VERSATEC
A XEROX COMPANY

Expert 1000 User's Guide

This document was prepared with the Xerox 8010 Information System.

Document Number 1-40B May 1983 Printed in USA

Table of Contents

Schematic Design

The Library Editor

Introduction

The Expert 1000 User's Guide is your learning guide and reference book for the Expert 1000 system. The User's Guide consists of three books. Book 1 describes the workstation, defines several basic terms and procedures, and tells you how to start and stop the system. Book 2 is a detailed guide to designing schematics. Book 3 describes creating and maintaining Expert 1000 library entries.

How to Use this Guide

Read the User's Guide before you use the system as an introduction to Expert 1000. Try procedures and commands as you read about them; they make more sense when you actually see them work. As you become more skilled, use the User's Guide as a reference guide. The index at the end of each book makes it easy for you to look up a term or a procedure you forget.

Which Books Should You Read?

All users should read Book 1 before reading any other books. Book 1 explains basic procedures not explained elsewhere. Read Book 2 if you want to create schematics with the system. If you've been assigned the task of system librarian, read Books 2 and 3.

User's Guide Conventions

Bullets (●) precede important procedures so that you can easily locate a procedure inside the text. Procedure steps are numbered. For example:

● **To select an object on the display:**

1) **Move the mouse until the cursor points to that object.**
2) **Press and release the mouse's left button.**

Menu commands appear in boldface with uppercase letters where they appear in the menu. For example:

Create Net
closest

Commands in command subwindows, and property values in property sheets appear in boldface as they appear on the display. For example:

Props!	**Selection Mode:**
Apply!	**Display Options:**

Keyboard keys are bold and all uppercase.

MOVE
DELETE

VERSATEC
A XEROX COMPANY

Expert Series
Documentation Comment Form

Please use this form to express your opinion of this manual. Your comments help us to improve our manuals, and make them more useful for you. All comments and suggestions become property of Versatec. Mail this form to Versatec, 2710 Walsh Avenue, Santa Clara, CA 95051.

Manual Title: ______________________________

Who are you?

Name: ______________________________

Company Name: ______________________________

Position: ______________________________

How do you use the manual?

As an introduction to the product it describes	☐
As a reference guide	☐
As a tutorial	☐
Other: ______________________	☐

How well does the manual meet your needs?	Yes	No
Is it easy to read?	☐	☐
Is it easy to understand?	☐	☐
Are the format and organization easy to follow?	☐	☐
Is it technically accurate?	☐	☐
Can you easily find the information you need?	☐	☐
Is it complete? Does it tell you everything you need to know?	☐	☐
Is it well illustrated?	☐	☐

Technical or editorial errors (please list page number):

____

____

____

____

____

Suggestions for improvement:

____

____

____

____

____

VERSATEC
A XEROX COMPANY

Expert 1000 User's Guide

The Basics

This document was prepared with the Xerox 8010 Information System.

Document Number 1-40B May 1983 Printed in USA

Table of Contents

1. Introduction

This book gives you an overview of your workstation. It contains an introduction to the workstation components, an explanation of terms and basic procedures common to Expert 1000 tools, and detailed instructions for starting and stopping the system.

Read this book before you read any of the others. Try the basic procedures as you read about them. You may want to read this book through once as an introduction, and a second time as you try each procedure.

1.1 The Workstation

Expert 1000's design tools run on a Xerox 8010 professional workstation. The workstation consists of a processor, a disk storage module, a high resolution graphics display, a keyboard, a pointing device called a *mouse*, and a connection to an Ethernet network.

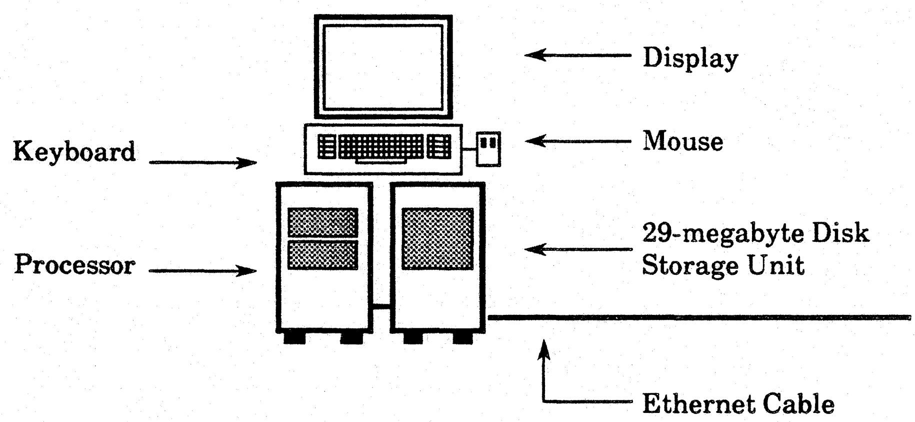


Figure 1.1

1.2 The Processor

The processor consists of two units that sit on the floor next to your desk. One unit contains the processor itself, and the other contains a rigid disk for storing your work. A floppy disk drive opens onto the front of the processor. A door below the disk drive hides the *maintenance panel.* The maintenance panel contains a four-character, LED display, two small switches marked "B Reset" and "Alt B," and a larger power switch.

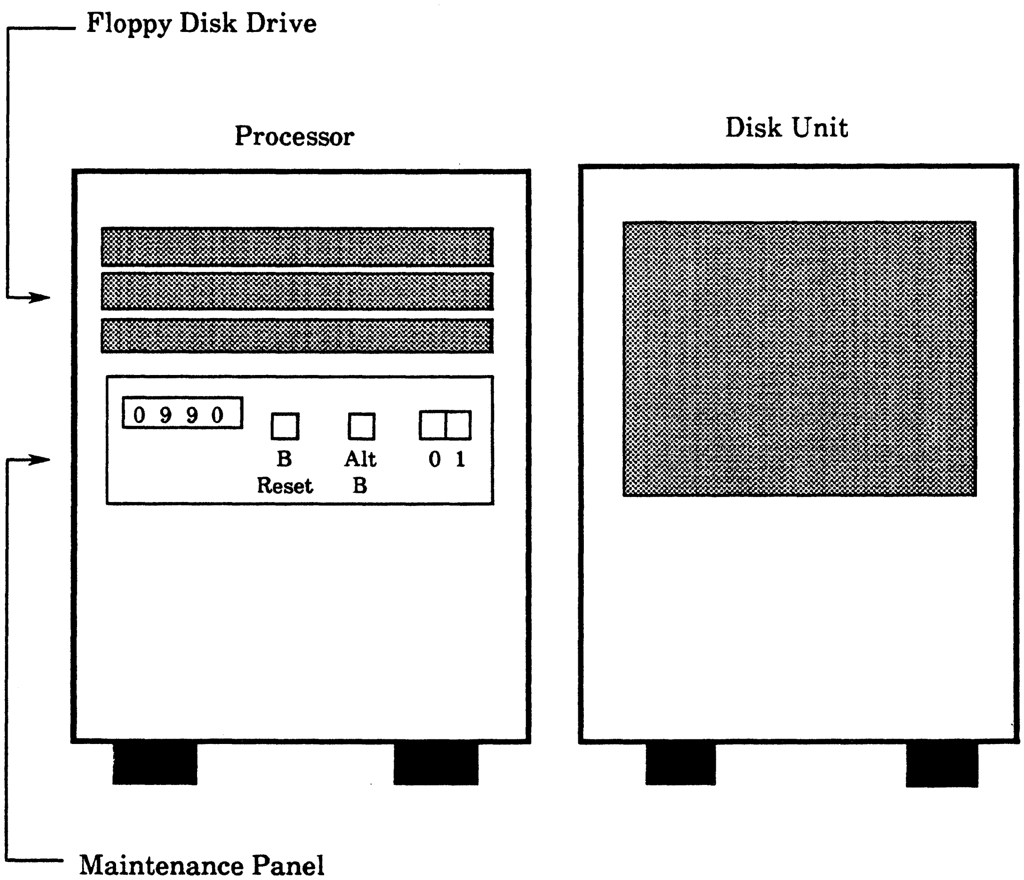


Figure 1.2

1.3 The Display

The workstation's special high-resolution display shows both graphics and text. The display screen is your electronic desktop. Working with Expert 1000 tools on the display is similar to working with paper, pencils, rulers, and templates on a desk or drawing board.

Under the left side of the display is an intensity control lever. Pull the intensity control lever toward you to make the display dimmer. Push it away from you to make the display brighter.

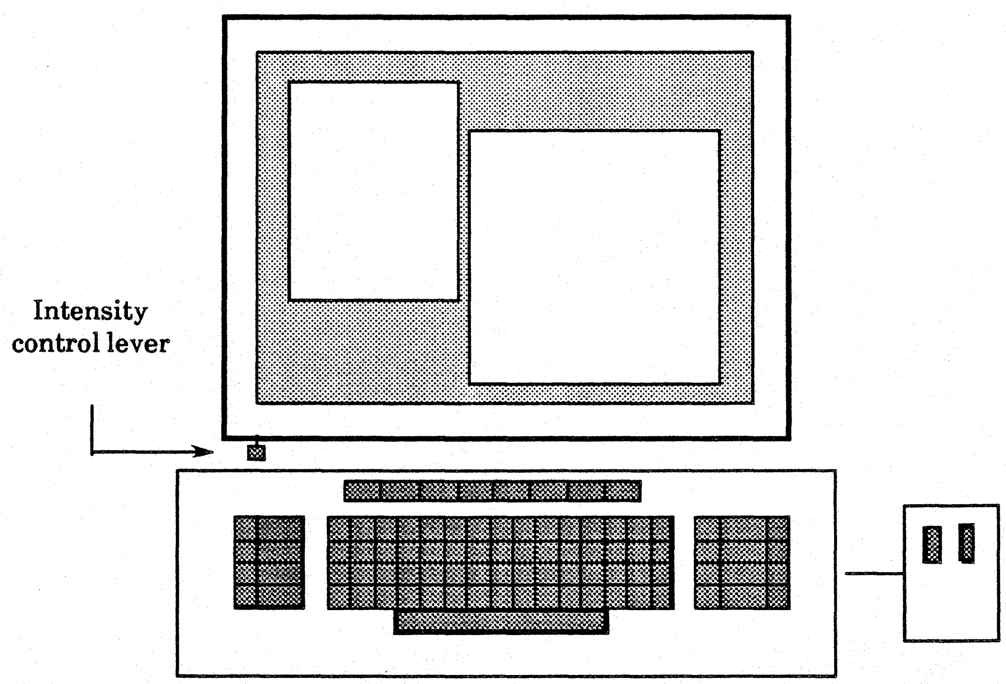


Figure 1.3

1.4 The Keyboard

The keyboard is one device you use to communicate with the system. You use the typewriter keypad to add text to your designs. You use the special function keys to the right, left, and above the typewriter keypad to issue commands.

Figure 1.4 shows the keyboard layout, and figure 1.5 shows actual keys.

Keyboard Layout

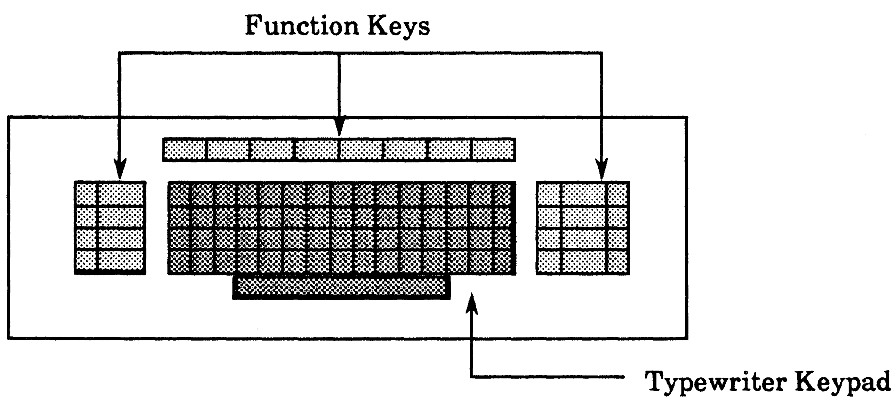


Figure 1.4

Keyboard Keys

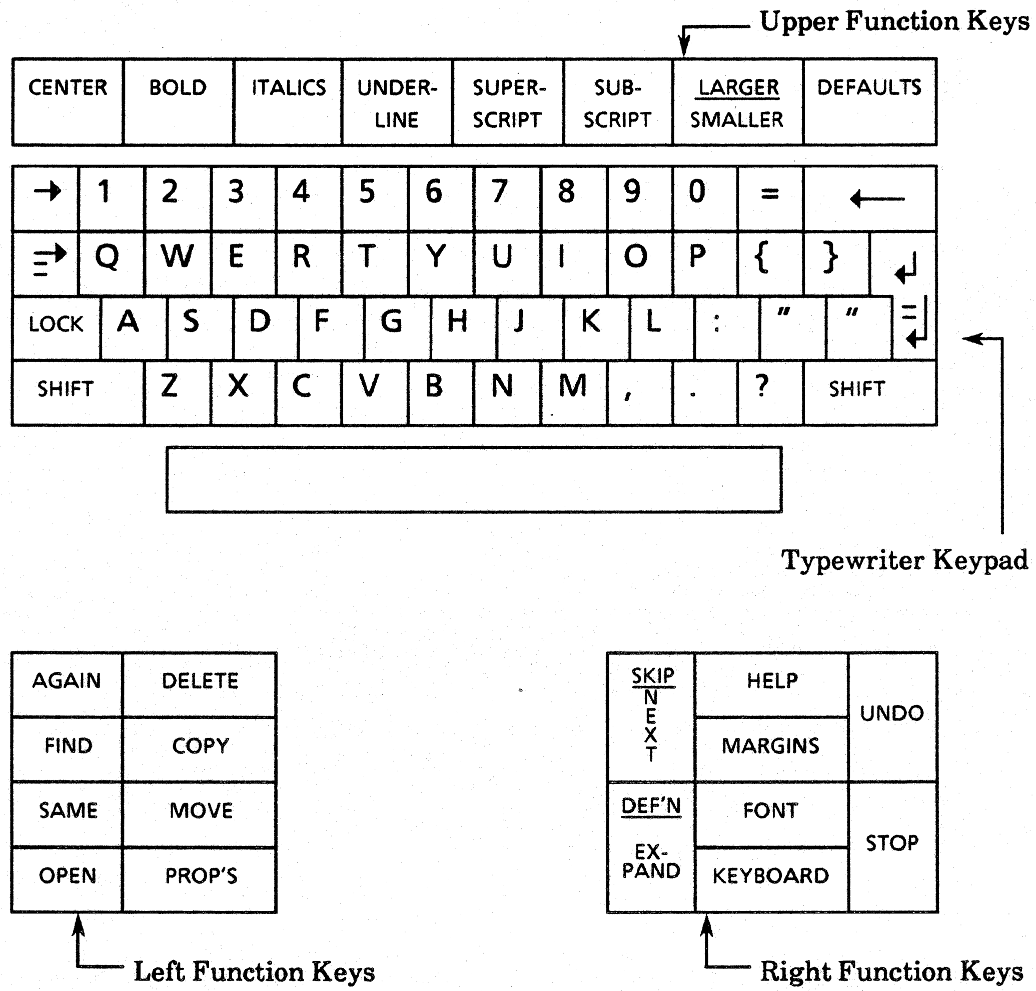


Figure 1.5

1.5 The Mouse

The mouse is a pointing device attached to the keyboard by a thin, two-foot-long cord. On the top surface of the mouse are two buttons. Like the keyboard, the mouse lets you communicate with the system. You use the mouse both to point to objects on the display, and to issue commands.

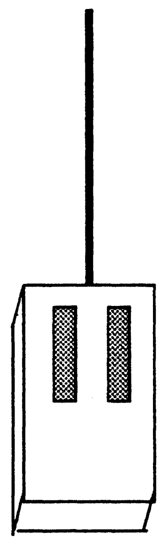


Figure 1.6

1.6 The Network

Your workstation is connected to others at your installation by an Ethernet local area network. The network gives you access to several resources beyond your own workstation. An output server lets you share an electrostatic printer/plotter and a magnetic tape drive with other users. You also share a file server for storing data bases and libraries. A communication server lets you communicate with users of other networks.

Your System Administrator has assigned a name to each server. You need to know the names of the servers you will use. For example, you need to know the name of a file server in order to store a file on that server. Your System Administrator can tell you which services are available on your network and the server names.

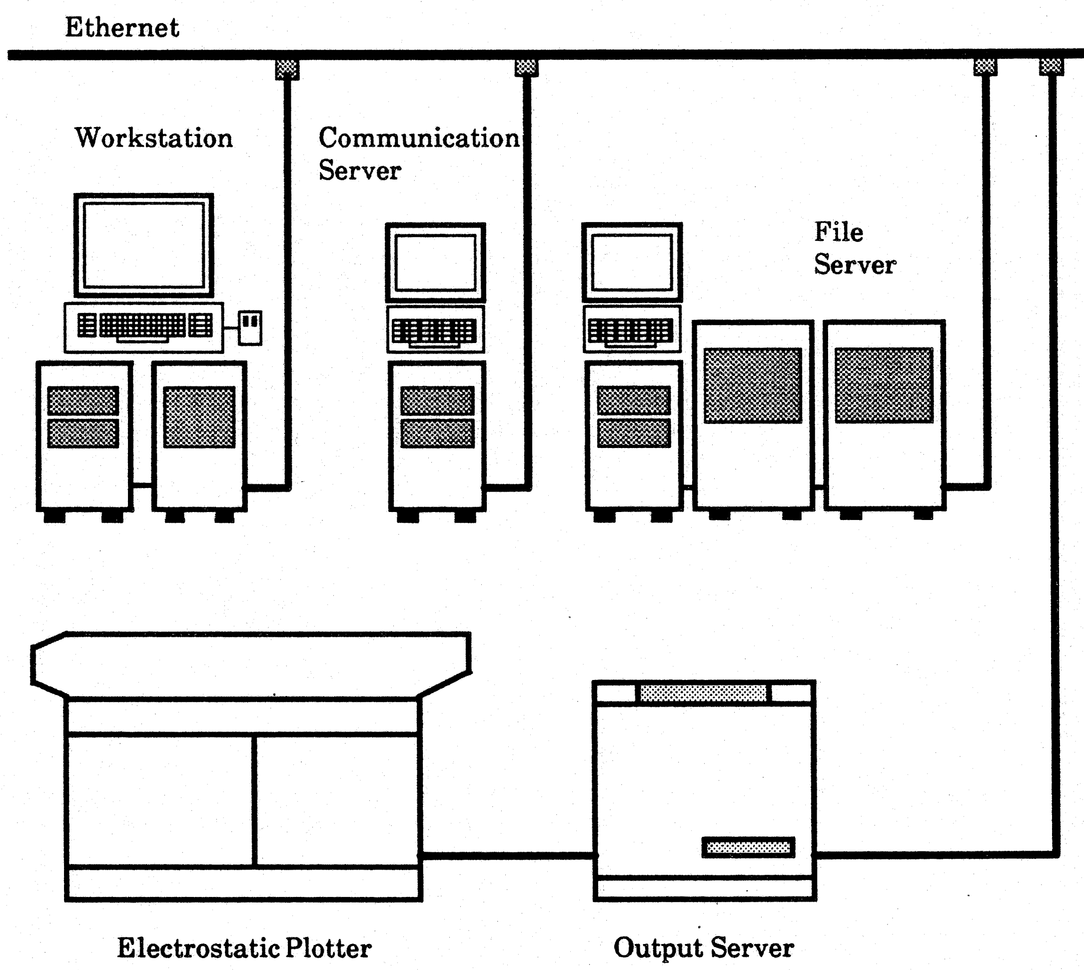


Figure 1.7

2. Starting a Session

2.1 An Idle Workstation

When you start a session the workstation should be *idle*. Idle means that the Expert software is running on the workstation and the workstation is ready for you to begin your work. The display should be blank except for a small white box containing the day of the week and the time. The small white box constantly moves from place to place on the display. See figure 2.1.

If your display is blank and you do not see a bouncing box, check the power switch on the maintenance panel; turn the switch to 1 to turn on the workstation, and wait a few minutes for the workstation to become idle. If the power is on, try adjusting the intensity of the display. Push the intensity control lever away from you as far as it will go. If you still do not see the bouncing box, *boot* your workstation. Chapter 7 of this book tells you how to boot a workstation. When you finish booting the workstation, you should see the bouncing box. If you still don't see the bouncing box, ask your System Administrator for help.

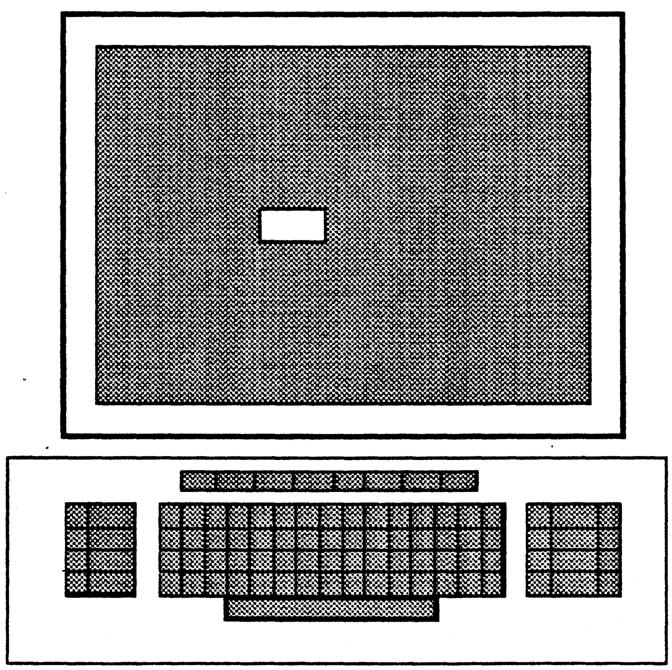


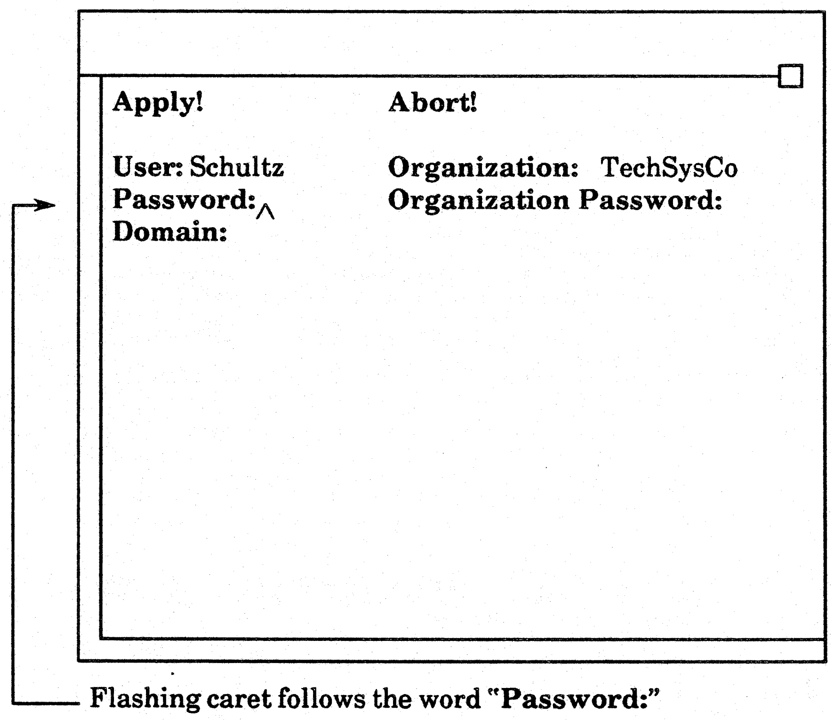
Figure 2.1

2.2 Logging In

You *log in* to tell the system who you are and to which tools and designs you have access. The system recognizes you by your *user name, organization, domain* and *password.* Your System Administrator assigns names and passwords. If your System Administrator has given you access to the system's library tools, he or she will also give you an *organization password.*

- To log in to an idle workstation:

1) Press the key marked **STOP**. The system displays the *login window.* The login window contains fields for entering your name, organization, password, organization password, and domain. Figure 2.2 illustrates the login window. In the login window you see the name and organization of the person who last used the workstation. Frequently that person is you.



Flashing caret follows the word "**Password:**"

Figure 2.2

2) Notice the flashing caret after the word "**Password:**" It shows you the *type-in* point. The system displays any text you type after the type-in point. If your name and organization are correct, type your password. The system does not display your password, but displays an asterisk for each character in the password. If your name is incorrect, move the mouse to move the cursor until it points to the word "**User:**" Press the mouse's right button. The system removes the existing name, and marks the type-in point after "**User:**." Type your name. If your organization name is incorrect, use the mouse to move the cursor until it points to the word "**Organization:**" Press the right mouse button. Type your organization

name. Move the cursor to "**Password:**," and press the right mouse button. Type your password.

3) If you want to make changes to the system's library, you must also enter an organization password. Usually you skip this step. Point the cursor at the words "**Organization Password**." Press the right mouse button. Type your organization password.

4) Point the cursor at the word "**Apply!**." Press either mouse button. The system removes the login window from the display. You see several *tiny windows*.

You can use the **SKIP NEXT** key in the right function group to automatically move the type-in point from field to field in the login window. As the system moves the type-in point to a field, it erases the text that was there. For example, after you enter your password you might want to enter an organization password. If you press the **SKIP NEXT** key, the system automatically marks the type-in point after "**Organization Password:**." You simply type the password.

3. Interacting with the System

This section defines several basic terms and procedures that you must know to use the system.

3.1 The Cursor, Pointing, and Selection

3.1.1 The Cursor

The *cursor* is a graphic object, initially a small arrow, that appears on the display screen. (See figure 3.1.) The cursor acts like your pencil on the display desktop; you use it to show the system where you want to work and what you want to do.

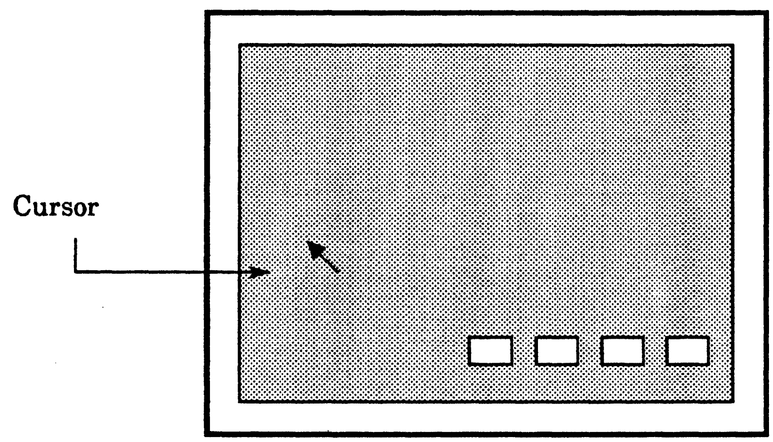


Figure 3.1

Moving the mouse along a flat surface moves the cursor on the display. When the cursor reaches an edge of the display screen it does not track the movements of the mouse.

3.1.2 Pointing

- To *point* to an object, (for example, a logic symbol in a schematic):

1) Move the cursor until the tip of the arrow "touches" any part of that object.

3.1.2 Selection

You *select* an object in order to do something to that object. For example, to delete an object, you select the object and press the key marked **DELETE**.

- To select an object on the display:

1) Move the mouse until the cursor points to that object.
2) Press and release the mouse's left button.

The system highlights selected objects in inverse-video; it marks selected objects in black on the white background.

3.1.4 Cursor Shapes

The cursor changes shape to tell you what the system is doing, and what you can tell it to do. When the system is busy, the cursor changes into an hourglass. It is processing your last command. (See figure 3.2.) The system stores commands you issue while the cursor is an hourglass, and processes them immediately after it completes processing the first.

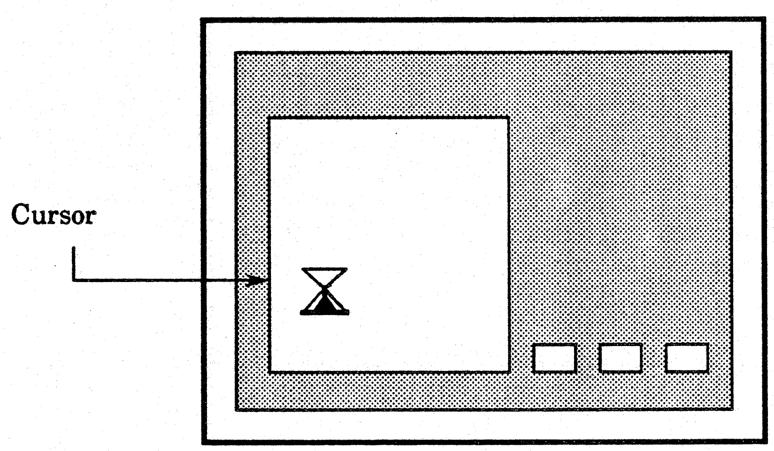


Figure 3.2

When the cursor is an arrow with a cross on its tail, you can select only objects or menu commands.

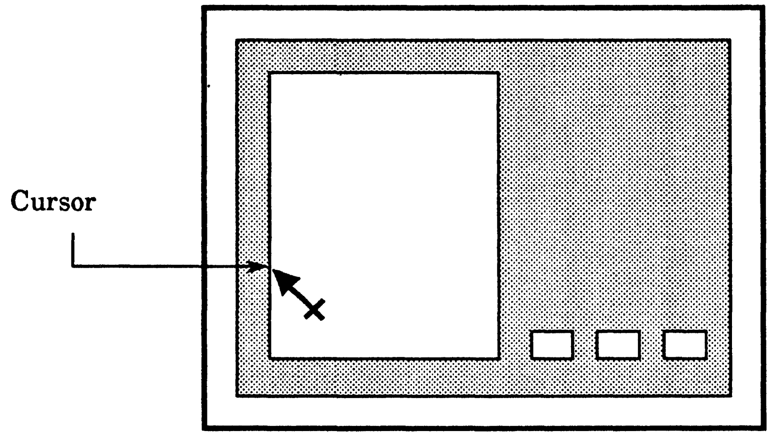


Figure 3.3

When the cursor is an arrow without a cross on its tail, you can select objects or menu commands, or type in text using the keyboard.

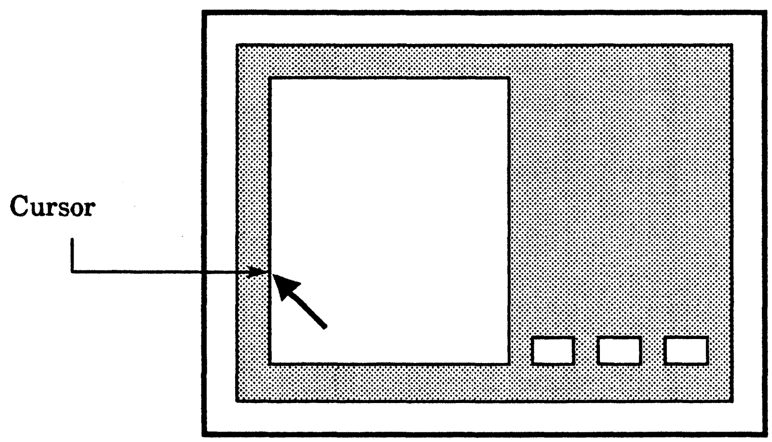


Figure 3.4

3.2 Windows

A *window* is a rectangular region on the display framed by a narrow black border. Windows are the sheets of paper on your display desktop.

You communicate with the system through windows. A window shows you messages from the system, commands available to you, and the actual schematic, net list, or logic symbol you are creating or editing. You interact with the system only when the cursor is within a window's boundary, as though the cursor was a pencil and the window a sheet of paper.

3.2.1 Active and Tiny Windows

Windows have two states: *active* and *tiny*. When a window is tiny, a small box containing only the window's name represents it on the display. Figure 3.5 shows active and tiny windows.

Making a window tiny is like putting a piece of paper aside on your desk; you aren't using it and want to get it out of your way, but you know you'll want to use it again soon. When you make a window tiny, you store its contents. The window looks exactly the same once you make it active again.

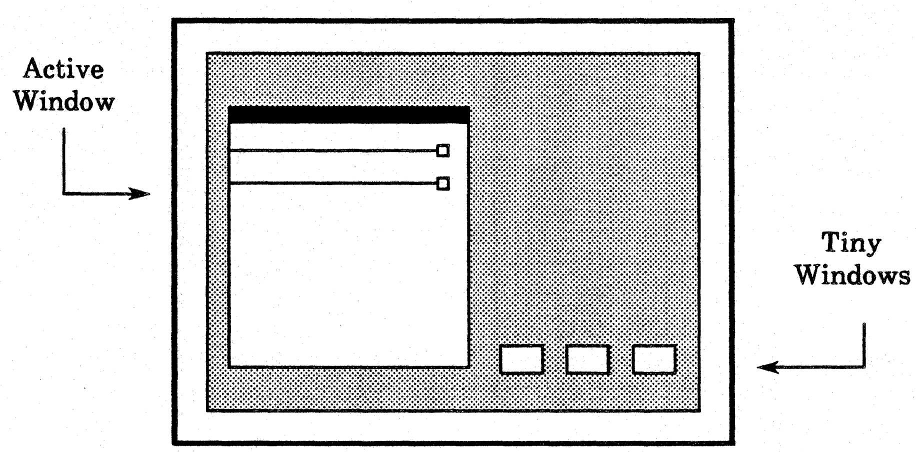


Figure 3.5

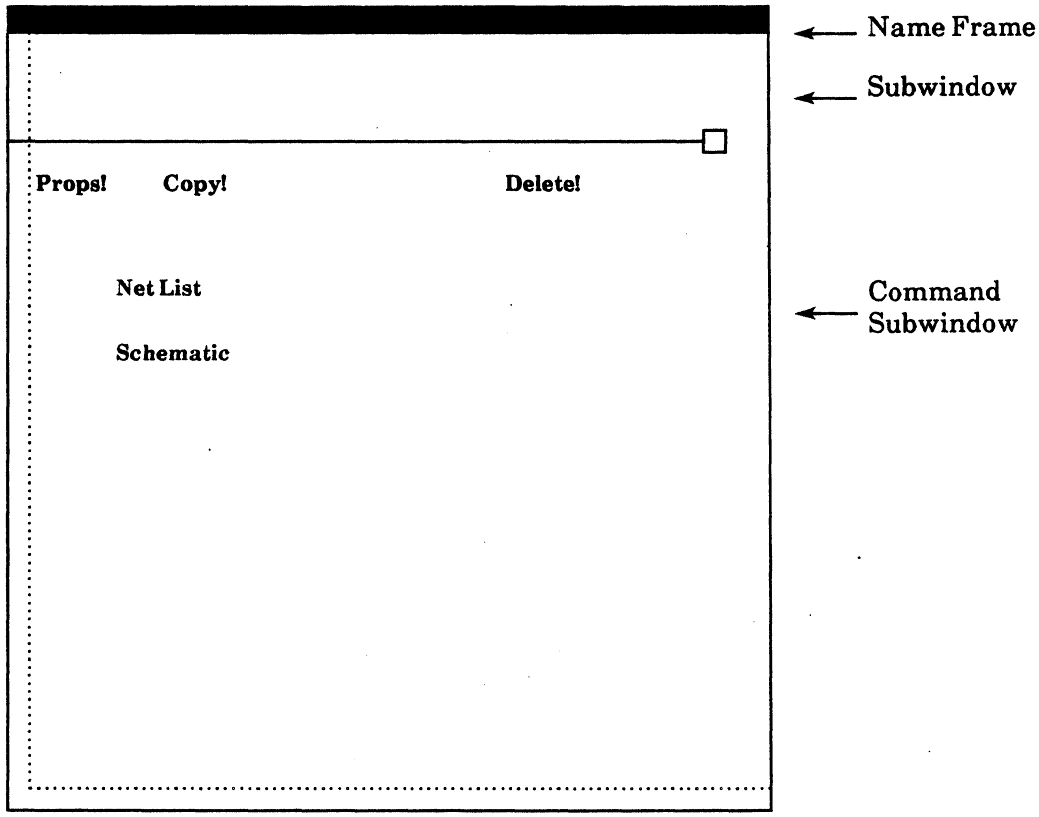


Figure 3.6

3.2.2 Name Frames

Every active window has a black, rectangular *name frame* across its top. (See figure 3.6.) The system displays the name of the window in white letters inside the black name frame.

3.2.3 Subwindows

Horizontal lines ending in a small box on the right divide the window into *subwindows*. Each subwindow represents a different function of the window. For example, one subwindow might display messages to you from the system, another might list commands, and another might display a schematic or report. (See figure 3.6.) The number of subwindows and their function varies from window to window.

3.2.4 Command Subwindows

A *command subwindow* is one type of subwindow common to most windows. Command subwindows contain lists of commands that affect the entire window, options that change the way the system displays data, and information such as page numbers. (See figure 3.6.)

All commands in a command subwindow end with an exclamation point (!). To issue a command, you *select* that command.

- To select a command:

1) Point the cursor at the command in the subwindow
2) Press the left mouse button.

The list of commands in a command subwindow may change with the context of your work. The system lists only commands that are relevant to what you are doing at a given time.

3.2.5 Adjusting the Borders of a Subwindow

You can change the size of a subwindow within a window to see more or less of the information inside it.

- To change the size of a subwindow:

1) Point the cursor inside the box at the end of the subwindow's division line.
2) Press and hold the mouse's left button.
3) While pressing the button, move the mouse up or down to move the division line. The subwindow size increases or decreases as you move the mouse. (See figure 3.7).
4) When the line is where you want it, release the mouse button. The adjoining subwindows adjust their size accordingly.

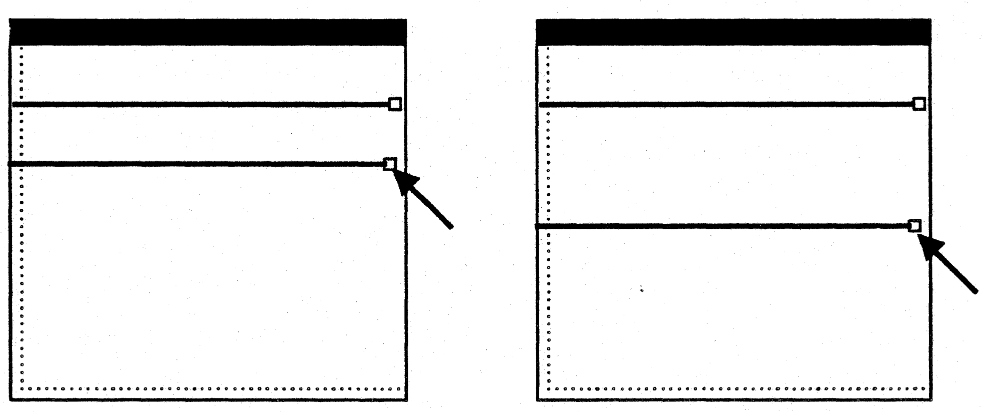


Figure 3.7

3.2.6 Scrolling

A subwindow can contain more information than its size lets it display at any one time. For example, a schematic page might be much larger than the portion of it you see through the subwindow. You can *scroll* the schematic in order to display different portions of it through the subwindow. You can scroll vertically (up and down) in all subwindows, and horizontally (left and right) in several subwindows.

You use a *scroll bar* to scroll a subwindow. Every window has a vertical scroll bar spanning its left border. Some windows also have horizontal scroll bars spanning their lower border. Figure 3.8 shows a vertical and a horizontal scrollbar.

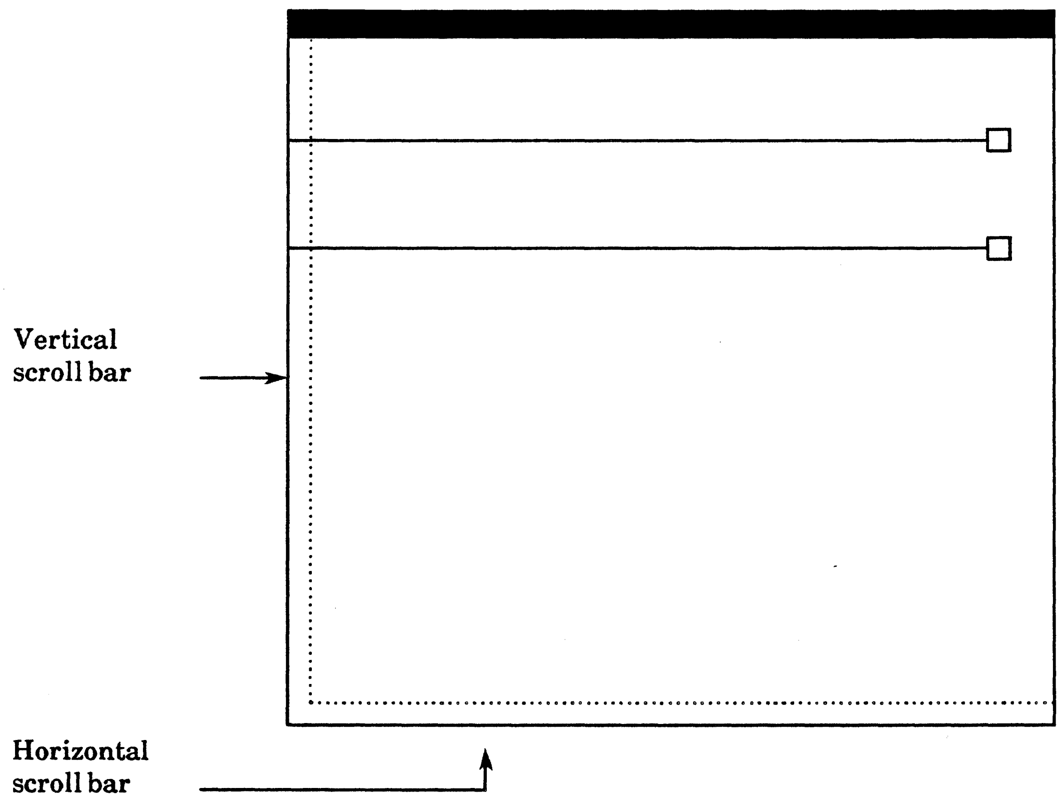


Figure 3.8

- To scroll:

  1) Move the cursor into the scrollbar. You see a shaded rectangle that spans the vertical length of the subwindow, and a smaller, darker rectangle. The lighter area represents the entire subwindow. See figure 3.9. The darker area represents that portion of the information displayed in the subwindow. The cursor changes to a double headed arrow.
  2) The cursor's distance from the the corner of the scrollbar tells the system the *amount to scroll.* Move the cursor so that its distance from the top or left corner is the amount you want to scroll.
  3) The mouse buttons control the *direction of scrolling.* In a vertical scrollbar, the left button scrolls up and the right mouse button scrolls down; in a horizontal scrollbar, the left button scrolls to the left, and the right button scrolls to the right. With the cursor inside the scrollbar, press the left or right mouse button to indicate which direction to scroll. As you press the button, the cursor changes to an arrow pointing in that direction.

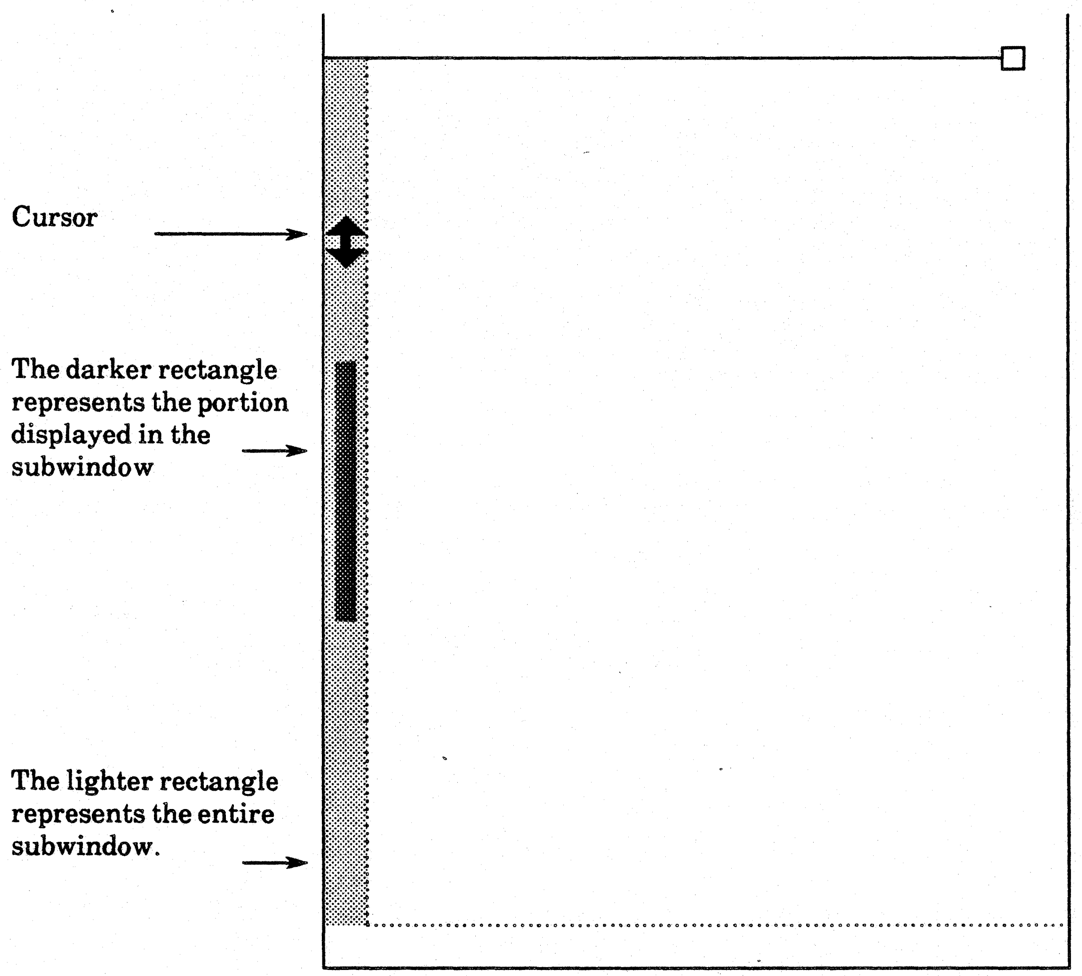


Figure 3.9

3.2.7 Thumbing

Scrolling is similar to skimming through the text of a book when you are looking for a certain passage. If you knew approximately where that passage was you could find it faster by thumbing through the book page by page. You can also *thumb* through a subwindow. When you thumb through a subwindow, you show the system which specific portion of the information inside that subwindow to display.

