-CAD symbol filename, you must use the PC-CAPS ATTR/ACOM command to assign the SNAME attribute to the symbol. This attribute is described below.

SNAME - Specifies the device name, in the format:

SNAME=<name>

where:

>   **name** is the name of the Calay part that the
>   component is to be packaged into.

For example, you might specify SNAME=L1022 for a
component.

The SNAME attribute is required for all symbols to be
packaged into nonhomogeneous parts. The next section
describes homogeneous and nonhomogeneous parts.

COMPONENT TYPES

NX-CALAY automatically packages PC-CAPS schematic
symbols into PCB parts. It supports four types of
components. To use some of these component types, you
must use the PC-CAPS ATTR/ACOM command to
assign special attributes to the PC-CAPS symbol. The
component types and their identifying attributes are
described in the following sections.

Components of Homogeneous Parts

A homogeneous part is composed of only one type of
gate. For example, a 7400 part is made up of four
2-input NAND gates. NX-CALAY requires no special
attributes for components that are going to be packaged
into homogeneous parts.

Components of Heterogeneous Parts

A heterogeneous part is composed of more than one type
of gate. For example, Figure 2-1 shows the 74LS265
part, which consists of two 2-input NAND/AND gates
and two buffer/inverter gates.
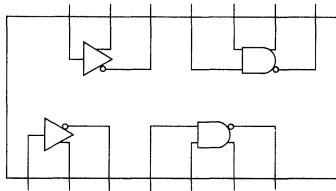
Figure 2-1. Heterogeneous Part

For each component that is going to be packaged into a heterogeneous part, you must assign the SNAME attribute, which is described above, and the MIX attribute, which is described below.

MIX - Specifies that the component is for a heterogeneous part or a part with a common pin. This attribute is assigned in the format:

    MIX=<num>/<section>

where:

   num is the number of gates on the part.

   section is the section of the part where the component is located.

For example, MIX=4/A specifies that the symbol is section A of a four-gate part.

You must also preassign a reference designator and explicit package pin numbers to each symbol to be used in a heterogeneous part. Refer to the "Packaging Preassignment" section below.

Components of Parts With a Common Pin

In some parts, two or more gates share one or more common pins. For example, Figure 2-2 shows the 5477 part, which consists of four transmission gates and in which two gates share one pin and the two other gates share another pin.
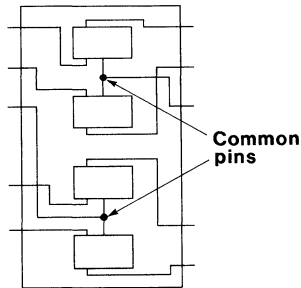
Figure 2-2. Part With Common Pins

For each component to be packaged into a part that has common pins, you must assign the SNAME and MIX attributes, which are described above, and the COMM attribute, which is described below.

COMM - Lists the common pins for the physical part, in the format:

COMM=(<pnum> <pnum> ...)

where:

> **pnum** is a pin number of a common pin for the part containing this component. All the common pins must be listed, with a space between pins. The parentheses are required only if the part has more than one common pin.

For example, you might specify COMM=1 for a part with one common pin and COMM=(1 6 9) for a part with three common pins.

You must also preassign a reference designator and explicit package pin numbers to each symbol to be used in a part that has common pins. Refer to "Packaging Preassignment" below.

Connectors

You can represent a single connector by one schematic symbol or by several symbols.

To represent a connector with a single schematic symbol, create the symbol in the same way you create a logical symbol.

To represent a connector with several symbols, create a separate symbol file for each unique symbol. You can represent a connector with as many or as few symbols as your application requires.

Figure 2-3 shows 1-pin and 6-pin connector symbols.

CONN1.SYM   CONN6.SYM
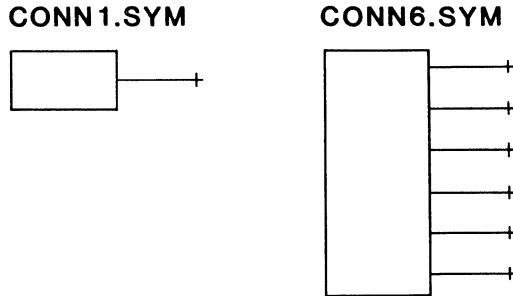
Figure 2-3. Connector Symbols

When placing a connector on a schematic, you must assign two attributes to each symbol in the connector. These attributes are the SNAME attribute described above and the CONN attribute, which is described below.

CONN - identifies the component as a connector symbol and specifies the number of pins in the entire physical connector part, in the format:

    CONN=<n>

where:

    n is the number of pins in the entire connector.