- To thumb:

1) Move the cursor into the scrollbar.
2) Press **both** of the mouse's buttons at once.
3) The cursor changes to a thick arrow pointing at the scrollbar. See figure 3.10. Remember, the scrollbar's lighter rectangle represents the entire subwindow, and the darker rectangle represents the portion of the subwindow you are viewing.
4) Use the cursor to point to the portion of the whole document that you want to display in the subwindow.
5) Release both buttons.

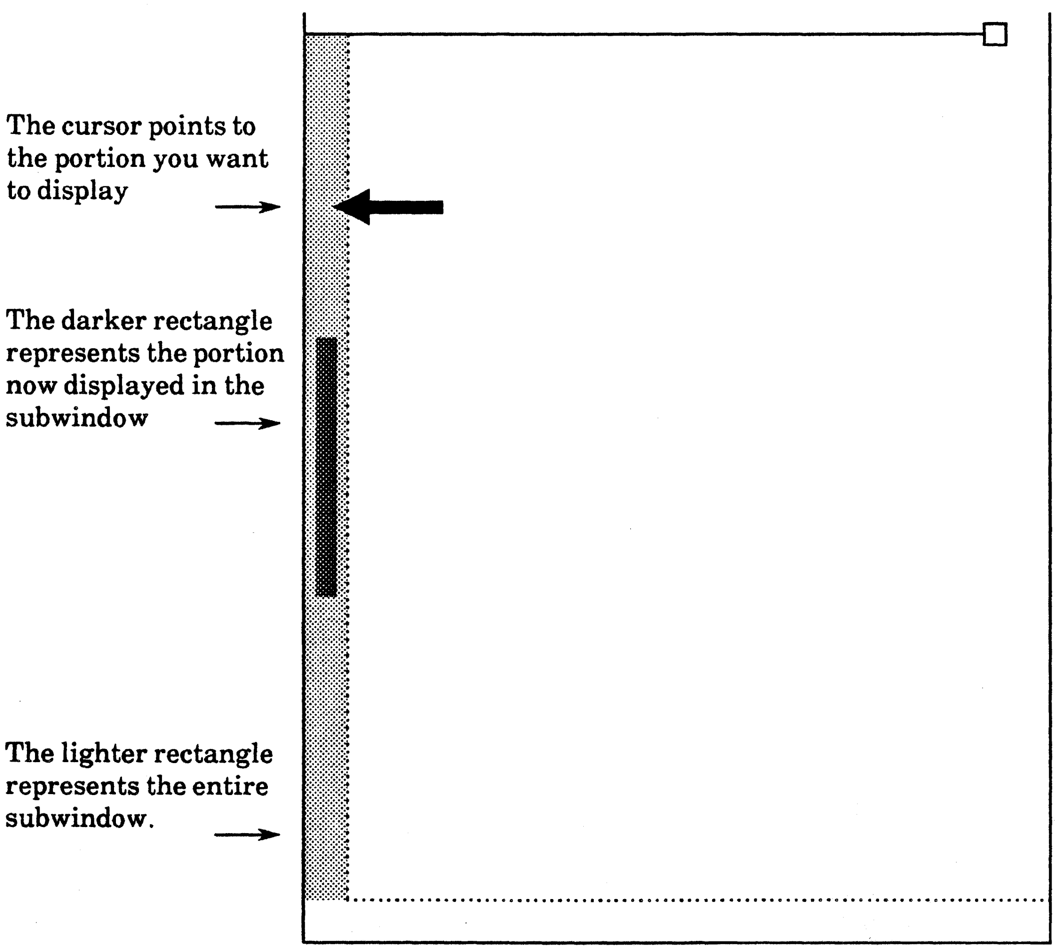


Figure 3.10

If you move the cursor inside the scrollbar once again, you can see that the darker region moved to where you pointed. See figure 3.11.

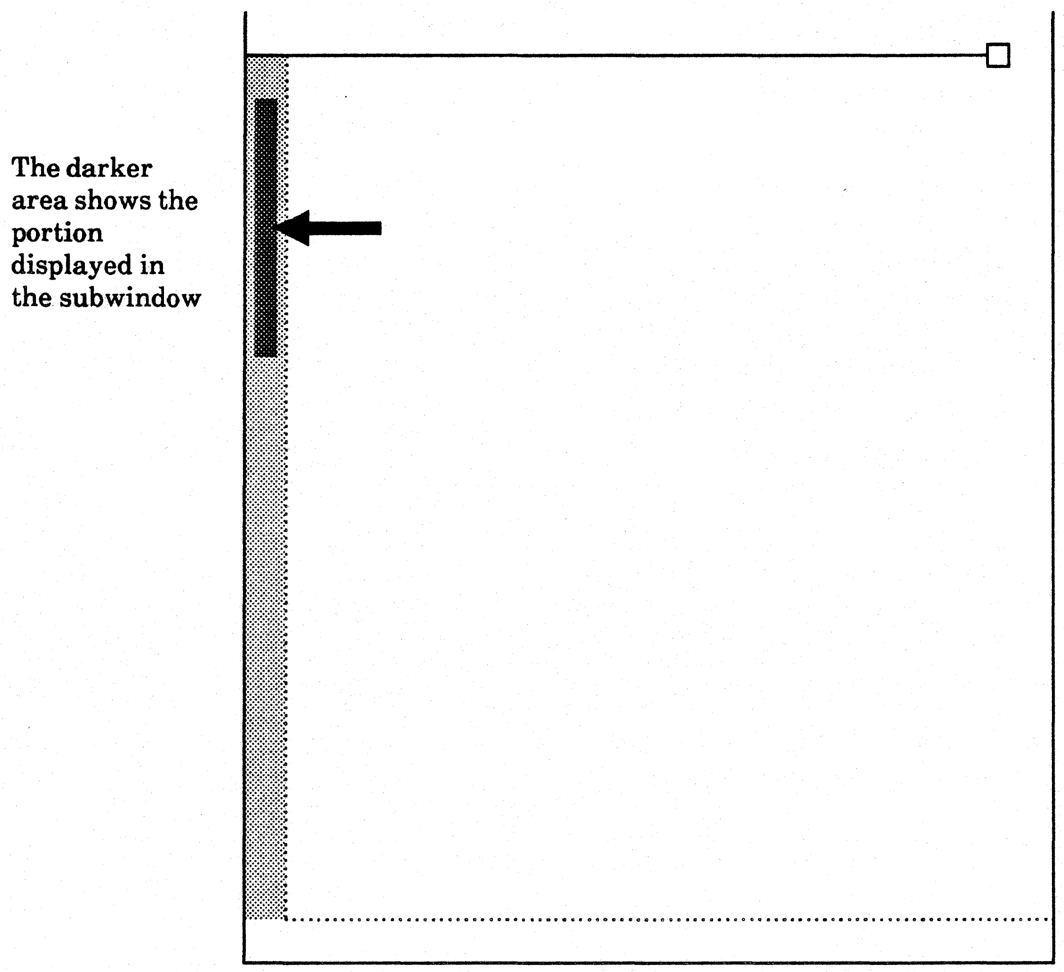


Figure 3.11

3.3 Dynamic Menus

A *dynamic menu,* or *pop-up menu,* is a list of options or commands. Each dynamic menu has a name inside a name frame; the menu's name describes the functions of the commands or options inside the menu. Figure 3.12 shows a dynamic menu inside a window.

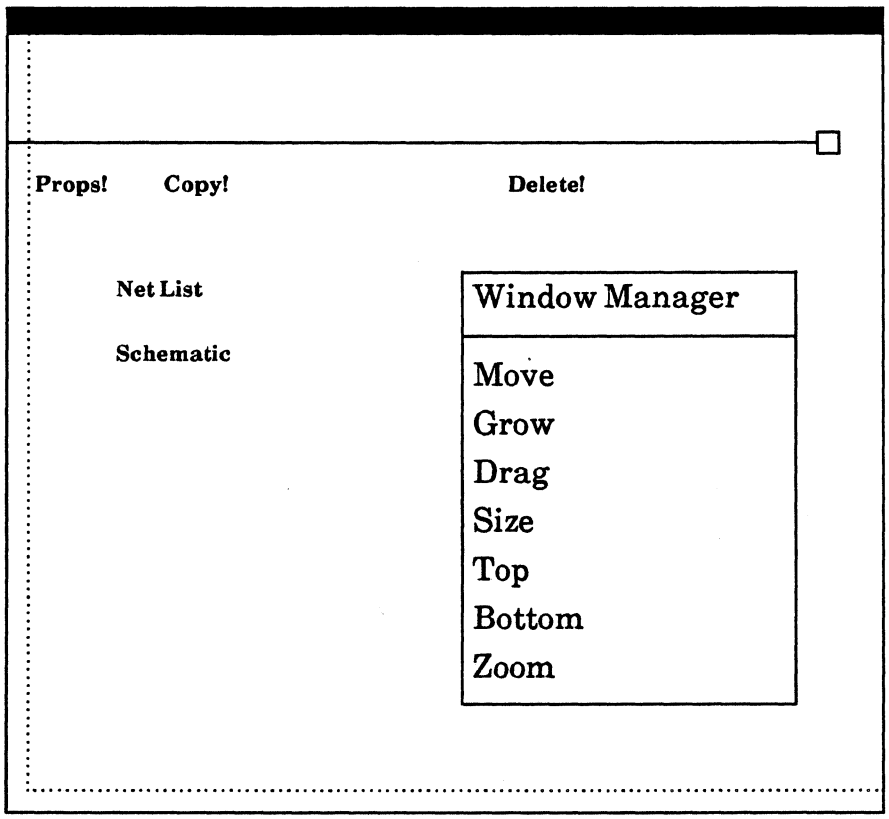


Figure 3.12

Dynamic menus are associated with windows, subwindows, and objects inside subwindows. The cursor's position inside a window or subwindow, or pointing at an object tells the system which menus to display. Thus, dynamic menus only list commands or options that are relevent to what you are doing at a given moment. Dynamic menus appear on the display only when you activate them. When you're not using them, they are out of your way.

- To activate dynamic menus:

1) Position the cursor inside the window or subwindow, or point to the object for which you want to display menus.
2) Press both mouse buttons simultaneously, or press the function key marked **CENTER**. The menu disappears when you release the mouse buttons or the **CENTER** key.

There is usually more than one menu associated with any cursor position on the display; when this is the case, dynamic menus are stacked on top of each other. The top menu covers all but the name frames of the other menus. Figure 3.13 shows a menu stack.

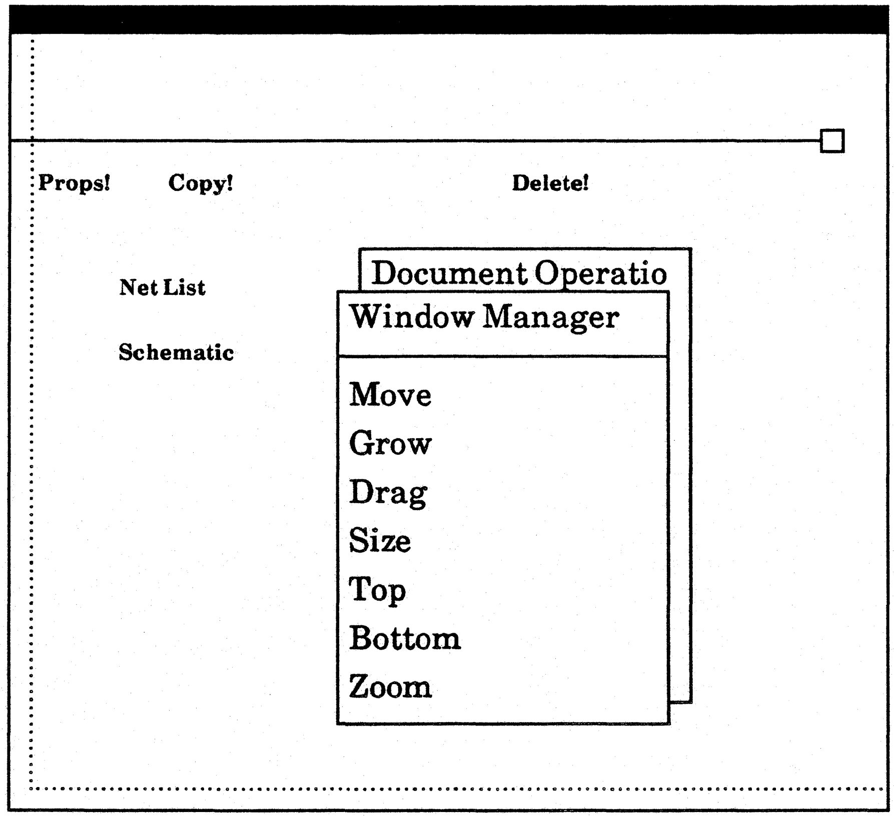


Figure 3.13

- To select a command from a menu that is not on top of the stack, you must first select that menu from the stack. Selecting a stacked menu brings that menu to the front of the stack. There are two ways to select a menu from the stack.

1) Press and hold both mouse buttons or the **CENTER** key to activate the stack of menus.
2) The cursor changes into an arrow pointing at the menu. Point the cursor at the name frame of the menu you wish to see. The system displays that name frame in inverse-video.
3) Release the mouse buttons or **CENTER** key.
4) Display the menus again. The menu you selected is on top.

Or

1) Press and hold both mouse buttons to activate the stack of menus.
2) The cursor changes into an arrow pointing at the menu. Release the left mouse button. As long as you hold the right mouse button, the menus remain active.
3) Point the cursor at the name frame of the menu you wish to see. The system displays that name frame in inverse-video.
3) Press and release the left mouse button.
4) The system displays the menu you selected at the top of the stack.

- There are also two ways to select a command from the chosen menu:

1) Press both mouse buttons or the **CENTER** key to activate the menus.
2) The cursor changes into an arrow pointing at the menu.
3) Point the cursor at the command you want to issue. The system highlights that command.
4) Release the mouse buttons or the **CENTER** key.

Or

1) Press both mouse buttons to activate the menus.
2) The cursor changes into an arrow pointing at the menu. Release the left mouse button.
3) Point the cursor at the command you want to issue. The system highlights that command.
4) Press and release the right mouse button. Notice that the menu remains active. Repeat this procedure to issue a second command, or move the cursor away from the menu and release the left mouse button.

3.3.1 The Window Manager Menu

Every window has at least one dynamic menu associated with it: the *Window Manager Menu*. The Window Manager lets you "shuffle" windows on the display as you shuffle papers on a desk. Using the Window Manager, you can move a window, change its shape and size, and make it active or tiny. Figure 3.14 shows you the Window Manager Menu.

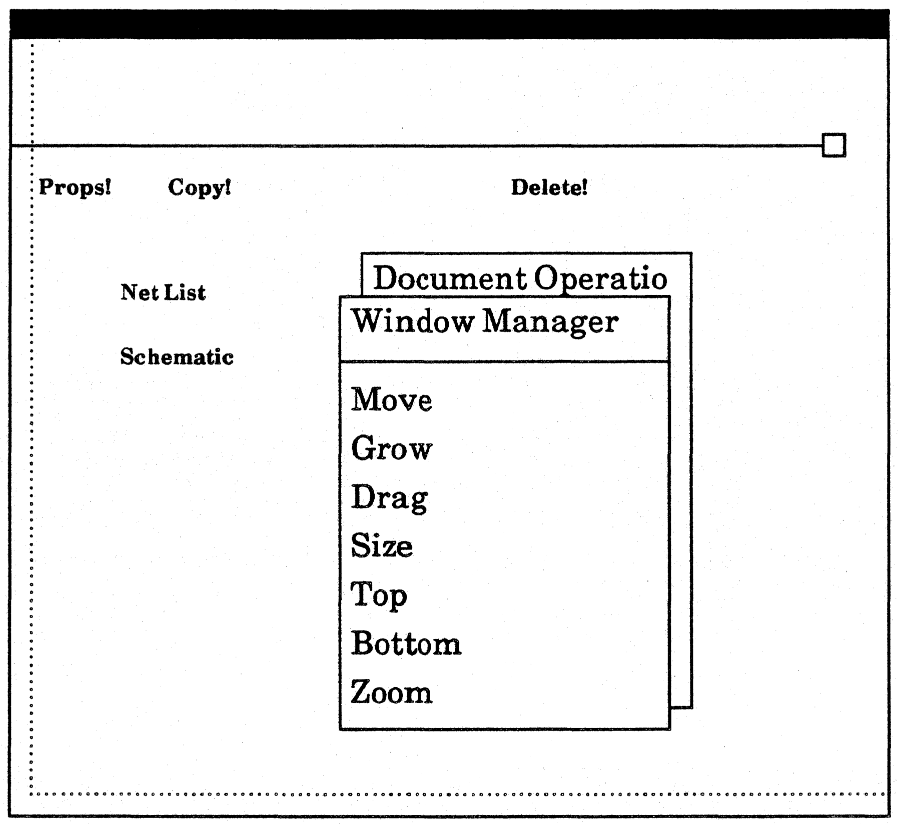


Figure 3.14

Move

- To move a window:

1) Select **Move** in the window's Window Manager Menu.
2) The cursor changes into the shape of an "M" inside a corner bracket.

3) Move the cursor to show the system where you want to move the window. As you move the cursor from one corner of the display to another, the cursor changes shape. The cursor's position and shape mark the new corner. See figure 3.15.

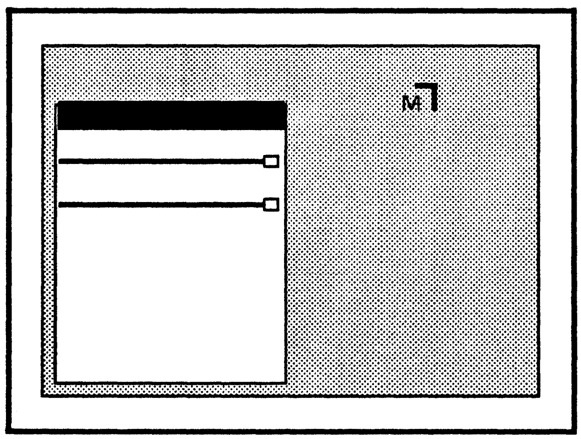


Figure 3.15

4) When you've determined where you want the window, press the left mouse button.
5) The window moves, and the cursor returns to its normal shape. The size and shape of the window remain unchanged. See figure 3.16.

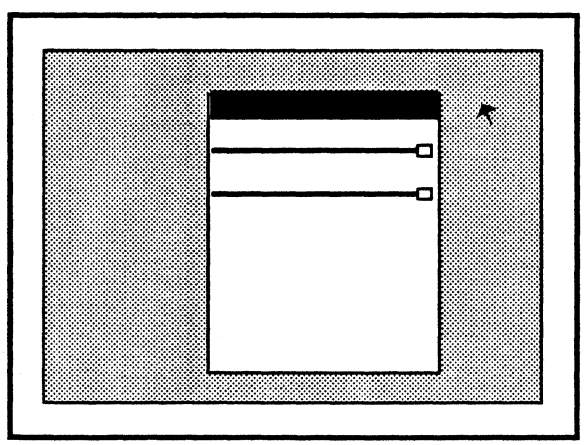


Figure 3.16

Grow

- The **Grow** command changes the size of a window by stretching two sides of the window.

1) Select **Grow** in the Window Manager Menu.
2) The cursor changes into the shape of a "G" inside a corner bracket.

3) As you move the cursor on the display, it changes shape to show you the new corner. (Figure 3.17).

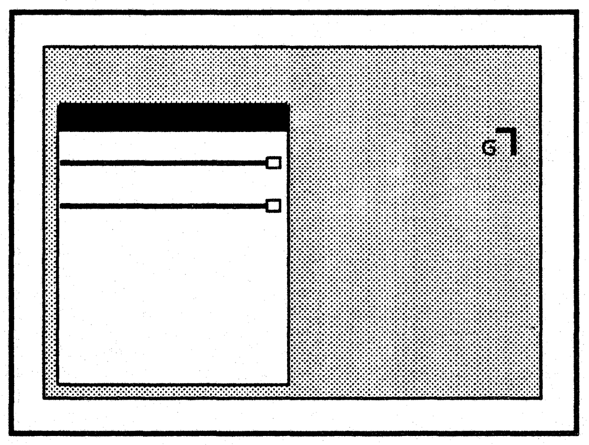


Figure 3.17

4) Press the left mouse button to stretch the sides to the new size. See figure 3.18.

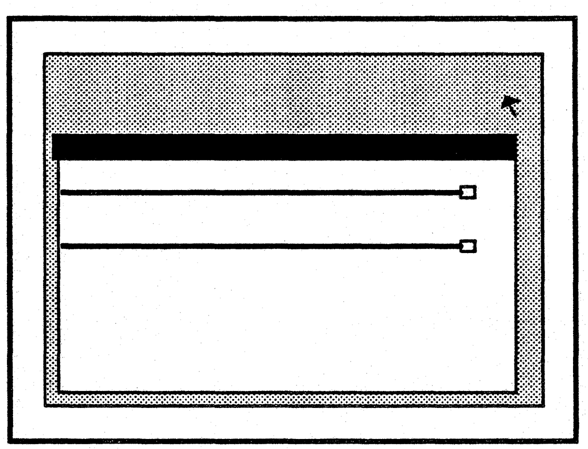


Figure 3.18

Drag

- The **Drag** command changes the size of a window by stretching only one side of the window.

1) Select **Drag** in the Window Manager Menu.
2) The cursor changes into an arrow pointing to a line.

3) Move the cursor across the border you want to stretch. As you move the cursor across the window's border, the cursor changes shape to show you that the system will drag that border. When the arrow points up, the upper border stretches. When the arrow points down, the lower border stretches. When the arrow points to the left, the left border stretches, and when it points to the right the right border stretches. The line to which the arrow points marks the new window border. See figure 3.19.

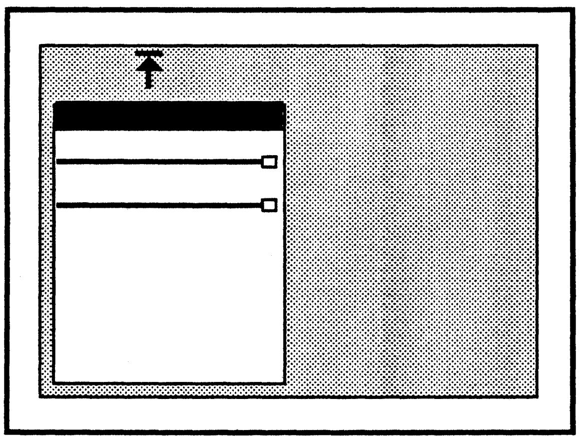


Figure 3.19

4) Press the left mouse button to stretch the side indicated to the new location (figure 3.20).

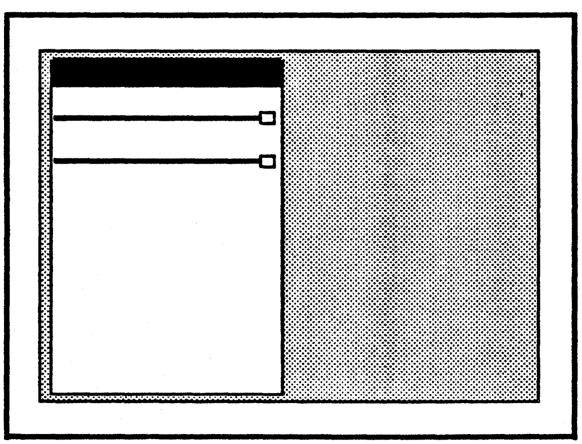


Figure 3.20

Size

- To make an active window tiny or a tiny window active:

1) Select **Size** in the Window Manager Menu. See figure 3.21.

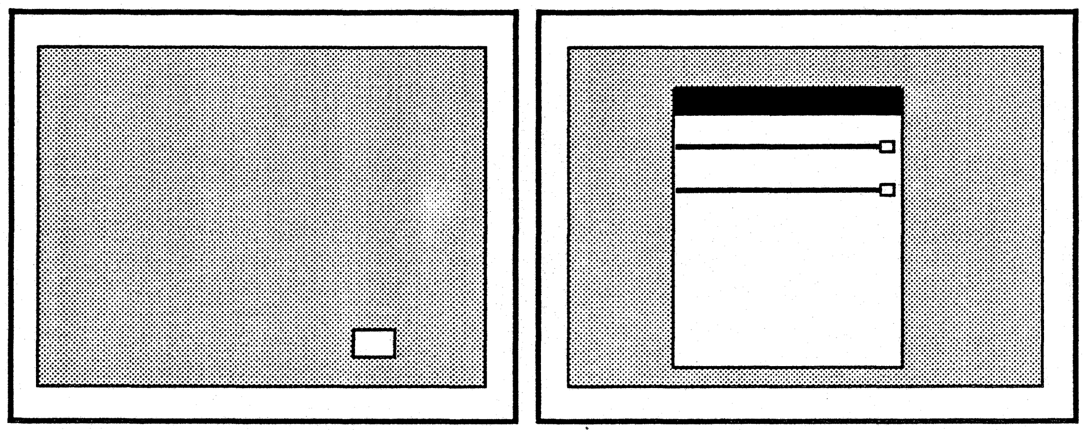


Figure 3.21

Top

- To place a window on top of all other windows on the display:

1) Select **Top** in the Window Manager Menu. See figure 3.22.

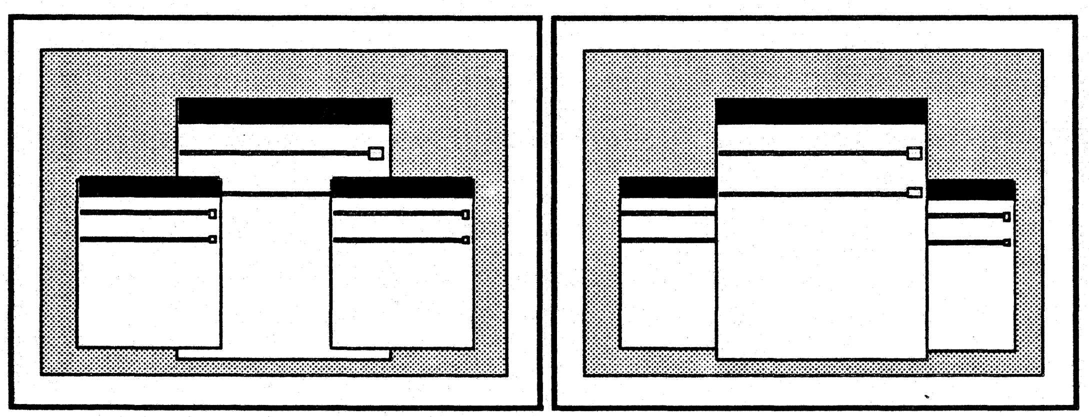


Figure 3.22

Bottom

- To place a window beneath all other windows on the display:

  1) Select **Bottom** in the Window Manager Menu. Figure 3.23 shows a window before and after you issue the **Bottom** command.

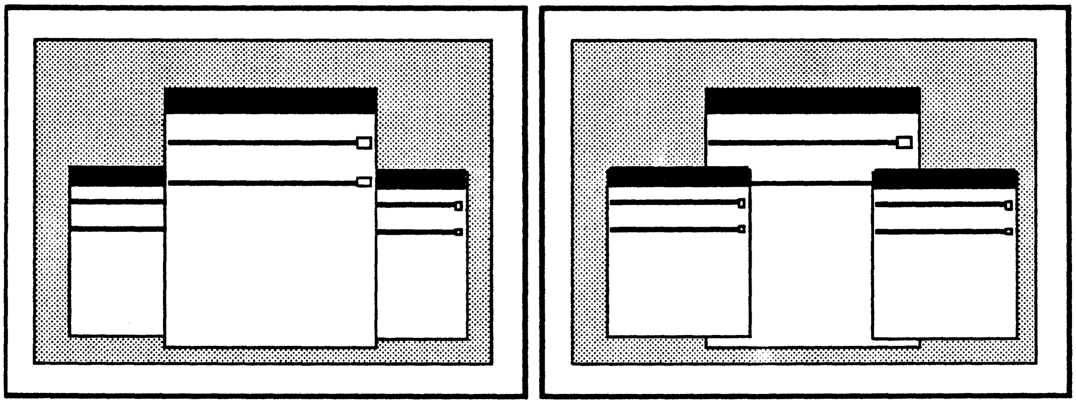


Figure 3.23

Zoom

- To make a window take up the entire display area, or to return it to its normal size from that state:

  1) Select **Zoom** in the Window Manager Menu. Figure 3.24 shows a window before and after you issue the **Zoom** command.

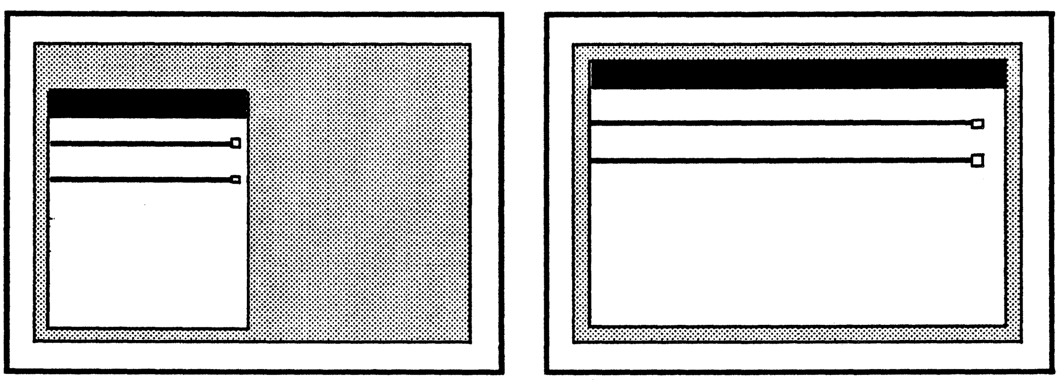


Figure 3.24

3.3.2 The Inactive Menu

The *Inactive Menu* is a special dynamic menu associated with the empty (grey) area on the display (figure 3.25). The Inactive Menu contains lists of Expert 1000 tools that you use less frequently, for example the File Tool and the Abbreviation Map. To open one of these tools, you select its name in the Inactive Menu.

- To activate the Inactive Menu:

1) Move the cursor to an empty area on the display, so that the cursor is not in any window.
2) Press both mouse buttons or the **CENTER** key.

You select an item in the Inactive Menu in the same way that you select an item in any menu. When you select an inactive menu item, the system creates a window displaying the item you selected. To store that window once again, select **Close!** in the window's command subwindow. Only select Inactive Menu items described in the *User's Guide* (**File Tool, Abbreviation Map, Library Editor, Bye**). Do not select other Inactive Menu items.

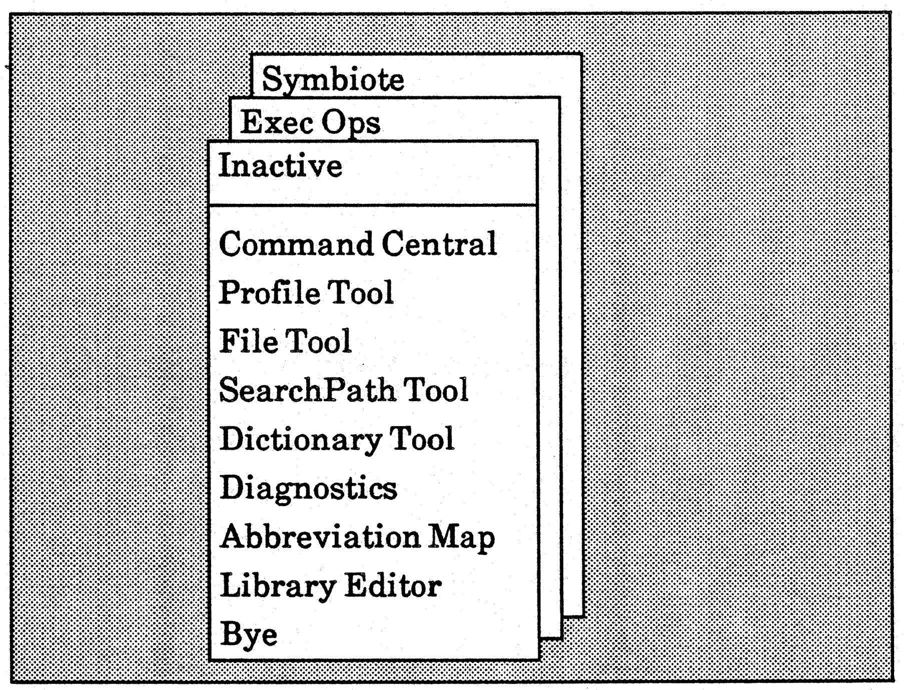


Figure 3.25

3.4 Accelerators

Accelerators are "shortcuts" that you use to give the system commands without using a menu. Accelerators are designed for users who are familiar with the system, use it frequently, and want a faster method of issuing commands. Unlike menu commands, accelerators never appear on the display; you must memorize them.

All of the Window Manager commands, for example, have accelerators. Like the Window Manager Menu, Window Manager accelerators are common to all windows.

3.4.1 Window Manager Accelerators

In general, you issue a Window Manager command with an accelerator by positioning the cursor in the left, middle, or right region of the window's name frame and pressing one of the mouse buttons. See the figure 3.26 below.

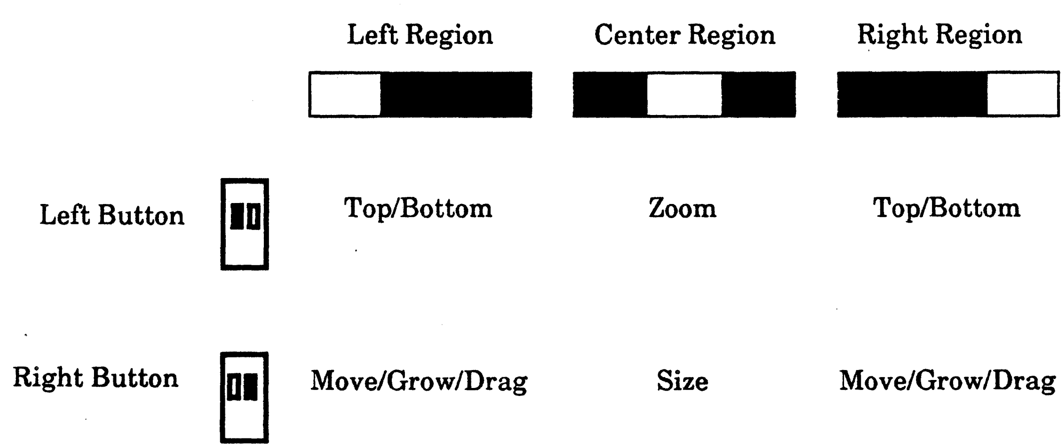


Figure 3.26

- To issue a Window Manager command with an accelerator:

  1) Move the cursor to the name frame region corresponding to the command. If the window is tiny, move the cursor to the left, right, or center third of the window near the upper border as if a name frame was there.
  2) The cursor changes its shape to a circle with a point in the middle.
  3) The system displays the region in inverse-video.
  4) Press the mouse button corresponding to the command you want to issue.

Figure 3.27 shows the cursor inside the middle region of the name frame.

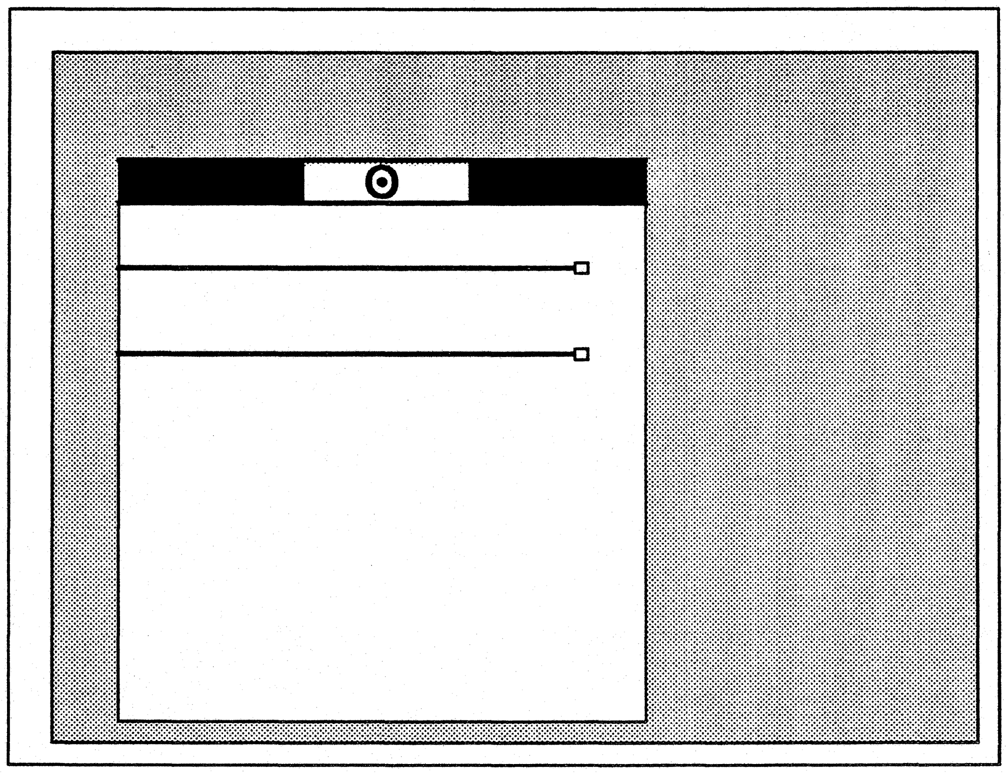


Figure 3.27

Top/Bottom

- To place a window that is on top of other windows beneath them, or to place a window that is beneath others on top of them:

1) Move the cursor to the left or right region of the name frame.
2) Press the left mouse button.

Move/Grow/Drag

- To **Move, Grow**, or **Drag** an active window:

1) Move the cursor to the left or right region of the name frame.
2) Press the right mouse button; do not release it.
3) The cursor changes to an "M" inside a corner bracket.

4) While holding down the right mouse button, press the left button.
5) The cursor changes to a "G" inside a corner bracket.

6) Press the left mouse button again.
7) The cursor changes to an arrow pointing to a line.

8) As you continue to press the left mouse button, the cursor continues cycling through the three shapes. Press the left button until you see the cursor corresponding to the Window Manager command (**Move**, **Grow**, or **Drag**) you want to issue.
9) Move the cursor to show the system where to move, grow, or drag the window.
10) Release the right button.

- You can also use the **Move/Grow/Drag** accelerator to move tiny windows on the display.

1) Move the cursor to the left or right third of the window below the upper border.
2) Press and hold the right mouse button.
3) The tiny window becomes the cursor.
4) While pressing the right button, move the cursor to the new location for the window.
5) Release the right button.

Zoom

- To make a window take up the entire display, or to make such a window return to normal size:

1) Move the cursor to the center region of the name frame or tiny window.
2) Press the left mouse button.

Size

- To make an active window tiny, or a tiny window active:

1) Move the cursor to the center region of the name frame or tiny window.
2) Press the right mouse button.

3.5 Text Editing

Some subwindows contain graphic items, and others contain text. You often need to edit text, for example to change a command parameter, alter a property value (section 3.6), or edit a text file. This section explains text files, file windows, and the techniques you use to edit text.

3.5.1 Text Files and File Windows

Text files are files of textual information that you can create and keep at your workstation. Creating a text file at your workstation is similar to writing a report on a piece of paper, and keeping it in a file drawer at your desk.

You create and edit text files in a special type of window known as a *file window*. A file window has two command subwindows, and a third *text subwindow* in which you display the text file. See figure 3.28.

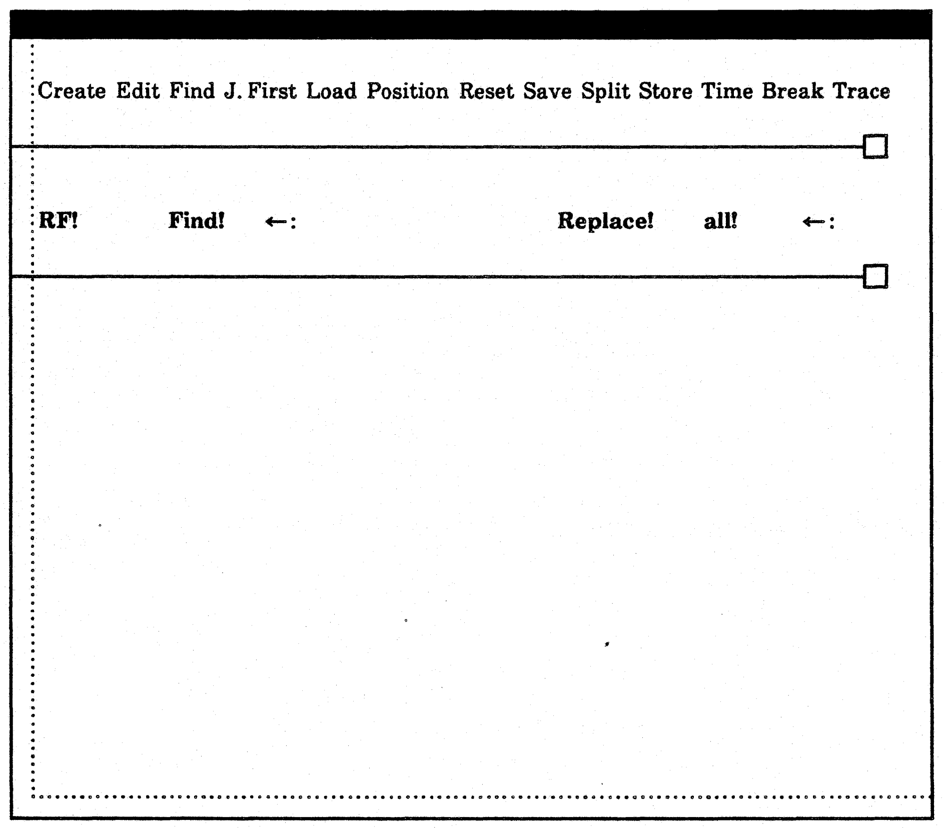


Figure 3.28

Creating a File Window

- To create a file window:

1) Move the cursor to an empty area on the display and press both mouse buttons to activate a menu stack. Select the Exec Ops Menu.
2) Activate the Exec Ops Menu. Select **File Window**.
3) The system asks you to confirm the **File Window** command. The cursor changes to the shape of a mouse. Press the left mouse button to confirm the command. The system creates an empty window at the point where you pressed the left mouse button.

The File Window Menu

The File Window Menu is associated with the third subwindow in the file window (the text subwindow). The File Window Menu contains commands for manipulating the text file displayed in the window. Figure 3.29 shows the File Window Menu.

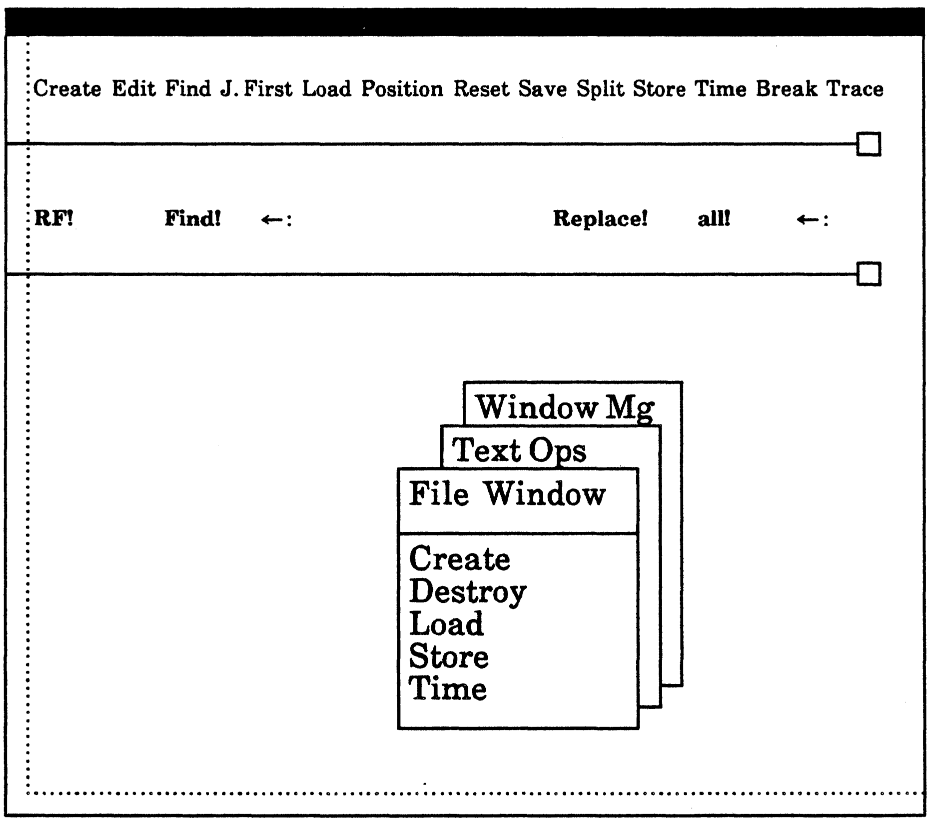


Figure 3.29

File Window Menu commands change as you perform different tasks in the file window. The File Window Menu displays only those commands relevant to what you are doing in the file window.

Create

The **Create** command creates a new file window.

- To create a file window with the **Create** command:

1) Bring up the File Window Menu.
2) Select **Create**.
3) The cursor changes to the shape of a mouse. Press the left mouse button to confirm the **Create** command, and to indicate the new window's position on the display. The system displays the new file window at the point where you pressed the left mouse button.

Destroy

The **Destroy** command removes the file window from the display.

- To remove a file window:

1) Bring up the File Window Menu.
2) Select **Destroy**.
3) The cursor changes to the shape of a mouse. Press the left mouse button to confirm the **Destroy** command. The system removes the file window from the display.

Time

When you select the **Time** command the system writes the date and time into the text subwindow. For example: 13-May-83 10:23:12.

Store

The **Store** command instructs the system to store a new text file.

To store a file:

1) Select the file name. If the name appears in the text subwindow, select it there; if not create an empty file window, type the file name it the new file window's text subwindow, and select it.
2) Select **Store**.

Load

The **Load** command instructs the system to load an existing text file into the file window.

- To load a file:

1) Type the file name in the file window's text subwindow.
2) Select the file name.
3) Select **Load**.

Edit

After loading an existing file, use the **Edit** command to tell the system you want to change the text in the file.

- To edit a file:

1) Load the file into a file window.
2) Select the **Edit** command.
3) Use the text manipulation techniques described in section 3.5.2 to change the text file.

When you finish editing the file, use the **Save** command to instruct the system to save your changes.

Save

The **Save** command appears in the File Window Menu only when you are editing a file. It instructs the system to save the changes that you made to the file.

- To save your changes:

1) Select **Save**.

Reset

If you are not editing a file, the **Reset** command removes all text from the text subwindow. If you are editing a text file (you have loaded the file and issued the **Edit!** command) the **Reset!** command restores the file to its original state before you began editing it.

- To reset the file window:

1) Select **Reset**.

The Command Subwindow

The first file window command subwindow lists Text Ops Menu and File Window Menu commands. The command name in the command subwindow acts as an accelerator for the command. For instance, you can **Split** the text subwindow without activating the Text Ops menu. You simply select **Split** in the command subwindow.

You cannot issue a command in this subwindow unless the command currently exists in the dynamic menu. For example, the **Edit** command is available only after you have loaded a file into the window. Therefore, even though the **Edit** command may appear in the command subwindow, you cannot issue it unless it also exists in the dynamic menu.

You can add menu items to the command subwindow, and delete items from the subwindow.

- To add an item:

1) Hold the **PROP'S** key and select the type-in point with the left mouse button.
2) Type the File Window Menu or Text Ops Menu command name.

- To remove a command from the list:

1) Hold the **PROP'S** key and select the type-in point with the left mouse button.
2) Use the backspace or backword key to delete the command name.

Creating a New File

- To create a new text file:

1) Create a new file window.
2) Move the cursor into the text subwindow. Press the left mouse button.
3) Enter the text. Use the text manipulation techniques discussed in section 3.5.2.
4) Store the file.

Editing an Existing File

- To edit an existing file:

  1) Move the cursor into the third subwindow of the empty window and press the left mouse button.
  2) Type the name of the file.
  3) Select the letters you typed.
  4) Select **Load** in the empty window's first subwindow. The system displays the text file.
  5) Select **Edit** in the first subwindow, and make your changes.
  6) When you finish editing the file, select **Save**.

3.5.2 Text Manipulation

You can edit the text in a text subwindow by selecting a portion of it and adding or deleting characters. In addition, the system supplies you with commands to manipulate the text subwindow, search for and replace patterns in the text.

Selecting Text

- To select a single character in the text body, point the cursor at the character and press the mouse's left button. Press the button twice to select the word, three times to select the line, four times to select the whole body of text, and five times to select just the character once again.

- You can *extend* a text selection to the left or right two ways:

  1) Point to the endpoint of the selection and press the mouse's right button.

  Or

  2) Move the cursor while holding the mouse's right button. Release the button when the cursor points at the endpoint of the selection.

The Current Selection

There is only one *current selection* at any time (not one per window); the system often uses the current selection as the argument to text editing commands you issue. The current selection applies only to text in text subwindows; a selected command, for example, is not a current selection.

Entering Text

- The system automatically marks a *type-in point* with a flashing caret at the end of the selected text. If you select just one character, the system marks the type-in point to the side of the character nearest the cursor, either before or after the character.

- To add text:

  1) Select the type-in point, then type the text using the keyboard.

  When entering text in an empty file window, move the cursor into the text subwindow and press the left mouse button. The system displays the type-in point at the top of the subwindow. Type the text.

Deleting Text

- To delete text, select the text you want to delete, and press the key marked **DELETE**. If you accidentally delete text, you can replace it by pressing the **PASTE** key (marked **SAME**). You can also use the backspace key (above the carriage return key) to delete the character immediately preceding the type-in point. The backword key (Press **SHIFT** and backspace simultaneously) deletes the word preceding the type-in point. You cannot replace text you delete with the backspace or backword keys.

Copying Text

- There are three ways to copy text within a text subwindow or from one subwindow to another:

  1) Select the text.
  2) Hold down the **PROP'S** key and use the left mouse button to select the type-in point where you want to copy the text. Release the **PROP'S** key.
  3) Press the **STUFF** key (marked **OPEN**).

  Or

  1) Select the text.
  2) Press **DELETE**.
  3) Press the **PASTE** key (marked **SAME**) to replace the text.
  4) Use the mouse to move the type-in point to where you want to copy the text.
  5) Press **SAME** once again.

  Or

  1) Move the type-in point to the desired location.
  2) Hold down the **COPY** key.
  3) Select the text you want to copy.
  4) Release the **COPY** key.

3.5.3 The Text Ops Menu

The *Text Ops Menu* provides commands for manipulating text placement inside a text subwindow. Figure 3.30 shows the Text Ops Menu.

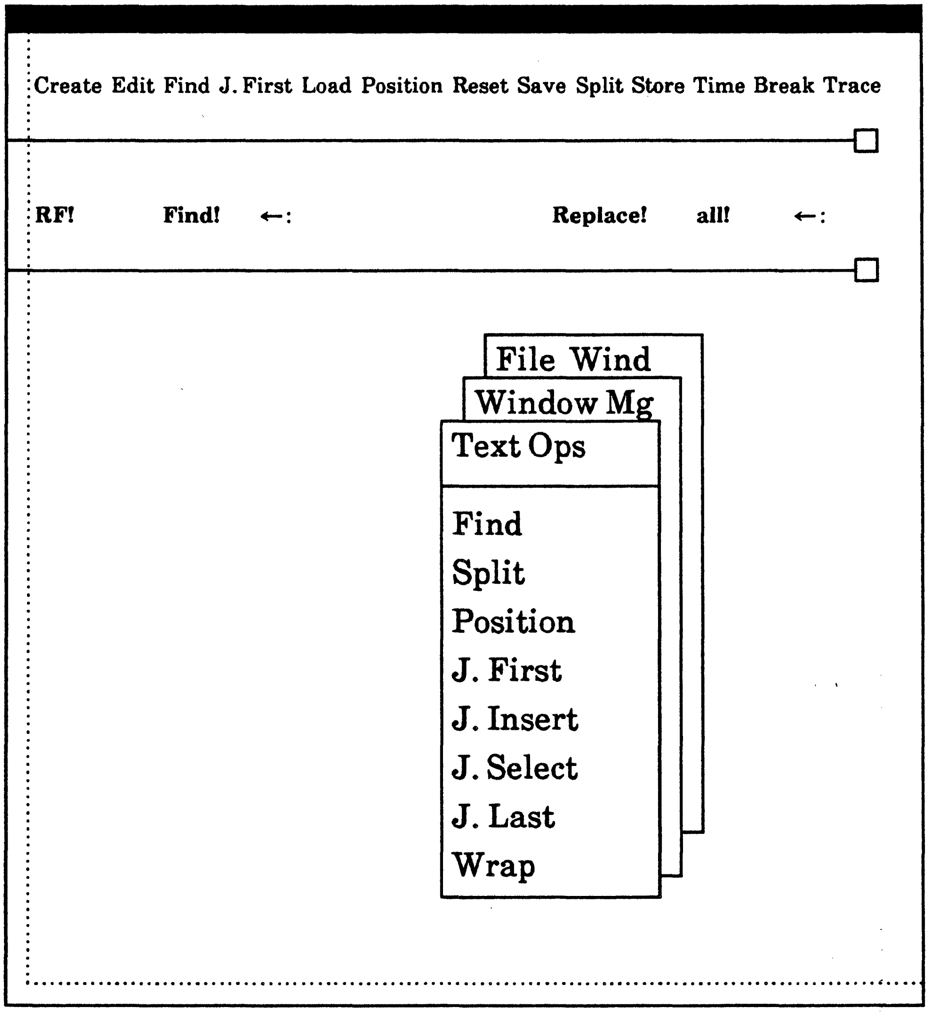


Figure 3.30

- To display the Text Ops Menu:

  1) Move the cursor inside the text subwindow.
  2) Press both mouse buttons or the **CENTER** key.

Find

The **Find** command finds the next occurrence of the current selection in the subwindow. If the current selection is in the subwindow, the search begins at the end of the selection; otherwise it begins at the first character visible in the subwindow. If the search is successful, the next occurrence of the text becomes the new selection. The search continues into text not visible on the screen; if found, the system scrolls the text to the top of the subwindow. If the system finds no further instances of the text, the display blinks.

If you hold down the **SHIFT** key while selecting the **Find** command, the system searches backwards from the current selection, or from the last character visible in the window.

Split

The **Split** command divides a region of the subwindow into two subwindows separated by a dashed line with a small box at the right end of the line. You can scroll the subwindows independently of each other, and move the border to adjust their sizes. Remove the line by moving it off the top or bottom of a subwindow. Figure 3.31 shows a split text subwindow.

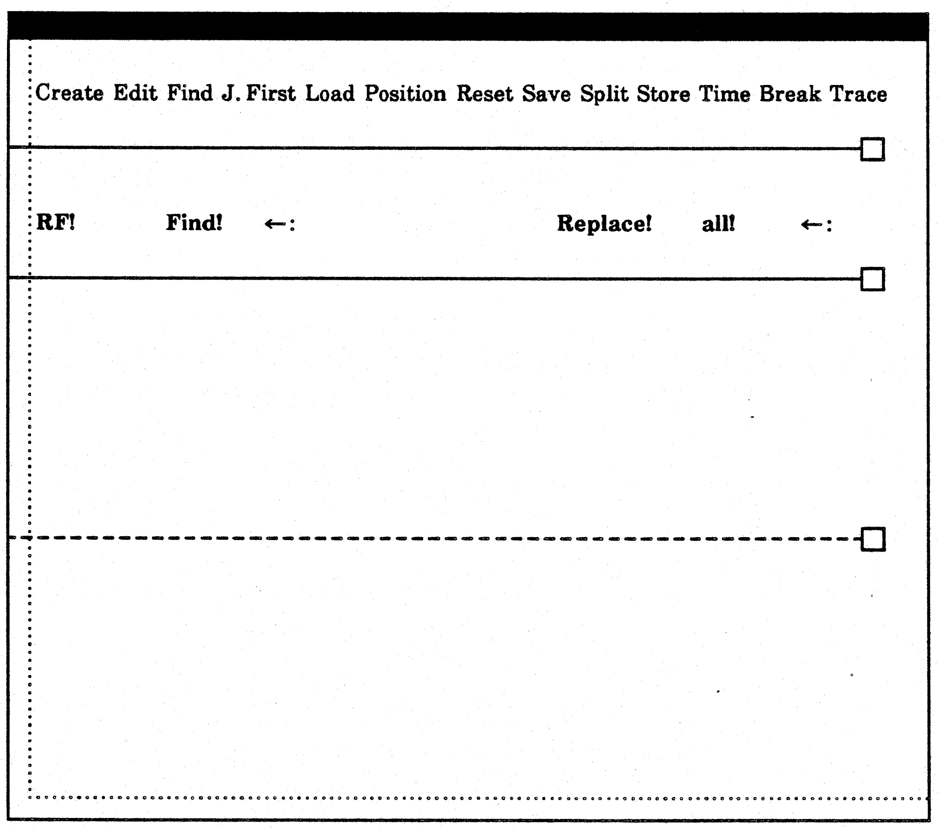


Figure 3.31

Position

Use the **Position** command to position the text in the subwindow so a specific character is at the top. You select a positive number in any text subwindow to specify a character in the file, then select the **Position** command. For example, to place the 275th character at the top of the subwindow, select the number 275 (either type it in the subwindow or in a second file window), and select **Position**.

J. First

The **J. First** command positions the text in a subwindow so that the first line of text is at the top of the subwindow.

J. Insert

The **J. Insert** command positions the text in the subwindow so that the type-in point is at the top.

J. Select

The **J. Select** command positions the text in the subwindow so that the line containing the leftmost character of the current selection is at the top.

J. Last

The **J. Last** command positions text in a subwindow so that the last line of text is at the top of the subwindow.

Wrap

Text subwindows have two states of line *wrap-around.* When wrapping is on, a line that you have not terminated with a carriage return by the time it reaches the right-hand edge of a subwindow continues onto the next line. When wrapping is off, the same line disappears off the right edge of the subwindow. The **Wrap** command reverses the current state of line wrap-around in all subwindows you created from the same subwindow with the **Split** command.

3.5.4 Pattern Matching: Find and Replace

The file window's second subwindow lets you perform pattern matching and replacement in text subwindows.

- You can instruct the system to look for a particular pattern of characters, for instance a word.

1) Use the right mouse button to select the arrow following the **Find!** command. Leave the cursor inside that subwindow.
2) Type the text for which you want to search.
3) Select **Find!** The system searches for the pattern, and selects it.

The pattern you entered after the arrow is called **Find!** *buffer*. If you issue the **Find!** command while the buffer is empty, the system searches for the current selection instead. In this case, the system fills the buffer with the current selection.

- You can also instruct the system to replace one pattern with another, for example, to substitute one word for another.

1) Select the pattern you want to replace, or enter the pattern you want to replace in the find buffer. (Use the right mouse button to select the arrow following the **Find!** command. Type the text you want to replace.)
2) Use the right mouse button to select the arrow following the **all!** command. Type the pattern you with which you want to replace the first pattern.
3) Select **Replace!**

The **all!** command does successive replaces until there are no more matches or until you press the **ABORT** key (marked **STOP**).

The **Find!** and **Replace!** commands operate on the window you are editing. If you have split the window, they operate on the split that contains the current selection; if the selection is in another window, or if there is no selection, they operate on the first split.

The **RF!** command is equivalent to **Replace!** followed by **Find!** It replaces the selected text with the text you enter following the **all!** command, then it finds and the next instance of the selected text.

3.6 Property Sheets

Most graphic and text objects inside subwindows have certain *properties* associated with them. For example, a logic symbol in a schematic has a library identification number, a library name, a symbol designator, a physical designator, a physical part name, pin numbers, physical device parameters, and a micro-note. Any window that displays a schematic or documentation set also has properties. You can change properties in a special type of window known as a *property sheet*. Figure 3.32 shows a logic symbol property sheet from an Expert 1000 schematic.

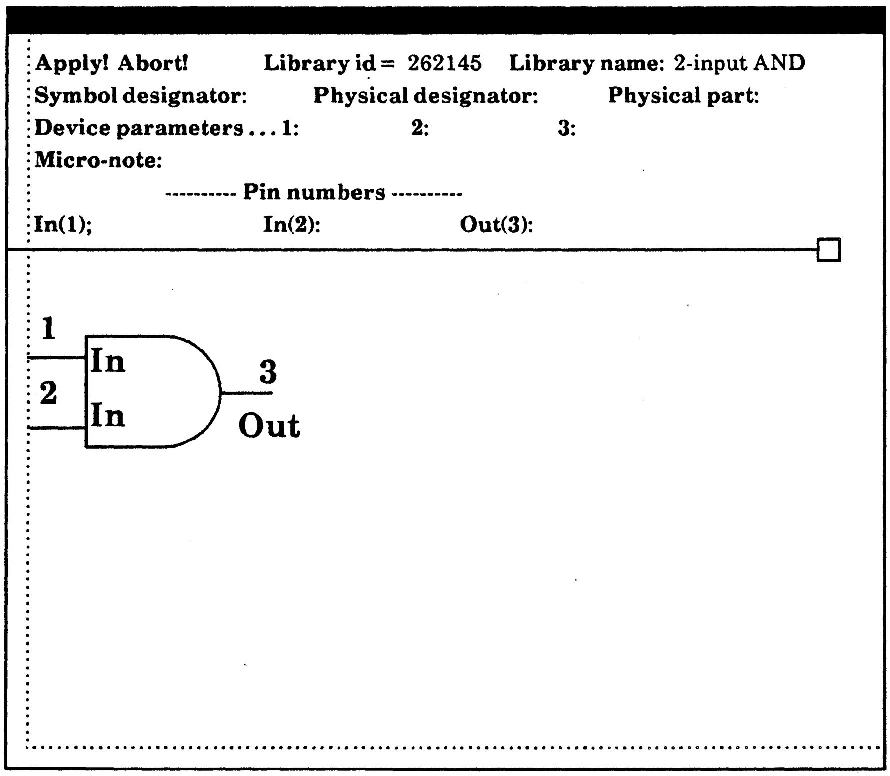


Figure 3.32

- To display the property sheet for a window:

  1) Select the **Props!** command in the window's command subwindow.

- To display the property sheet for a graphic or text object inside a subwindow:

  1) Select the object
  2) Press the function key marked **PROP'S**.

A property sheet window usually has just one subwindow. This subwindow lists the names and values of the object or window properties. The property name appears in bold type, and the property value appears in regular type.

While you display a property sheet for an item inside a window, the system may not let you issue commands associated with the window. You usually enter the item's properties and close the property sheet before issuing other commands.

There are four types of properties. The procedure for changing a property differs for each property type.

3.5.1 Boolean Properties

Some properties are simply listed in the property sheets. These are properties that you can turn on or off, like a toggle switch. Often a description followed by three dots precedes a boolean property. Figure 3.33 shows boolean properties. If the name of the property is in inverse-video, the property is on, otherwise it is off.

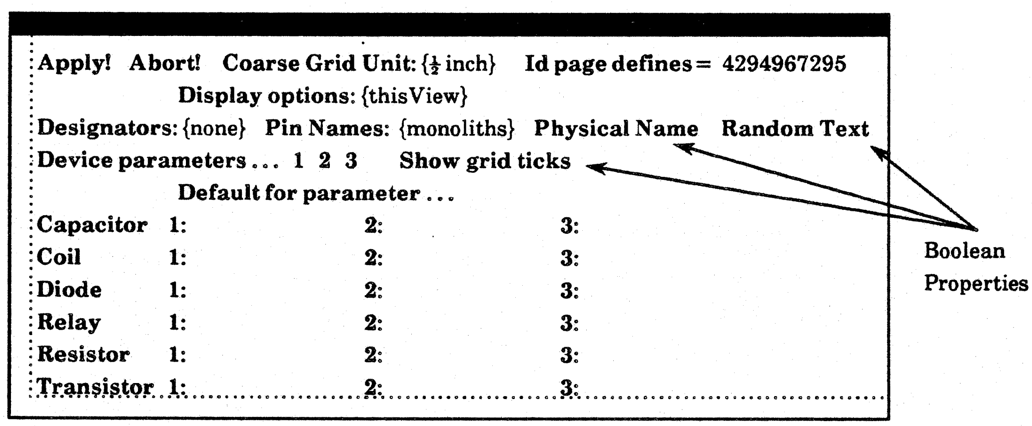


Figure 3.33

- To turn a boolean property on or off:

1) Point the cursor at the property.
2) Press either mouse button.

3.5.2 Property Lists

A *property list* consists of a property name and several possible property values. (See figure 3.34.) You choose a property value from the list of possibilities. There are two ways to do this, depending on how Expert 1000 displays the property list.

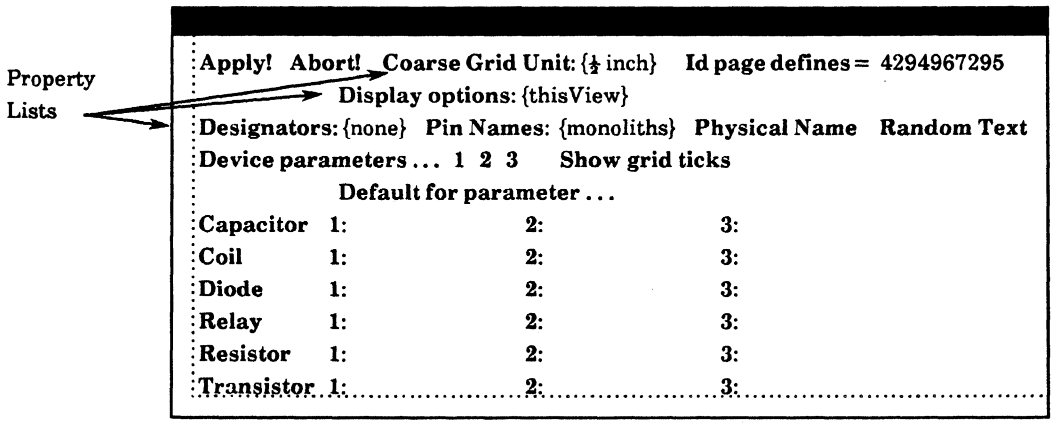


Figure 3.34

- If there is only one value inside the brackets, the other possible values are contained in a dynamic menu associated with the property name.

  1) Point the cursor at the property name and activate the menu.
  2) Select a value from the menu. Notice that Expert 1000 displays the value you selected inside the brackets.

- If the property name is followed by a list of values inside brackets, you can select one value from that list, or select the value from an associated menu. To select the value in the list:

  1) Turn the value on as if it were a boolean property. The system video-inverts that value.

3.5.3 Text Properties

A *text property* consists of a property name followed by one or several words. The text might be the name or a description of the object with which the property sheet is associated. Figure 3.35 shows text properties.

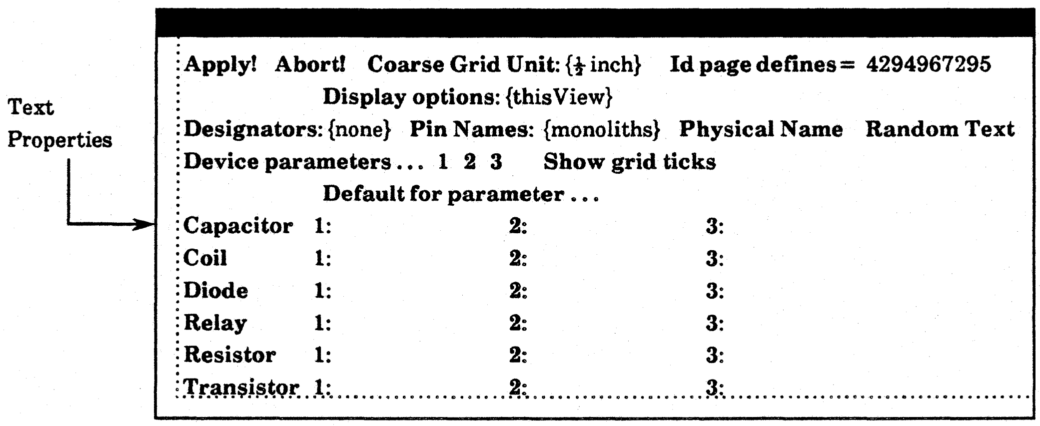


Figure 3.35

Use the text manipulation techniques described in section 3.5.2 to change text properties. You can also use these special text editing techniques for property sheets:

- When you select any character in the property name with the left mouse button, the system automatically selects the whole body of text following the name.
- You can use the **SKIP NEXT** key in the right function group to automatically move the type-in point to the next text or numeric property in the property sheet. The system automatically deletes that property's value. You type the new value.
- When you select any character in the property name with the right mouse button, the system automatically deletes the whole body of text following the name.

3.5.4 Numeric Properties

Numeric properties are properties whose values may be only numbers. For example, the width of a line in a schematic is a numeric property. An equal sign (=) following a property name distinguishes a numeric type from a text type (see figure 3.36). Use the text editing techniques above to change the value of a numeric property type.

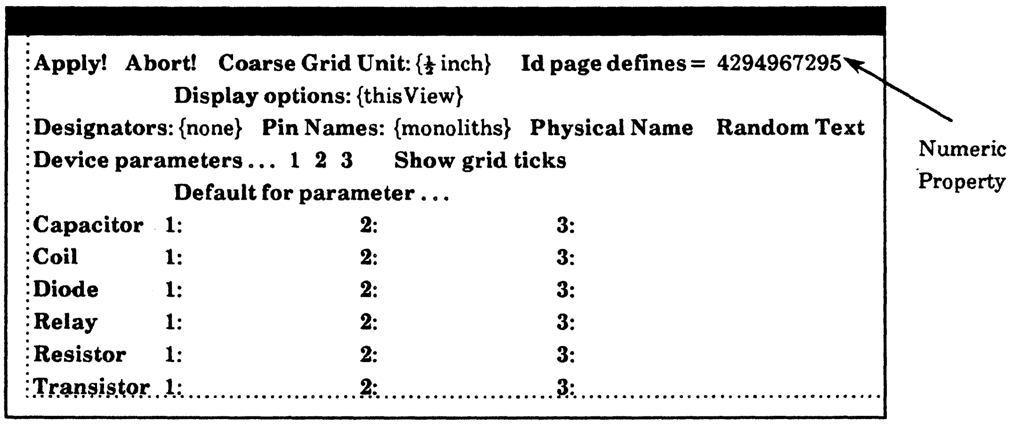


Figure 3.36

3.5.5 Apply! and Abort!

Below the property sheet's name frame are two commands: **Apply!** and **Abort!** Both close the property sheet.

- To apply the changes that you made in the property sheet:

  1) Select **Apply!**

- To erase the changes you made:

  1) Select **Abort!**

4. The Documentation Set

A documentation set is the collection of data for a particular design. An Expert 1000 documentation set window resembles a folder in which you place a schematic and its associated net list, parts lists, and other reports. Each Expert 1000 schematic has its own documentation set.

When you first log on to the system, several tiny windows appear at the bottom of the display. These are documentation set windows. Each has a name, such as D-Size-Template, or E-Size-Template. Most likely, there is one documentation set for each schematic format you use. Use these empty documentation sets as templates. When you begin a new schematic, you copy the documentation set containing a schematic with the format you want to use.

An active documentation set window contains a system message subwindow, and a command subwindow that lists the contents of the documentation set. Figure 4.1 shows a documentation set window with the Document Operations Menu.

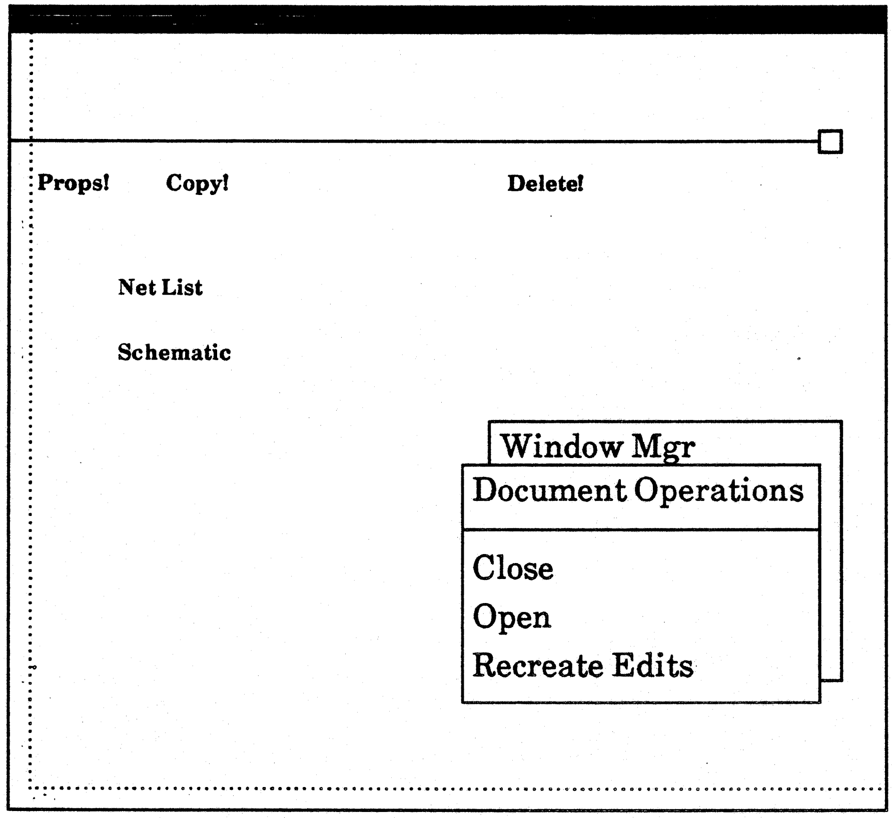


Figure 4.1

4.1 Copying a Documentation Set

- To copy a documentation set:

1) Decide which of the empty, tiny documentation sets contains a schematic with the format you need. Use the Window Manager Menu or the Window Manager accelerators to activate that documentation set window.
2) Select the **Copy!** command in the second subwindow. The system creates a new, tiny documentation set window at the bottom of the display, and gives it a temporary name. (The system names the new documentation set "Temporary000" if it is the first temporarily-named documentation set at your workstation, "Temporary001" if it is the second, "Temporary002" if it is the third, etc.)

4.2 Naming a Documentation Set

- To name a documentation set:

1) Select the **Props!** command in the documentation set's second subwindow. The system displays the documentation set's property sheet.
2) The property sheet contains only one property: the documentation set name. Use the text editing techniques described in section 3.5 to change the documentation set name. Documentation set names may not include periods, spaces, or brackets.
3) When you finish entering the name, select **Apply!**. The system closes the property sheet, and displays the new name in the documentation set's name frame.

4.3 Deleting a Documentation Set

- To delete a documentation set:

1) Select the **Delete!** Command in the documentation set's second subwindow.
2) The system asks you to confirm the **Delete!** command. The cursor changes into the shape of a mouse. If you are certain that you want to delete the documentation set, press the left mouse button. Press the right mouse button if you do not want to delete the documentation set.
3) The system asks you to confirm the **Delete!** command again. If you are still certain that you want to delete the documentation set, press the left mouse button. Press the right mouse button if you do not want to delete the documentation set.

4.4 The Document Operations Menu

The Document Operations Menu lets you manipulate the net list and schematic stored in the documentation set.

- To activate the Document Operations Menu:

1) Place the cursor inside the documentation set command subwindow.
2) Press both mouse buttons or the **CENTER** key.

The Document Operations Menu lists three commands: **Close**, **Open**, and **Recreate Edits**. You use the **Open** command to open the documents stored in the documentation set. You use the **Close** command to close documents. The **Recreate Edits** command lets you replay changes that you made in a document. Book 2 discusses these commands in context and in more detail.

5. The File Tool and the Tape Tool

5.1 The File Tool

The File Tool lets you transfer files from your local disk to remote servers, and from servers to your local disk. You use the File Tool, for example, to store a completed documentation set on a file server. That file server acts as a large filing cabinet in which there are several drawers known as directories. When you store a documentation set you specify the directory name as well as the file server name.

When you first start the system, the File Tool is a tiny window on your display. Use the Window Manager's **Size** command to make the File Tool window active.

5.1.1 The File Tool Window

The File Tool window contains four subwindows. The first displays messages from the system. The second contains fields for entering command parameters. The third contains File Tool commands, and the fourth displays file lists. The List Options window is a property sheet window associated with the File Tool. Figure 5.1 shows the File Tool and List Options windows.

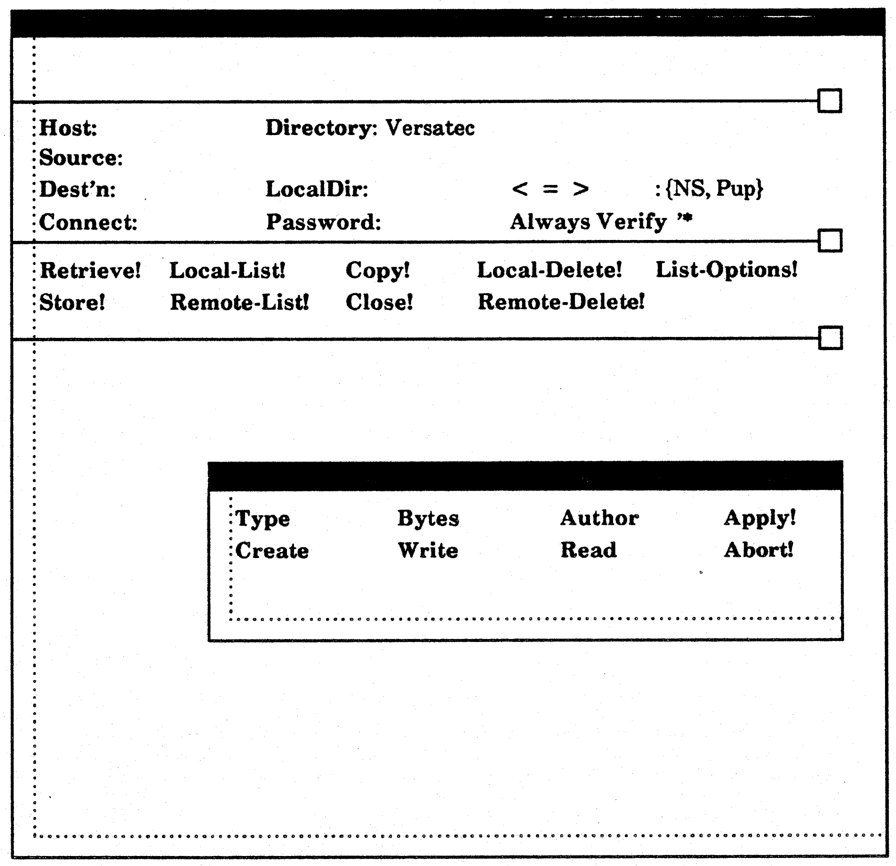


Figure 5.1

5.1.2 Command Parameters

The second, command parameter, subwindow acts as a property sheet containing text and boolean properties. Each text parameter has a name. You enter the parameter value after its name. Use the text editing procedures described in section 3.5 to enter text parameters as if they were text properties. A caret shows you the current type-in point, the point where the system will display the text you type.

- To move the type-in point from one parameter to another:

  1) Press the **SKIP NEXT** key. The system moves the type-in point to the next parameter and deletes the existing parameter value. For instance if the type-in point is at **Host:**, pressing **SKIP NEXT** moves the type-in point to **Directory:**.

  Or

  1) Select any character in the parameter name. The system selects the parameter value.

  Or

  1) Select the characters you want to change in the parameter name. Delete them, and type the new characters.

Host

The **Host** parameter is the server with which you want to transfer a file. For example, to store a file on the server named "File Cabinet," you enter "File Cabinet" as the host.

Directory

Servers are partitioned into directories. A directory is similar to a file drawer inside a file cabinet. You enter the directory name for the file you wish to transfer after the word "**Directory**."

Source

The **Source** is a list of files (separated by spaces or returns) for the next command to act upon. File names may include *expansion characters*. Expansion characters tell the system to expand the file name. An asterisk (*) in a file name tells the system to replace the file with a list of files matching that file name where the asterisk matches zero or more characters in the file name. For example, "*.df" matches all files that end with the characters ".df."

A single quote (') tells the system to treat the following character as if it were not a file name expansion character. The system removes the single quote from the file list. For example, a single quote before an asterisk tells the system not to treat the asterisk as a file expansion character; "'*.df" specifies the file named "*.df."

Destination, Local Directory, Connect and Password

The **Destination, LocalDir, Connect** and **Password** parameters are for use by Versatec personnel only.

Boolean Parameters

The parameter subwindow also contains several boolean parameters. Boolean parameters displayed in inverse-video are true. Turn boolean parameters on and off as you would turn boolean properties on and off.

The "*" parameter tells the system to treat asterisks in **Source** as if they were quoted when you issue remote commands, (**Retrieve**, **RemoteList**, **RemoteDelete**). The default value of this parameter is true.

The >, <, and = parameters tell the system to compare creation dates of source and destination files when destination files already exist. The greater-than parameter (>) tells the system to store or retrieve the source file only if the destination file exists and the source file is newer than the destination. The default value is false. The less-than parameter (<) tells the system to store or retrieve the source file only if the destination file exists and the source file is older than the destination. The default value is false. The equal-sign parameter (=) tells the system to store or retrieve the source file only if the destination file exists and the source file is the same as the destination. The default value is false. You can specify "not equal" by turning on both < and >.

The **Always** parameter tells the system to heed the >, <, and = parameters even when the destination file does not exist.

The **Verify** parameter tells the system to request a confirmation of each file transfer. The default is false.

The **NS** parameter tells the system that the host server understands Xerox NS protocol. This parameter is *always* true. The **PUP** parameter is always false.

5.1.3 File Tool Commands

After entering command parameters in the parameter subwindow, you select a command in the command subwindow. The actual file transfer is a background process. You may enter different parameters in the parameter subwindow, or work with other windows while the system is transfering files. The system clears the command subwindow so that you cannot issue a second File Tool command.

Retrieve!

The **Retrieve!** command transfers the file you specified in the **Source** parameter from the remote file system to your local disk. If the **Dest'n** parameter is blank the local file name is the same as the source file name.

Store!

The **Store!** command transfers the file specified in the **Source** field from the local disk to the remote host.

Local-List!

The **Local-List!** command lists all files on the local disk corresponding to the **Source** value. If **Source** is empty, the **Local-List!** command lists all local files.

Local-Delete!

The **Local-Delete!** command deletes all files specified in **Source**. If, for any reason, the system cannot delete a file, it skips that file and continues processing the rest of the files in the list.

Remote-List!

The **Remote-List!** command lists all files on the remote file system corresponding to the name in **Source**.

Remote-Delete!

The **Remote-Delete!** command deletes the names specified in the **Source** parameter from the remote file system.

Verification Commands

When the **Verify** parameter is true, the system displays three additional commands: **Confirm!**, **Deny!**, and **Stop!**. Select **Confirm!** to instruct the system to carry out the command you issued. Select **Deny!** to cancel the command. Select **Stop!** to terminate the command.

5.1.4 List Options

The **List-Options!** command displays the List Options property sheet. List Options properies control the way the system displays file listings in the File Tool's fourth subwindow. All List Option properties are booleans. After changing the options, select **Apply!** to effect those changes. Select **Abort!** to restore the commands to their status before you opened the List Options property sheet. Both **Apply!** and **Abort!** close the property sheet window.

Type

The **Type** property controls whether the system displays the file type.

Create

When **Create** in on, the system displays file creation dates.

Bytes

Turn on **Bytes** to display the length of files in bytes.

Write

Turning on the **Write** property instructs the system to display the last date the file was read.

Author

When **Author** is on, the system displays the name of the person who created the file.

Read

Turn on the **Read** property to display the date the file was last read.

5.1.5 Storing a Documentation Set

You use the File Tool to store documentation sets in your directory on a file server.

- To store a documentation set:

1) Enter the file server name in the **Host** parameter field. Press the **SKIP NEXT** key.
2) Enter the name of your directory in the **Directory** field. Press the **SKIP NEXT** key.
3) Enter the name of the documentation set followed by the characters ".df." For example, if the documentation set is named NewDesign, you type NewDesign.df.
4) If you want the documentation set on the file server to have a different name, press the **SKIP NEXT** key and enter the destination name.
5) Select **Store!**.

- To list the documentation sets stored in your directory:

1) Enter the file server name in the **Host** parameter field. Press the **SKIP NEXT** key.
2) Enter the name of your directory in the **Directory** field. Press the **SKIP NEXT** key.
3) In the **Source** field, enter "*.df."
4) Select **RemoteList!**.

5.2 The Tape Tool

The Tape Tool lets you transfer files to and from the magnetic tape unit on an Output Server. Initially, the Tape Tool is a tiny window on your display.

- To make the Tape Tool active:

1) Select **Size** in the Window Manager Menu.

5.2.1 The Tape Tool Window

The Tape Tool appears and operates much like the File Tool. It has four subwindows: a message subwindow, a parameter subwindow, a command subwindow, and a fourth subwindow in which it lists file transfers. Figure 5.2 shows the Tape Tool window.

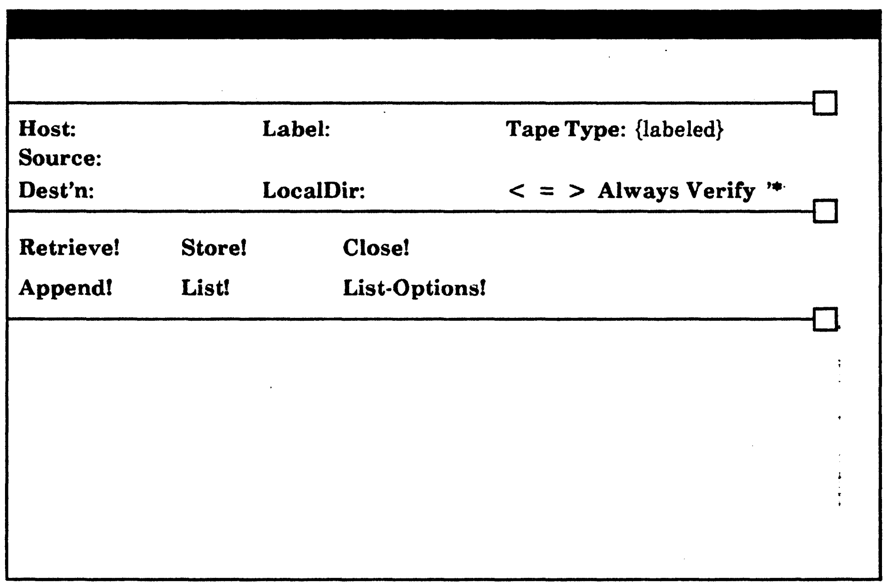


Figure 5.2

- In general, to transfer a file to or from a magnetic tape unit:

1) Enter the appropriate parameters in the second subwindow.
2) Select the file transfer command in the command subwindow.

The Tape Tool is set up to transfer files to and from three different types of tapes. Tape types are *labeled, unlabeled,* and *COMp80*. The parameters in the Tape Tool's second subwindow and the commands in the third subwindow change when you change the type.

- To change the type:

  1) Point the cursor at the words "**Tape Type:**" in the parameter subwindow, and bring up the Tape Type Menu.
  2) Select a type from the options in the menu.

5.2.2 Labeled Tapes

Labeled tapes are tapes that conform to ANSI standard X3.27-1978 format for magnetic tapes. The labeled format lets you write several files to one tape. The Tape Tool lets you retrieve a particular file from a labeled tape. You can store a file at the beginning of the tape or add it to the end of the tape, and you can list all files on a tape. Figure 5.3 shows labeled tape commands and parameters.

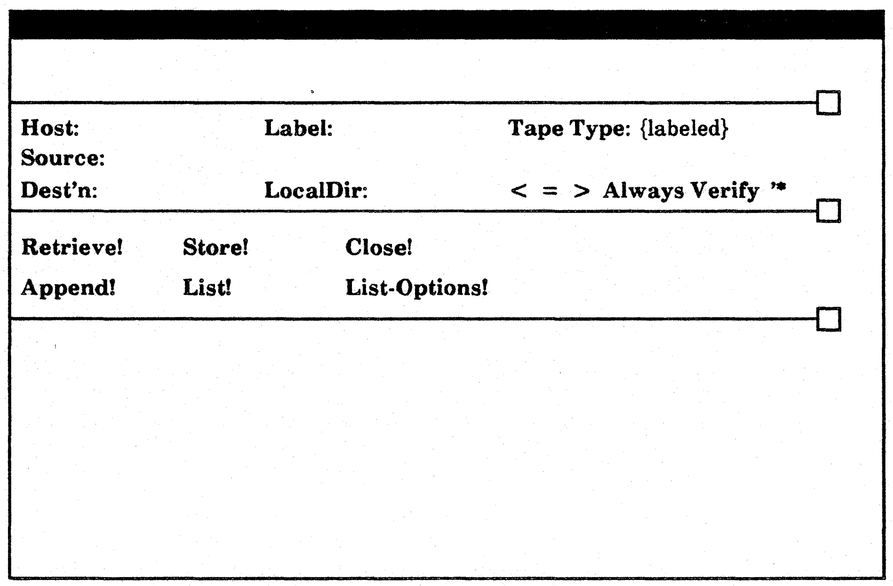


Figure 5.3

Command Parameters

The **Host** parameter is the output server with which you want to transfer a file. For example, to write a file to the server named "Elephant," you enter "Elephant" as the host.