You must also preassign a reference designator and explicit package pin numbers to each connector symbol. Refer to "Packaging Preassignment" below.

PACKAGING PREASSIGNMENT

NX-CALAY packages the symbols on the PC-CAPS
schematic into physical parts. For homogeneous parts,
this process is entirely automatic. For heterogenous
parts, parts with common pins, and connectors, you must
preassign packaging information. You can also
preassign packaging information in cases where you
want to preassign several logical symbols to the same
physical package or to assign a symbol to a specific gate
in the package.

Preassign these components using the two PC-CAPS
commands described below. See the *PC-CAPS User's
Manual* for more information.

SCMD/PNLC (system command/pin number location) is
used when defining basic symbols in SYMB mode. It
lets you set the number of gates that are to be assigned
to each physical package and the pin numbers which
correspond to each gate location. It also allows you to
specify locations for the reference designator and pin
numbers. All symbols used in the schematic, except
connector symbols, must have this information assigned.
All P-CAD library symbols include this information.

SCMD/PNUM (system command/pin number) is used in
DETL mode to preassign a reference designator to a
symbol in a schematic. You can use SCMD/PNUM only
if SCMD/PNLC was used for the symbol in SYMB mode.

To conform to Calay conventions, reference designators
can be no longer than six characters and can contain
only letters, numbers, and the + and - characters.

The SCMD/PNUM command also allows you to assign
package pin numbers to the symbol, by assigning either
a section or actual pin numbers.

For example, if you specify U3/A, U3 is the reference designator and the pins will be numbered as defined using the SCMD/PNLC command when the symbol was created.

NX-CALAY requires explicit pin numbers to be assigned to each symbol to be packaged into a heterogenous part or a part with a common pin and to each connector symbol. To assign pin numbers explicitly, use the format shown in the example below:

    U3(2 4 6 8 10)

For example, if you use three 3-pin connector symbols to represent a 9-pin connector, you would use the SCMD/PNUM command to assign the following information to the symbols:

    P3(1 2 3)
    P3(4 5 6)
    P3(7 8 9)

Figure 2-4 shows pin number assignments for three symbols that constitute a 10-pin connector.

P3(9 10)

P3(6 7 8)

P3(1 2 3 4 5)
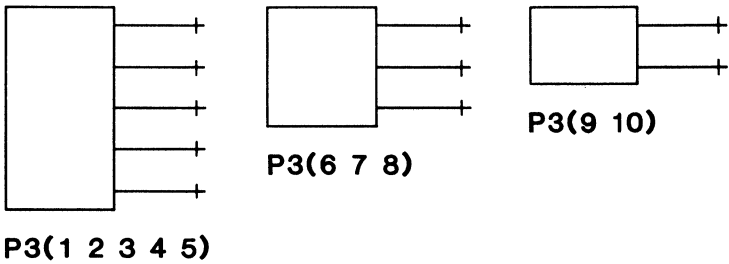
Figure 2-4. Connector Pin Number Preassignment

SIGNAL NAMES

To follow Calay conventions, signal names can be no
longer than six characters and can contain only letters,
numbers, and the + and - characters.

You can name signals using the NAME/NET command
in PC-CAPS DETL mode. In PC-CAPS, an apostrophe in
a signal name signifies that the previous character is to
have a bar over it. NX-CALAY substitutes the -
character for the apostrophe.

For unnamed signals, PC-NODES assigns a name in the
format UNsssnnn, where sss is the value of the SHEET
attribute assigned to the page (000 if the attribute was
not assigned) and nnn is an identifying number assigned
sequentially starting with 000. To conform to the Calay
six-character limit, NX-CALAY removes the UN prefix
from these names.

When naming signals, be sure not to use any names that might be the same as names assigned by NX-CALAY.

NX-CALAY reports an error if a signal name is the same as the reference designator of a component to which the signal is connected.

SUMMARY OF SCHEMATIC REQUIREMENTS

This section summarizes the PC-CAPS attributes and packaging commands that NX-CALAY requires for schematics.

You assign an attribute by using the ATTR/ACOM command and entering:

    <keyword>=<value>

where:

keyword is the name of the attribute.

value is the value you specify for it. If the value contains embedded blanks, you must enclose it in quotation marks or parentheses.

With the exception of the SHEET attribute, which is assigned to an entire schematic, all the attributes used by NX-CALAY can be assigned to a symbol either when it is created in SYMB mode or when it is placed on the schematic in DETL mode.