The **Source** is the list of files (separated by spaces or carriage returns) you want to transfer. File names may include the *expansion characters* described in section 5.1.2.

Dest'n is the name of the file resulting from the transfer. When transfering a file from your workstation to magnetic tape, the destination is the file's name on the tape. When transfering a file from a tape to your workstation, the destination is the name of the file at your workstation. If you leave **Dest'n** empty, the file retains its name, as if you entered the same name for both the **Source** and **Dest'n**.

The **Label** parameter lets you assign a label to a tape. Some systems use this label to control access to the information on the tape.

The **LocalDir** parameter is for use by Versatec personnel only.

Section 5.1.2 describes the boolean parameters <, =, >, **Always**, **Verify**, and **'***.

Commands

The **Retrieve!** command retrieves the file you specified in the **Source** parameter from the magnetic tape and writes it to your local disk. If the **Dest'n** parameter is blank, the local file name is the same as the source file name.

The **Store!** command writes the file specified in the **Source** field from your local disk to the beginning of the magnetic tape. The **Store!** command writes over any existing files.

The **Close!** command closes the connection to the output server.

The **Append!** command writes the files specified in the **Source** field to the tape. It writes the files after any other files that already exist.

The **List!** command instructs the system to list all files on the tape. The system lists files in the Tape Tool's fourth subwindow.

The **List-Options** command displays the List Options property sheet. List Options properties control the way the system displays file listings. (See section 5.1.4.)

5.2.3 Unlabeled Tapes

You usually use unlabeled tapes when you want to transfer design information to other systems. Because files are unlabeled, you cannot list them as you can with labeled tapes. Therefore it is best to write just one file to a tape. Figure 5.4 shows the Tape Tool when the tape type is unlabeled.

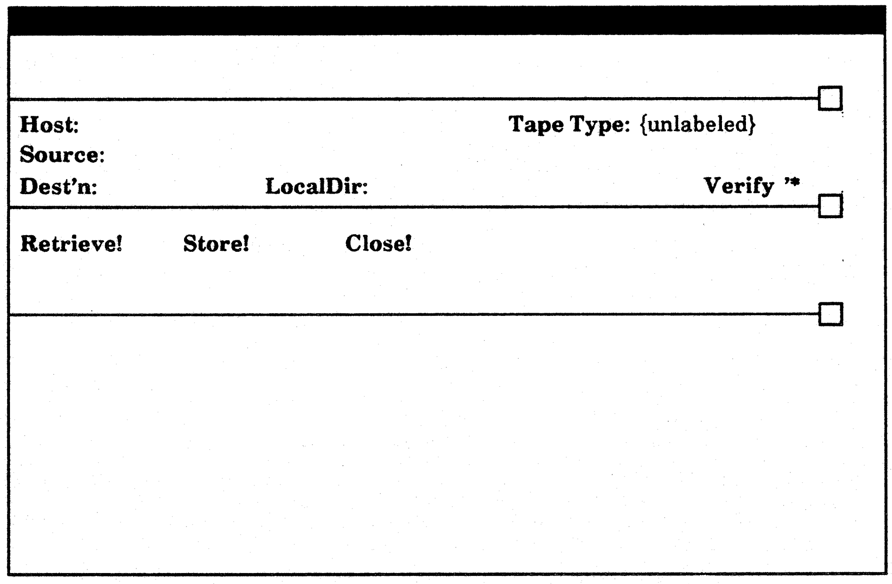


Figure 5.4

Command Parameters

The **Host** parameter is the output server with which you want to transfer a file. The **Source** is the list of files (separated by spaces or carriage returns) you want to transfer. **LocalDir** is the local directory for the transfer. **Verify** and **'*** are boolean parameters described in section 5.1.2.

You use the **Dest'n** parameter to specify the record length and file number of the magnetic tape file. Other systems require magnetic tape files to have various record lengths. Learn the apropriate record lengths for the systems you use. The file number lets you keep more than one file on a tape. In order to prevent writing over files, keep a list of file names and numbers for each tape.

- To specify the record length and file number:

1) Select any character in the word "**Dest'n.**" Type the word "magtape," followed by a period, followed by the magnetic tape file number, followed by a period, followed by the record length. See figure 5.5.

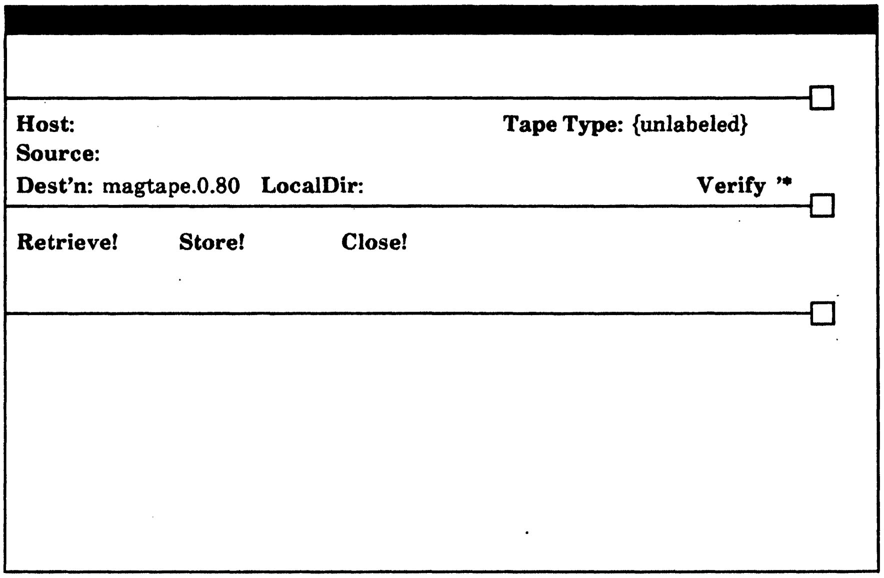


Figure 5.5

Commands

The **Retrieve!** command retrieves the file you specified in the **Source** parameter from the magnetic tape to your local disk. If **Dest'n** is blank the local file name is the same as the source file name.

The **Store!** command writes the files you specified in the **Source** field at the beginning of magnetic tape. The **Store!** command writes over any existing files.

The **Close!** command closes the connection to the output server.

5.2.4 COMp80 Tapes

COMp80 tapes conform to a special format for transfer to a COMp80 Universal Page Setter/COM Graphics Recorder. COMp80 commands and parameters are the same as unlabeled tapes. Figure 5.6 shows COMp80 commands and parameters. Book 2, section 2.8.3 describes converting designs to COMp80 format..

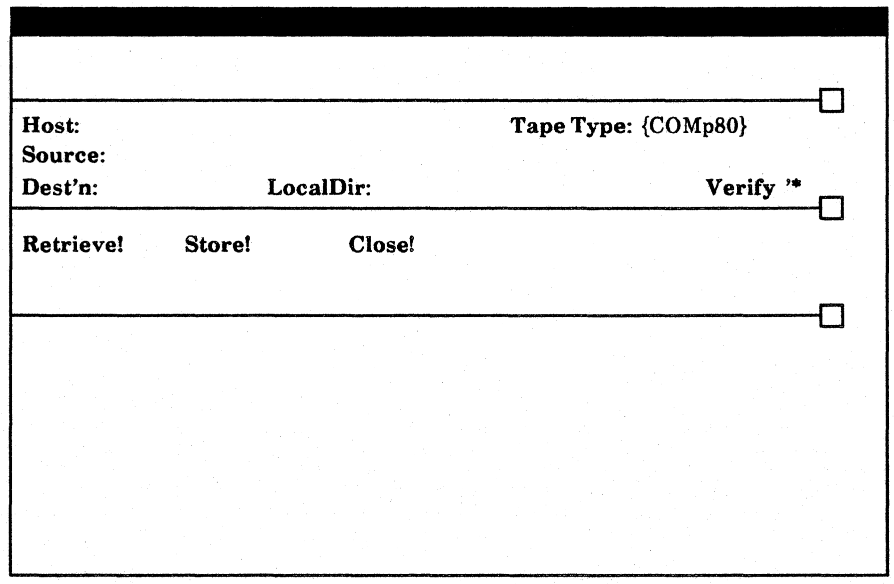


Figure 5.6

- To write a COMp80 file to magnetic tape from your workstation:

  1) Enter the name of your output server as the **Host**.
  2) Enter the COMp80 file or files as the **Source**.
  3) Select **Store!**

6. Ending A Session

When you complete your work at the workstation, you return the workstation to its idle mode.

- To end a session:

1) Activate the Inactive Menu.
2) Select **Bye**. The workstation returns to idle mode (figure 6.1).

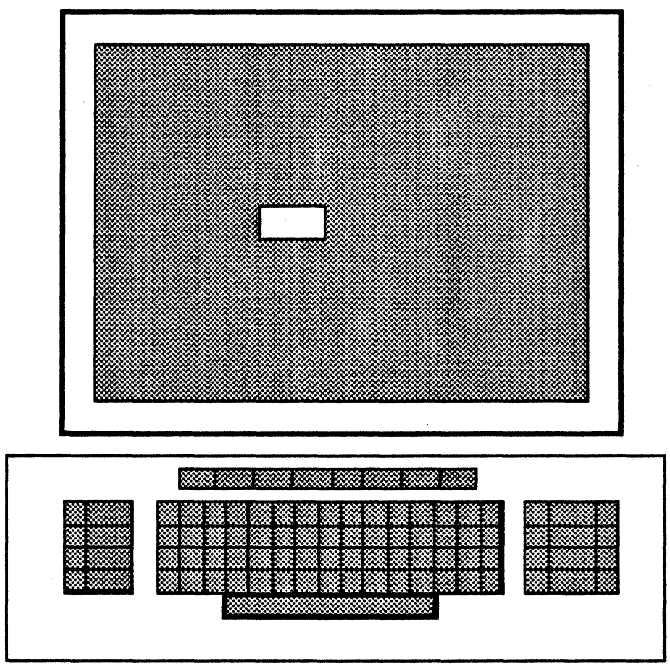


Figure 6.1

You can leave the workstation running in idle mode, or you can turn off the workstation. The workstation's power switch is located on its maintenance panel. Figure 6.2 shows the location of the power switch.

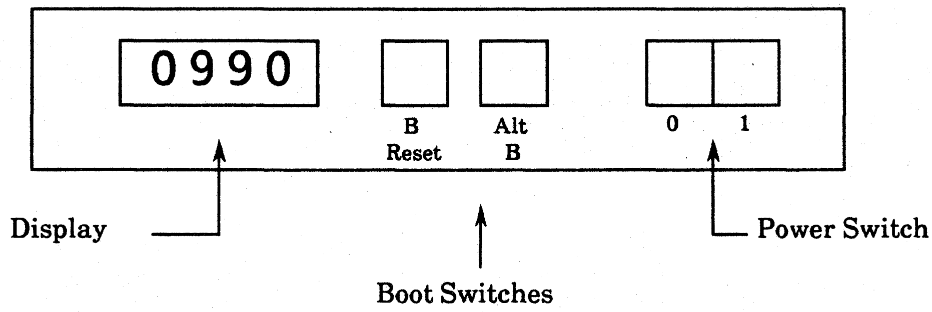


Figure 6.2

- To turn off the workstation:

  1) Turn the power switch to **0**.

- To turn on the workstation:

  1) Turn the power switch to **1**. After you turn it on, the workstation runs a series of diagnostic programs before returning to idle mode.

7. Booting a Workstation

Booting a workstation means starting the the workstation's Expert software. You boot a workstation after turning its power on, after power failures, and after hardware or software failures.

There are two ways to boot a workstation, depending on whether you choose to run a series of short, diagnostic programs before starting the Expert software. In most cases you'll want to run a diagnostic boot, to insure that the workstation is running properly before you begin your work. You may even want to boot the workstation, running the diagnostics, each morning.

7.1 The Maintenance Panel

Each workstation has a *maintenance panel* below the floppy disk drive. A small door covers the maintenance panel when you're not using it. You use the *boot switches*, marked "B Reset" and "Alt B," on the maintenance panel to boot the workstation. The maintenance panel display shows you diagnostic codes. Figure 7.1 shows the location of the boot switches and display on the maintenance panel.

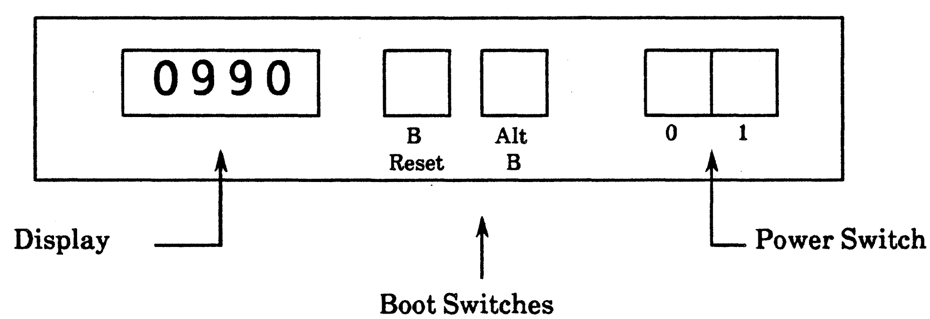


Figure 7.1

7.2 Diagnostic Boot

A diagnostic boot can assure you that your workstation is functioning properly, or alert you to the nature of a workstation malfunction.

- To perform a diagnostic boot:

1) Open the maintenance panel door.
2) Press the switch marked "B Reset."
3) The maintenance panel displays the number "8888," and then cycles through other numbers as the system runs the diagnostics. When the diagnostics are complete, the maintenance panel display reads "0990," and the workstation becomes idle.

If the maintenance panel display shows a number other than "0990" for longer than two minutes, report that number to your System Administrator. Your System Administrator can show you how to run other diagnostic programs to test your workstation.

7.3 Booting without Diagnostics

The second method of booting bypasses the diagnostic programs. Use this method when you believe your workstation to be operating properly.

- To boot a workstation without running the diagnostics:

1) Open the maintenance panel door.
2) Press the switch marked "B Reset" and the switch marked "Alt B" at the same time. Do not release them.
3) Release "B Reset." Watch the maintenence panel display.
4) When the display reads "0001," release "Alt B."
5) The maintenance panel display quickly cycles through a few numbers then displays "0990." The workstation becomes idle.

8. Index

VERSATEC
A XEROX COMPANY

EIS MEMORANDUM

TO: **Distribution**

FROM: **Lisa Schultz**

DATE: **June 1, 1983**

SUBJECT: **Expert 1000 User's Guide: Release 1.5 Update**

cc: **Janet Gilmore, Mark Maltese**

File: [Blue Flamingo]<drawer>Folder>Document

Attached is an updateto the Expert 1000 User's Guide. The update describes the features added to the system in release 1.5. Please use the Documentation Comment Form to list any errors or suggestions.

You will receive the final User's Guide for release 1.5 during the first few weeks of July.

DISTRIBUTION:

EIS Application Analysts
Walt Chun
Jim Cucinitti

VERSATEC
A XEROX COMPANY

Expert Series
Documentation Comment Form

Please use this form to express your opinion of this manual. Your comments help us to improve our manuals, and make them more useful for you. All comments and suggestions become property of Versatec. Mail this form to Versatec, 2710 Walsh Avenue, Santa Clara, CA 95051.

Manual Title: ______________________________

Who are you?

Name: ______________________________

Company Name: ______________________________

Position: ______________________________

How do you use the manual?

- As an introduction to the product it describes ☐
- As a reference guide ☐
- As a tutorial ☐
- Other: ______________ ☐

How well does the manual meet your needs?	Yes	No
Is it easy to read?	☐	☐
Is it easy to understand?	☐	☐
Are the format and organization easy to follow?	☐	☐
Is it technically accurate?	☐	☐
Can you easily find the information you need?	☐	☐
Is it complete? Does it tell you everything you need to know?	☐	☐
Is it well illustrated?	☐	☐

Technical or editorial errors (please list page number):

____

____

____

____

____

Suggestions for improvement:

____

____

____

____

____

VERSATEC
A XEROX COMPANY

Expert 1000 User's Guide

Release 1.5 Update

Draft
May 25, 1983

This document was prepared with the Xerox 8010 Information System.

Document Number 1-40B July 1983 Printed in USA

1. Introduction

Sections in this update replace sections of the Expert 1000 User's Guide for release 1.0. The first page of each replacement section corresponds to the first page of the section it replaces in the original.

This update is for internal use only. Customers who purchase release 1.5 will receive a revised User's Guide with updates included as part of the text.

Change Summary

The following table summarizes the changes documented in this update to the Expert 1000 User's Guide.

Topic	Section
The Basics	
Text Editing	3.5
The File Tool and the Tape Tool	5
Schematic Design	
The Schematic Window Property Sheet	2.1.5
Net Properties	2.3.6
Output	2.8
Interfacing to SCICARDS	3.5
Interfacing to Computervision	3.6
Interfacing to LASAR	3.7
The Library Editor	
Displaying Text Fonts	2.2
Assigning Fonts to Text Items	2.7.5

The Basics

3.5 Text Editing

Some subwindows contain graphic items, and others contain text. You often need to edit text, for example to change a command parameter, alter a property value (section 3.6), or edit a text file. This section explains text files, file windows, and the techniques you use to edit text.

3.5.1 Text Files and File Windows

Text files are files of textual information that you can create and keep at your workstation. Creating a text file at your workstation is similar to writing a report on a piece of paper, and keeping it in a file drawer at your desk.

You create and edit text files in a special type of window known as a *file window*. A file window has two command subwindows, and a third *text subwindow* in which you display the text file. See figure 3.28.

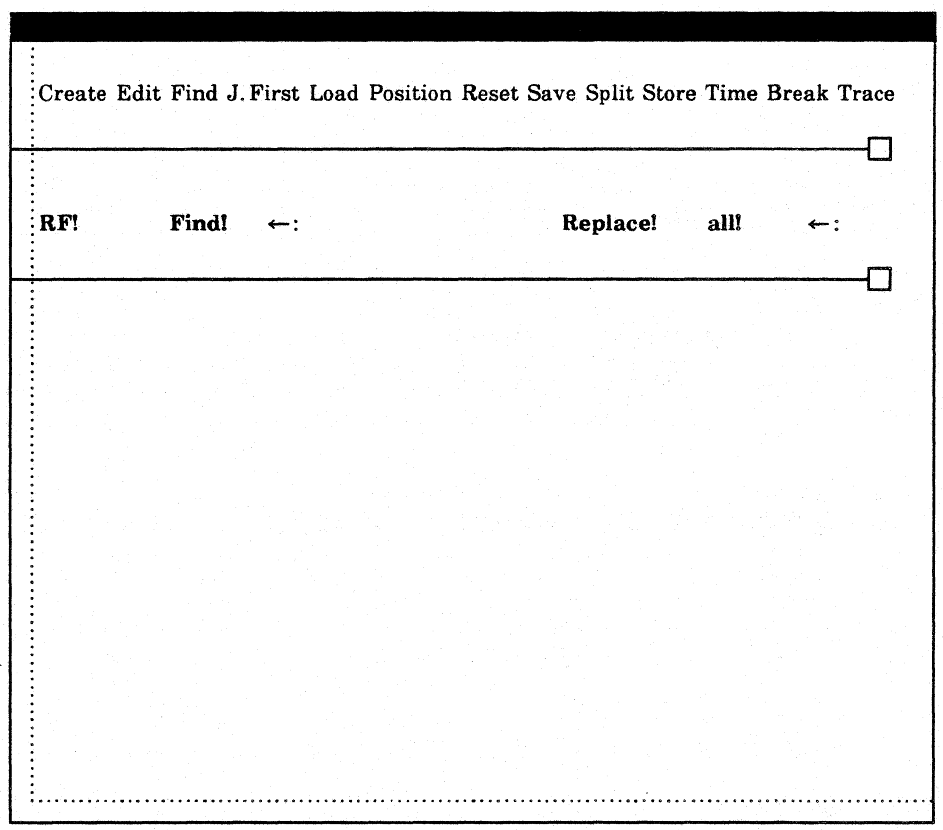


Figure 3.28

Creating a File Window

- To create a file window:

1) Move the cursor to an empty area on the display and press both mouse buttons to activate a menu stack. Select the Exec Ops Menu.
2) Activate the Exec Ops Menu. Select **File Window**.
3) The system asks you to confirm the **File Window** command. The cursor changes to the shape of a mouse. Press the left mouse button to confirm the command. The system creates an empty window at the point where you pressed the left mouse button.

The File Window Menu

The File Window Menu is associated with the third subwindow in the file window (the text subwindow). The File Window Menu contains commands for manipulating the text file displayed in the window. Figure 3.29 shows the File Window Menu.

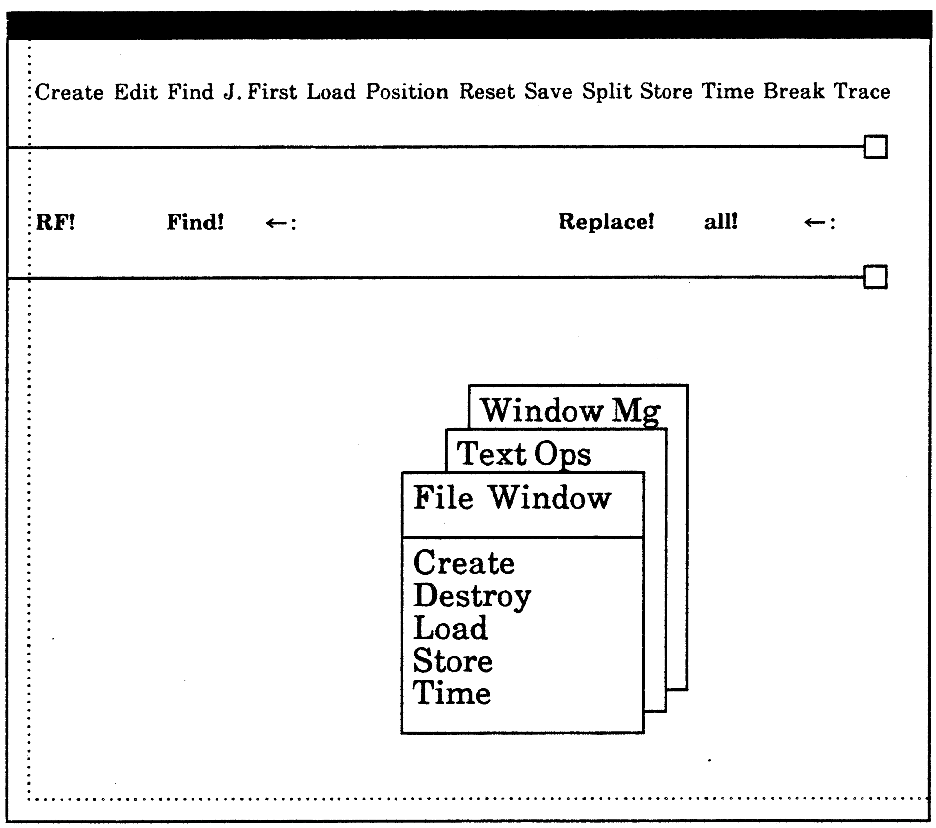


Figure 3.29

File Window Menu commands change as you perform different tasks in the file window. The File Window Menu displays only those commands relevant to what you are doing in the file window.

Create

The **Create** command creates a new file window.

- To create a file window with the **Create** command:

1) Bring up the File Window Menu.
2) Select **Create**.
3) The cursor changes to the shape of a mouse. Press the left mouse button to confirm the **Create** command, and to indicate the new window's position on the display. The system displays the new file window at the point where you pressed the left mouse button.

Destroy

The **Destroy** command removes the file window from the display.

- To remove a file window:

1) Bring up the File Window Menu.
2) Select **Destroy**.
3) The cursor changes to the shape of a mouse. Press the left mouse button to confirm the **Destroy** command. The system removes the file window from the display.

Time

When you select the **Time** command the system writes the date and time into the text subwindow. For example: 13-May-83 10:23:12.

Store

The **Store** command instructs the system to store a new text file.

To store a file:

1) Select the file name. If the name appears in the text subwindow, select it there; if not create an empty file window, type the file name it the new file window's text subwindow, and select it.
2) Select **Store**.

Load

The **Load** command instructs the system to load an existing text file into the file window.

- To load a file:

1) Type the file name in the file window's text subwindow.
2) Select the file name.
3) Select **Load**.

Edit

After loading an existing file, use the **Edit** command to tell the system you want to change the text in the file.

- To edit a file:

1) Load the file into a file window.
2) Select the **Edit** command.
3) Use the text manipulation techniques described in section 3.5.2 to change the text file.

When you finish editing the file, use the **Save** command to instruct the system to save your changes.

Save

The **Save** command appears in the File Window Menu only when you are editing a file. It instructs the system to save the changes that you made to the file.

- To save your changes:

1) Select **Save**.

Reset

If you are not editing a file, the **Reset** command removes all text from the text subwindow. If you are editing a text file (you have loaded the file and issued the **Edit!** command) the **Reset!** command restores the file to its original state before you began editing it.

- To reset the file window:

  1) Select **Reset**.

The Command Subwindow

The first file window command subwindow lists Text Ops Menu and File Window Menu commands. The command name in the command subwindow acts as an accelerator for the command. For instance, you can **Split** the text subwindow without activating the Text Ops menu. You simply select **Split** in the command subwindow.

You cannot issue a command in this subwindow unless the command currently exists in the dynamic menu. For example, the **Edit** command is available only after you have loaded a file into the window. Therefore, even though the **Edit** command may appear in the command subwindow, you cannot issue it unless it also exists in the dynamic menu.

You can add menu items to the command subwindow, and delete items from the subwindow.

- To add an item:

  1) Hold the **PROP'S** key and select the type-in point with the left mouse button.
  2) Type the File Window Menu or Text Ops Menu command name.

- To remove a command from the list:

  1) Hold the **PROP'S** key and select the type-in point with the left mouse button.
  2) Use the backspace or backword key to delete the command name.

Creating a New File

- To create a new text file:

  1) Create a new file window.
  2) Move the cursor into the text subwindow. Press the left mouse button.
  3) Enter the text. Use the text manipulation techniques discussed in section 3.5.2.
  4) Store the file.

Editing an Existing File

- To edit an existing file:

1) Move the cursor into the third subwindow of the empty window and press the left mouse button.
2) Type the name of the file.
3) Select the letters you typed.
4) Select **Load** in the empty window's first subwindow. The system displays the text file.
5) Select **Edit** in the first subwindow, and make your changes.
6) When you finish editing the file, select **Save**.

3.5.2 Text Manipulation

You can edit the text in a text subwindow by selecting a portion of it and adding or deleting characters. In addition, the system supplies you with commands to manipulate the text subwindow, search for and replace patterns in the text.

Selecting Text

- To select a single character in the text body, point the cursor at the character and press the mouse's left button. Press the button twice to select the word, three times to select the line, four times to select the whole body of text, and five times to select just the character once again.

- You can *extend* a text selection to the left or right two ways:

1) Point to the endpoint of the selection and press the mouse's right button.

Or

2) Move the cursor while holding the mouse's right button. Release the button when the cursor points at the endpoint of the selection.

The Current Selection

There is only one *current selection* at any time (not one per window); the system often uses the current selection as the argument to text editing commands you issue. The current selection applies only to text in text subwindows; a selected command, for example, is not a current selection.

Entering Text

- The system automatically marks a *type-in point* with a flashing caret at the end of the selected text. If you select just one character, the system marks the type-in point to the side of the character nearest the cursor, either before or after the character.

- To add text:

  1) Select the type-in point, then type the text using the keyboard.

  When entering text in an empty file window, move the cursor into the text subwindow and press the left mouse button. The system displays the type-in point at the top of the subwindow. Type the text.

Deleting Text

- To delete text, select the text you want to delete, and press the key marked **DELETE**. If you accidentally delete text, you can replace it by pressing the **PASTE** key (marked **SAME**). You can also use the backspace key (above the carriage return key) to delete the character immediately preceding the type-in point. The backword key (Press **SHIFT** and backspace simultaneously) deletes the word preceding the type-in point. You cannot replace text you delete with the backspace or backword keys.

Copying Text

- There are three ways to copy text within a text subwindow or from one subwindow to another:

  1) Select the text.
  2) Hold down the **PROP'S** key and use the left mouse button to select the type-in point where you want to copy the text. Release the **PROP'S** key.
  3) Press the **STUFF** key (marked **OPEN**).

  Or

  1) Select the text.
  2) Press **DELETE**.
  3) Press the **PASTE** key (marked **SAME**) to replace the text.
  4) Use the mouse to move the type-in point to where you want to copy the text.
  5) Press **SAME** once again.

  Or

  1) Move the type-in point to the desired location.
  2) Hold down the **COPY** key.
  3) Select the text you want to copy.
  4) Release the **COPY** key.

3.5.3 The Text Ops Menu

The *Text Ops Menu* provides commands for manipulating text placement inside a text subwindow. Figure 3.30 shows the Text Ops Menu.

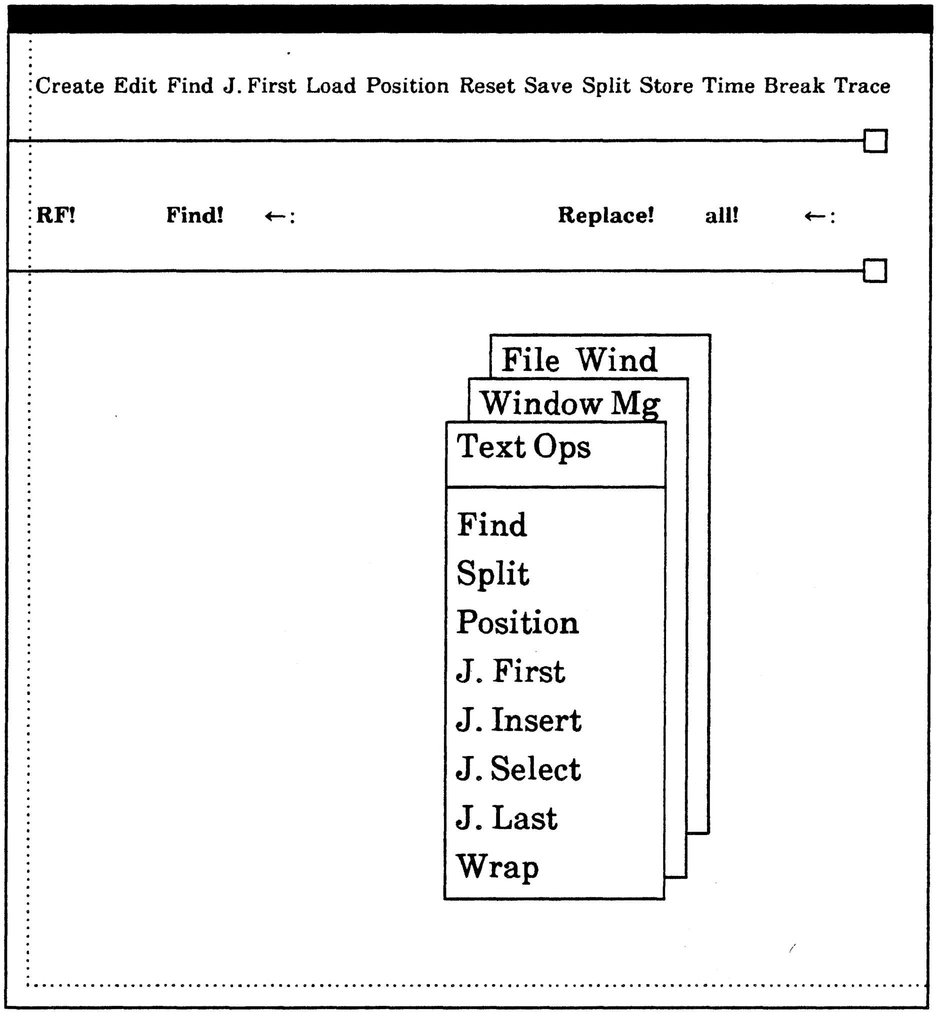


Figure 3.30

- To display the Text Ops Menu:

1) Move the cursor inside the text subwindow.
2) Press both mouse buttons or the **CENTER** key.

Find

The **Find** command finds the next occurrence of the current selection in the subwindow. If the current selection is in the subwindow, the search begins at the end of the selection; otherwise it begins at the first character visible in the subwindow. If the search is successful, the next occurrence of the text becomes the new selection. The search continues into text not visible on the screen; if found, the system scrolls the text to the top of the subwindow. If the system finds no further instances of the text, the display blinks.

If you hold down the **SHIFT** key while selecting the **Find** command, the system searches backwards from the current selection, or from the last character visible in the window.

Split

The **Split** command divides a region of the subwindow into two subwindows separated by a dashed line with a small box at the right end of the line. You can scroll the subwindows independently of each other, and move the border to adjust their sizes. Remove the line by moving it off the top or bottom of a subwindow. Figure 3.31 shows a split text subwindow.

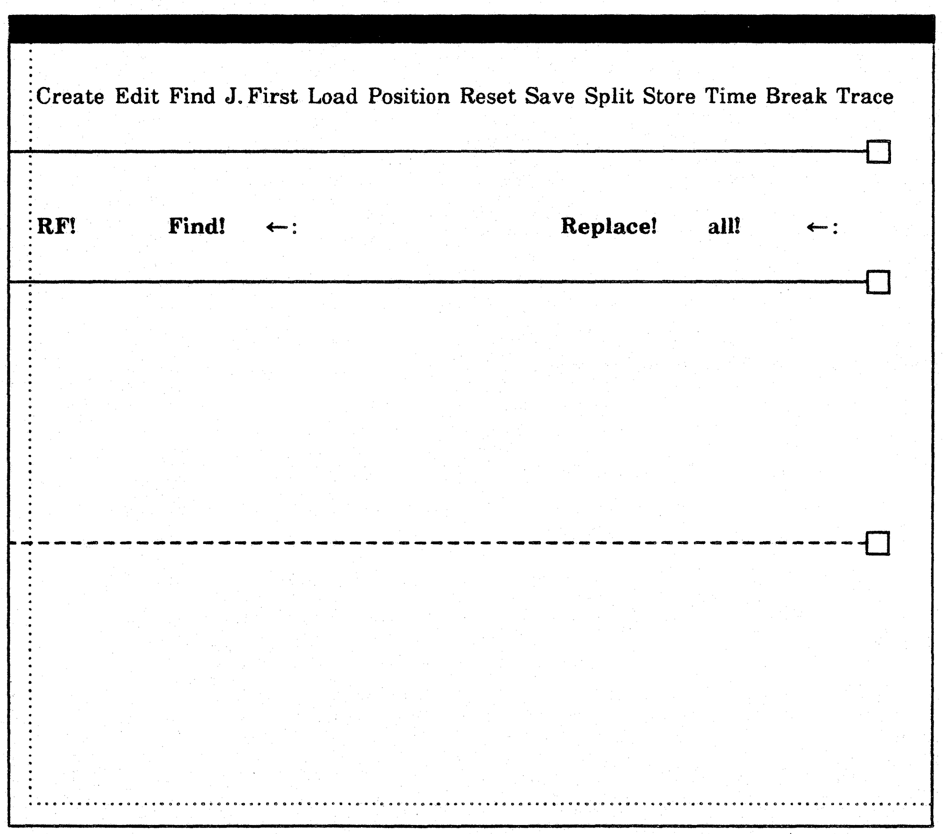


Figure 3.31

Position

Use the **Position** command to position the text in the subwindow so a specific character is at the top. You select a positive number in any text subwindow to specify a character in the file, then select the **Position** command. For example, to place the 275th character at the top of the subwindow, select the number 275 (either type it in the subwindow or in a second file window), and select **Position**.

J. First

The **J. First** command positions the text in a subwindow so that the first line of text is at the top of the subwindow.

J. Insert

The **J. Insert** command positions the text in the subwindow so that the type-in point is at the top.

J. Select

The **J. Select** command positions the text in the subwindow so that the line containing the leftmost character of the current selection is at the top.

J. Last

The **J. Last** command positions text in a subwindow so that the last line of text is at the top of the subwindow.

Wrap

Text subwindows have two states of line *wrap-around*. When wrapping is on, a line that you have not terminated with a carriage return by the time it reaches the right-hand edge of a subwindow continues onto the next line. When wrapping is off, the same line disappears off the right edge of the subwindow. The **Wrap** command reverses the current state of line wrap-around in all subwindows you created from the same subwindow with the **Split** command.

3.5.4 Pattern Matching: Find and Replace

The file window's second subwindow lets you perform pattern matching and replacement in text subwindows.

- You can instruct the system to look for a particular pattern of characters, for instance a word.

1) Use the right mouse button to select the arrow following the **Find!** command. Leave the cursor inside that subwindow.
2) Type the text for which you want to search.
3) Select **Find!** The system searches for the pattern, and selects it.

The pattern you entered after the arrow is called **Find!** *buffer*. If you issue the **Find!** command while the buffer is empty, the system searches for the current selection instead. In this case, the system fills the buffer with the current selection.

- You can also instruct the system to replace one pattern with another, for example, to substitute one word for another.

1) Select the pattern you want to replace, or enter the pattern you want to replace in the find buffer. (Use the right mouse button to select the arrow following the **Find!** command. Type the text you want to replace.)
2) Use the right mouse button to select the arrow following the **all!** command. Type the pattern you with which you want to replace the first pattern.
3) Select **Replace!**

The **all!** command does successive replaces until there are no more matches or until you press the **ABORT** key (marked **STOP**).

The **Find!** and **Replace!** commands operate on the window you are editing. If you have split the window, they operate on the split that contains the current selection; if the selection is in another window, or if there is no selection, they operate on the first split.

The **RF!** command is equivalent to **Replace!** followed by **Find!** It replaces the selected text with the text you enter following the **all!** command, then it finds and the next instance of the selected text.

3.6 Property Sheets

Most graphic and text objects inside subwindows have certain *properties* associated with them. For example, a logic symbol in a schematic has a library identification number, a library name, a symbol designator, a physical designator, a physical part name, pin numbers, physical device parameters, and a micro-note. Any window that displays a schematic or documentation set also has properties. You can change properties in a special type of window known as a *property sheet.* Figure 3.32 shows a logic symbol property sheet from an Expert 1000 schematic.

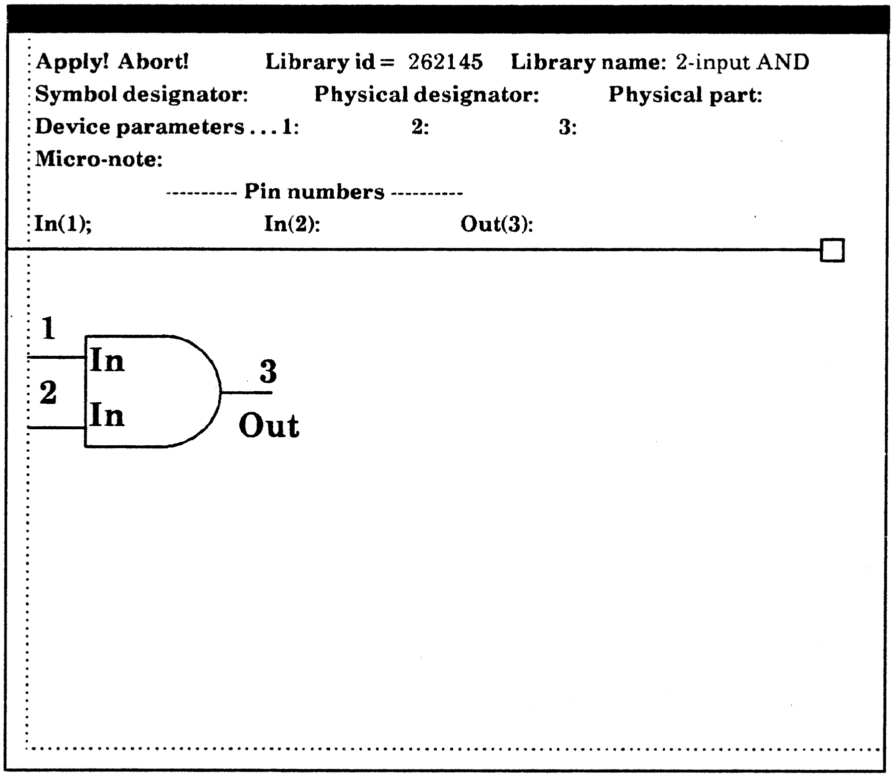


Figure 3.32

- To display the property sheet for a window:

  1) Select the **Props!** command in the window's command subwindow.

- To display the property sheet for a graphic or text object inside a subwindow:

  1) Select the object
  2) Press the function key marked **PROP'S**.

A property sheet window usually has just one subwindow. This subwindow lists the names and values of the object or window properties. The property name appears in bold type, and the property value appears in regular type.

While you display a property sheet for an item inside a window, the system may not let you issue commands associated with the window. You usually enter the item's properties and close the property sheet before issuing other commands.

There are four types of properties. The procedure for changing a property differs for each property type.

3.5.1 Boolean Properties

Some properties are simply listed in the property sheets. These are properties that you can turn on or off, like a toggle switch. Often a description followed by three dots precedes a boolean property. Figure 3.33 shows boolean properties. If the name of the property is in inverse-video, the property is on, otherwise it is off.

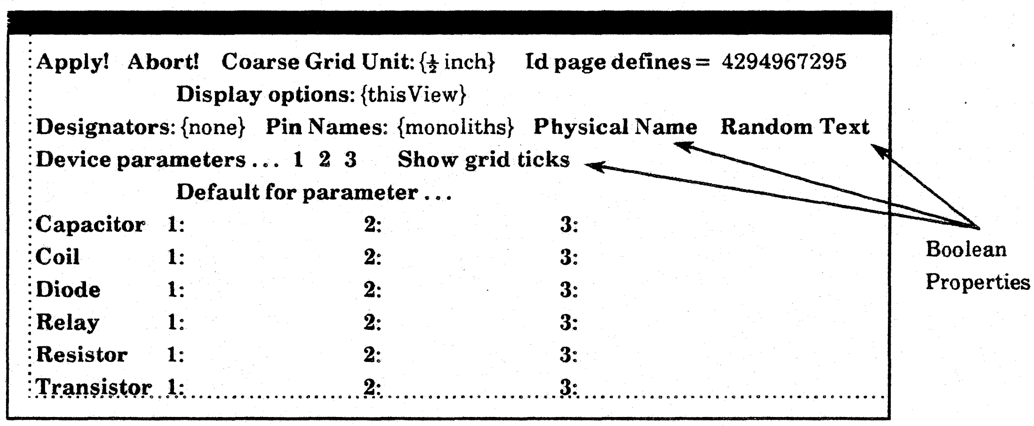


Figure 3.33

- To turn a boolean property on or off:

1) Point the cursor at the property.
2) Press either mouse button.

3.5.2 Property Lists

A *property list* consists of a property name and several possible property values. (See figure 3.34.) You choose a property value from the list of possibilities. There are two ways to do this, depending on how Expert 1000 displays the property list.

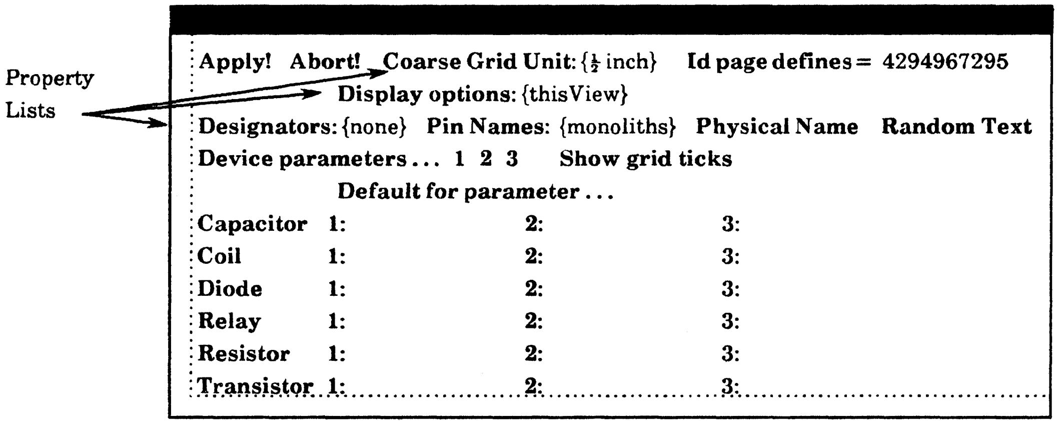


Figure 3.34

- If there is only one value inside the brackets, the other possible values are contained in a dynamic menu associated with the property name.

  1) Point the cursor at the property name and activate the menu.
  2) Select a value from the menu. Notice that Expert 1000 displays the value you selected inside the brackets.