For each attribute or command, Table 2-1 describes the
function and shows which symbols require this
information. The following codes are used for symbols:

   mix -   symbol to be packaged into a heterogeneous
              part

   comm - symbol to be packaged into a part with
              common pins

   conn - connector symbol

Table 2-1. Schematic Requirements Summary

Attribute or Command	Function	Required For
Attributes:		
COMM	Identifies symbol used in common-pin part and gives pin number of common pin(s)	comm
CONN	Identifies connector symbol and gives number of gates for the entire connector	conn
MIX	Identifies symbol used in heterogeneous or common-pin part, and gives number of gates in package and section number of symbol	mix, comm
SHEET	Identifies multiple sheets	multisheet design
SNAME	Device type (to be used instead of symbol filename)	mix, comm, conn [1]
Commands:		
SCMD/ PNLC	Gates per package, pins per gate, reference designator location	all symbols
SCMD/ PNUM	Package pin numbers, reference designator	mix, comm, conn [1,2]

1.   Can be used for other symbols as needed.
2.   Package pin numbers must be explicit for connectors and symbols used in heterogeneous and common-pin parts.

CHAPTER 3. PREPARING THE INPUT FILES

This chapter describes how to prepare the files that are to be input to NX-CALAY.

After your schematic is completed, you must use PC-NODES to extract the schematic netlist(s). If your schematic consists of just one schematic file, you can input the netlist directly into NX-CALAY. If the schematic has multiple sheets, you must use PC-LINK to combine the netlists into one expanded netlist file.

You can also create a library file to indicate the reference designator prefixes to be used for the P-CAD symbols in the design.

The following sections contain instructions for creating and expanding the schematic netlist and for creating the library file.

EXTRACTING AND EXPANDING THE SCHEMATIC NETLIST

Run PC-NODES to extract schematic netlists from each sheet of your schematic. Use the instructions in the *PC-NODES User's Manual.*

If your schematic has multiple sheets, run PC-LINK to link together the netlists. Use the instructions in the *PC-LINK User's Manual.*

> NOTE: As explained in "Schematic Database Structure" in Chapter 2, NX-CALAY does not support hierarchical structure. If your schematic does use hierarchical symbols, you can extract a netlist compatible with NX-CALAY.

However, this netlist will include only the symbols
and not the subcircuits that they represent. You
might use this procedure if you want to use a
subcircuit that is hierarchical (that is, it has both
DETL and SYMB representations) and you do not
require detail information of the subcircuit for
Calay.

First, use PC-NODES to extract the netlist for the
schematic. Next, ensure that the netlist files for the
hierarchical symbols are not present in any
directory in the library path that PC-LINK
searches. Then run PC-LINK. PC-LINK reports the
missing netlists as errors, but produces the
expanded netlist, which includes the desired
symbols without the subcircuits.

CREATING THE LIBRARY FILE

The library file is an ASCII file that you can use to tell
NX-CALAY the prefixes to use when it assigns
reference designators to unpreassigned PC-CAPS
schematic symbols.

You can create the library file using a text editing
program. This file must be named NXCALAY.LBR.

The format of the library file is shown below.

    PARTS

    <filename> <prefix>
    <filename> <prefix>
    .
    .
    .
    <filename> <prefix>

    ENDP

where:

    PARTS is a required keyword that signifies the
    beginning of the file.

    filename is the filename of the P-CAD symbol. If a
    filename extension is not entered, NX-CALAY
    assumes the extension is .SYM.

    prefix is the prefix to be used for reference
    designators assigned by NX-CALAY. If a reference
    designator is preassigned in PC-CAPS, the prefix is
    not used.

    ENDP is a required keyword that signifies the end
    of the file.

The filename and prefix for each component must be on
one line. Any text before the PARTS keyword, after
the ENDP keyword or to the right of a prefix is
ignored. In addition, all text to the right of a percent
sign (%) is considered a comment and is ignored.

The default prefix is U. If a component is not listed in
the library file, NX-CALAY uses the letter U as its
default reference designator prefix.

CHAPTER 4. USING NX-CALAY

This chapter describes the required conditions and procedures for running NX-CALAY.

Before running NX-CALAY, be sure that:

● Your system is correctly configured.

● You have installed the NX-CALAY program file (NXCALAY.EXE).

● You have assembled the schematic circuit.

● You have extracted and, if necessary, linked the netlist files.

● You have prepared the library file if necessary.

NX-CALAY has two operating modes: interactive and command line. In interactive mode, NX-CALAY prompts you for input and output filenames. In command line mode, you specify filenames on the command input line when you start the program, and NX-CALAY processes the input file automatically.

NX-CALAY outputs the Calay-compatible netlist file and the back-annotation command file. Before you input the output netlist into the Calay system, you must sort it. You can input the back-annotation command file into PC-CAPS to update the schematic with packaging information assigned by NX-CALAY.

The following sections describe how to run NX-CALAY using interactive mode and command line mode, how to sort the Calay-format netlist, and how to input the back-annotation command file into PC-CAPS.

INTERACTIVE MODE

To run NX-CALAY, be sure you are in the appropriate
project directory and follow the steps below.

1. Type:

     NXCALAY [Return]

   When the NX-CALAY Title Screen appears, press
   any key to continue.

   The system displays the NX-CALAY Program
   Screen and prompts for the input netlist filename as
   shown in Figure 4-1.

<div align="center">NX-CALAY</div>

Input net list   :   <filename>.XNL

Enter filename; press [Return] to accept, [Esc] to exit

Figure 4-1. NX-CALAY Program Screen

2. Type the filename of the netlist to be read by
    NX-CALAY and press [Return]. If you do not enter
    the filename extension, NX-CALAY adds the .XNL
    extension.

    The system prompts for the output Calay netlist
    filename. The default is the input netlist filename
    with the .NET extension.

3. Press [Return] to accept the default filename or
    type another filename and press [Return]. If you do
    not enter the filename extension, NX-CALAY adds
    the .NET extension.

    The system prompts for the output back-annotation
    command filename. The default is the input netlist
    filename with the .CMD extension as shown in
    Figure 4-2.

NX-CALAY

Input net list   : TESTFILE.XNL
Output net list   : TESTFILE.NET
Annotation File   : TESTFILE.CMD

Enter filename; press [Return] to accept, [Esc] to exit

Figure 4-2. Sample Program Screen

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

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

When processing is complete, the system returns you to
DOS.

COMMAND LINE MODE

Command line mode allows you to specify input and output filenames when you start NX-CALAY.

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

The format of the command line is:

NXCALAY <infile> <outfile> <cmd>

where:

NXCALAY initiates the program.

infile is the name of the input netlist file and is required. If you do not include the filename extension, NX-CALAY uses the .XNL extension.

outfile is the name of the Calay-format netlist file to be output. This name is optional. If you specify a name with no extension, NX-CALAY uses the .NET extension. If you do not specify a name, NX-CALAY uses the input netlist filename with the .NET extension.

cmd is the name of the back-annotation command file to be output. This name is optional and can be entered only if the output Calay-format filename is also entered on the command line. If you specify a name with no extension, NX-CALAY uses the .CMD extension. If you do not specify a name, NX-CALAY uses the input netlist filename with the .CMD extension.

Three examples of command lines are shown below.

        NXCALAY TESTFILE
        NXCALAY TESTFILE.NLT TEST
        NXCALAY TESTFILE TEST BACK

The first example causes NX-CALAY to create a Calay
netlist from the TESTFILE.XNL file. The output
netlist will have the default filename TESTFILE.NET
and the back-annotation file will have the default
filename TESTFILE.CMD.

The second example causes NX-CALAY to create a
Calay netlist from the TESTFILE.NLT file and give the
output netlist the filename TEST.NET. The
back-annotation command file will have the default
filename TESTFILE.CMD.

The third example causes to create a Calay netlist from
the TESTFILE.XNL file and give the output netlist the
filename TEST.NET and the back-annotation command
file the filename BACK.CMD.

After you enter the command line, the program runs
automatically with no further input from you. First,
the NX-CALAY Title Screen appears, followed by the
Program Screen.

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

SORTING A CALAY NETLIST

Before inputting the NX-CALAY output netlist into the Calay system, you must sort the file by signal name as required by Calay. Use the DOS SORT command as shown below:

SORT <<filename>.NET ><filename>.CON

where:

< signifies that the next filename is the file to be sorted.

filename.NET is the name of the NX-CALAY output file.

> signifies that the next filename is to be used for the sorted file.

filename.CON is the name of the sorted file.

For example, to sort the TESTFILE.NET file, you would type:

SORT <TESTFILE.NET >TESTFILE.CON

UPDATING A SCHEMATIC

You can input the back-annotation command file
produced by NX-CALAY into PC-CAPS to update the
schematic with packaging information assigned by
NX-CALAY. Follow the steps 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.ext> [Return]

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

CHAPTER 5. VIEWING THE OUTPUT FILES

NX-CALAY has three types of output files: the Calay-compatible netlist file (<filename>.NET), the back-annotation command file (<filename>.CMD), and the log file (NXCALAY.LOG).