- If the property name is followed by a list of values inside brackets, you can select one value from that list, or select the value from an associated menu. To select the value in the list:

  1) Turn the value on as if it were a boolean property. The system video-inverts that value.

3.5.3 Text Properties

A *text property* consists of a property name followed by one or several words. The text might be the name or a description of the object with which the property sheet is associated. Figure 3.35 shows text properties.

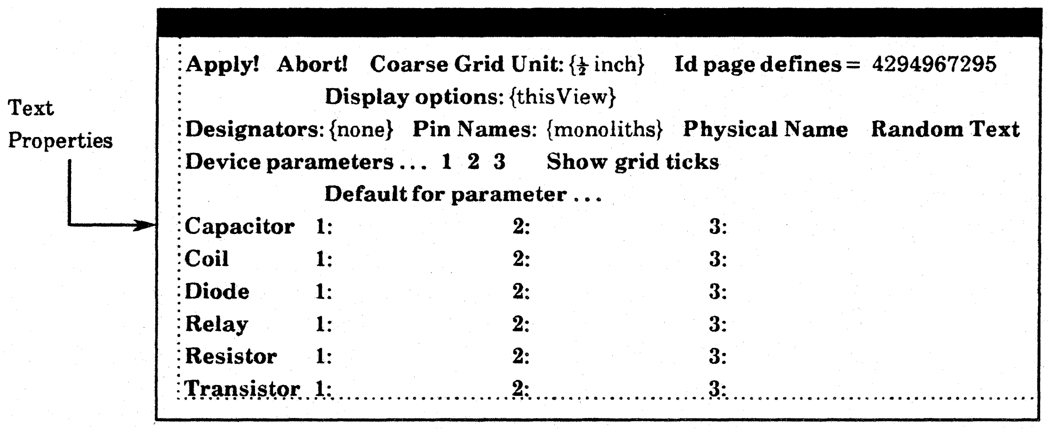


Figure 3.35

Use the text manipulation techniques described in section 3.5.2 to change text properties. You can also use these special text editing techniques for property sheets:

- When you select any character in the property name with the left mouse button, the system automatically selects the whole body of text following the name.
- You can use the **SKIP NEXT** key in the right function group to automatically move the type-in point to the next text or numeric property in the property sheet. The system automatically deletes that property's value. You type the new value.
- When you select any character in the property name with the right mouse button, the system automatically deletes the whole body of text following the name.

3.5.4 Numeric Properties

Numeric properties are properties whose values may be only numbers. For example, the width of a line in a schematic is a numeric property. An equal sign (=) following a property name distinguishes a numeric type from a text type (see figure 3.36). Use the text editing techniques above to change the value of a numeric property type.

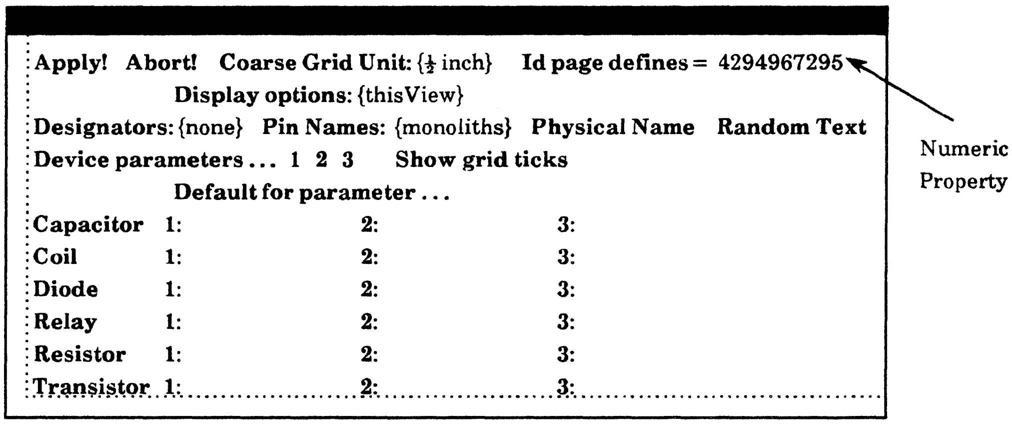


Figure 3.36

3.5.5 Apply! and Abort!

Below the property sheet's name frame are two commands: **Apply!** and **Abort!** Both close the property sheet.

- To apply the changes that you made in the property sheet:

  1) Select **Apply!**

- To erase the changes you made:

  1) Select **Abort!**

5. The File Tool and the Tape Tool

5.1 The File Tool

The File Tool lets you transfer files from your local disk to remote servers, and from servers to your local disk. You use the File Tool, for example, to store a completed documentation set on a file server. That file server acts as a large filing cabinet in which there are several drawers known as directories. When you store a documentation set you specify the directory name as well as the file server name.

When you first start the system, the File Tool is a tiny window on your display. Use the Window Manager's **Size** command to make the File Tool window active.

5.1.1 The File Tool Window

The File Tool window contains four subwindows. The first displays messages from the system. The second contains fields for entering command parameters. The third contains File Tool commands, and the fourth displays file lists. The List Options window is a property sheet window associated with the File Tool. Figure 5.1 shows the File Tool and List Options windows.

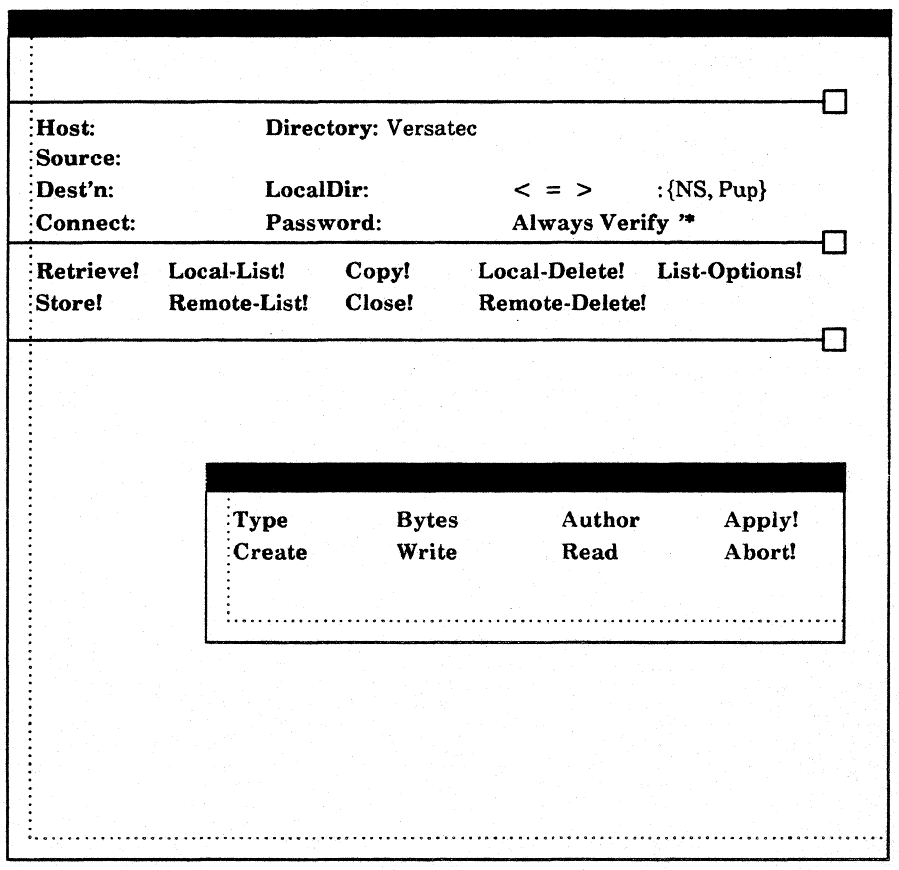


Figure 5.1

5.1.2 Command Parameters

The second, command parameter, subwindow acts as a property sheet containing text and boolean properties. Each text parameter has a name. You enter the parameter value after its name. Use the text editing procedures described in section 3.5 to enter text parameters as if they were text properties. A caret shows you the current type-in point, the point where the system will display the text you type.

- To move the type-in point from one parameter to another:

  1) Press the **SKIP NEXT** key. The system moves the type-in point to the next parameter and deletes the existing parameter value. For instance if the type-in point is at **Host:**, pressing **SKIP NEXT** moves the type-in point to **Directory:**.

  Or

  1) Select any character in the parameter name. The system selects the parameter value.

  Or

  1) Select the characters you want to change in the parameter name. Delete them, and type the new characters.

Host

The **Host** parameter is the server with which you want to transfer a file. For example, to store a file on the server named "File Cabinet," you enter "File Cabinet" as the host.

Directory

Servers are partitioned into directories. A directory is similar to a file drawer inside a file cabinet. You enter the directory name for the file you wish to transfer after the word "**Directory**."

Source

The **Source** is a list of files (separated by spaces or returns) for the next command to act upon. File names may include *expansion characters*. Expansion characters tell the system to expand the file name. An asterisk (*) in a file name tells the system to replace the file with a list of files matching that file name where the asterisk matches zero or more characters in the file name. For example, "*.df" matches all files that end with the characters ".df."

A single quote (') tells the system to treat the following character as if it were not a file name expansion character. The system removes the single quote from the file list. For example, a single quote before an asterisk tells the system not to treat the asterisk as a file expansion character; "'*.df" specifies the file named "*.df."

Destination, Local Directory, Connect and Password

The **Destination, LocalDir**, **Connect** and **Password** parameters are for use by Versatec personnel only.

Boolean Parameters

The parameter subwindow also contains several boolean parameters. Boolean parameters displayed in inverse-video are true. Turn boolean parameters on and off as you would turn boolean properties on and off.

The "*" parameter tells the system to treat asterisks in **Source** as if they were quoted when you issue remote commands, (**Retrieve, RemoteList, RemoteDelete**). The default value of this parameter is true.

The >, <, and = parameters tell the system to compare creation dates of source and destination files when destination files already exist. The greater-than parameter (>) tells the system to store or retrieve the source file only if the destination file exists and the source file is newer than the destination. The default value is false. The less-than parameter (<) tells the system to store or retrieve the source file only if the destination file exists and the source file is older than the destination. The default value is false. The equal-sign parameter (=) tells the system to store or retrieve the source file only if the destination file exists and the source file is the same as the destination. The default value is false. You can specify "not equal" by turning on both < and >.

The **Always** parameter tells the system to heed the >, <, and = parameters even when the destination file does not exist.

The **Verify** parameter tells the system to request a confirmation of each file transfer. The default is false.

The **NS** parameter tells the system that the host server understands Xerox NS protocol. This parameter is *always* true. The **PUP** parameter is always false.

5.1.3 File Tool Commands

After entering command parameters in the parameter subwindow, you select a command in the command subwindow. The actual file transfer is a background process. You may enter different parameters in the parameter subwindow, or work with other windows while the system is transfering files. The system clears the command subwindow so that you cannot issue a second File Tool command.

Retrieve!

The **Retrieve!** command transfers the file you specified in the **Source** parameter from the remote file system to your local disk. If the **Dest'n** parameter is blank the local file name is the same as the source file name.

Store!

The **Store!** command transfers the file specified in the **Source** field from the local disk to the remote host.

Local-List!

The **Local-List!** command lists all files on the local disk corresponding to the **Source** value. If **Source** is empty, the **Local-List!** command lists all local files.

Local-Delete!

The **Local-Delete!** command deletes all files specified in **Source**. If, for any reason, the system cannot delete a file, it skips that file and continues processing the rest of the files in the list.

Remote-List!

The **Remote-List!** command lists all files on the remote file system corresponding to the name in **Source**.

Remote-Delete!

The **Remote-Delete!** command deletes the names specified in the **Source** parameter from the remote file system.

Verification Commands

When the **Verify** parameter is true, the system displays three additional commands: **Confirm!**, **Deny!**, and **Stop!**. Select **Confirm!** to instruct the system to carry out the command you issued. Select **Deny!** to cancel the command. Select **Stop!** to terminate the command.

5.1.4 List Options

The **List-Options!** command displays the List Options property sheet. List Options properies control the way the system displays file listings in the File Tool's fourth subwindow. All List Option properties are booleans. After changing the options, select **Apply!** to effect those changes. Select **Abort!** to restore the commands to their status before you opened the List Options property sheet. Both **Apply!** and **Abort!** close the property sheet window.

Type

The **Type** property controls whether the system displays the file type.

Create

When **Create** in on, the system displays file creation dates.

Bytes

Turn on **Bytes** to display the length of files in bytes.

Write

Turning on the **Write** property instructs the system to display the last date the file was read.

Author

When **Author** is on, the system displays the name of the person who created the file.

Read

Turn on the **Read** property to display the date the file was last read.

5.1.5 Storing a Documentation Set

You use the File Tool to store documentation sets in your directory on a file server.

- To store a documentation set:

1) Enter the file server name in the **Host** parameter field. Press the **SKIP NEXT** key.
2) Enter the name of your directory in the **Directory** field. Press the **SKIP NEXT** key.
3) Enter the name of the documentation set followed by the characters ".df." For example, if the documentation set is named NewDesign, you type NewDesign.df.
4) If you want the documentation set on the file server to have a different name, press the **SKIP NEXT** key and enter the destination name.
5) Select **Store!**.

- To list the documentation sets stored in your directory:

1) Enter the file server name in the **Host** parameter field. Press the **SKIP NEXT** key.
2) Enter the name of your directory in the **Directory** field. Press the **SKIP NEXT** key.
3) In the **Source** field, enter "*.df."
4) Select **RemoteList!**.

5.2 The Tape Tool

The Tape Tool lets you transfer files to and from the magnetic tape unit on an Output Server. Initially, the Tape Tool is a tiny window on your display.

- To make the Tape Tool active:

  1) Select **Size** in the Window Manager Menu.

5.2.1 The Tape Tool Window

The Tape Tool appears and operates much like the File Tool. It has four subwindows: a message subwindow, a parameter subwindow, a command subwindow, and a fourth subwindow in which it lists file transfers. Figure 5.2 shows the Tape Tool window.

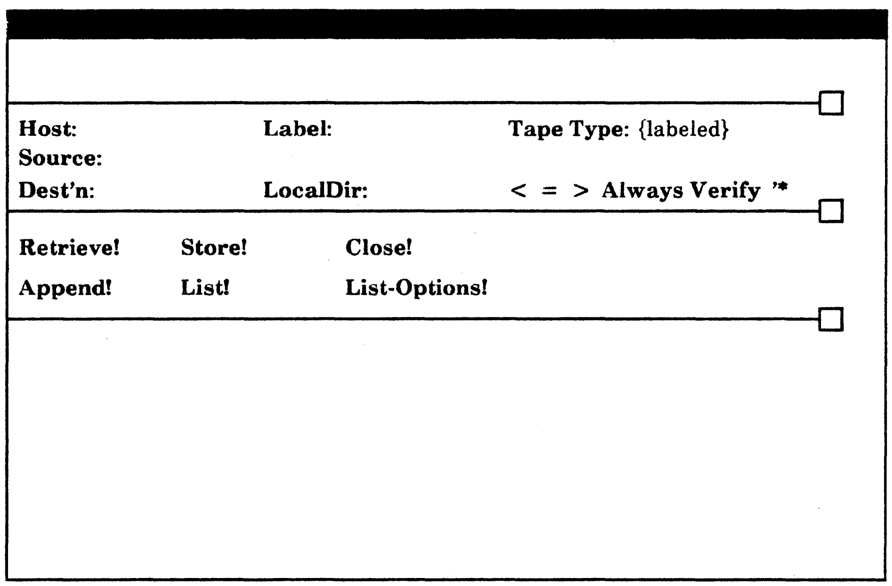


Figure 5.2

- In general, to transfer a file to or from a magnetic tape unit:

  1) Enter the appropriate parameters in the second subwindow.
  2) Select the file transfer command in the command subwindow.

The Tape Tool is set up to transfer files to and from three different types of tapes. Tape types are *labeled*, *unlabeled*, and *COMp80*. The parameters in the Tape Tool's second subwindow and the commands in the third subwindow change when you change the type.

- To change the type:

1) Point the cursor at the words "**Tape Type:**" in the parameter subwindow, and bring up the Tape Type Menu.
2) Select a type from the options in the menu.

5.2.2 Labeled Tapes

Labeled tapes are tapes that conform to ANSI standard X3.27-1978 format for magnetic tapes. The labeled format lets you write several files to one tape. The Tape Tool lets you retrieve a particular file from a labeled tape. You can store a file at the beginning of the tape or add it to the end of the tape, and you can list all files on a tape. Figure 5.3 shows labeled tape commands and parameters.

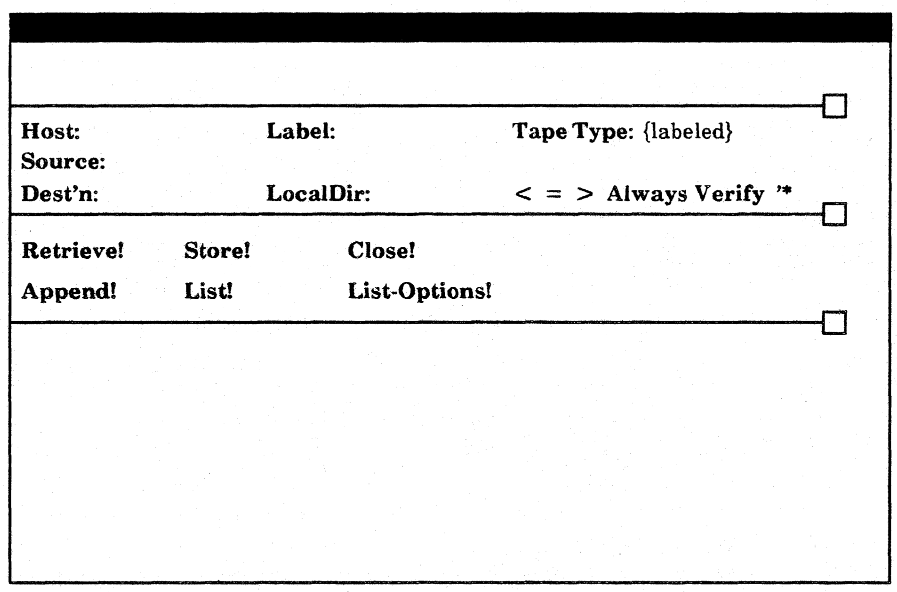


Figure 5.3

Command Parameters

The **Host** parameter is the output server with which you want to transfer a file. For example, to write a file to the server named "Elephant," you enter "Elephant" as the host.

The **Source** is the list of files (separated by spaces or carriage returns) you want to transfer. File names may include the *expansion characters* described in section 5.1.2.

Dest'n is the name of the file resulting from the transfer. When transfering a file from your workstation to magnetic tape, the destination is the file's name on the tape. When transfering a file from a tape to your workstation, the destination is the name of the file at your workstation. If you leave **Dest'n** empty, the file retains its name, as if you entered the same name for both the **Source** and **Dest'n**.

The **Label** parameter lets you assign a label to a tape. Some systems use this label to control access to the information on the tape.

The **LocalDir** parameter is for use by Versatec personnel only.

Section 5.1.2 describes the boolean parameters <, =, >, **Always**, **Verify**, and '*.

Commands

The **Retrieve!** command retrieves the file you specified in the **Source** parameter from the magnetic tape and writes it to your local disk. If the **Dest'n** parameter is blank, the local file name is the same as the source file name.

The **Store!** command writes the file specified in the **Source** field from your local disk to the beginning of the magnetic tape. The **Store!** command writes over any existing files.

The **Close!** command closes the connection to the output server.

The **Append!** command writes the files specified in the **Source** field to the tape. It writes the files after any other files that already exist.

The **List!** command instructs the system to list all files on the tape. The system lists files in the Tape Tool's fourth subwindow.

The **List-Options** command displays the List Options property sheet. List Options properties control the way the system displays file listings. (See section 5.1.4.)

5.2.3 Unlabeled Tapes

You usually use unlabeled tapes when you want to transfer design information to other systems. Because files are unlabeled, you cannot list them as you can with labeled tapes. Therefore it is best to write just one file to a tape. Figure 5.4 shows the Tape Tool when the tape type is unlabeled.

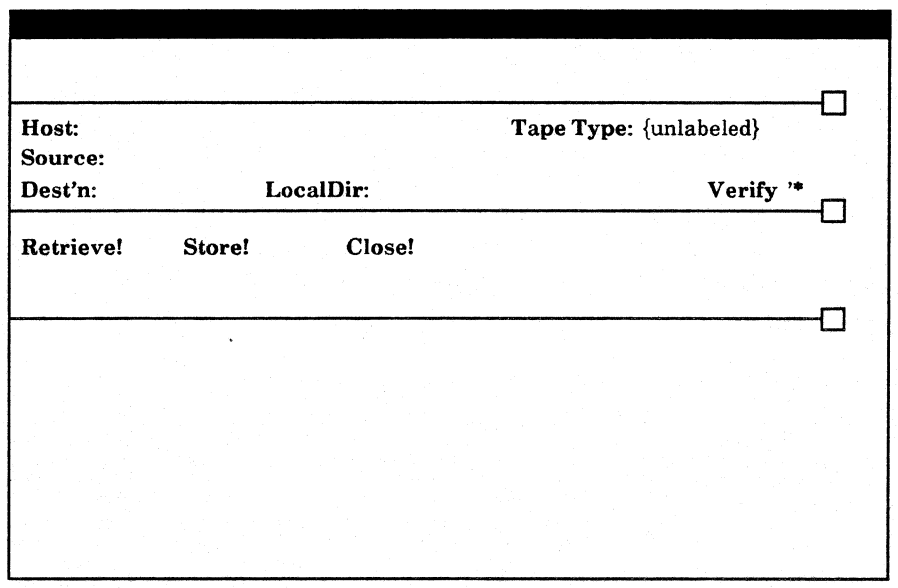


Figure 5.4

Command Parameters

The **Host** parameter is the output server with which you want to transfer a file. The **Source** is the list of files (separated by spaces or carriage returns) you want to transfer. **LocalDir** is the local directory for the transfer. **Verify** and **'*** are boolean parameters described in section 5.1.2.

You use the **Dest'n** parameter to specify the record length and file number of the magnetic tape file. Other systems require magnetic tape files to have various record lengths. Learn the apropriate record lengths for the systems you use. The file number lets you keep more than one file on a tape. In order to prevent writing over files, keep a list of file names and numbers for each tape.

- To specify the record length and file number:

1) Select any character in the word "**Dest'n**." Type the word "magtape," followed by a period, followed by the magnetic tape file number, followed by a period, followed by the record length. See figure 5.5.

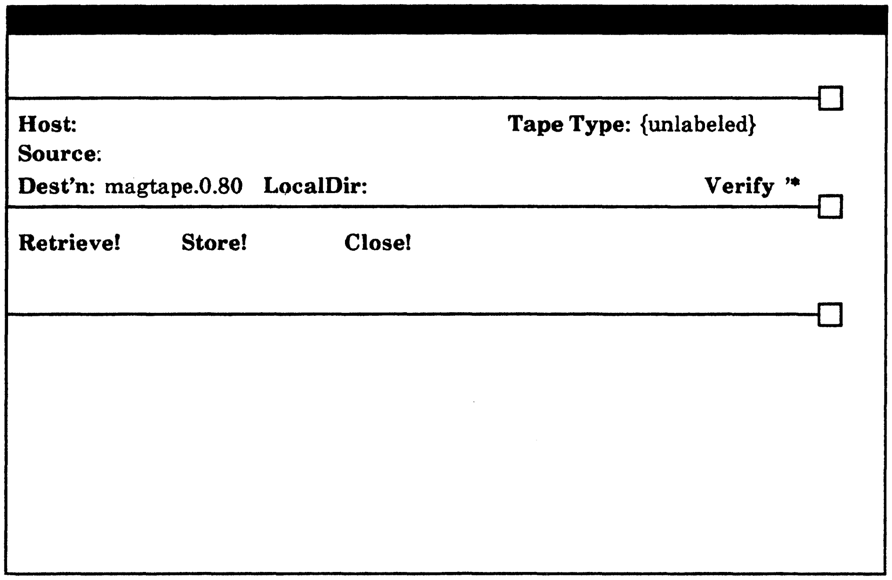


Figure 5.5

Commands

The **Retrieve!** command retrieves the file you specified in the **Source** parameter from the magnetic tape to your local disk. If **Dest'n** is blank the local file name is the same as the source file name.

The **Store!** command writes the files you specified in the **Source** field at the beginning of magnetic tape. The **Store!** command writes over any existing files.

The **Close!** command closes the connection to the output server.

5.2.4 COMp80 Tapes

COMp80 tapes conform to a special format for transfer to a COMp80 Universal Page Setter/COM Graphics Recorder. COMp80 commands and parameters are the same as unlabeled tapes. Figure 5.6 shows COMp80 commands and parameters. Book 2, section 2.8.3 describes converting designs to COMp80 format.

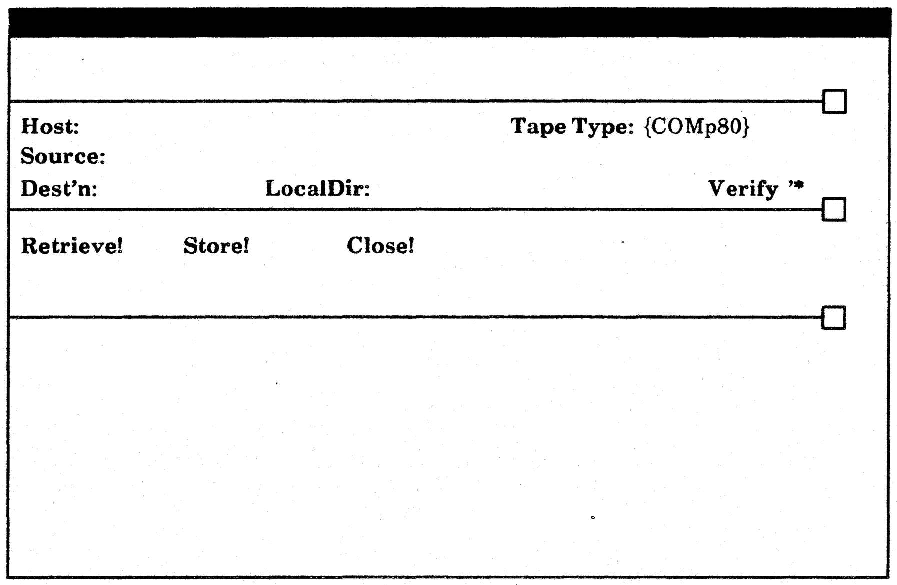


Figure 5.6

- To write a COMp80 file to magnetic tape from your workstation:

1) Enter the name of your output server as the **Host**.
2) Enter the COMp80 file or files as the **Source**.
3) Select **Store!**

Schematic Design

------*This section replaces section 2.1.5 in the original Expert 1000 User's Guide*------

2.1.5 The Schematic Window Property Sheet

The schematic window property sheet lets you control the way the system displays the schematic. Figure 2.3 shows the schematic window property sheet.

Apply! **Abort!** **Coarse Grid Unit:** {½ inch} **Id page defines** =
Display options: {thisView}
Designators: {none} **Pin Names:** {monoliths} **Physical Name** **Random Text**
Device parameters . . . **1 2 3** **Show grid ticks**
Default for parameter . . .

Capacitor	**1:**	**2:**	**3:**
Coil 1	**1:**	**2:**	**3:**
Diode	**1:**	**2:**	**3:**
Relay	**1:**	**2:**	**3:**
Resistor	**1:**	**2:**	**3:**
Transistor	**1:**	**2:**	**3:**

Figure 2.3

Coarse Grid Unit

Each schematic has a coarse grid for controlling the placement of objects in the schematic. You set the grid units in the schematic window's property sheet. Coarse grid unit options are: **2 grid units, 3 grid units, ½ inch**, and **1 inch**.

Id page defines

Id page defines is a numeric property. You use it to tell the system which symbol a schematic page defines in a hierarchical schematic. Section 2.7 describes hierarchy.

Display Options

The remainder of the schematic window properties are options for displaying the schematic inside the window. You choose whether the system displays certain symbol properties on the schematic. For instance, to build a logical schematic, you display symbol designators for each logic symbol. To build a physical schematic, you display physical designators, pin numbers, physical names, and pin names. You may also choose to display device parameters and grid ticks.

The **Display Options** property lets you control whether the system displays grid ticks and the symbol properties you select in all views of the schematic or just in the view in which you are working.

Designators

Each logic symbol has unique symbol and physical designators in its property sheet. The **Designators** property lets you control whether the system displays no designators, both symbol and physical designators, just symbol designators, or just physical designators on the schematic.

Pin Names

A logic symbol's pin names are properties of that symbol. You can instruct the system to display the pin names on the schematic. When you first open the schematic, the system displays pin names only for *monoliths*. Monoliths are symbols more complex than small-scale integrated circuits. You can change this property so that the system displays no pin names or pin names for all symbols.

Pin Numbers

Pin Numbers is a boolean property. When the property is on, the system displays pin numbers for each symbol. When it is off, the system does not display pin numbers.

Physical Name

Physical Name is also a boolean property. When it is on, the system displays physical names for logic symbols.

Random Text

Random Text is a boolean property. When it is on, the system displays random text associated with the symbol.

Grid Ticks

Turn on the **Show Grid Ticks** property to display grid marks on the schematic. Turn it off to remove grid marks.

Device Parameters

You may assign device parameters to symbols in their property sheets, for example, to specify the ohm value, wattage, and tolerance of a resistor. Device parameters are numbered one through three. To display device parameter one for all symbols, turn on the boolean property **1**. To display device parameter two for all symbols, turn on the boolean property **2**. To display device parameter three for all symbols, turn on the boolean property **3**.

Device Parameter Defaults

You can set default parameters for discrete devices in the schematic window property sheet. Each device listed is followed by text properties **1**, **2**, and **3**. The system automatically assigns default device parameters to the corresponding symbols. For example, if you enter default parameter values for resistors, the system automatically assigns those parameters to each resistor you place in the schematic. You can change device parameter values for an individual device in the device's symbol property sheet.

2.3.6 Net Properties

A net has two properties, its name, and the set of net rules that it follows.

- To activate a net's property sheet:

1) Select the net.
2) Press the **PROP'S** key.

Figure 2.? shows a net's property sheet.

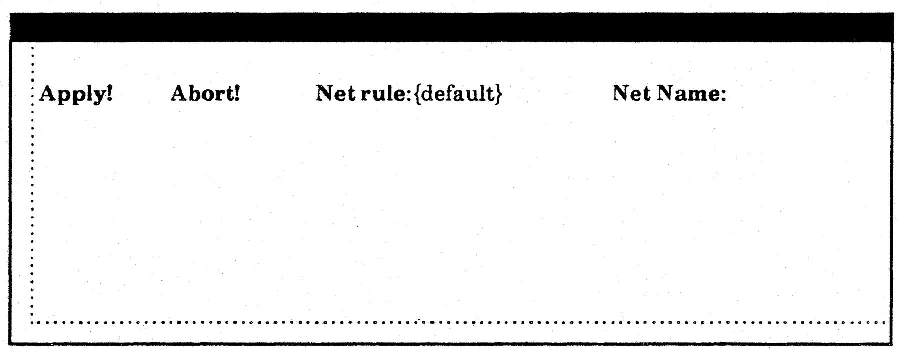


Figure 2.30

Net Rule

The **Net rule** value tells the system which set of electrical rules to follow when routing the net on a pc board with future releases of Expert 1000. The current release does not utilize the **Net rule** property.

Net Name

Unless the net has a connectivity symbol at one of its endpoints, its name must be unique. (Section 2.3.8 describes connectivity symbols.) You can enter the net name in the property sheet, and copy it onto the schematic, or you can name a net with the **Name Net** command described in section 2.3.7.

- To copy a net name from a property sheet to the schematic:

1) Mark the location on the grid at which you want to place the net name with the destination marker.
2) Press the key marked **COPY**. The system copies the name to the location you marked.

- You can display the net name on the schematic in as many places as you want. To copy the name to another location:

  1) Set the destination again.
  2) Press **COPY**.

  When you finish copying names onto the schematic, apply the name. Select **Apply!** in the net's property sheet.

- To change the name of a net:

  1) Change the name in the net's property sheet.
  2) Select **Apply!**. The system automatically changes the name on the schematic.

  While the property sheet is open, you can select and edit the net names individually. When you close the property sheet, the net name becomes part of the net; you cannot select the name without selecting the net.

2.8 Output

You can output a schematic in one of two ways. You can plot the schematic on a Versatec plotter, or you can write the schematic to magnetic tape in COMp80 format and use the tape to plot the schematic on microfiche with computer-output microfilm (COM) equipment.

2.8.1 The Output Options Window

The Output Options window is a property sheet associated with the schematic command subwindow. Properties in the Output Options window act as parameters for the **Output!** command in the command subwindow. Figure 2.47 shows an Output Options window.

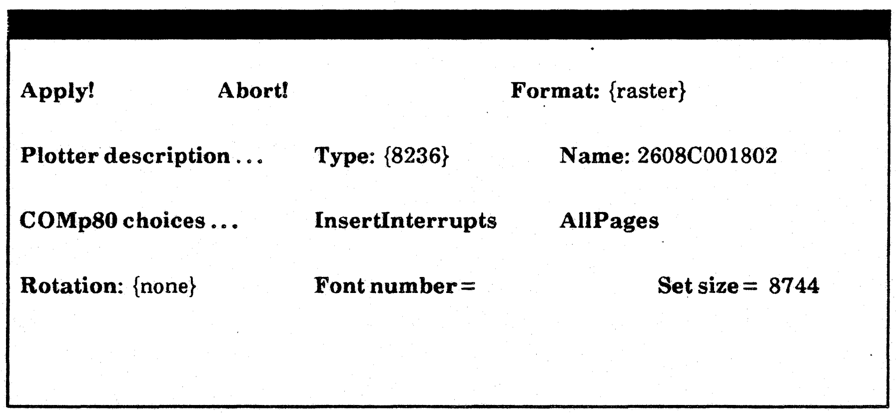


Figure 2.47

- To display the Output Options window:

    1) Select the **OutputOptions!** command in the schematic's command subwindow.

The value of the **Format** property controls the type of output. To plot a schematic, set the **Format** value to **raster**. To create a COMp80 file, set the value to **COMp80**. **Plotter description** properties tell the system on which plotter to plot the schematic. **COMp80 choices** give the system information about the particular format necessary for your COM equipment. Both **Plotter description** properties and **COMp80 choices** have default values that match the equipment in your system.

2.8.2 Plotting the Schematic

- To plot a schematic page on an electrostatic plotter:

1) Display the page you want to plot in the schematic subwindow.
2) Select the scale at which you want to plot the schematic. The scale must be less than or equal to normal.
3) Set the **Display Options** in the schematic window property sheet, so that the system displays the symbol properties you want to plot.
4) Select **OutputOptions!** in the command subwindow. The **Format** value must be **raster**. Enter the correct **Plotter description** and **Name**. The name of the plotter is the hexadecimal address of the output server to which it is connected.
5) Select **Apply!**
6) Select **Output!** in the schematic's command subwindow.

2.8.3 Translating the Schematic to COMp80 Format

You may want to store schematics on microfiche instead of keeping paper copies of each design. Expert 1000 provides interfaces to computer-output microfilm (COM) equipment using COMp80 format. You can use either an FR80, COMp80, or COMp80/2 Universal Page Setter/COM Graphics Recorder to make microfiche plots of Expert 1000 schematics. Expert automatically scales drawings to industry standards for microfiche.

1) If you want to output just one page of the schematic, display that page in the schematic window. Set the **Display Options** in the schematic window property sheet, so that the system displays the symbol properties you want.
2) Select **OutputOptions!** Select the COMp80 options you want to use, entering the appropriate parameters for your COM system.

   InsertInterrupts is a boolean property. If you turn it on, Expert inserts a level 9 interrupt between schematic pages in the COMp80 file. The operator of the COMp80 system can instruct the system to stop and send a message when it encounters a level 9 interrupt, or set the system to ignore level 9 interrupts.

   AllPages is also a boolean property. When you turn it on, it tells the system to write all pages of the schematic to the COMp80 file. If you turn it off, the system writes only the page you are displaying to the COMp80 file.

   Because COM systems have cameras mounted differently, you may need to specify an angle of rotation for your system. **Rotation** options are **none, rotate90, rotate180**, and **rotate270**. **None** instructs the system not to rotate the image. **Rotate90** rotates the image 90 degrees counter-clockwise, **Rotate180** rotates the image 180 degrees, and **Rotate270** rotates the image 270 degrees. Ask your System Administrator for the proper rotation for your system. He or she can get that information from a Versatec Application Analyst.

Some COM systems have a set of numbered fonts. You can assign one of these fonts to the text in the schematic. Use the **Font** parameter to assign a font number. If you assign an invalid number, the COM system uses its default font.

Each COM system has a certain resolution, measured in *scope points*. Enter the number of scope points per inch as the value of the **Set size** parameter.

3) Select **Apply!**
4) Select the **Output!** command in the schematic subwindow. The system creates a local file in COMp80 format. If you selected **All pages**, it names the file with the documentation set name, a period, and the word "COMp80," for example: **Demo.COMp80**. If **All pages** was false, the system adds a dash and the page number after the documentation set name, for example: **Demo-1.COMp80**.

- Use the Tape Tool to write the COMp80 file to a magnetic tape.

1) Find the **Tape Tool** window. If it is tiny, make it active.
2) Set the **Type** parameter to COMp80.
3) Enter the name of the COMp80 file as the **Source**. If you want to write several files to the same tape, type each file name. Separate file names by a space.
4) Select **Store!**

Use the magnetic tape as input to your COM system.

3.5 Interfacing to SCICARDS™

Design services departments with the SCICARDS printed circuit board design system can design printed circuit board layouts from the schematics you create with Expert 1000. You design the schematic, convert it to SCICARDS format, and write it to a magnetic tape on an output server. The magnetic tape serves as input to the SCICARDS system.

- The SCICARDS system must meet the following requirements:

1) SCICARDS version 82.01.
2) VAX/VMS version V3.1 or Prime version 18.3

3.5.1 Designing a Schematic to Interface with the SCICARDS System

- To design a schematic to interface with the SCICARDS system:

1) Check that your symbol library is compatible with the SCICARDS library. Expert symbol library pin names must match the SCICARDS part library pin names for all SCICARDS functional parts.
2) In symbol property sheets, enter pin numbers for all "fixed pin devices." SCICARDS defines fixed pin devices as devices that perform a single function. If you do not enter all pin numbers, Expert passes pin names but is unable to back-annotate pin numbers.
3) Enter SCICARDS part names as physical name values. If no comparable SCICARDS part name exists, enter the SCICARDS device type and shape (separate the two with a space).
4) Enter the SCICARDS part designator as the physical designator for all fixed-pin parts. If you do not enter the part designator, Expert uses its symbol designator as the SCICARDS part designator. If you do not enter the symbol designator, Expert generates a number preceded by "ND" (for example: ND330) and uses it as the SCICARDS part designator. Do not rename SCICARDS part designators; Expert does not back-annotate physical designators from SCICARDS part designators you rename.
5) Enter symbol designators for all SCICARDS *functional parts*. SCICARDS uses the Expert symbol designator as its function designator. If you do not enter a symbol designator, Expert generates a number preceded by "ND" (for example: ND220), and uses it as the SCICARDS function designator. If you entered a SCICARDS part designator as the Expert physical designator, Expert assigns the function to the first available slot in the part. Expert leaves the function free and swapable for SCICARDS users. Do not enter pin numbers; if you do SCICARDS cannot swap functions and pins. Expert assigns nodes to the designated pin numbers and fixes that function.
6) Save the schematic.

3.5.2 Converting the Schematic to SCICARDS Format

- To convert a net list to SCICARDS format:

1) Open the net list window. If it is already open and you have listed a net list, part list, or pin list in it, close the net list and reopen it.
2) Select the **SciCards Interface!** command in the command subwindow. The system displays the net list in SCICARDS format (figure 3.6). Do not select any other commands.
3) Close the net list window. Write the net list to magnetic tape before reopening the window.

Close! **Net List!** **Part List!** **Pin List!** **SCICARDS Interface!**
Aquarius Wirelist! **LASAR Wirelist!** **Computervision Wirelist!**

```
--SCICARDS Interface FOR DOCUMENTATION SET: Demo

PARTS
7400                U11
7404                U10
7476                U9
EOS
FUNCTIONS
7476                ND541       ND540
ASSIGN  ND541 U9
ASSIGN  ND540 U9
EOS
NET LIST
NODE   280 $
    ND539-Nam  ND541      Q
NODE   279 $
    ND538-Nam  Nd541     QN
NODE   278 $
    ND537-Nam  ND540      Q
NODE   277 $
    ND536-Nam  ND540     QN
NODE   286 $
U10          5            P1-14
NODE   285 $
U11          1  U11       10  U11          4         P1-15
NODE   281 $
ND541        K  ND540      K  U11          3
NODE   282 $
ND540        J  U11        8
NODE   284 $
U11          2  U11        9  U11          5  U11       6
NODE   287 $
ND541        J  U11        6
EOS
NAMED NODES
   280  FF00
   279  FF01
   278  FF02
   277  FF03
   286  INC
   285  IND
   281  NN281
   282  NN282
   284  NN284
   287  NN287
EOS
```

Figure 3.6

- Before writing the net list to magnetic tape, Prime users must replace carriage returns created by Expert with line feed characters. To edit the net list:

1) Create an empty file window. See Book 1 section 3.5.
2) Move the cursor into the third subwindow of the file window and press the left mouse button.
3) Type the name of the net list's documentation set, followed by a period, followed by the letters ".nl." See figure 3.7.

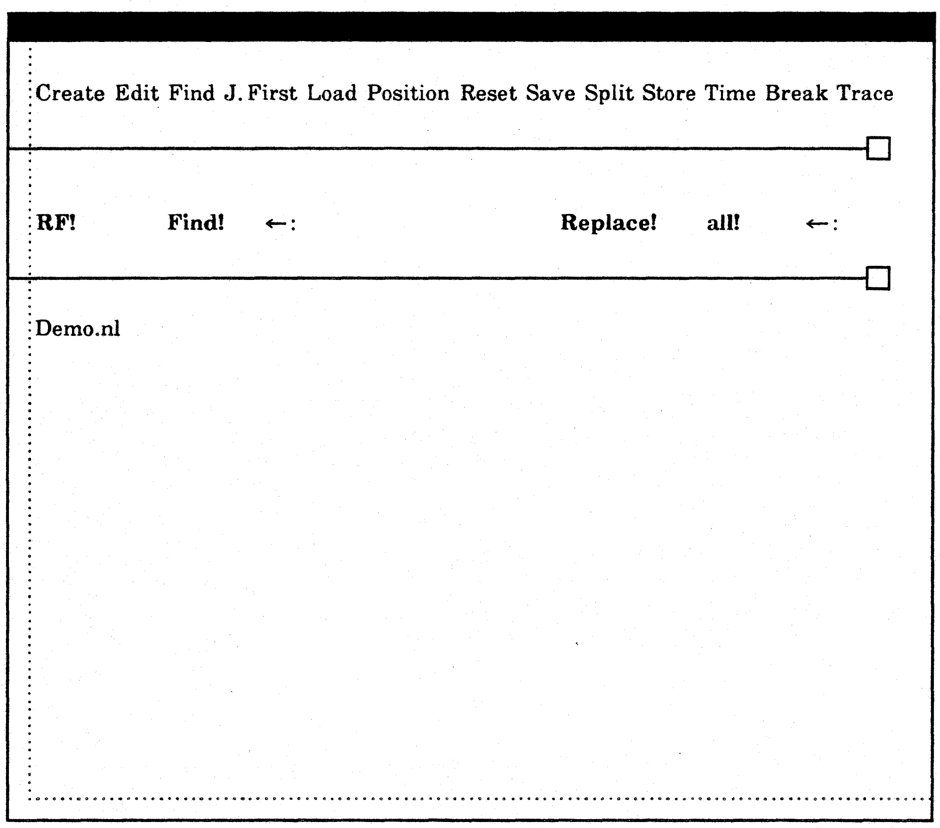


Figure 3.7

6) Select the letters you typed.
7) Load the SCICARDS interface list. Select **Load** in the file window's first subwindow. The system displays the net list in the third subwindow in SCICARDS interface format.
8) Select **Edit** in the first subwindow.
9) Select the colon following the **Find!** command in the second subwindow. Leave the cursor inside that subwindow. Type a carriage return; press the carriage return key.
10) Scroll back to the top of the subwindow. Select the colon after the **all!** command. Leave the cursor inside the subwindow. Type **PROP'S** and **J** simultaneously.
11) Select the **all!** command.
12) Save the file. Select the **Save** command in the first subwindow.

13) Select the **Destroy** command in the first subwindow to remove the window from the display.

3.5.3 Writing the Net List to Magnetic Tape

- To write the net list to magnetic tape:

1) Make the Tape Tool window active.
2) Change the **Tape Type** to **unlabeled.**
3) Enter the name of the output server to which you want to send the net list as the **Host.**
3) Enter the name of the net list's documentation set, followed by a period, followed by the letters "nl" as the **Source.**
4) Select any character in the word "**Dest'n.**" Type the word "magtape," followed by a period, followed by the magnetic tape file number, followed by a period, followed by the record length. Figure 3.8 shows the tape tool with a sample **Host, Source,** and **Dest'n** entered.