All these files are ASCII files that you can examine as necessary.

This chapter describes how to view and print the output files and how to interpret the Calay netlist file.

The back-annotation command file is in standard P-CAD executable format, which is explained in the *PC-CAPS User's Manual.*

The log file contains error and information messages displayed during the generation of the other output files. Appendix A describes the error messages.

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.

NX-CALAY 5-2

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

THE CALAY NETLIST FILE

The Calay netlist file output by NX-CALAY consists of a series of records, one for each component pin. Each record is in the format:

    <signal> <loc> <pin> <device>

where:

> **signal** is the name of the net connected to the pin, assigned using the NAME/NET command in PC-CAPS DETL mode or assigned by PC-NODES and NX-CALAY as described in Chapter 2 in the section "Signal Names." For unconnected pins, a name of NONCON is used.

> **loc** is the reference designator, assigned using the SCMD/PNUM command in PC-CAPS DETL mode or assigned by NX-CALAY using the prefix specified in the library file (or the default prefix U) and an identifying number assigned sequentially starting with 1.

> **pin** is the physical pin number assigned using the SCMD/PNUM command in PC-CAPS DETL mode or assigned by NX-CALAY.

> **device** is the Calay device name, assigned using the SNAME attribute. If the attribute is not assigned, the symbol filename without the extension is used.

SAMPLE CALAY NETLIST FILE

This section shows the sample Calay netlist file that is included on the NX-CALAY diskette and the schematic files that the netlist describes. The netlist, CALAY.NET, was produced from the CALAY.XNL expanded netlist file, which was produced from a multisheet design composed of two schematic files, CSHEET1.SCH and CSHEET2.SCH.

Calay Netlist File: CALAY.NET

```
001006    P2       1    CONN
001007    P2       2    CONN
001008    P2       3    CONN
001009    P2       4    CONN
001015    P2       5    CONN
001014    P2       6    CONN
001013    P2       7    CONN
001012    P2       8    CONN
001011    P2       9    CONN
001010    P2      10    CONN
001000    P1       1    CONN
001002    P1       2    CONN
001003    P1       3    CONN
001004    P1       4    CONN
001005    P1       5    CONN
SET       P2      11    CONN
CLOCK     P2      12    CONN
001006    U5       1    NAND2
001007    U5       2    NAND2
N5        U5       3    NAND2
001008    U5       4    NAND2
001009    U5       5    NAND2
N6        U5       6    NAND2
001015    U5      10    NAND2
001014    U5       9    NAND2
RESET     U5       8    NAND2
001013    U5      12    NAND2
001012    U5      13    NAND2
SET       U5      11    NAND2
001011    U6       1    NAND2
001010    U6       2    NAND2
CLOCK     U6       3    NAND2
001001    U1      12    NAND2
001005    U1      13    NAND2
N4        U1      11    NAND2
001001    U1      10    NAND2
```
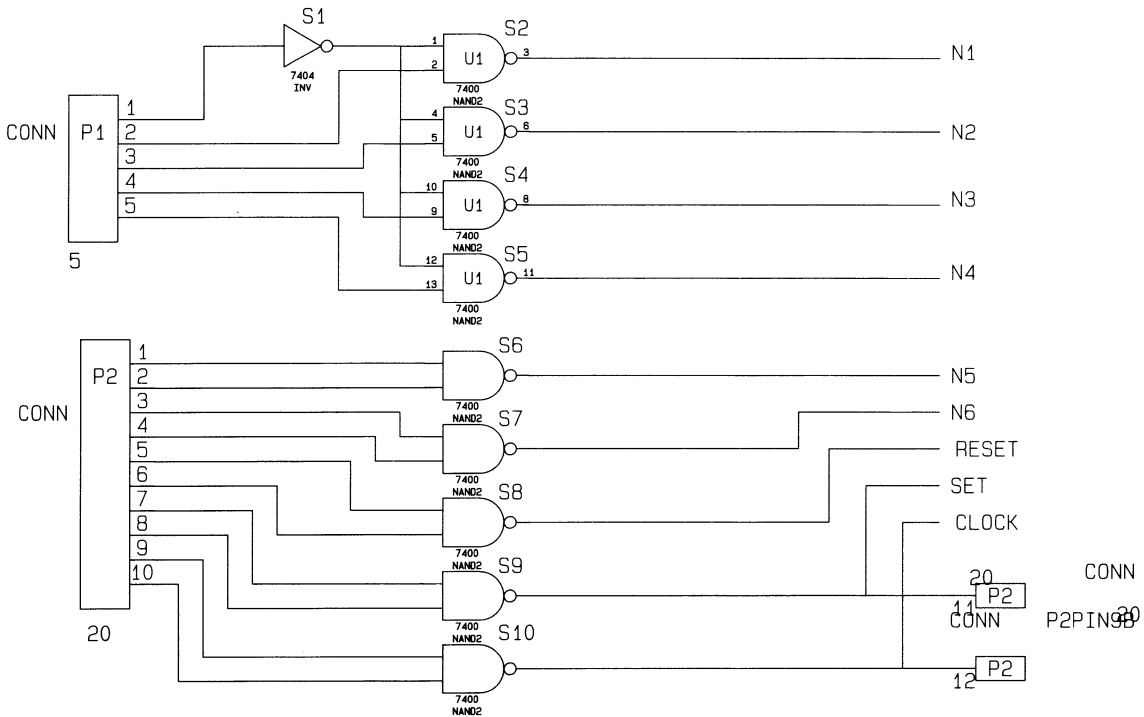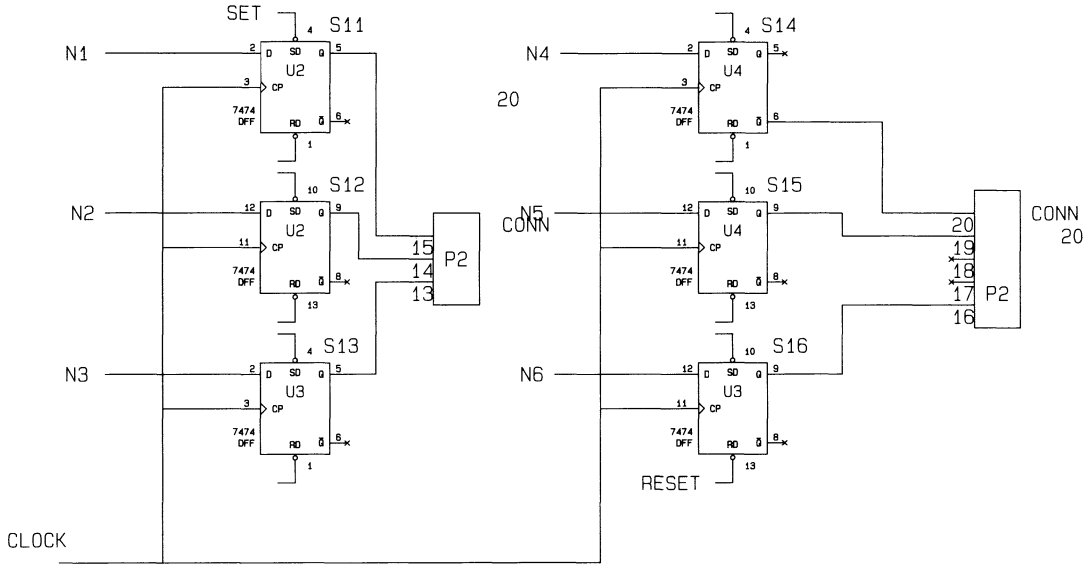
001004	U1	9	NAND2
N3	U1	8	NAND2
001001	U1	4	NAND2
001003	U1	5	NAND2
N2	U1	6	NAND2
001001	U1	1	NAND2
001002	U1	2	NAND2
N1	U1	3	NAND2
001001	U7	2	INV
001000	U7	1	INV
CLOCK	U3	3	DFF
RESET	U3	1	DFF
NONCON	U3	6	DFF
002002	U3	5	DFF
N3	U3	2	DFF
SET	U3	4	DFF
CLOCK	U2	11	DFF
RESET	U2	13	DFF
NONCON	U2	8	DFF
002001	U2	9	DFF
N2	U2	12	DFF
SET	U2	10	DFF
CLOCK	U2	3	DFF
RESET	U2	1	DFF
NONCON	U2	6	DFF
002000	U2	5	DFF
N1	U2	2	DFF
SET	U2	4	DFF
CLOCK	U4	3	DFF
RESET	U4	1	DFF
002003	U4	6	DFF
NONCON	U4	5	DFF
N4	U4	2	DFF
SET	U4	4	DFF
CLOCK	U4	11	DFF
RESET	U4	13	DFF
NONCON	U4	8	DFF
002004	U4	9	DFF
N5	U4	12	DFF
SET	U4	10	DFF
CLOCK	U3	11	DFF
RESET	U3	13	DFF
NONCON	U3	8	DFF
002005	U3	9	DFF
N6	U3	12	DFF
SET	U3	10	DFF
002002	P2	13	CONN
002001	P2	14	CONN
002000	P2	15	CONN
002005	P2	16	CONN

```
NONCON    P2        17    CONN
NONCON    P2        18    CONN
002004    P2        19    CONN
002003    P2        20    CONN
```