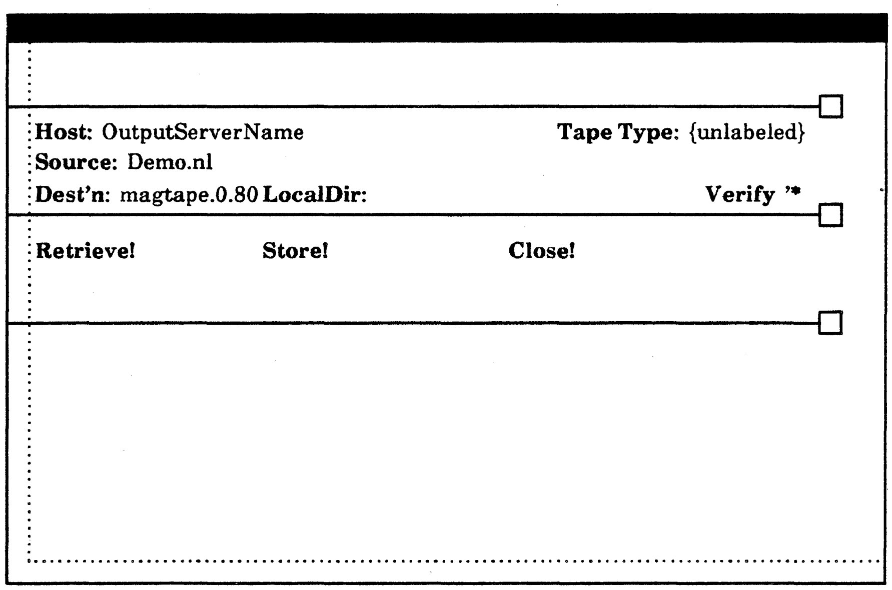


Figure 3.8

5) Select **Store!** in the Tape Tool command subwindow.

3.5.4 Designing the PC Board Layout with SCICARDS

- To design a printed circuit board with the SCICARDS system from an Expert 1000 schematic, SCICARDS users must follow several procedures.

  1) Users must rename SCICARDS connectors to match Expert connector names and numbers. Connectors must have an Expert symbol type of **Connector**.
  2) The user must check that there is a sufficient number of parts for all assignable functions. Use the SETUP COUNT command. Add additional parts to the SCICARDS subfile as required.
  3) Before reading the Expert file into the SCICARDS system, users must rename connectors to Expert designators and pin numbers. CARD, GRID, etc. should exist.

- To read the Expert magnetic tape:

  1) Using VAX/VMS, read the tape as /FOR/DENS=1600 (foreign density=1600 BPI), and create a VMS file. The following paragraph shows you the commands you use to instruct the VAX to read the magnetic tape. Prompts that the system displays are in regular type. Commands you enter are in bold type. Enter a carriage return at the end of each line. Replace the word **filename** with the name you want to use for your design file.

     $ **ALL MT:**
     $ **MOU MT: /FOR/DENS=1600**
     $ **COPY MT: filename.DAT**

     When using Prime, read the tape with MAGNET, and create a PRIMOS file. The following paragraph shows you the commands to use. Prompts that the system displays are in regular type. Commands you enter are in bold type. Enter a carriage return at the end of each line. Replace the word **filename** with the name you want to use for your design file.

     OK, **AS MT1**
     OK, **MAGNET**
     OPTION: **READ**
     MTU#= **1**
     MT FILE=**1**
     LOGICAL RECORD SIZE= **80**
     BLOCKING FACTOR= **1**
     ASCII, BCD, BINARY, OR EBCDIC? **ASCII**
     OUTPUT FILE: **filename**

  2) Complete the procedures above before reading the file to SCICARDS.
  3) Use the INPUT CR command to read the VMS or PRIMOS file into SCICARDS.

3.5.5 Back-Annotating from SCICARDS

To back-annotate the schematic with Expert, the SCICARDS user must first write the file to magnetic tape. He or she uses the OUTPUT CP command to create a VMS or PRIMOS file of LIST ALL. The VAX user writes the file to magnetic tape with VMS answering "yes" to the carriage control question, and entering the commands in the paragraph below.

$ ALL MT:
$ RUN UTIL: TAPECRE

The Prime user uses the TOVAX command at the SC command level as in the example below.

OK, **PROFILE**
SC COMMAND: **TOVAX filename**

- The Expert user uses the Tape Tool to read the magnetic tape.

1) Make the Tape Tool window active. Set the **Tape Type** to **unlabeled.**
2) Enter the name of your output server as the host.
3) Enter "MAGTAPE.*filenumber*" as the source. The file number is the number of the file you want to retrieve from the magnetic tape.
4) Enter a file name of your choice as the destination.

- To back-annotate the schematic.

1) Open the schematic.
2) Enter the file name you entered as the destination in the Tape Tool after the words "**FromFile:**" in the schematic command subwindow.
3) Select **BackAnnotate!**

3.6 Interfacing to Computervision™

You can use schematics created with Expert 1000 as input to a Computervision printed circuit board design system. The Computervision interface requires you to build your symbol library and schematic within certain limitations. When you complete the schematic, you convert it to Computervision format and write it to a magnetic tape. Give the tape to Computervision users to complete the board design. Expert 1000 currently does not support back-annotation from the Computervision system.

3.6.1 Building a Symbol Library to Interface with Computervision

- If you plan to use a Computervision system to design pc boards from Expert 1000 schematics, you must build your library with the following requirements:

  1) Symbol pin names must be less than or equal to 10 characters.

3.6.2 Building a Schematic to Interface with Computervision

Computervision must know symbol designators, physical part names, physical designators, physical pin numbers, and net names, and each has a maximum length. If you do not enter these properties, Expert automatically assigns default values.

- Use the following guidelines to create a schematic for use with a Computervision system:

  1) Expert automatically generates symbol designators for interfacing with Computervision. If you enter a symbol designator the system ignores it.
  2) Net names cannot exceed 20 characters. If you do not enter a net name, the system automatically generates a name such as "NN305." The prefix "NN" tells the Computervision user that the name is an Expert default name.
  3) Physical part names must be no longer than 12 characters. If you do not enter a physical part name, Expert assigns the symbol function from the Expert symbol library.
  4) Physical designators must be no longer than 7 characters. You enter the physical designator for each symbol or else Expert marks the symbol as an unassigned symbol awaiting assignment by the Computervision user.
  5) Physical pin numbers must be less than or equal to 10 characters. If you do not assign physical pin numbers, they are translated to Computervision with unassigned status.
  6) Device-specific parameters must be no longer than 4 characters. DEV1 must be value, DEV2 must be wattage or voltage, and DEV3 must be tolerance.

Computervision is a registered trademark of Computervision Corporation, Bedford, Massachusetts.

3.6.3 Creating a Computervision Wire List

- To convert the net list to Computervision format:

  1) Open the net list window.
  2) Select the **Computervision Wirelist!** command. The system displays the wire list in Computervision format. Figure 3.9 shows a Computervision wire list.
  3) Select the **Close!** command to close the net list window.

Close! **Net List!** **Part List!** **Pin List!** **SCICARDS Interface!**

Aquarius Wirelist! **LASAR Wirelist!** **Computervision Wirelist!**

1536	FF03	CONNECTOR	?	Nam	Nam	
1537	FF02	CONNECTOR	?	Nam	Nam	
1538	FF01	CONNECTOR	?	Nam	Nam	
1539	FF00	CONNECTOR	?	Nam	Nam	4
1547	IND	P1	P1	Nam	15	
1546	INC	P1	P1	Nam	14	2
1545	IND	7400	U11	In	4	
1545	NN284	7400	U11	In	5	
1545	NN287	7400	U11	Out	6	
1544	IND	7400	U11	In	10	
1544	NN282	7400	U11	Out	8	
1544	NN284	7400	U11	In	9	
1543	IND	7400	U11	In	1	
1543	NN281	7400	U11	Out	3	
1543	NN284	7400	U11	In	2	
1542	INC	7404	U10	in	5	
1542	NN284	7404	U10	out	6	1
1541	FF00	7476	U9	Q	Q	
1541	FF01	7476	U9	Q#	Q#	
1541	NN281	7476	U9	K	K	
1541	NN287	7476	U9	J	J	
1540	FF02	7476	U9	Q	Q	
1540	FF03	7476	U9	Q#	Q#	
1540	NN281	7476	U9	K	K	
1540	NN282	7476	U9	J	J	2

Figure 3.9

3.6.4 Writing the Computervision Wire List to Magnetic Tape

- Use the Tape Tool to write the Computervision wire list to a magnetic tape.

  1) Activate the Tape Tool window.
  2) Set the **Tape Type** to **unlabeled**.
  3) Enter the name of the documentation set, followed by a period and the letters "nl" as the **Source**.
  4) Select **Store!**

3.7 Interfacing to LASAR

You can format the net list for input to LASAR, and output it to magnetic tape with the output server. Use that tape as input to the LASAR program running on a mainframe computer.

3.7.1 LASAR Map

Before you can build schematics to interface with LASAR, you must have a special text file named "LASARMap.$Teton" stored on your local disk. The LASAR Map file maps Expert symbol descriptions to LASAR terminology.

The LASAR Map file consists of several *sections*. Each section describes a symbol. Each section has a name. The section name is either the symbol's Expert library function or physical part name from the symbol's property sheet in the schematic. The section name is enclosed in brackets. For example:

[74LS14]

Each section contains *key:item* pairs that map the Expert terminology to LASAR terminology. Each pair is on a separate line in the file. One pair maps the the section name to the LASAR component code. The *key* for this pair is "LASAR Alias." The *item* is the LASAR component code. For example:

LASAR Alias: AM25LS14

The remaining pairs form a *pin table*. The pin table maps either Expert pin names or pin numbers to LASAR pin codes. The first pair in the pin table tells the system whether the mapping is by Expert pin name or pin number. The key for this pair is "Pin Table." The item is "Names" if you are describing the following pins by their Expert library names, or "Numbers" if you are describing the pins by their Expert library pin numbers. For example:

Pin Table: Numbers

"Names" is the default case; if you omit the "Pin Table" pair, the system assumes that you are describing pins with their Expert names. The two options exist because different pins can have the same name, but pin numbers are unique.

The remaining pairs map individual Expert pins to the LASAR pin code. The Expert pin name or number is the key, and the corresponding LASAR pin code is the item. For example:

A:1

Separate LASAR Map sections with an empty line. Figure 3.10 shows a LASAR Map section that describes Expert symbol 74LS14.

[74LS14]	←	The section name is the Expert library name.
LASARAlias: AM25LS14	←	The LASAR component code for Expert symbol 74LS14.
Pin Table: Names	←	Pin table listing is by Expert names.
A:1 B:2 Y:3	←	Expert pin A is equivalent to LASAR pin 1. Pin B is equivalent to LASAR pin 2. Pin Y is equivalent to LASAR pin 3.

Figure 3.10

The LASAR Map file must have one section named "Global Definitions" that describes sections associated with input/output type functions. For example, you must define connectors and test points in this section so that Expert can interpret them correctly when generating the LASAR wire list.

The Global Definitions section maps function types to sections. Figure 3.11 shows a Global Definitions section.

		[Global Definitions]		
The type of function	→	I/O: CONNECTOR	←	LASAR Map section associated with the function

Figure 3.11

Creating a LASAR Map File

Although individual users can create their own LASAR Map files, it is more efficient to have one person, perhaps the system librarian, create a single LASAR Map file that contains LASAR descriptions of all symbols in your installation's library. You can store that file on a central file server. Users can retreive it with the File Tool.

- To create a LASAR Map file:

1) Compile the information you need from the Expert symbol library, and from the LASAR library.
2) Create an empty file window. See Book 1 section 3.5.
3) Use text manipulation techniques to enter each section in the file window. Use carriage returns to create new lines. Leave an empty line between sections.
4) Create a second file window. In its text subwindow type the word "LASARMap.$Teton" Do not type the quotation marks. Select the letters you typed. Select **Store** in the LASAR Map file window.

Editing the LASAR Map File

If you are responsible for updating the LASAR Map file, you might want to add a section to the file each time you add a symbol to your library. You then store the new LASAR Map on a central file server, and notify other users.

- To edit the LASAR Map file:

1) Create an empty file window.
2) In its text subwindow type the word "LASARMap.$Teton" Do not type the quotation marks. Select the letters you typed, and select **Load** in the empty file window's command subwindow or File Window Menu. The system loads the LASAR Map file into the text subwindow.
3) Use the text editing techniques described in Book 1, section 3.5 to edit the file.
4) Save your changes by selecting **Save** in either the command subwindow or File Window Menu.

3.7.2 Building a Schematic to Interface with LASAR

- To build a schematic to interface with LASAR follow these rules:

  1) All symbol properties and net names must be three characters long. The first character must be alphabetic, and the second two numeric. For example:

     C55

     The simplest way to construct a schematic to interface with LASAR, is to construct it without entering any symbol properties (except for connectors) or net names. The system automatically assigns LASAR codes.
  2) You must enter physical designators and pin numbers for all connectors. Physical Designators must also be three characters long. The first character must be alphabetic, and the second two numeric.
  3) The maximum number of symbols in the schematic must be less than 2,600.

3.7.3 Converting the Net List to LASAR Format

- To convert the net list to LASAR format:

1) Open the net list window.
2) Select the **LASAR Wirelist!** command. The system displays the wire list in LASAR format. It marks truncated fields with a tilde (~). Figure 3.? shows a LASAR wire list.

```
Close!   Net List!        Part List!    Pin List!     SCICARDS Interface!
Aquarius Wirelist!   LASAR Wirelist!  Computervision Wirelist!

$ALEC
PRINT
MODEL DEBUG
WIRELIST TERADYNE
$EOD
Demo   /CREATED ON 24-May-83 14:44:39
7400    AAA     SN7400D
7404    AAB     7404
7476    AAC     SN7476D
CIRCUIT
P0 P2T1,f41P3   /FF00
P0 P3T1,f41P4   /FF01
P0 P4T1,f40P3   /FF02
P0 P5T1,f40P4   /FF03
P1 P1T14,f42P1  /INC
P1 P1T15,f43P1,f44P1,f45P1    /IND
   f41P2,f40P2,f43P3 /NN281
   f40P1,f44P3   /NN282
   f43P1,f44P1,f45P1,f42P2    /NN284
   f41P1,f45P3   /NN287
DEVICE
f43,f44,f45 = 7400
f42 = 7404
f40,f41 = 7476
$EOD
```

Figure 3.12

3) Check the wire list for errors. Errors occur when the LASAR description of a pin is not in the LASAR Map. The system flags errors with the word "ERROR," and a description of the error. If there are errors, close the net

list window, and edit the LASAR Map file. Then repeat steps 1 through 3.

4) When the system displays the wire list without errors, select the **Close!** command to close the net list window.

3.7.4 Writing the LASAR Wire List to Magnetic Tape

- Use the Tape Tool to write the LASAR wire list to a magnetic tape.

1) Activate the Tape Tool window.
2) Set the **Tape Type** to **unlabeled**.
3) Enter the name of the documentation set, followed by a period and the letters "nl" as the **Source**.
4) Select **Store!**

The Library Editor

2.2 Displaying Text Fonts

When you create a symbol, you assign font numbers to the text items in its definition. You assign properties (family and height) to each font when you create a schematic format. Font properties control the appearance of the text when you display the schematic and when you plot it. (See Book 2, section 2.11.)

Unless you instruct the Library Editor otherwise, the system applies default font properties to text items inside the Library Editor window. The Library Editor's **Documentation Set:** command lets you instruct the Library Editor to display text items as they would actually appear in a particular schematic. For instance if you assign font 3 to a pin name, the Library Editor looks at the schematic you specify to see with which properties to display font 3. It then displays the pin name with those properties.

- To specify font properties from a particular schematic:

1) Enter the name of the schematic's documentation set after the words "**Documentation Set:**" in the Library Editor's command subwindow.

2.3 Reading Symbols

Before you can alter a symbol you must transfer a copy of that symbol from the library to the Library Editor window; this is called *reading* the symbol from the library. To read a symbol, you must know its identification number. If you don't know the symbol's identification number, set the entry type to **library** and use the **List!** command to list all entries (see section 3.1), or use the *Abbreviation Map* to list symbols and their identification numbers (see section 2.9.1).

- To read a symbol from the library:

1) Replace the entry identification number with the symbol's identification number.
2) Set the grid and template sizes.
3) Select the **Read!** command.

The system displays the symbol in a third subwindow, and adds several commands to the command subwindow. See figure 2.1.

2.7.5 Assigning Fonts to Text Items

You assign a font number to each text item, including symbol property location markers, pin names, pin numbers, and random text. Font numbers range from 1 to 16. A format designer assigns each font number a family and height in each schematic. For example, if you want the system to display pin number text the same as pin name text, assign the same font number to all pin names and pin numbers.

A new symbol has default font settings. The default for the logic uniqueness text is font 2. The default for physical name text is font 3. The default for device parameter text is font 4. Pin numbers default to font 5, pin names to font 6, and random text to font 7.

- To assign a font number to a text item:

1) Select the text item.
2) Press the **PROP'S** key to display the text item's property sheet.
3) Delete the current font value and replace it with the value you want to assign.
4) Select **Apply!**.

VERSATEC
A XEROX COMPANY

Expert 1000 User's Guide

Schematic Design

This document was prepared with the Xerox 8010 Information System.

Document Number 1-40B May 1983 Printed in USA

Table of Contents

1. Introduction

Book 2 tells you how to create a schematic using Expert 1000. It describes the schematic window, commands for building a schematic inside the window, and the net list window. It assumes that you have read Book 1 and are familiar with Expert 1000's basic terms and procedures.

2. Building a Schematic

- To build a schematic, you must first create a new documentation set for the schematic.

  1) Find an empty documentation set containing a schematic with the format you want to use. Use the Window Manager to make that window active.
  2) Select **Copy!** The system creates a new, tiny documentation set at the bottom of the display. Make the original documentation set window tiny.
  3) Make the new documentation set window active.
  4) Select **Props!** to bring up the documentation set property sheet. Delete the temporary documentation set name, and enter a more descriptive name for the documentation set. Select **Apply!**

- Once you have created the documentation set, *open* the schematic.

  1) Select any character in the word "**Schematic**" in the documentation set window.
  2) Select **Open** in the Document Operations Menu, or, with the cursor inside the documentation set window, press the function key marked **OPEN**.

2.1 The Schematic Window

The schematic window has three subwindows. The first, the system message subwindow, gives you feedback from the system. The second is a command subwindow, and the third displays the schematic.

The system displays the Browsing Schematic Menu in the third subwindow. The Browsing Schematic Menu lets you view a certain part of the schematic in the third subwindow, and select objects in the subwindow, for example to copy them to another schematic. Section 2.1.7 explains how to use the Browsing Schematic Menu commands: **Center Source**, and **Add to Selection**. The Browsing Schematic Menu does not let you make changes to the schematic in the schematic subwindow. To make changes to a schematic you must enable editing.

Figure 2.1 shows the schematic window with the Browsing Schematic Menu.

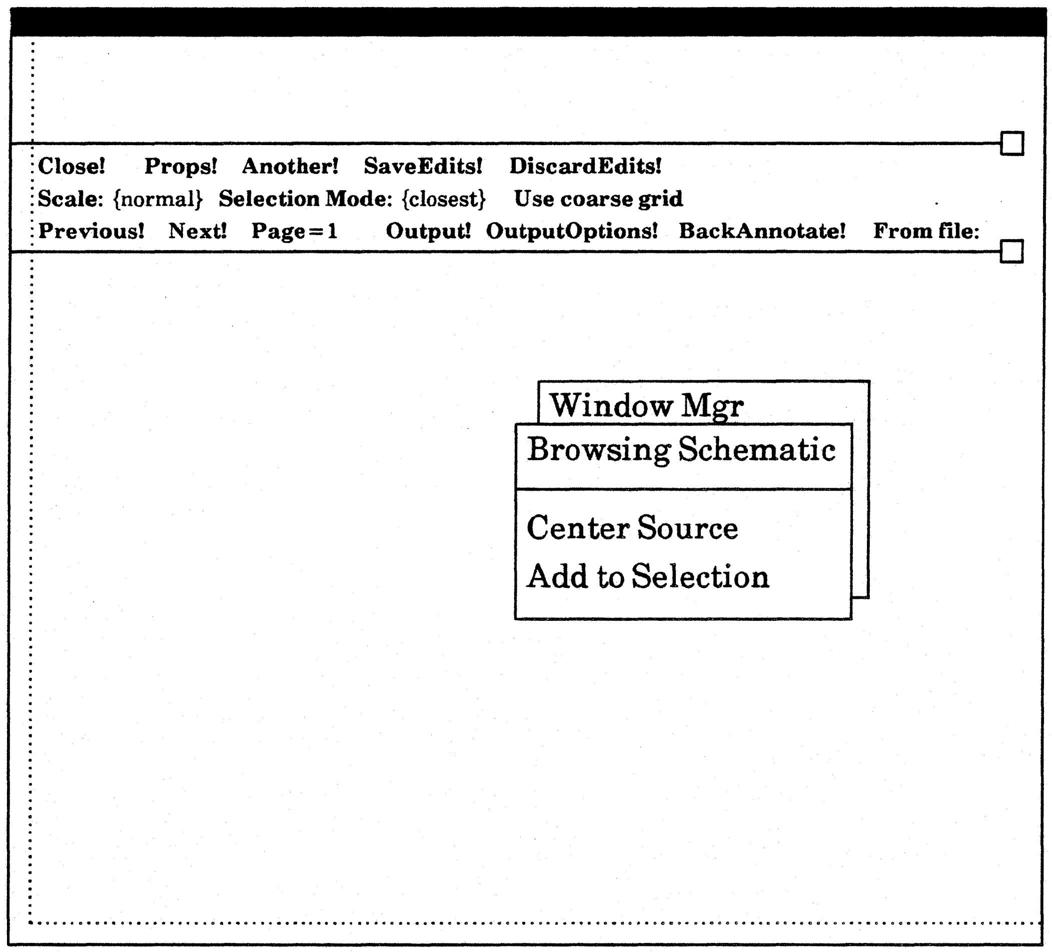


Figure 2.1

2.1.1 Enabling Editing

You cannot make changes to a schematic until you enable editing.

- To enable editing:

1) Select **EditSchematic!** in the schematic window's command subwindow.

The system removes the **EditSchematic!** command (it is no longer relevant), and replaces it with commands for saving or discarding your edits once you complete them. After you enable editing, the system displays the Editing Schematic Menu in the schematic subwindow. The Editing Schematic Menu contains commands you use to create a schematic diagram inside the subwindow. Figure 2.2 shows the Editing Schematic Menu.

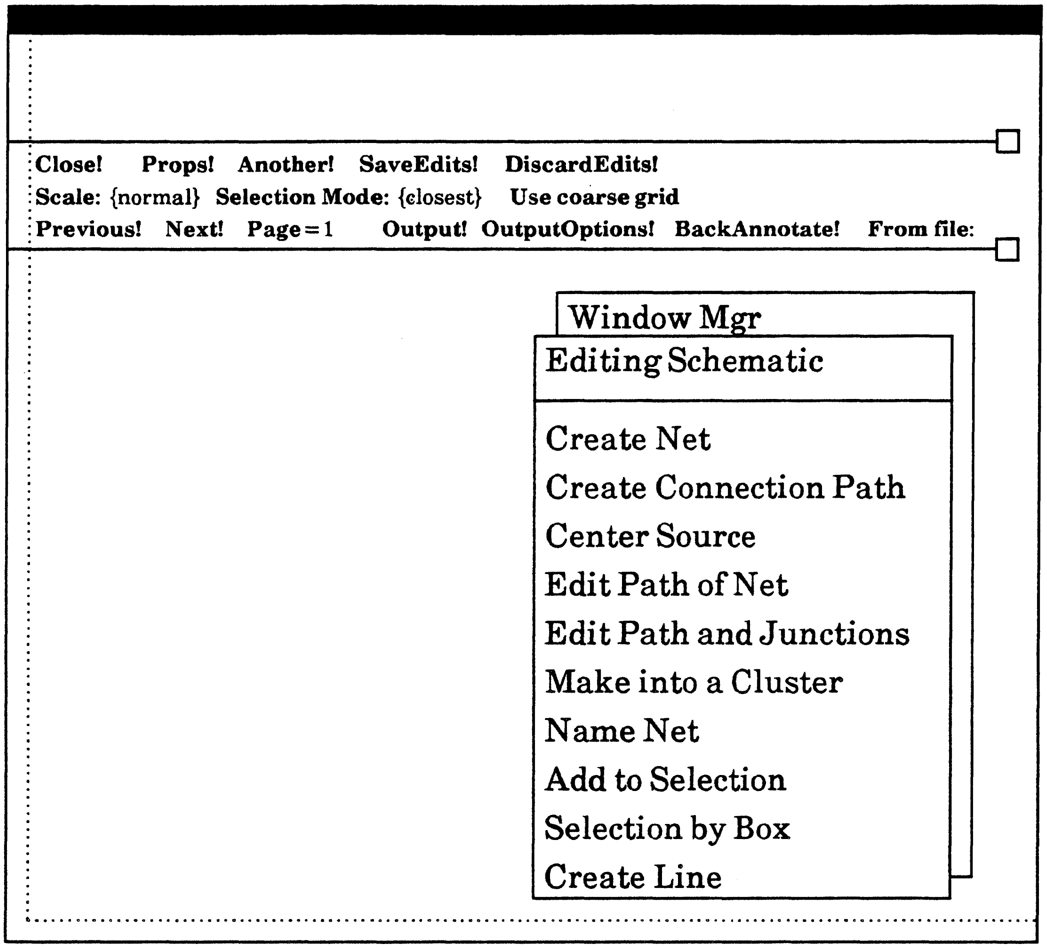


Figure 2.2

2.1.2 Multiple Page Schematics

An Expert 1000 schematic can be as many pages as you need. The number of the page you are looking at follows the word "**Page=**" in the command subwindow. You move from page to page by selecting the **Next!** and **Previous!** commands in the command subwindow.

- To view the next page:

  1) Select **Next!** in the command subwindow.

- To view the previous page:

  1) Select **Previous!** in the command subwindow.

2.1.3 Changing the Scale of a Schematic

When you first open the schematic, the system displays the schematic at normal scale. You can change the scale to one-eighth, one-fourth, one-half, twice, or four times the normal scale. The system shows you the current scale inside the brackets after the word "**Scale:**" in the command subwindow.

- To change the scale of a schematic:

1) Point the cursor at the word "**Scale:**" in the command subwindow.
2) Activate the Scale Menu. The Scale Menu contains the scale options: **eighth, fourth, half, normal, twice** and **four**.
3) Select a scale from the menu. The system displays the schematic at the scale you requested, and displays the new scale after the word "**Scale:**" in the command subwindow.

2.1.4 Multiple Views

You can create as many views of the schematic window as you want. In these other windows, you can display different pages of the schematic, or display the same page at different scales. All the windows reflect the changes that you make in any one window. If you have two windows displaying the same part of the schematic, you can actually see the system make changes in both windows.

- To create another view of a schematic:

1) Select **Another!** in the command subwindow.

- To remove a schematic window from the display:

1) Select **DestroyView!** in that window's command subwindow.

The system will not let you remove the last window from the display without closing the schematic. It automatically removes the **DestroyView!** command from that window's command subwindow.

2.1.5 The Schematic Window Property Sheet

The schematic window property sheet lets you control the way the system displays the schematic. Figure 2.3 shows the schematic window property sheet.

Apply! **Abort!** **Coarse Grid Unit:** {$\frac{1}{2}$ inch} **Id page defines** =
Display options: {thisView}
Designators: {none} **Pin Names:** {monoliths} **Physical Name** **Random Text**
Device parameters ... **1 2 3** **Show grid ticks**
Default for parameter ...

Capacitor	**1:**	**2:**	**3:**
Coil	**1:**	**2:**	**3:**
Diode	**1:**	**2:**	**3:**
Relay	**1:**	**2:**	**3:**
Resistor	**1:**	**2:**	**3:**
Transistor	**1:**	**2:**	**3:**

Figure 2.3

Coarse Grid Unit

Each schematic has a coarse grid for controlling the placement of objects in the schematic. You set the grid units in the schematic window's property sheet. Coarse grid unit options are: **2 grid units**, **3 grid units**, **$\frac{1}{2}$ inch**, and **1 inch**.

Id page defines

Id page defines is a numeric property. You use it to tell the system which symbol a schematic page defines in a hierarchical schematic. Section 2.7 describes hierarchy.

Display Options

The remainder of the schematic window properties are options for displaying the schematic inside the window. You choose whether the system displays certain symbol properties on the schematic. For instance, to build a logical schematic, you display symbol designators for each logic symbol. To build a physical schematic, you display physical designators, pin numbers, physical names, and pin names. You may also choose to display device parameters and grid ticks.

The **Display Options** property lets you control whether the system displays grid ticks and the symbol properties you select in all views of the schematic or just in the view in which you are working.

Designators

Each logic symbol has unique symbol and physical designators in its property sheet. The **Designators** property lets you control whether the system displays no designators, both symbol and physical designators, just symbol designators, or just physical designators on the schematic.

Pin Names

A logic symbol's pin names are properties of that symbol. You can instruct the system to display the pin names on the schematic. When you first open the schematic, the system displays pin names only for *monoliths*. Monoliths are symbols more complex than small-scale integrated circuits. You can change this property so that the system displays no pin names or pin names for all symbols.

Pin Numbers

Pin Numbers is a boolean property. When the property is on, the system displays pin numbers for each symbol. When it is off, the system does not display pin numbers.

Physical Name

Physical Name is also a boolean property. When it is on, the system displays physical names for logic symbols.

Random Text

Random Text is a boolean property. When it is on, the system displays random text associated with the symbol.

Grid Ticks

Turn on the **Show Grid Ticks** property to display grid marks on the schematic. Turn it off to remove grid marks.

Device Parameters

You may assign device parameters to symbols in their property sheets, for example, to specify the ohm value, wattage, and tolerance of a resistor. Device parameters are numbered one through three. To display device parameter one for all symbols, turn on the boolean property **1**. To display device parameter two for all symbols, turn on the boolean property **2**. To display device parameter three for all symbols, turn on the boolean property **3**.

Device Parameter Defaults

You can set default parameters for discrete devices in the schematic window property sheet. Each device listed is followed by text properties **1**, **2**, and **3**. The system automatically assigns default device parameters to the corresponding symbols. For example, if you enter default parameter values for resistors, the system automatically assigns those parameters to each resistor you place in the schematic. You can change device parameter values for an individual device in the device's symbol property sheet.

2.1.6 Graphic Objects

An Expert 1000 schematic contains four types of graphic objects: symbols, text, lines, and nets. (See figure 2.4.) These objects are different from each other; they have different properties, and different electrical meanings. For example, a line is different from a net. Nets show electrical connections between symbols. When you delete a symbol the system automatically deletes the nets that connect to that symbol. A line has no electrical meaning at all.

Symbols, text, lines, and nets are the only graphic objects that you need to use to create an Expert 1000 schematic.

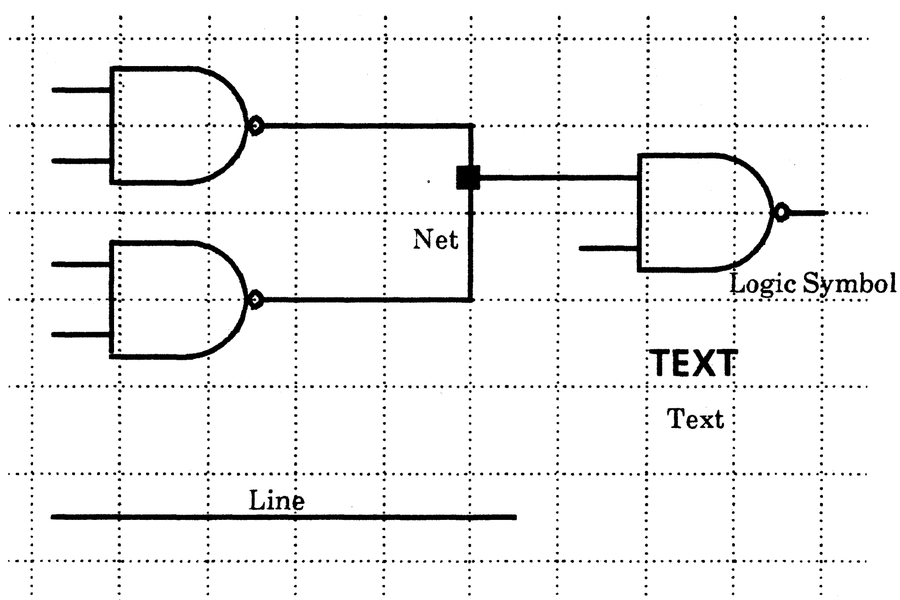


Figure 2.4

2.1.7 Selection

Book 1 explained the concept of selection. This section describes selection methods unique to the schematic window.

Each time you select an object in the schematic with the left mouse button, you also select a point on the schematic grid. The system video-inverts the selected object, and marks the selected grid point with a *source marker*. You can select a second point on the grid with the right mouse button. The system marks that point with the *destination marker*. Figure 2.5 shows source and destination markers.

The source and destination markers let you place objects on the schematic. When you want to add a logic symbol to the schematic, for instance, you mark the point on the grid at which you want to place that symbol with the source marker. When you move an object, you select the object, and then mark the point to which you want to move it with the destination marker.

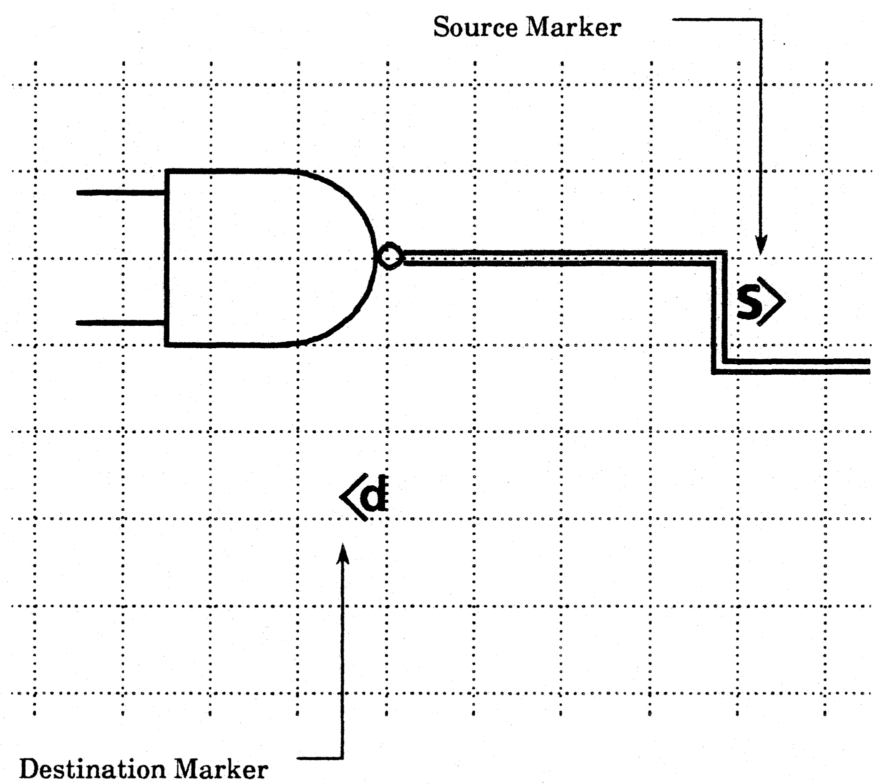


Figure 2.5

Using the Coarse Grid

You can use the a *coarse grid* to control how closely you place objects on the schematic. When you use the coarse grid, you can only select grid points a certain distance apart, for instance every half-inch. When you first open a schematic window the coarse grid is off. The default coarse grid unit is one-half inch.

- To turn on the coarse grid:

  1) Point the cursor at the words "**Use coarse grid**" in the schematic command subwindow.
  2) Press the left mouse button. The system video-inverts the words "**Use coarse grid.**"

- To turn off the coarse grid:

  1) Point the cursor at a the words "**Use coarse grid**" in the schematic command subwindow.
  2) Press the left mouse button.

- To change the coarse grid unit:

  1) Select **Props!** in the command subwindow to display the window's property sheet.
  2) Point the cursor at the words "**Coarse Grid Unit,**" and press both mouse buttons or the **CENTER** key.
  3) The system displays the Coarse Grid Unit Menu containing the coarse grid options: **2 grid units, 3 grid units, $\frac{1}{2}$ inch**, and **1 inch**.
  4) Select one of these grid units.
  5) Select **Apply!**.

Centering the Source Marker

- You can center the source marker in the schematic subwindow. The **Center Source** command is in both the Editing Schematic and Browsing Schematic menus.

  1) Use the left mouse button to set the source marker in the schematic subwindow.
  2) With the cursor inside the schematic subwindow, activate the dynamic Menus.
  3) Select the **Center Source** command from the Editing Schematic or Browsing Schematic Menu.

This command is particularly useful when you want to display a certain part of a large schematic. Scale the window to one-half or one-fourth its normal scale so that you can see the whole schematic. Select a grid point in the area of the schematic that you want to display in the subwindow. Make the scale larger, and issue the **Center Source** command.

Selection Mode

Initially, when you select any point on the grid, the system selects the closest graphic object. The schematic's selection mode lets you tell the system to select only a particular type of object: a net, a line, a symbol, text, or the closest object. If you set the selection mode to **line**, for example, the system selects the closest line when you select any point on the grid. The system displays the selection mode in brackets after the words "**Selection Mode:**" in the command subwindow.

- To change the selection mode:

1) Point the cursor at the the words "**Selection Mode:**" in the command subwindow.
2) Activate the Selection Mode Menu. The Selection Mode Menu displays the selection mode options: **closest**, **net**, **line**, **symbol**, and **text**.
3) Select the type of symbol you want the system to select. The system displays the new selection mode following the words "**Selection Mode:**" in the command subwindow.

Selecting Several Objects at Once

You select an object because you want to do something with that object. For example, to delete a logic symbol, you select the symbol then press the **DELETE** key. You can also select several objects at once. For example, you delete several logic symbols quickly by selecting all the logic symbols you want to delete, and then pressing the **DELETE** key just once. There are two ways to select several objects: adding to a selection, and selection by box.

Adding to the Selection

- To add individual objects to the selection:

1) Select one object.
2) Select **Add to Selection** in the Editing Schematic or Browsing Schematic Menu.
3) Select each of the objects you want to add to the selection. The system video-inverts all of the objects you select.
4) Bring up the Editing Schematic or Browsing Schematic Menu and select **Stop Adding to Selection**.

The selection mode limits the **Add to Selection** command. If you've set the selection mode to net, for instance, you can only select nets with the **Add to Selection** command.

Selection by Box

- To select all the objects within a rectangular region of the schematic.

  1) Mark the opposite corners of the rectangular region with the source and destination markers.
  2) Select **Selection by Box** in the Editing Schematic Menu.

The system video-inverts all of the objects wholly contained inside the rectangular region. For example, if a connection is not completely within the box, the system does not select it. Like the **Add to Selection** command, the system only selects objects specified by the selection mode.

2.2 Logic Symbols

2.2.1 The Abbreviation Map

The system stores the physical and functional description of every logic symbol you use in a special symbol library. In the library, the symbol has a unique identification number and name. In addition, each symbol may have one or several nicknames. You must know a symbol's nickname or identification number in order to place the symbol in the schematic. The *Abbreviation Map* is the tool you use to list symbol nicknames and identification numbers. See figure 2.6.

- To open the Abbreviation Map window:

1) Move the cursor to an empty area on the display.
2) Bring up the Inactive Menu.
3) Select **Abbreviation Map**. The system displays the Abbreviation Map window.

The Abbreviation Map window contains a system message subwindow, a command subwindow, and a third subwindow in which it lists symbol nicknames and numbers.

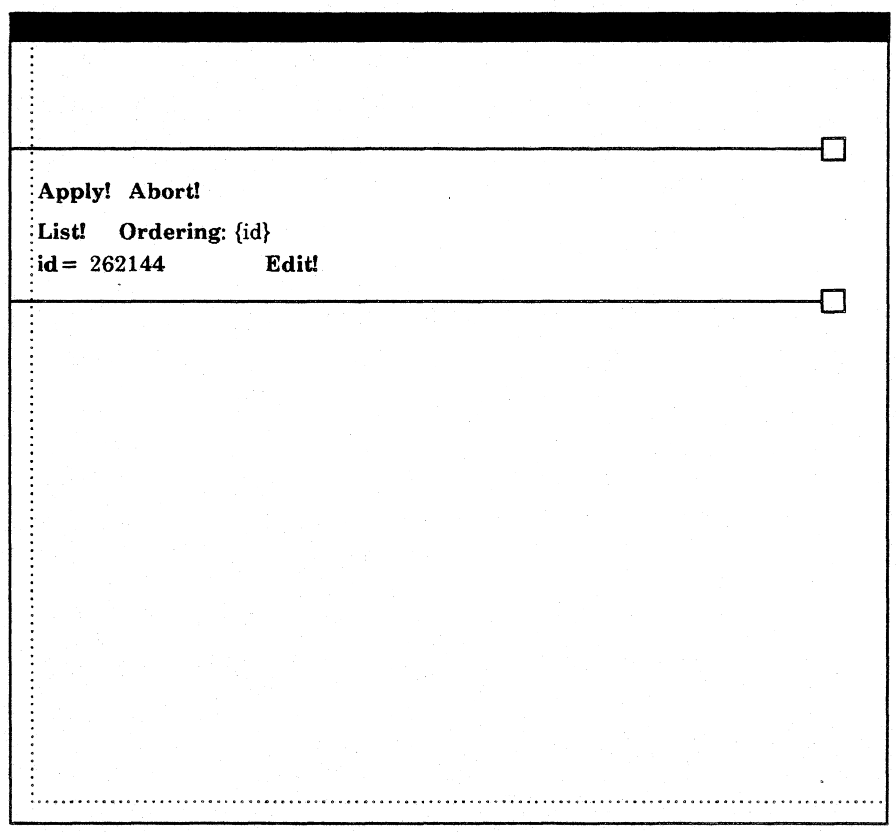


Figure 2.6

- To display a list of symbols in the abbreviation map in the order of their identification numbers:

  1) Select the **List!** command in the Abbreviation Map command subwindow.

  The system lists each symbol with all of its nicknames. You may use the symbol's identification number or any one of its nicknames to identify a particular symbol. Figure 2.7 shows symbols listed by identification number.

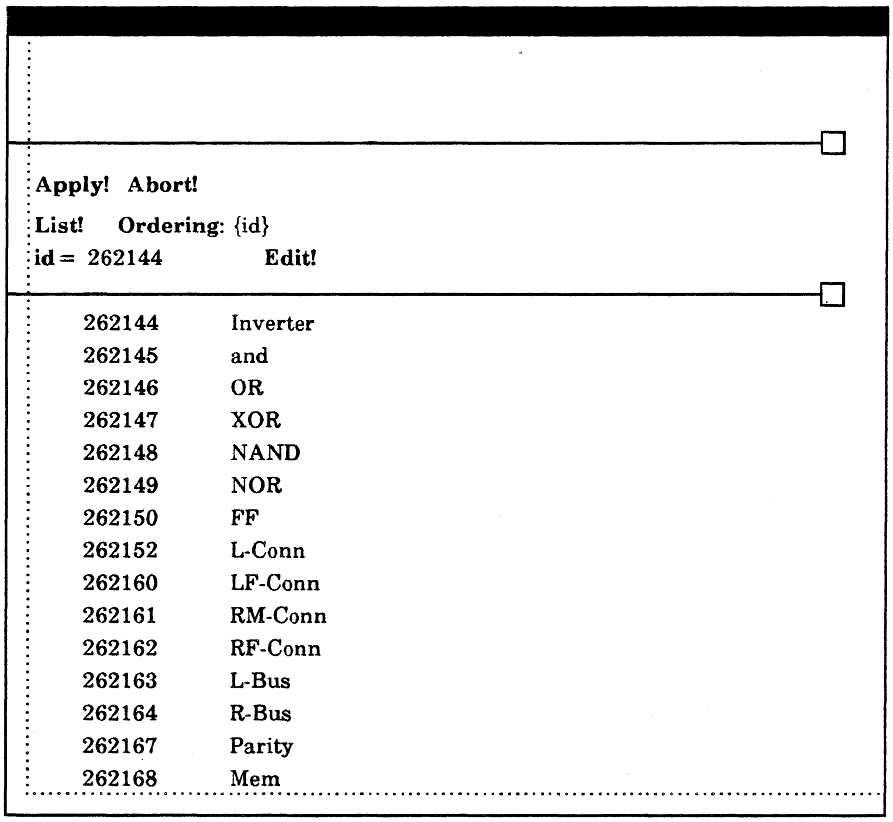


Figure 2.7

- To list symbols alphabetically by nicknames:

1) Point the cursor at the word "**Ordering**" in the command subwindow. Activate the Ordering Menu.
2) Select **nickname**.
3) If you want to list only the symbol with a particular nickname. Type that nickname after the words "**NicknameListKey**."
4) Select the **List!** command.

Figure 2.8 shows symbols listed alphabetically.