Schematic File: CSHEET1.SCH

Schematic File: CSHEET2.SCH

SET

S11

N1

U2

7474
DFF

S12

N2

U2

7474
DFF

15
14
13
P2

N3

S13

U3

7474
DFF

CLOCK

S14

N4

U4

7474
DFF

20

S15

N5
CONN

U4

7474
DFF

20
19
18
17
16
P2

CONN
20

N6

S16

U3

7474
DFF

RESET

APPENDIX A. ERROR MESSAGES

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

This appendix lists and describes NX-CALAY error messages. Errors are grouped into three types: file access and system errors, netlist errors, and packaging 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: Cannot open <filename>

Cause: NX-CALAY 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: Cannot use reserved file extension

Cause: A filename with the .WKF extension was specified.

Action: Specify a different extension.

Message: Database error

Cause: Access to the input netlist failed.

Action: Verify that the file specified as the input netlist actually is a PCAD netlist. Specify the correct file or, if necessary, recreate the netlist using PC-NODES and/or PC-LINK.

Message: Disk is Full
Press Any Key to Exit

Cause: The disk is full.

Action: Delete some files, and then rerun NX-CALAY.

Message: Duplicated file name : <filename>

Cause: You specified the same filename for two files.

Action: Specify a different filename for one of your output files.

Message: Reserved file name : <filename>

Cause: You specified NXCALAY.LOG as one of your filenames. This name is reserved for the log file.

Action: Change the filename.

Message: System error
Memory Allocation Error

Cause: The system could have insufficient memory. NX-CALAY requires 640K.

Action: Be sure you have 640K. If so, reboot your system.

Message: System error
Memory Deallocation Error

Cause: This message indicates a system error.

Action: Turn your computer off and then on again. Use the CHKDSK to check your hard disk. Check all your input files and regenerate them if necessary. If the error persists, call P-CAD.

NETLIST ERRORS

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

Message: Duplicated name : <signal/refdes>

Cause: The specified name was used both for the net name and reference designator.

Action: Use PC-CAPS to change one of the names, then rerun PC-NODES and NX-CALAY.

Message: Hierarchical Component : <device>

Cause: The named component was hierarchical.

Action: The output file is not generated. Correct your
          schematic, then rerun PC-NODES and
          NX-CALAY.

Message: Invalid device name : <device>

Cause: The specified name has more than six
          characters or contains a character other than
          letters, numbers, +, and -.

Action: Correct the name.

Message: Invalid reference designator : <refdes>

Cause: The specified reference designator has more
          than six characters or contains a character
          other than letters, numbers, +, and -.

Action: Correct the reference designator.

Message: Invalid signal name : <signal>

Cause: The specified name does not conform to the
          conventions described in "Signal Names" in
          Chapter 2.

Action: Correct the name.

Message: No pin assignments for <device> <refdes>

Cause: The named component does not have preassigned package pin numbers.

Action: Use PC-CAPS to preassign explicit pin numbers to all components of nonhomogeneous parts, then use PC-NODES to extract an updated netlist and input the netlist to NX-CALAY.

Message: No Reference Designator for <device>

Cause: The named component was not preassigned a reference designator.

Action: Use PC-CAPS to preassign reference designators to all components of nonhomogeneous parts, then use PC-NODES to extract an updated netlist and input the netlist to NX-CALAY.

PACKAGING ERRORS

The following errors are produced by NX-CALAY during netlist translation while it is packaging the components. In these messages, **compname** is the P-CAD symbol filename and **location** is the X and Y coordinates of the component in the P-CAD schematic database.

Message: **Common pin found - pin number <n>**
         Component <compname> at <location>

Cause: The same pin number was assigned more than once for a reference designator. The named component is not included on the output netlist.

Action: Use PC-CAPS to correct the schematic. If the component is to be packaged into a part with common pins, add the COMM attribute. Otherwise, correct the pin numbers. Then rerun PC-NODES and NX-CALAY.

Message: **Connector pins exceed CONN attribute**
         Component <compname> at <location>

Cause: The number of connector pins for the same reference designator exceeded the number specified with the CONN attribute. The named component is not included on the output netlist.

Action: Use PC-CAPS to change the CONN attribute or to correct the component preassignment. Then rerun PC-NODES and NX-CALAY.

Message: **Duplicate reference designator <refdes>**
Component <compname> at <location>
corrected with <refdes> (<pinnum>)

Cause: The named reference designator is used more
times than the number of gates per package
assigned to the named component. If the
component in homogeneous, NX-CALAY
corrected the error as shown.

Action: Check the output file to see whether
corrections are necessary. If so, use PC-CAPS
to make corrections, then rerun PC-NODES
and NX-CALAY. If corrections are not
necessary, use PC-CAPS to add the MIX
attribute.

Message: **Error in assignment <refdes> (<pinnum>)**
Component <compname> at <location>
corrected with <refdes> (<pinnum>)

Cause: This component contains duplicate sections or
the section assignment does not match the
assignment made using the SCMD/PNLC
command. If the component is homogeneous,
NX-CALAY corrected the error as shown.

Action: Check the output file to see whether
corrections are necessary. If so, use PC-CAPS
to make corrections, then rerun PC-NODES
and NX-CALAY. If corrections are not
necessary, use PC-CAPS to add the MIX
attribute.

Message: **Hierarchical component detected**
Component <compname> at <location>

Cause: The named component is hierarchical.
NX-CALAY cannot process the database.

Action: Do not input to NX-CALAY a design with
any hierarchical components.

Message: **Missing CONN attribute**
Component <compname> at <location>

Cause: The named component, which was not
assigned the CONN attribute, had the same
reference designator as another component
that was assigned the CONN attribute. The
component was not included on the output
netlist.

Action: Use PC-CAPS to change the reference
designator of one of the components or to add
the CONN attribute, then rerun PC-NODES
and NX-CALAY.

Message: **Missing MIX attribute**
Component <compname> at <location>

Cause: The named component has the same reference
designator as another component that was
assigned the MIX attribute. The component is
not included on the output netlist.

Action: Use PC-CAPS to change the component's
reference designator or to add the MIX
attribute, then rerun PC-NODES and
NX-CALAY.

Message: **Number of components exceeds MIX attribute**
Component <compname> at <location>

Cause: The number of components assigned to a heterogeneous part is greater than the number of gates specified in the MIX attribute for the specified part. The named component is not included on the output lists.

Action: Use PC-CAPS to correct the component's MIX attribute or to correct the preassignment. Then rerun PC-NODES and NX-CALAY.

Message: **Pin number not preassigned to connector component <compname> at <location>**

Cause: The named component was assigned the CONN attribute but was not preassigned pin numbers. The component is not included on the output netlist.

Action: Use PC-CAPS to assign pin numbers to the component, then rerun PC-NODES and NX-CALAY.

Message: **Pin number not preassigned to mixed-part component**
Component <compname> at <location>

Cause: The named component was assigned the MIX attribute but was not assigned package pin numbers. The component is not included on the output netlist.

Action: Use the PC-CAPS SCMD/PNUM command to assign package pin numbers to the component, then rerun PC-NODES and NX-CALAY.

Message: **PNLC not defined for <compname>**
Database not processed

Cause: The named symbol does not have the information assigned with the SCMD/PNLC command. NX-CALAY cannot process the database.

Action: Use PC-CAPS to add the appropriate information to the symbol, then replace all instances of the symbol in the schematic with the corrected symbol. Then rerun PC-NODES and NX-CALAY.

Message: **Same reference designator for CONN/no-CONN component <compname> at <location>**

Cause: The named component, which was assigned the CONN attribute, has the same reference designator as another component that was not assigned the CONN attribute. The component is not included on the output netlist.

Action: Use PC-CAPS to change the reference designator of one of the components or to add the CONN attribute, then rerun PC-NODES and NX-CALAY.

Message: Same reference designator for different
symbols
Component <compname> at <location>

Cause: The named component has the same reference
designator as another component of a
different type. The component is not
included on the output netlist.

Action: Use PC-CAPS to change the reference
designator of one of the components, then
rerun PC-NODES and NX-CALAY.

Message: Same reference designator for MIX/no-MIX
Component <compname> at <location>

Cause: The named component, which was assigned
the MIX attribute, has the same reference
designator as another component that was not
assigned the MIX attribute. The component is
not included on the output netlist.

Action: Use PC-CAPS to change the reference
designator of one of the components or to add
the MIX attribute, then rerun PC-NODES and
NX-CALAY.

TOOLS FOR
MODERN
DESIGN™