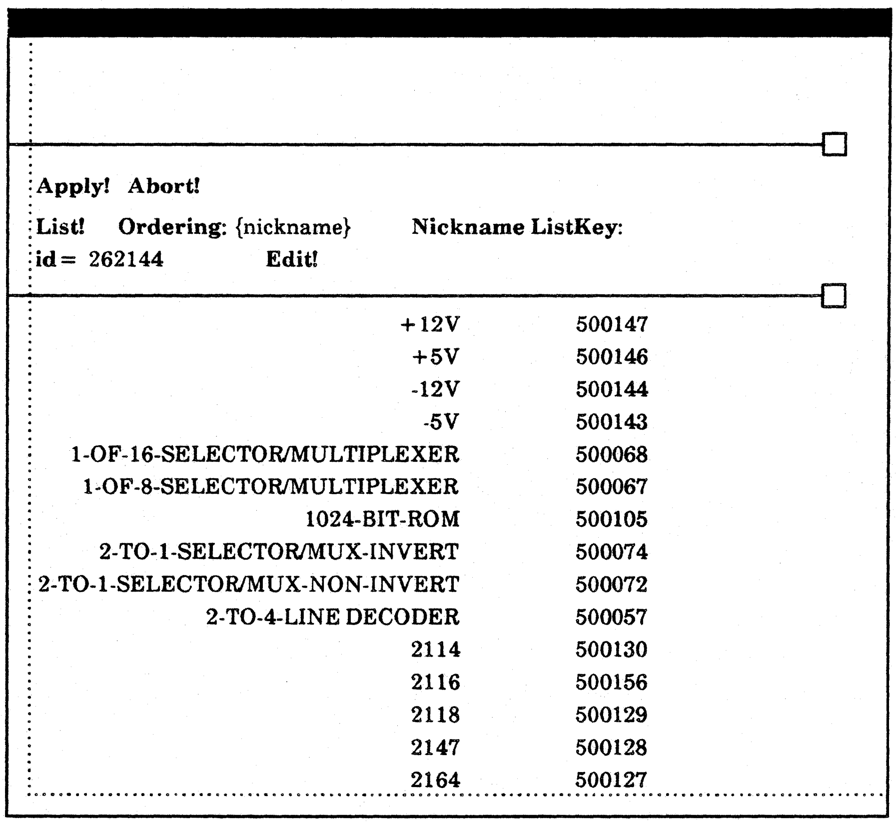


Figure 2.8

You may want to keep the Abbreviation Map open on the display while you are working. Like any other window, you can make it tiny, and change its shape or location. You can also close the Abbreviation Map, returning it to its inactive state.

- To close the Abbreviation Map:

1) Select **Abort!** or **Apply!** in the Abbreviation Map command subwindow.

2.2.2 Placing a Symbol

- To place a symbol on the schematic:

  1) Select the point on the grid at which you want to place the symbol. The system marks the point with the source marker.
  2) Type the symbol nickname or identification number.
  3) The system automatically selects (video-inverts) the text as you type it. See figure 2.9.
  4) Select **Make into a Symbol** in the Editing Schematic Menu, or press the key marked **EXPAND**.

  The system replaces the text with the corresponding logic symbol. See figure 2.10.

  You can also *copy* symbols to create new ones. For example, you might create a template schematic containing the symbols you use most often. To place a symbol in a new schematic, you select it in the template, and copy it to the new schematic. You can also copy symbols within schematics. Section 2.7 describes the **COPY** command and other schematic editing commands.

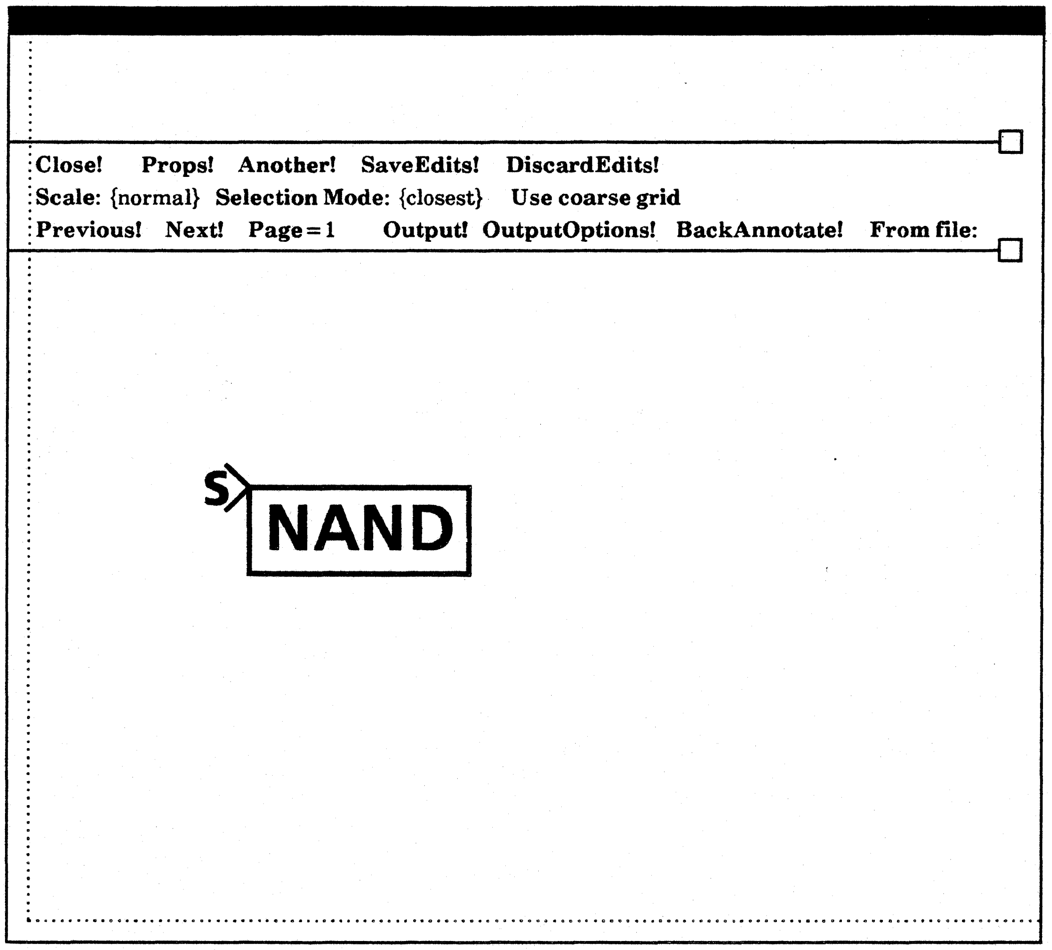

Close! Props! Another! SaveEdits! DiscardEdits!
Scale: {normal} Selection Mode: {closest} Use coarse grid
Previous! Next! Page=1 Output! OutputOptions! BackAnnotate! From file:
S
NAND

Figure 2.9

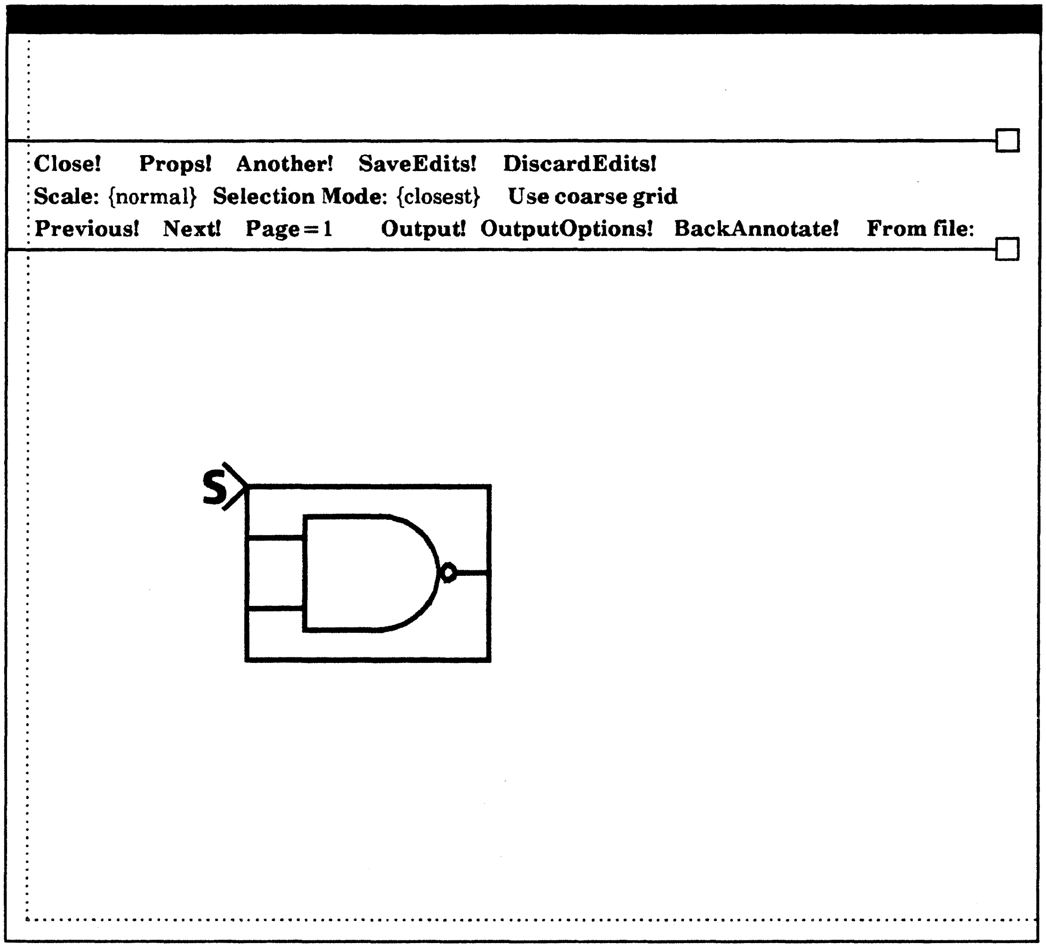


Figure 2.10

2.2.3 Logic Symbol Properties

The logic symbol property sheet lets you view and change the characteristics of individual logic symbols. You can assign symbol designators, physical designators, physical names, physical pin numbers, physical part names, and device parameters. You choose which of these characteristics you want to assign. Use the schematic window property sheet to display certain symbol properties as annotation.

- To display a logic symbol's property sheet:

  1) Select the symbol.
  2) Press the **PROP'S** key.

Figure 2.11 shows a logic symbol property sheet.

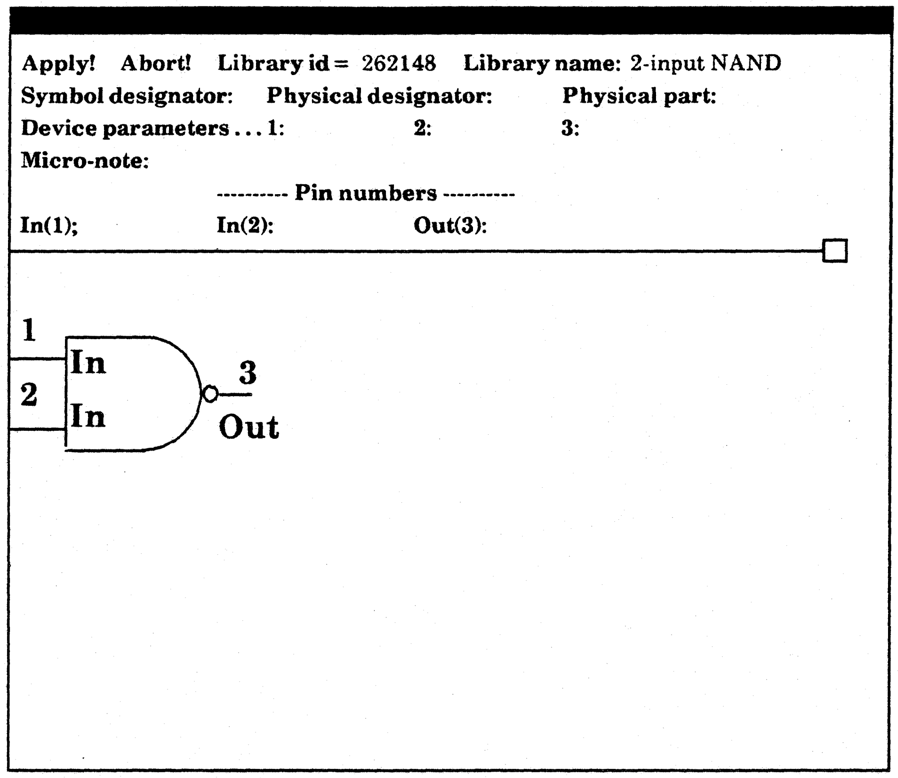


Figure 2.11

Library Id

The library identification number is the number that the system uses to recognize the symbol. Each logic symbol has its own identification number. You cannot change a symbol's identification number.

Library Name

The symbol's library name is the name corresponding to the symbol's identification number. For example, "Generic 2-input NAND" might be the library name for the symbol whose identification number is 10000042. You cannot change the library name.

Symbol Designator

You can label each individual symbol with a symbol designator. No two symbols in the same schematic can have the same symbol designator. Use the schematic window property sheet to display the designator on the schematic.

Physical Designator

Physical designators show symbol packaging and placement on the printed circuit board. Several symbols can have the same physical designator. Use the schematic window property sheet to display physical designators.

Physical Part

In addition to assigning a physical designator to a symbol, you can assign it to a physical part type, for example 7400. Enter the physical part name in the symbol's property sheet.

Device Parameters

Device parameter properties let you describe particular electrical characteristics of a symbol. For example, you might enter the ohm value (10K) of a resistor as device parameter one, its wattage ($\frac{1}{4}$) as device parameter two, and its tolerance (5%) as device parameter three.

In the schematic property sheet, you set default device parameters, and instruct the system to display any or all parameters.

Micro-note

You can keep notes about a symbol in its property sheet. Simply type the note after the word "Micro-note:" in the property sheet. In general, a micro-note is additional information that you want the symbol to have, but that you do not want to display on the schematic itself. The micro-note is a way to leave information for the board designer as well as for yourself or other engineers with whom you might be sharing the design.

Pin Numbers

The second subwindow in the logic symbol property sheet shows you a diagram of the symbol with its pin names and numbers. You can assign physical pin numbers to each pin.

2.3 Connecting Symbols

Nets show the electrical connection between symbols. Nets consist of one or more *connection paths*. A connection path shows the connection between two symbols, two connection paths, or a connection path and a symbol. You can use *connectivity symbols* to draw nets in segments. You create and edit net segments as if they were individual nets, but the system recognizes them as part of the same net.

You have two ways to create nets. You can create a net by selecting its endpoints and instructing the system to draw the connections between the endpoints, or you can create connection paths to merge two nets, connect a symbol to an already-existing net, or connect two symbols. The system also supplies you with tools to edit net paths and name nets.

2.3.1 Creating Nets

- To create a net between logic symbols:

1) Select **Create Net** in the Editing Schematic Menu.
2) As you move the cursor on the display, the system places a box at the end of the closest symbol connection point (figure 2.12). Move the box to one of the symbol connection points for the net you want to create, and press the left mouse button. The system marks that point with a smaller, solid box (figure 2.13). Mark all of the symbol connection points that are on the net (figure 2.14). If you make a mistake, or decide that you do not want to include a marked connection point on the net, move the larger box back to that connection point and press the right mouse button. The system removes the solid box.

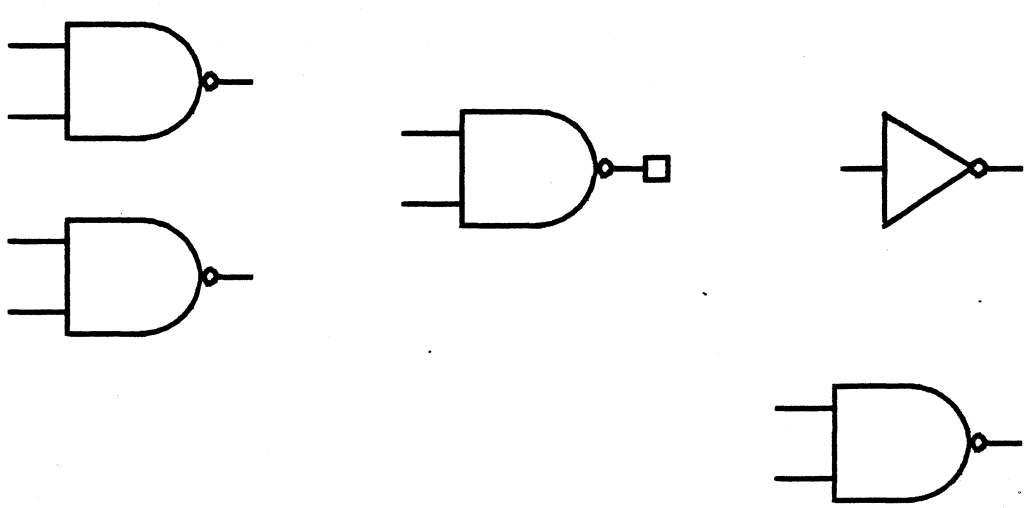


Figure 2.12

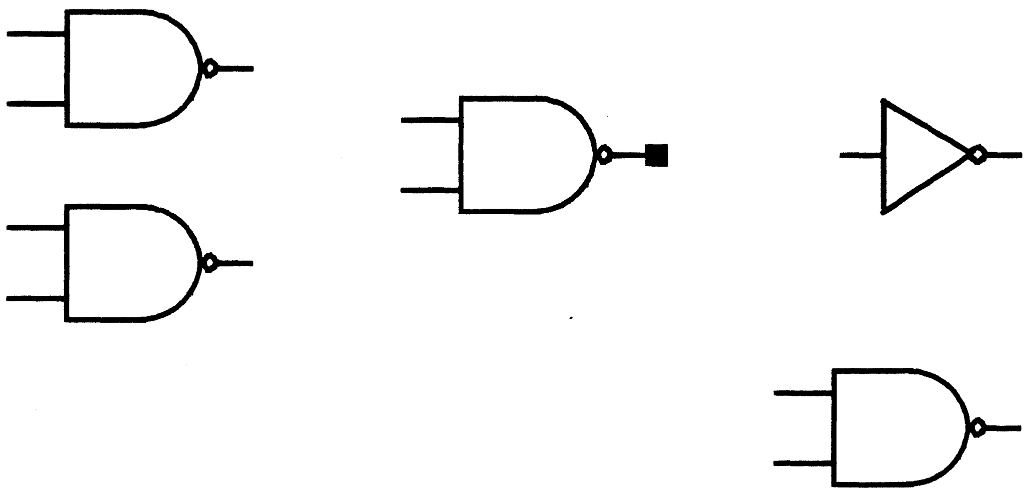


Figure 2.13

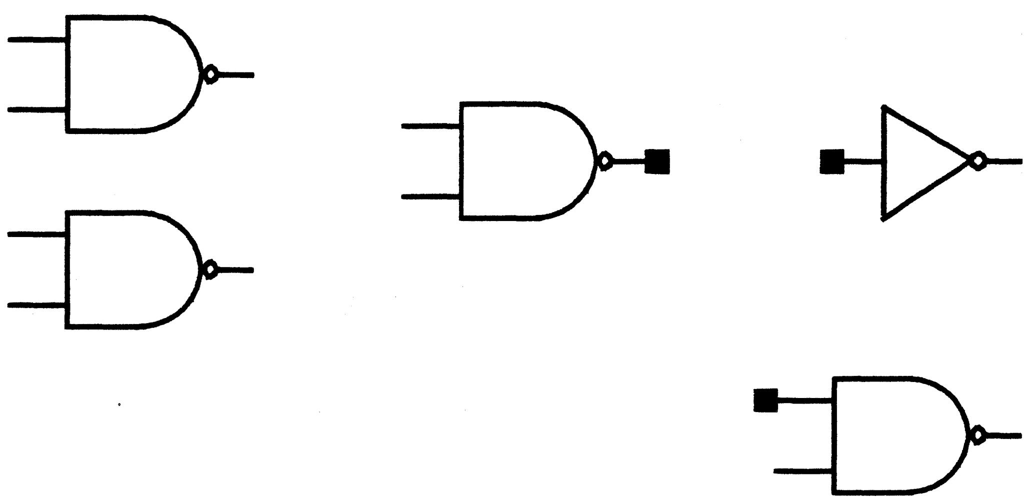


Figure 2.14

3) Once you have marked all the endpoints for the net, bring up the Create/Modify Net Menu. Notice that the system displays only this one menu; it is the only menu relevant to your current task, creating nets. The Create/Modify Net Menu gives you the options of applying the net between the marked connection points or aborting the net. If you select **Apply Net Edit**, the system draws the net path between the connection points marking junctions with solder dots. Selecting **Abort Net Edit** aborts the net creation.

4) The system draws a path between the connection points, and marks junctions with solder dots (figure 2.15).

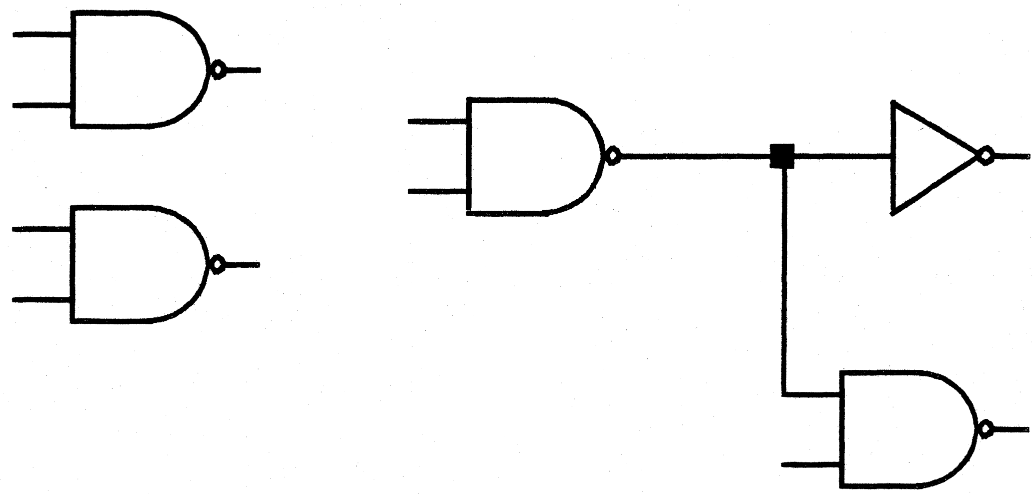


Figure 2.15

- To create several nets at once:

1) Select **Create Net** in the Editing Schematic Menu.
2) Select the endpoints for the first net.
3) Press the key marked **AGAIN**. The system draws the net.
4) Select the endpoints for the second net.
5) Press **AGAIN**. The system draws the second net.
6) Continue this procedure until you have selected all of the endpoints for the last net you want to create.
7) Bring up the Create/Modify Net Menu. Select **Apply Net Edit** or **Abort Net Edit.**

2.3.2 Creating Connection Paths

- In addition to creating complete nets, you can create connection paths between two symbols, two nets, or a symbol and a net. When you create a connection path, you draw the path between the connection's two endpoints.

1) Select **Create Connection Path** in the Editing Schematic Menu.
2) As you move the cursor on the display, the system places a small box at the end of the closest connection point, or along the closest net. Move the box to the first endpoint for your net (either a symbol connection point or a point on a net) and press the left mouse button. The box remains stationary at that point (figure 2.16).

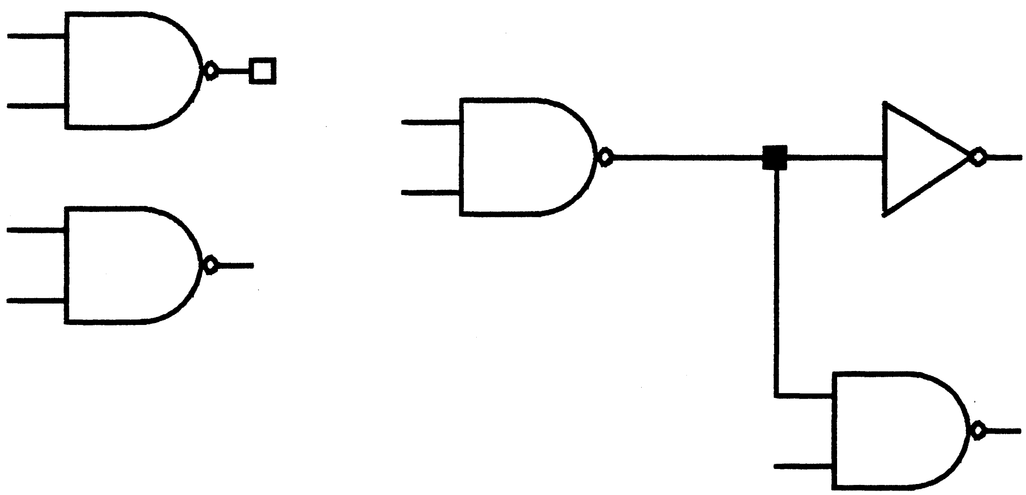


Figure 2.16

3) Move the cursor along the path of the net. The system draws the net path. The small box moves to the closest symbol connection point or net point at which to end the net (figure 2.17). When you want to make a 90° bend in the net press the left mouse button (figure 2.18). If you want to erase the previous bend, press the right mouse button, or activate the Create Connection Path Menu and select **Forget Last Bend**.

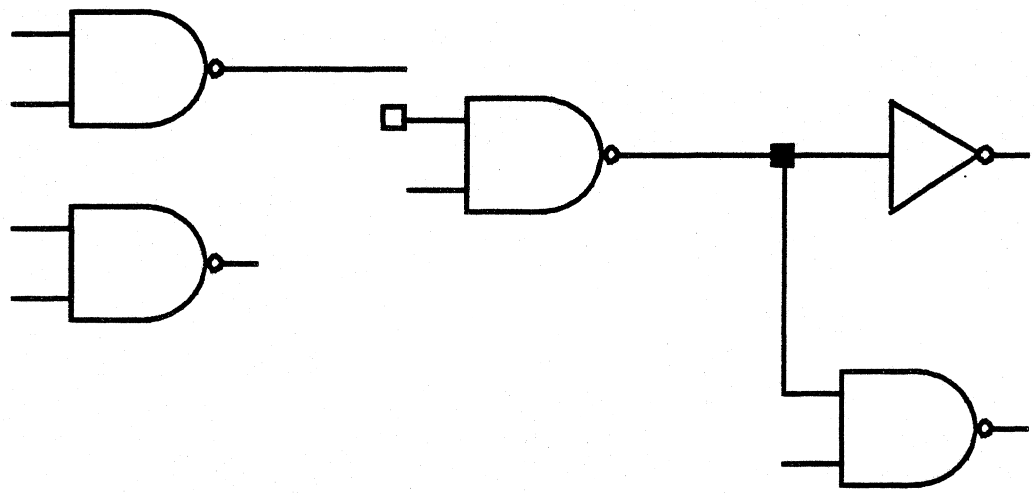


Figure 2.17

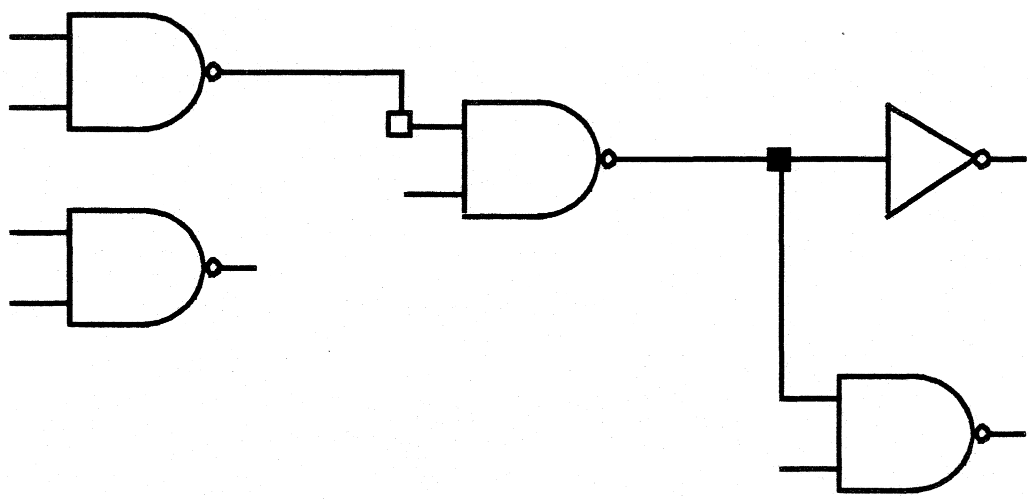


Figure 2.18

4) When you finish drawing the net path, activate the Create Connection Path Menu. The Create Connection Path Menu gives you three options: **Apply Create/Merge**, **Abort Create/Merge**, and **Forget Last Bend**. Select **Apply Create/Merge** to apply net. If you apply the net before you connect two symbols or a symbol and a net, the system completes the net to the point marked by the small box (figure 2.19). Select **Abort Create/Merge** to abort the net.

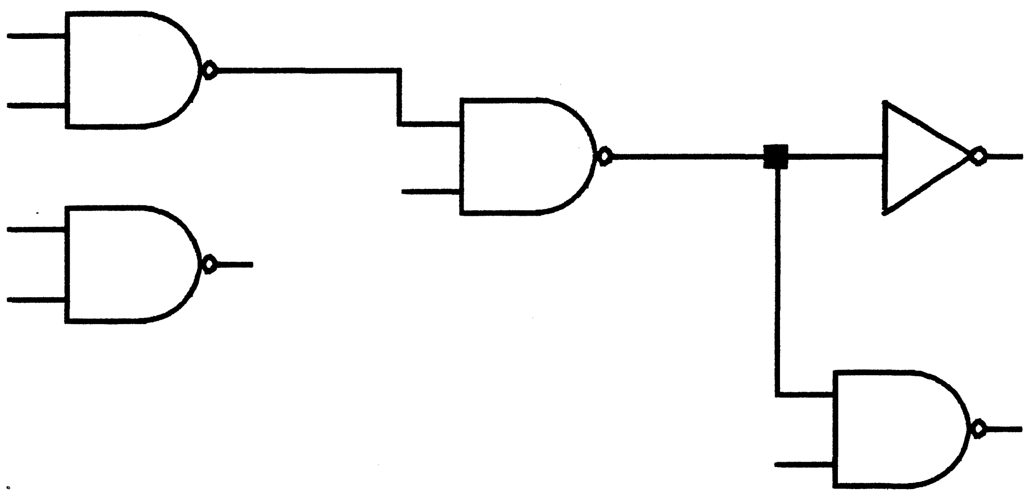


Figure 2.19

2.3.3 Editing the Path of a Net

- You can edit the path of a net segment by drawing a new path, made up of horizontal and vertical lines, that begins and ends at any two points along the old net.

1) Select the connection path you want to edit. You cannot edit past solder dots. See figure 2.20.

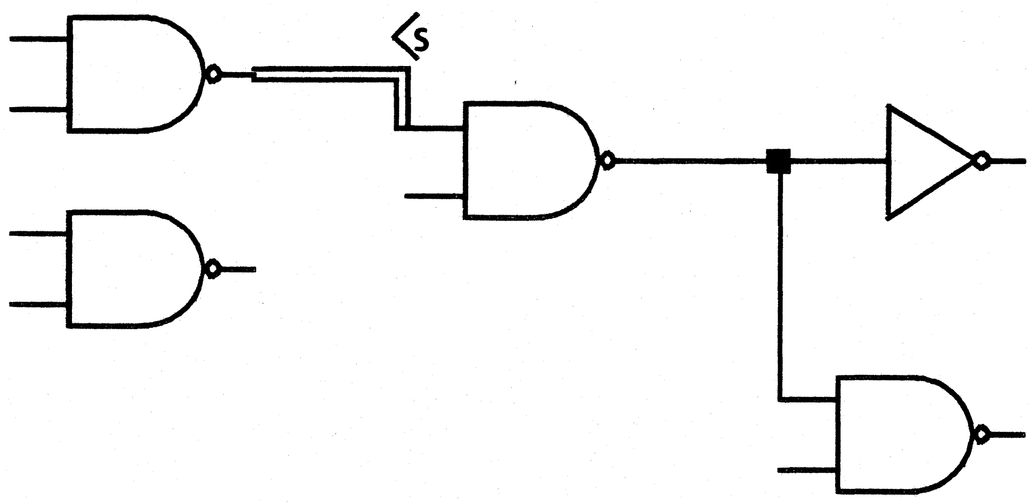


Figure 2.20

2) Select **Edit Path of Net** in the Editing Schematic Menu.

3) The system places an "X" on the net. Moving the cursor moves the "X" along the net. Move the "X" to the position on the net at which you want to begin the edit (figure 2.21).

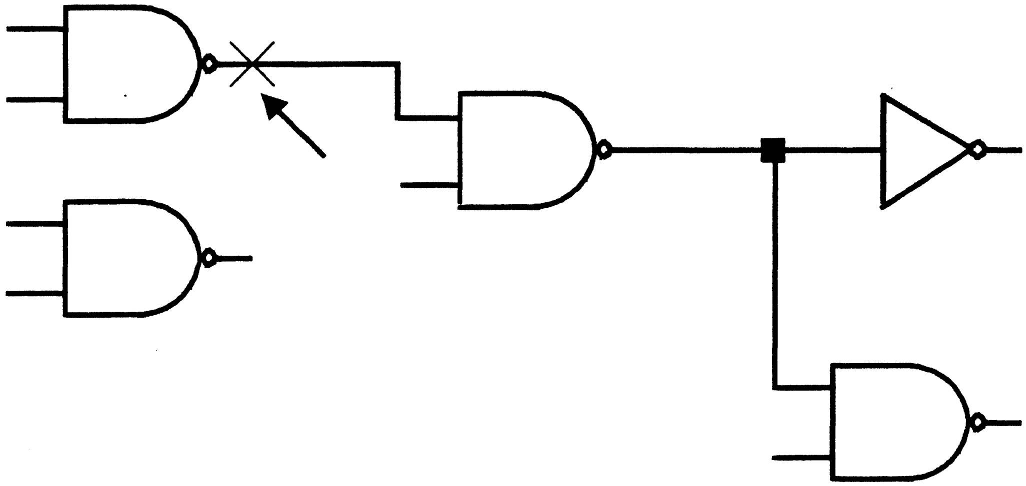


Figure 2.21

4) Press the left mouse button to mark the starting point of your edit. The "X" remains stationary. Moving the cursor moves a line that is anchored at the "X," and perpendicular to the net segment.
5) Move the cursor to stretch the line in either direction. The line represents the new path of the net. Stretch the line to the length you want. Press the left mouse button to mark the line's endpoint (figure 2.22). Press the right mouse button to "unmark" the endpoint, and adjust the length of the line.

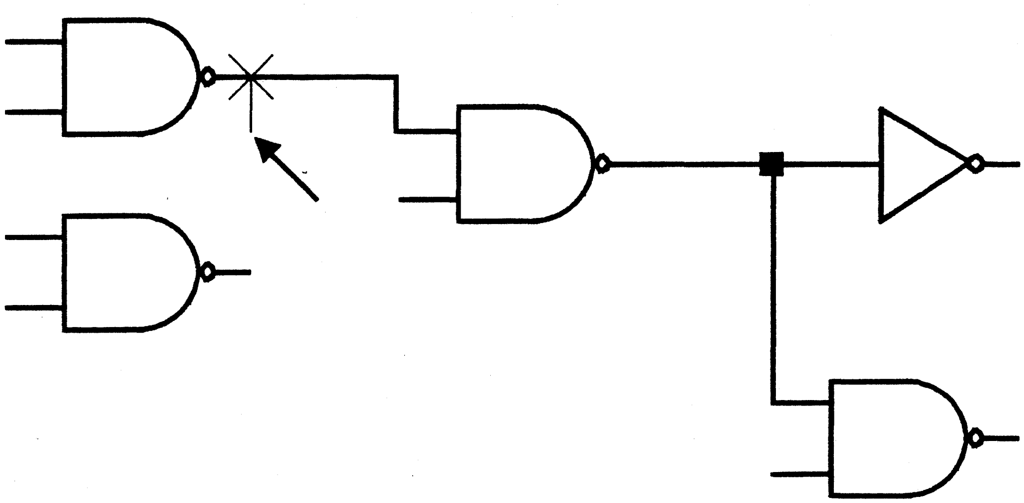


Figure 2.22

6) Now you can draw a second line beginning at the endpoint of the first. This line is perpendicular to the first line. Move the cursor to stretch the line to the length you want (figure 2.23). Notice that the "X" moves along the net. It shows you the most logical place to rejoin your lines to the net. Press the left button to mark the endpoint of the line.

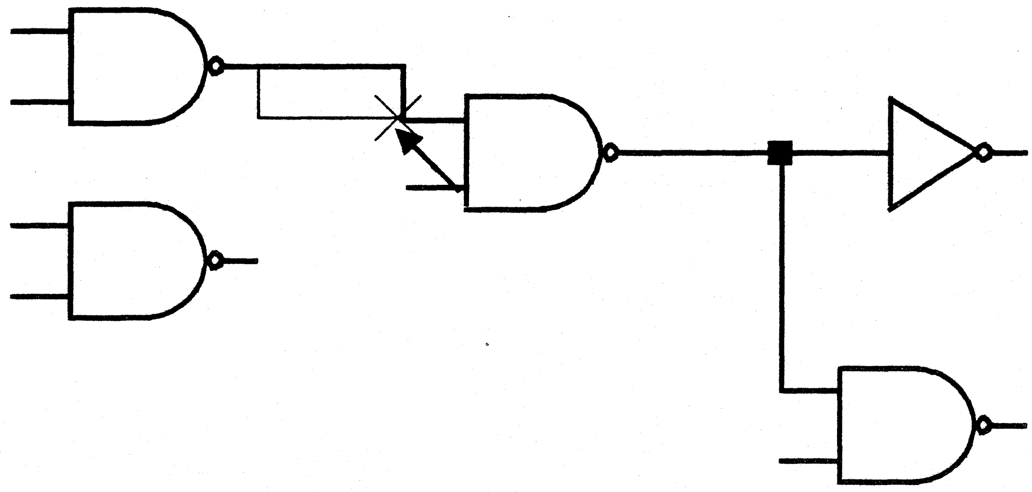


Figure 2.23

7) Continue drawing lines until you have connected your lines to the original net. If you decide to change the path that you are drawing, you can press the right mouse button to "erase" the endpoint of the previous line.

8) Bring up the EditNet Menu. The EditNet Menu gives you three options: **Apply Edit**, **Abort Edit**, and **Forget Last**. Select **Apply Edit** to apply the changes you made in the net's path (figure 2.24). Select **Abort Edit** to abort the changes. Like the right mouse button, **Forget Last** erases the last endpoint that you marked.

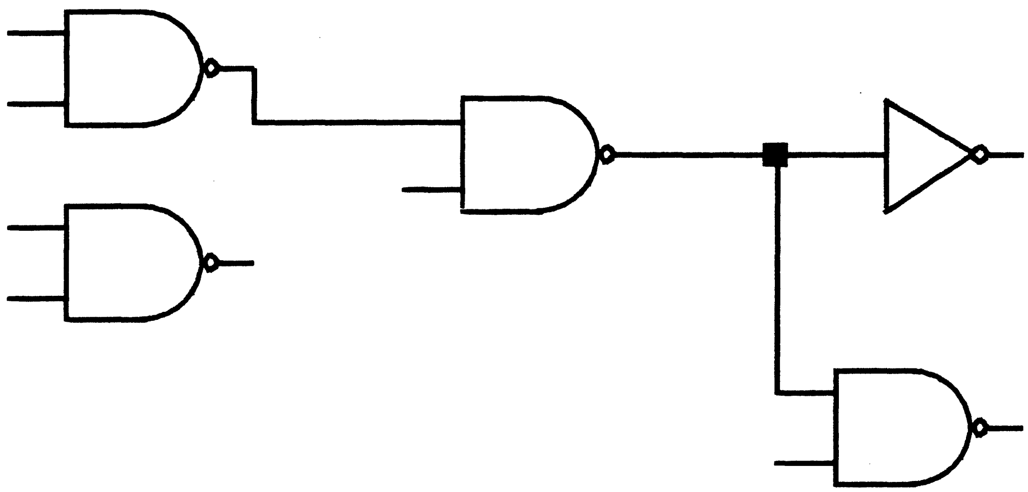


Figure 2.24

If you apply the edit before you connect the new net path to the original path, the system completes the new path for you by drawing the shortest path between the end of your net path and the original net path.

2.3.4 Deleting a Connection Path

- To delete a connection path:

1) Select the net, placing the source marker on the connection path you want to delete.
2) Select **Edit Path of Net** in the Editing Schematic Menu.
3) Press the **DELETE** key. The system removes the connection path.
4) Activate the Delete Net Path Menu. Select **Apply**. Select **Abort** if you decide not to delete the connection path.

2.3.5 Editing Net Paths and Junctions

- You can edit the path of a connection moving the solder dots that connect it to other connection paths.

1) Select the connection path you want to edit. See figure 2.25.

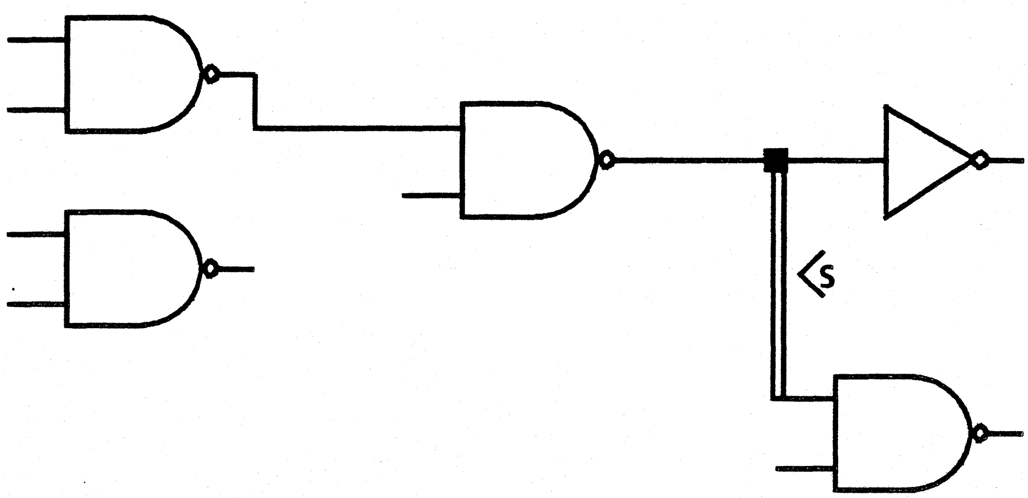


Figure 2.25

2) Select **Edit Path and Junctions** in the Editing Schematic Menu.

3) The system deletes the net segment, and displays an "X." Moving the cursor moves the "X" along the net segments, or symbol connection points the segment connects. Move the "X" to the first new endpoint of the net segment, either a junction or connection point. Press the left mouse button. The "X" remains stationary (figure 2.26).

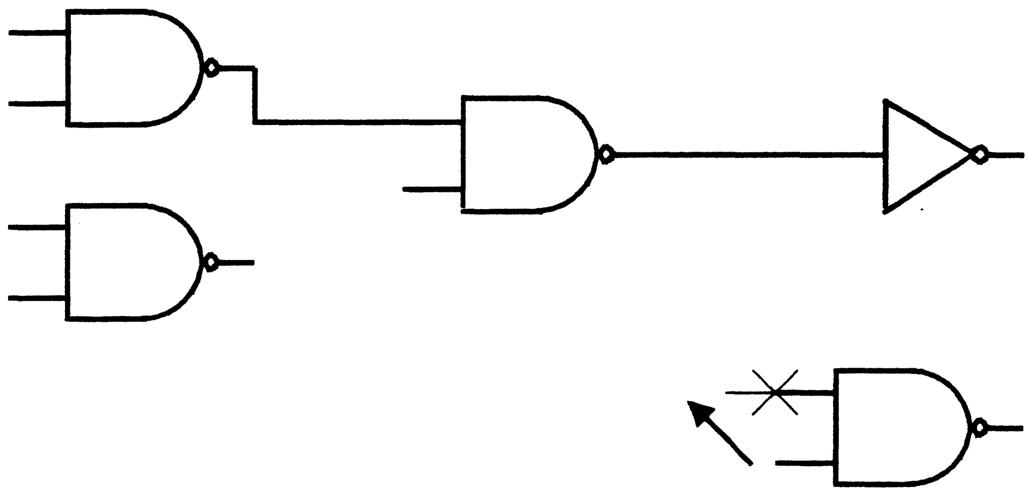


Figure 2.26

3) Move the cursor to draw the path of the net segment. The "X" moves to the closest junction point or symbol connection point (figure 2.27). When you want to make a 90° bend in the net, press the left mouse button (figure 2.28). If you want to erase the previous bend, press the right mouse button.

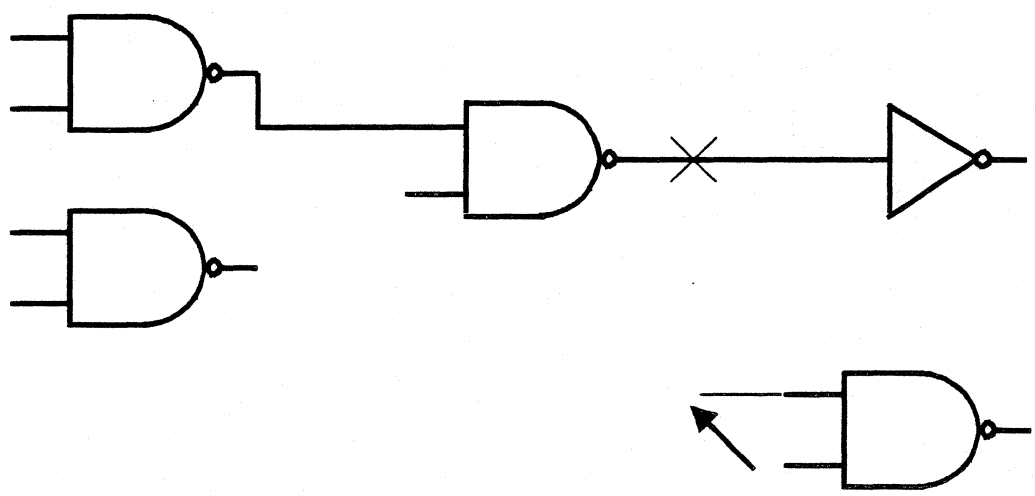


Figure 2.27

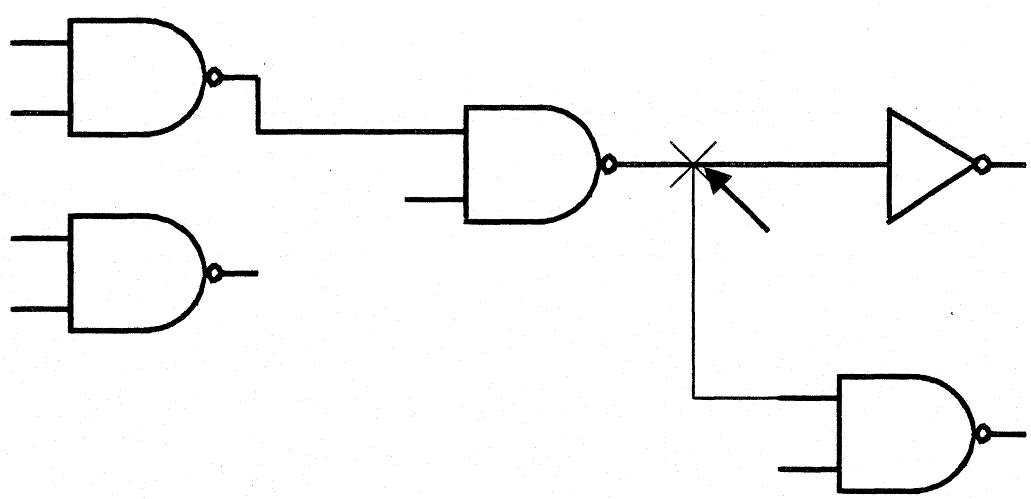


Figure 2.28

4) When you complete the segment between connection points or junctions, activate the Edit Path and Junctions Menu. Select **Apply Edit** to apply net (figure 2.29). If you apply the net before you connect two junctions, symbol connection points, or a symbol and a junction, the system completes the net to the point marked by the "X." Select **Abort Edit** to abort your edit. Like the right mouse button, **Forget Last** erases the last bend in the net path.

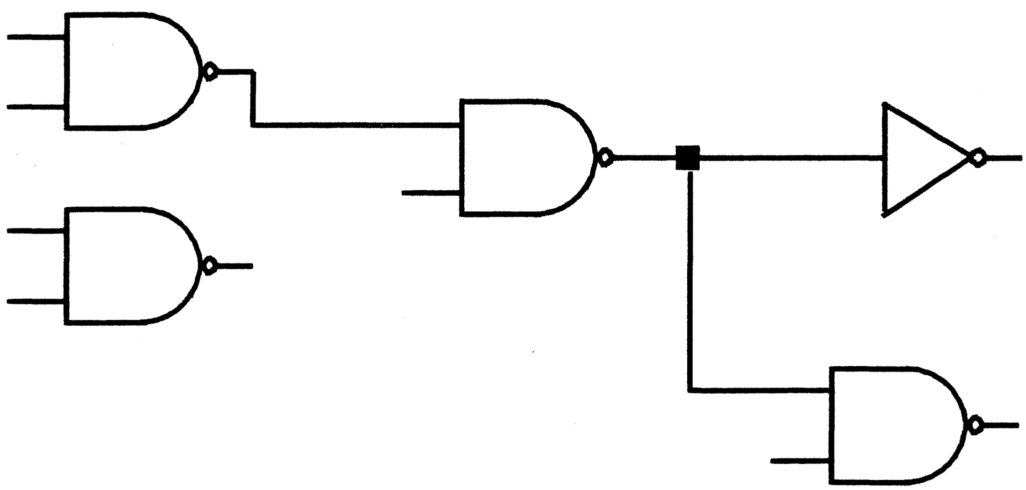


Figure 2.29

2.3.6 Net Properties

A net has two properties, its name, and the set of net rules that it follows.

- To activate a net's property sheet:

1) Select the net.
2) Press the **PROP'S** key.

Figure 2.3 shows a net's property sheet.

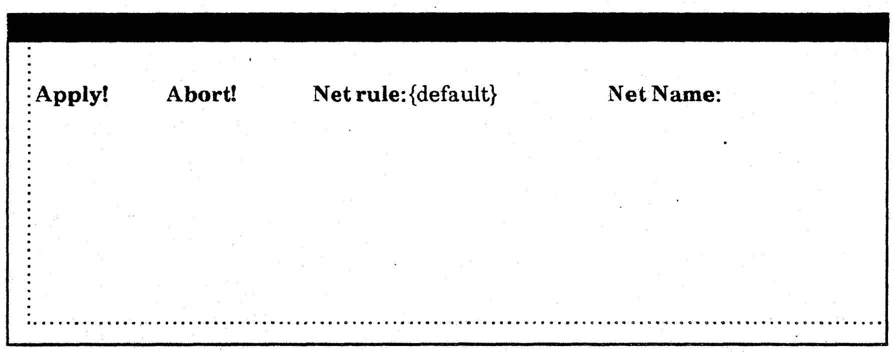


Figure 2.30

Net Rule

The **Net rule** value tells the system which set of electrical rules to follow when routing the net on a pc board with future releases of Expert 1000. The current release does not utilize the **Net rule** property.

Net Name

Unless the net has a connectivity symbol at one of its endpoints, its name must be unique. (Section 2.3.8 describes connectivity symbols.) You can enter the net name in the property sheet, and copy it onto the schematic, or you can name a net with the **Name Net** command described in section 2.3.7.

- To copy a net name from a property sheet to the schematic:

1) Mark the location on the grid at which you want to place the net name with the destination marker.
2) Press the key marked **COPY**. The system copies the name to the location you marked.

- You can display the net name on the schematic in as many places as you want. To copy the name to another location:

  1) Set the destination again.
  2) Press **COPY**.

  When you finish copying names onto the schematic, apply the name. Select **Apply!** in the net's property sheet.

- To change the name of a net:

  1) Change the name in the net's property sheet.
  2) Select **Apply!**. The system automatically changes the name on the schematic.

  While the property sheet is open, you can select and edit the net names individually. When you close the property sheet, the net name becomes part of the net; you cannot select the name without selecting the net.

2.3.7 Naming Nets

The **Name Net** command lets you name a net without displaying the net's property sheet.

- To name a net:

  1) Select a grid point near the net with the source marker.
  2) Type the net name.
  3) Select **Name Net** in the Editing Schematic Menu. The system names the net closest to the source marker with the selected text. The name becomes a part of the schematic. You cannot select the name without also selecting the net. Use the net property sheet to change the net name, and move, copy, or delete the net name text on the schematic.

2.3.8 Connectivity Symbols

There are two types of special symbols called connectivity symbols. The first type automatically names the net that connects to it. Examples of this type are power and ground symbols. The second type of connectivity symbols is off-page connectors. Off-page connectors do not name nets automatically; you must name the nets.

Connectivity symbols let you create a net in several segments, without drawing the connection paths between each segment. You connect each segment to a connectivity symbol, and then give each segment the same net name in order to tell the system that the segments are part of the same net. Connectivity symbols have no electrical meaning. They simply mark connection points. Figure 2.31 shows a connectivity symbol.

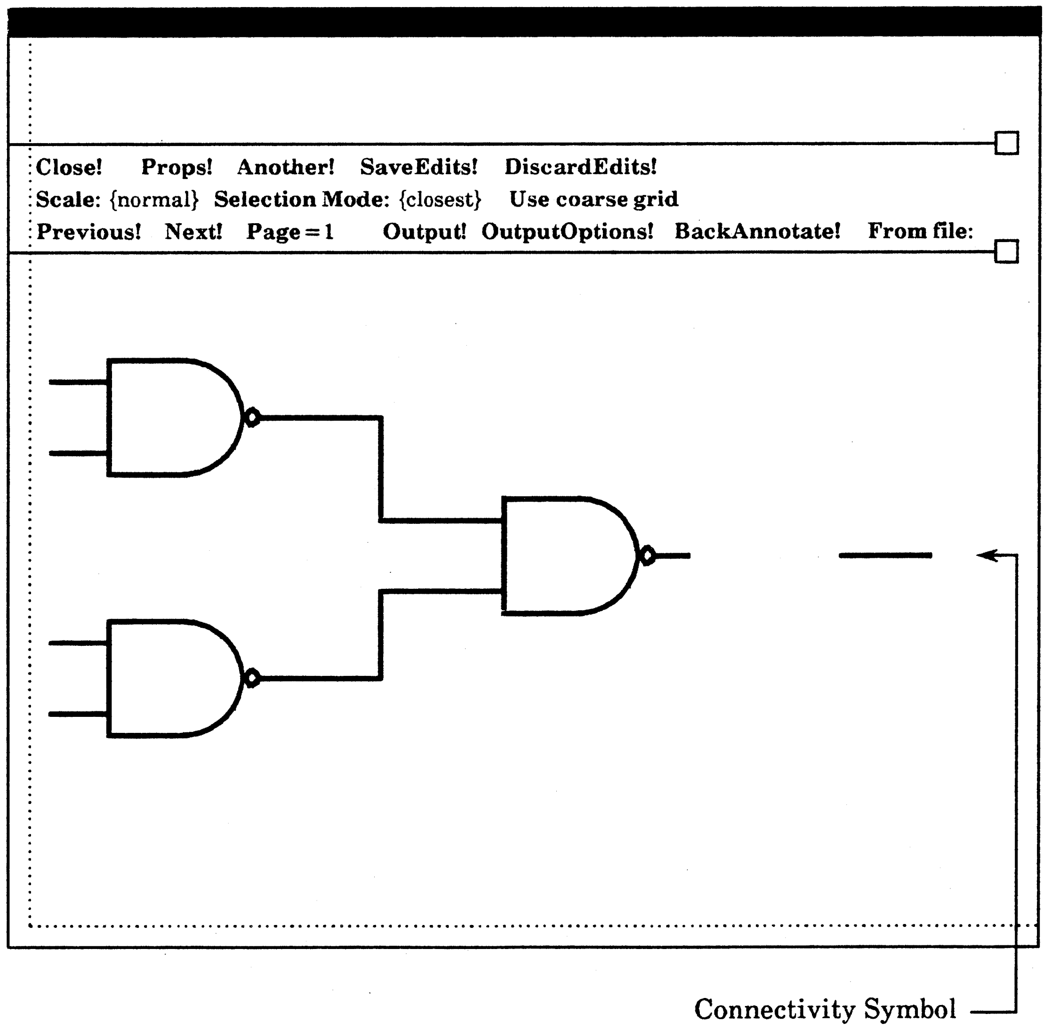


Figure 2.31

An address bus, for example, connects the same address bit on several memory chips. A simple way to draw an address bus is to connect address bit Zero on each chip to a connectivity symbol, and name that net segment "Address0." Repeat this process for each address bit. To show that the net segments are part of a bus, display the net name for each segment (figure 2.32).

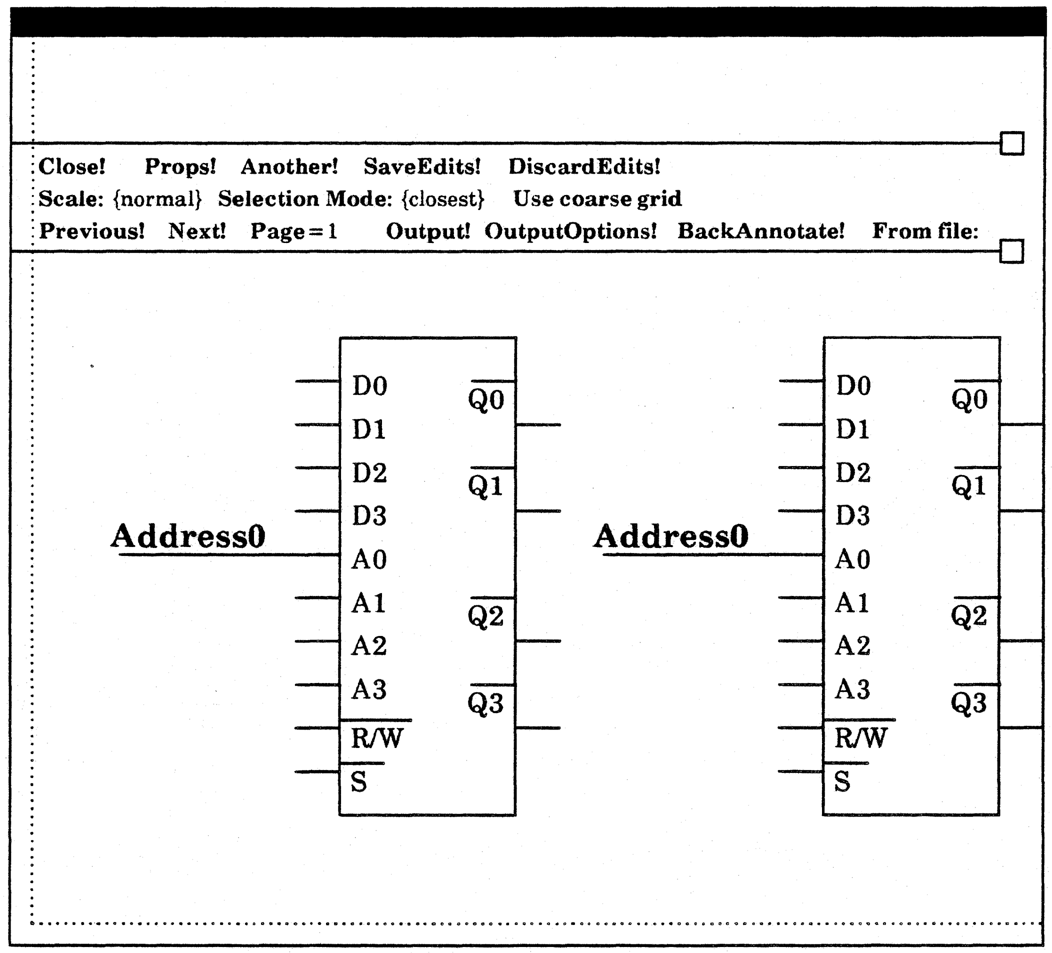


Figure 2.32

Connectivity Symbols as Off-page Connectors

- You use connectivity symbols to connect symbols on different schematic pages:

1) Place a connectivity symbol on the first page. Connectivity symbol nicknames are listed in the Abbreviation Map.
2) Use the standard method to create the net with the symbols on that page. Include the connectivity symbol as if it were any other symbol.
3) Name the net.
4) Place another connectivity symbol on the second page.
5) Create the net on that page. Connect the connectivity symbol as if it were any other symbol.
6) Give the net the same name as the net on the first page.

2.4 Annotation

In addition to displaying properties such as pin names, designators, physical names, pin numbers, device parameters, and net names, you can annotate the schematic with horizontal and vertical lines and text.

2.4.1 Lines

- To create a horizontal or vertical line:

1) Mark the line's two endpoints with the source and destination markers.
2) Select **Create Line** in the Editing Schematic Menu.

Line Properties

A line has style and width properties. Figure 2.33 shows a line property sheet.

Apply! Abort!
Style: {solid, dashed} **Width**= 6

Figure 2.33

Style

A line's *style* can be solid or dashed. To change a line's style, change the style property.

Width

One line width unit is one-four-hundredth of an inch. To make a line wider, replace the width value with a larger number. To make it narrower, replace the width value with a smaller number.

The system "remembers" line properties. It creates a new line with the properties you assigned to the last line it created.

2.4.2 Text

- To place text on a schematic:

1) Mark the point where you want to place the text with the source marker.
2) Type the text.

Text Properties

You change the appearance of text by changing the text's properties. Figure 2.34 shows a text property sheet.

Apply! Abort!
Families: {Typewriter, SansSerif} **Height=** 10
Layout: {portrait, portraitR90, portraitR270, down}
Text Body: TEXT

Figure 2.34

Family

The text *family* is the type style. Select either Typewriter or SansSerif.

Height

The *height* property controls the size of the text. You may enter any number as the text height, but the system recognizes only valid combinations of height and family. If you enter an invalid height, the system displays the text at the closest valid height less than the height you entered. Figure 2.35 shows you valid text families and their text sizes.

The system creates text with the properties you gave the last text it created.

Figure 2.35

Layout

The *layout* property controls the text's orientation on the schematic. Portrait layout is a normal text layout, from left to right across the page. PortraitR90 rotates the text 90 degrees. PortraitR270 rotates the text 270 degrees, and down writes the text vertically, from top to bottom. Expert 1000 Release 1.0 supports only portrait layout.

Text Body

The system displays the body of the text in the property sheet. You can change the text in the schematic by changing the text body in the property sheet.

2.5 Clusters

You can group objects into *clusters*. Clusters make it possible to treat related objects as if they were a single object. For example, you could group the circuitry for a device driver into a cluster, then easily move or copy the driver to another part of the schematic.

The system automatically makes objects associated with the symbols in the cluster part of the cluster. For example, a net is part of a cluster if all symbols at its endpoints are. A cluster can span multiple pages (although the system will not let you move or copy a multiple-page cluster) and is a permanent part of the schematic.

2.5.1 Creating a Cluster

- To create a cluster:

1) Use either the **Add to Selection** or **Selection by Box** command in the Editing Schematic Menu to select the objects that you want to group into the cluster. (Section 2.1.8 explains both **Add to Selection** and **Selection by Box**.)
2) Select **Make into Cluster** in the Editing Schematic Menu.

2.5.2 Selecting a Cluster

You can select a cluster as if it was a single object, or select the objects in the cluster individually.

- To select just one object in the cluster:

  1) Point the cursor at the object, and press the left mouse button once (figure 2.36).

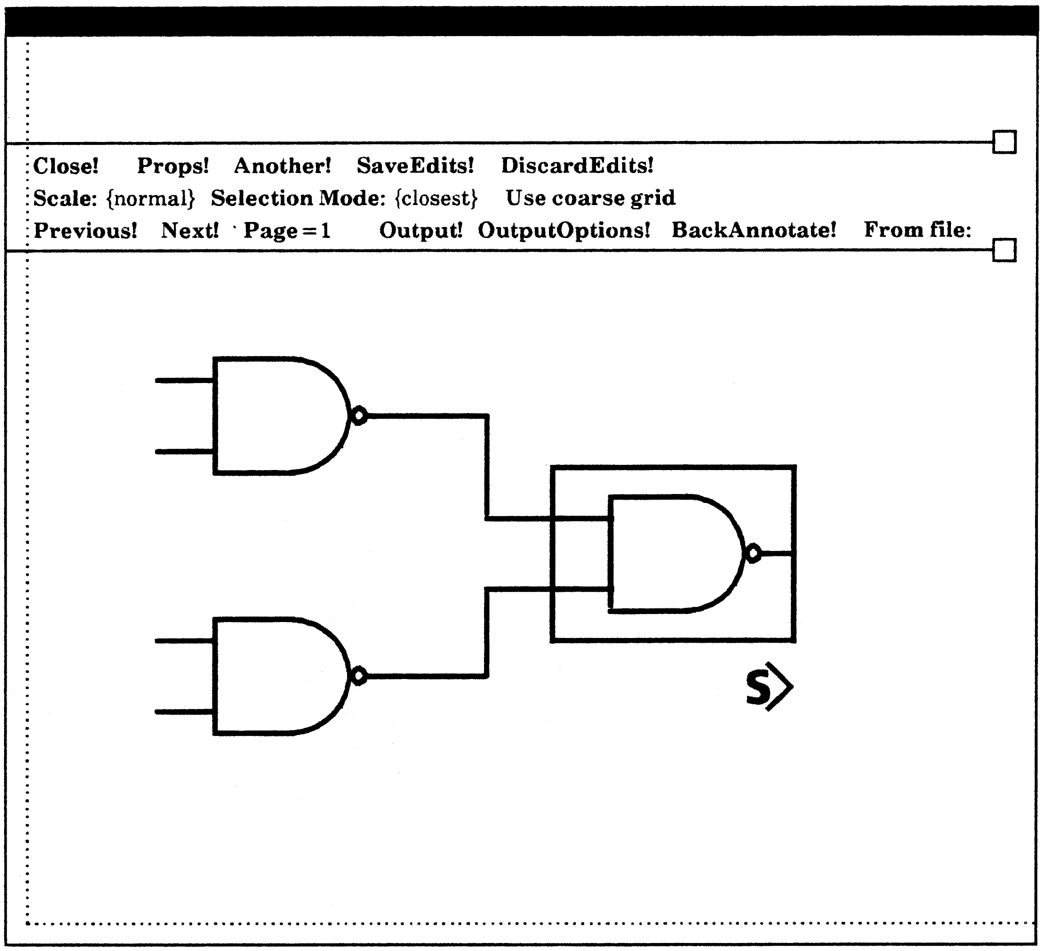


Figure 2.36

- To select a cluster:

  2) Point the cursor at any object in the cluster, and press the left mouse button twice without moving the cursor (figure 2.37).

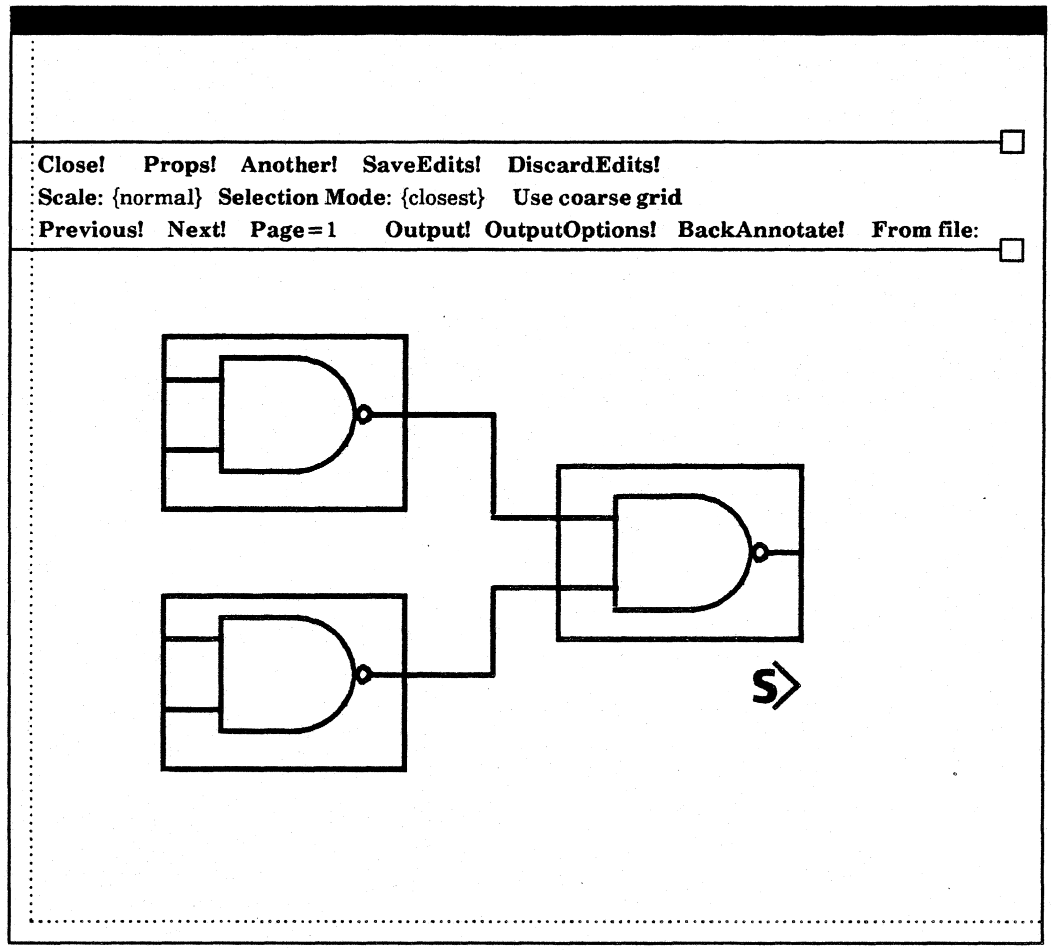


Figure 2.37

2.5.3 Cluster Properties

You can give each cluster a name. For example, the cluster name might describe the subsystem that the cluster represents.

- To name a cluster:

1) Select the cluster.
2) Press the **PROP'S** key.
3) Point the cursor at the words "**Cluster Name**," and press the left mouse button.
4) Type the cluster name.
5) Select **Apply!**

2.6 Editing the Schematic

You edit a schematic by *deleting, moving,* and *copying* objects inside the schematic. If you accidentally delete an object, you can *paste* that object back in.

In general, to delete, move, or copy an object, you select the object and press the corresponding function key. You edit a group of objects by extending the selection. You can edit a cluster as if it was a single logic symbol, and you can edit the individual objects inside the cluster as if they were not part of the cluster.

You control the type of object that you edit by setting the selection mode. If, for some reason, you decided to remove several nets in a schematic, you could set the selection mode to **net** to assure that you delete only nets.

2.6.1 Delete

- To delete an object:

  1) Select the object.
  2) Press the **DELETE** key.

When you delete a logic symbol, the system automatically deletes any connections to the symbol. The connections have no electrical meaning without the logic symbol.

2.6.2 Paste

The paste function undoes the last deletion that you made. If you accidentally delete a line, for example, you press the PASTE key to recover that line.

- To recover the last object you deleted:

  1) Press the **PASTE** key (marked **SAME**).

Use caution when pasting nets. The system places a net at its location when you deleted it. For example, if you delete a net, move the symbols at its endpoints, and then paste it back, the net no longer connects the symbols.

2.6.3 Move

- To move an object:

1) Select the object.
2) Mark the location to which you want to move the object with the destination marker (figure 2.38). The source and destination markers act as anchor points showing the distance to move the selected object. When the system moves the object, it moves the source marker to the destination location, and moves the item so that it retains its relationship with the source marker.

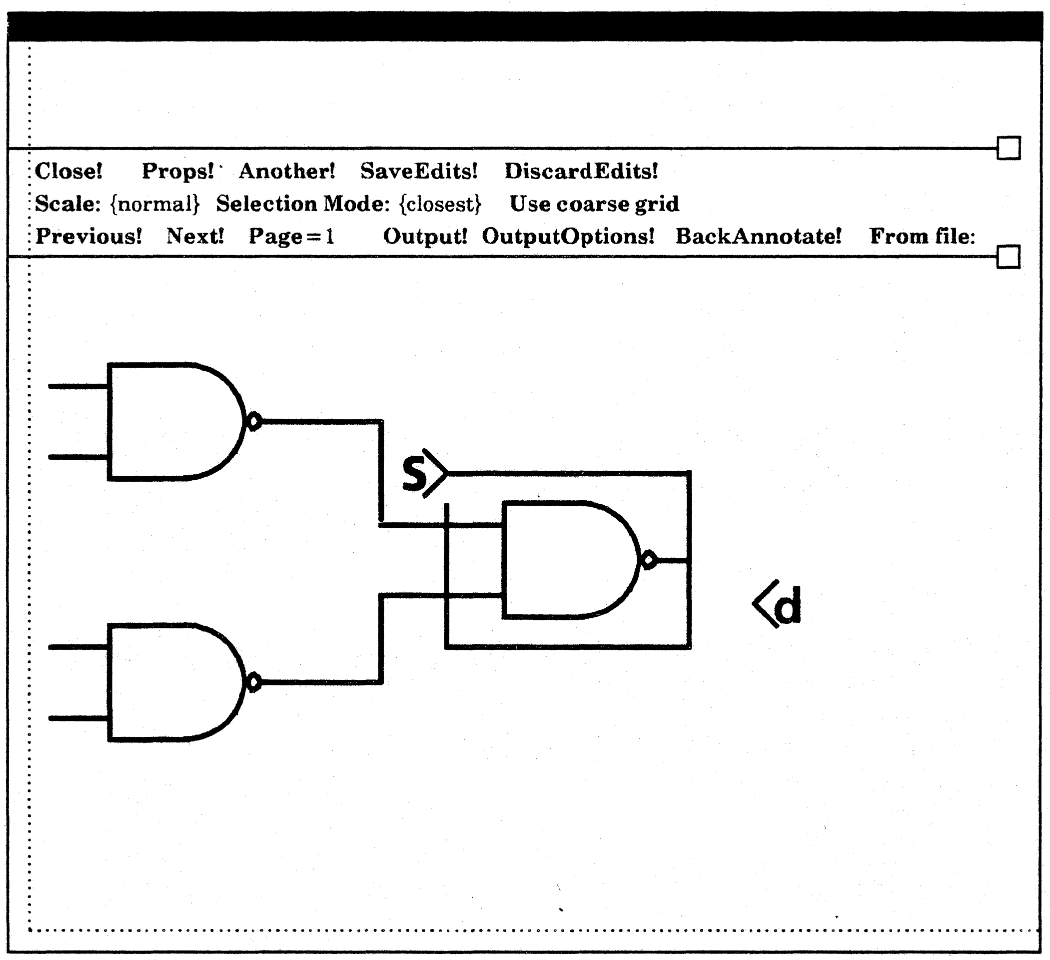


Figure 2.38

3) Press the **MOVE** key. The system moves the object to the destination, moves the source marker to the destination point, and the destination marker to the source (figure 2.39).

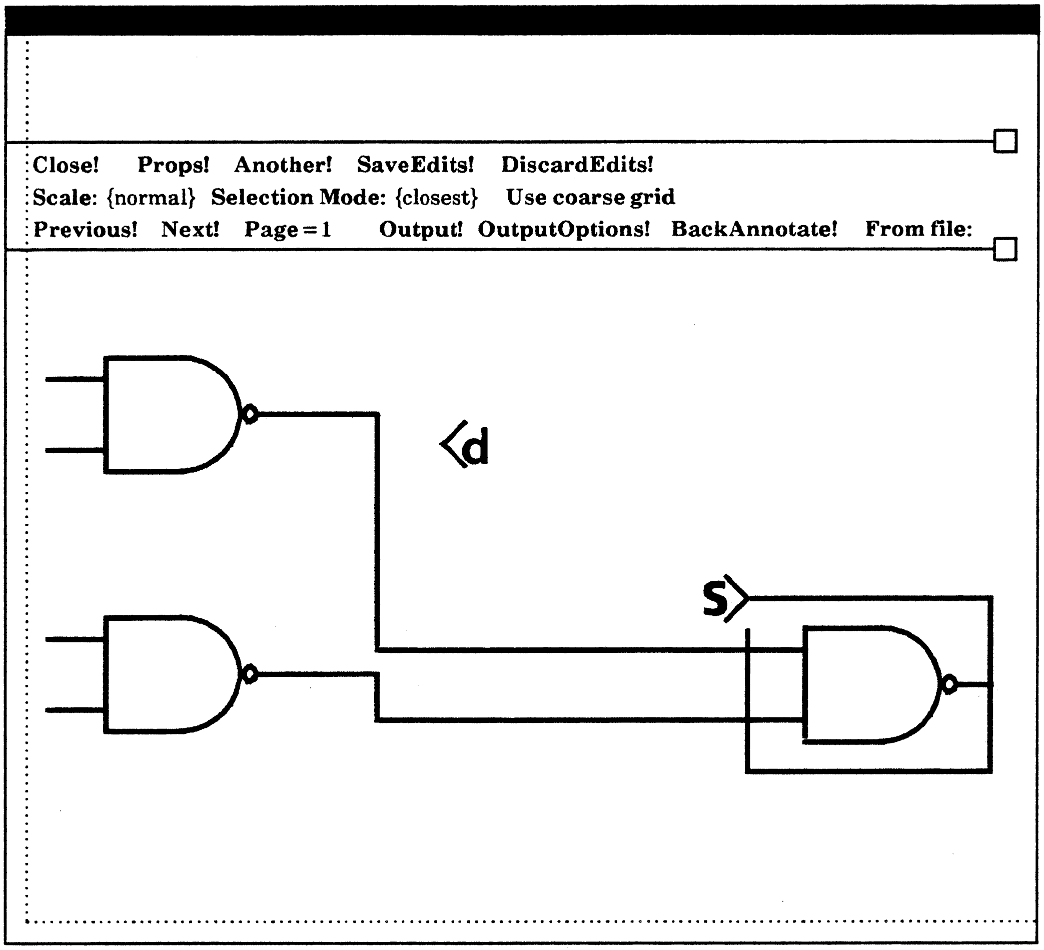


Figure 2.39

When you move a logic symbol, the system also moves the nets associated with it. You can see the system "stretch" the nets. When you move a net, the system moves the symbols at the net's endpoints as well. You cannot move clusters that span multiple pages.

2.6.4 Copy

1) Select the object you want to copy.
2) Mark the location to which you want to copy the object with the destination marker (figure 2.40). The system moves the source marker to the destination location, and copies the item so that it retains its relationship with the source marker.

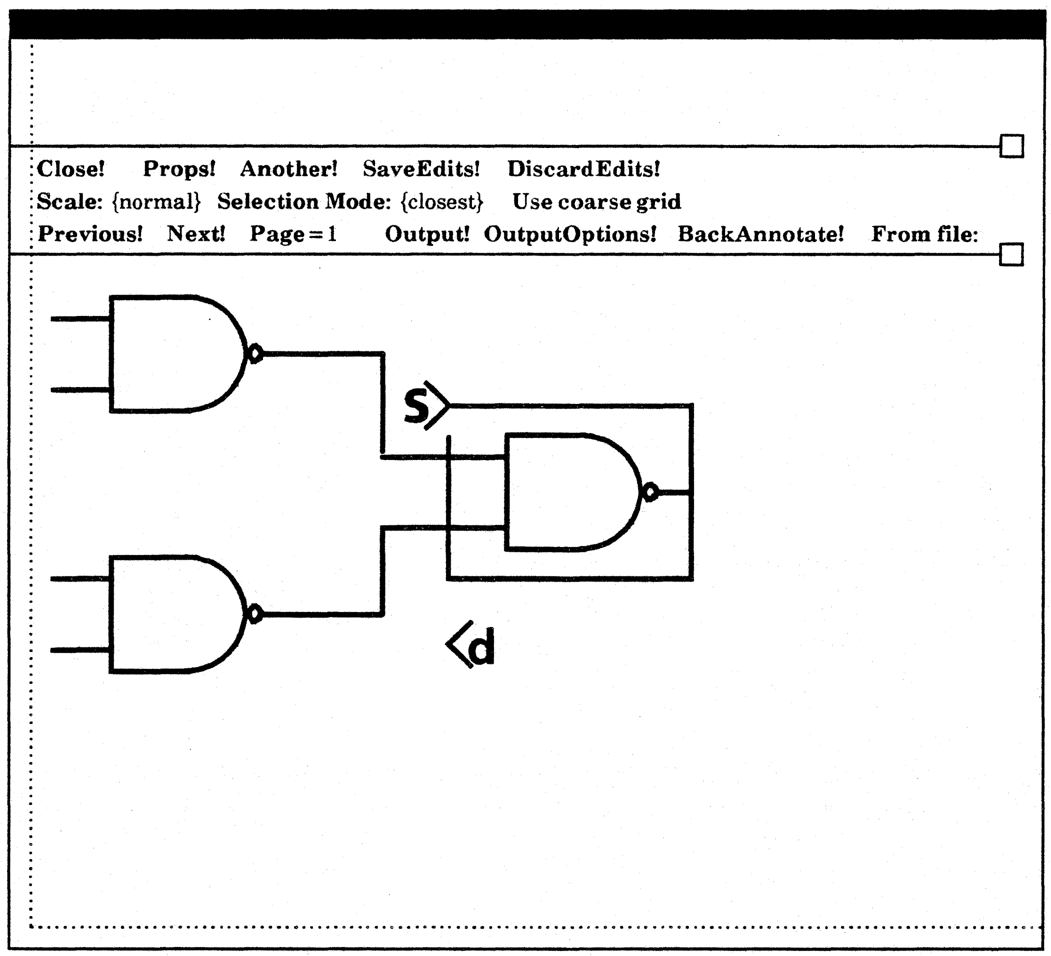


Figure 2.40

3) Press the **COPY** key. The system copies the object to the destination point, moves the source marker to the destination point, and moves the destination marker so that it keeps its same position relative to the source marker (figure 2.41).

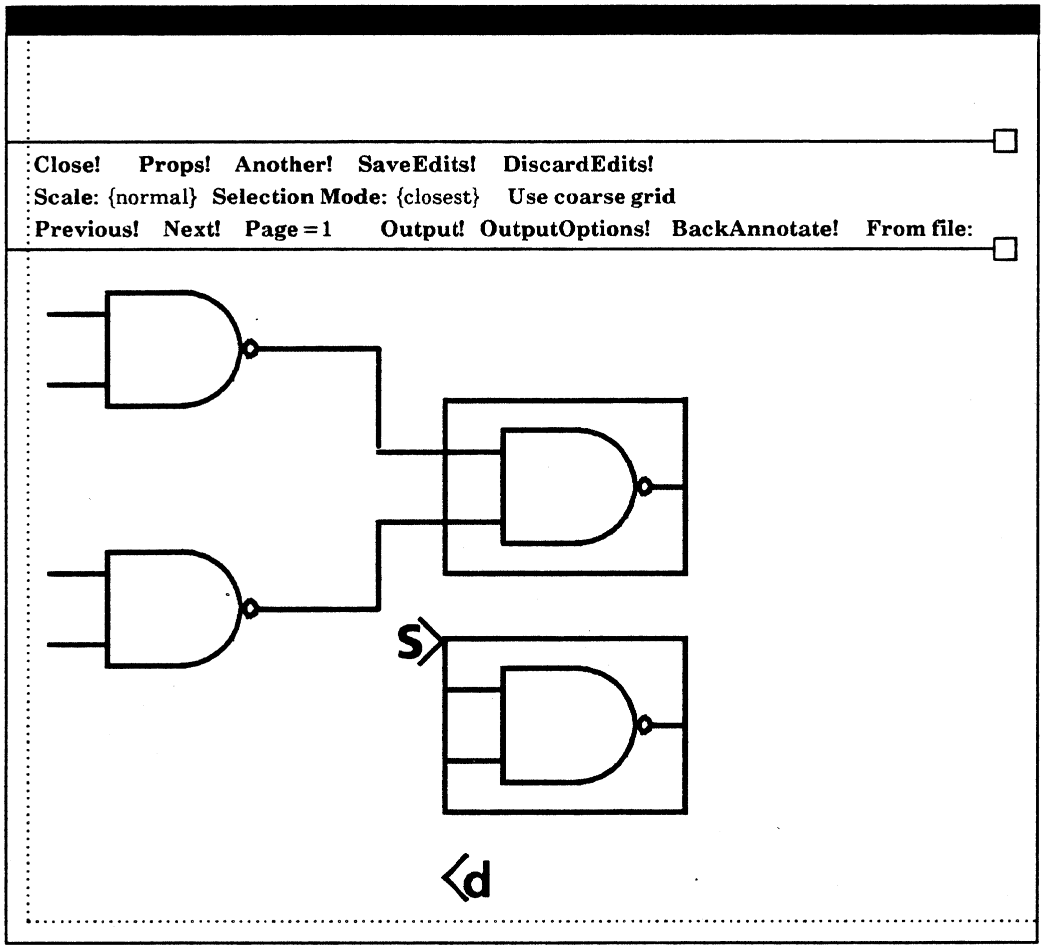


Figure 2.41

When you copy a net, the system also copies the logic symbols at its endpoints. When you copy a cluster, the system copies all of the attributes of the symbols within the cluster, but the copied symbols are not a cluster. You cannot copy clusters that span multiple pages.

Copying Properties

The system copies all properties assigned to individual objects except for symbol designators and micro-notes.

The system copies a net name only if there is a connectivity symbol on the net.

Copying from Multiple Schematics

- To copy one or several objects from one schematic to another:

1) Open both schematics.
2) In the first window, select the object or objects you want to copy.
3) In the second window, mark the location to which you want to copy the object with the destination marker. Leave the cursor inside the destination window.
4) Press **COPY**.

2.7 Hierarchy

Using Expert 1000's hierarchical data base, you can design a schematic as a series of functional blocks. The top level of a hierachical design is a block diagram representing the entire design. You draw this block diagram by placing symbols representing the functional blocks, and connecting them with nets. Next you *define* the symbols in the block diagram, by drawing the circuitry for each symbol. Symbol definitions may contain hierarchical symbols that you define at a lower level in the design. When you have defined all hierarchical symbols, you instruct the system to *expand* the hierarchy.

2.7.1 Drawing a Functional Diagram

You draw the top level of a hierarchical schematic as you would draw any schematic. You use symbols to show functional blocks, and nets to show the connections between blocks. In a top-level diagram nets represent *busses*, groups of connections. To create the functional diagram, you can use any of the symbols in your symbol library. You will probably use symbols that represent functional blocks such as the symbols in the figure 2.42.

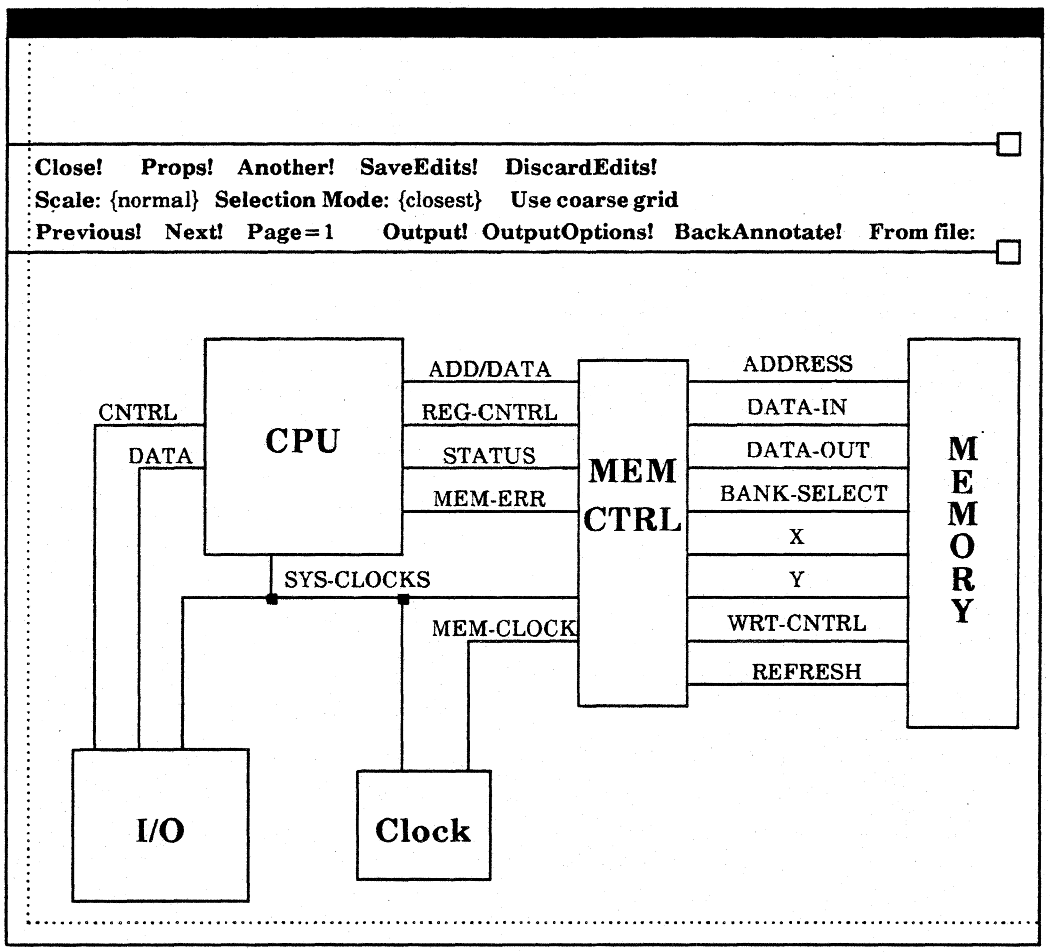


Figure 2.42

2.7.2 Defining a Symbol

- You *define* the circuitry for a symbol on empty schematic pages. You must tell the system which pages define the symbol. To label a page as defining a particular symbol:

  1) Display the symbol's property sheet to find the identification number of the symbol.
  2) Use the **Next!** or **Previous!** command to display an empty schematic page.
  3) Select **Props!** to display the property sheet for that schematic page.
  4) Enter the symbol's identification number as the value of the **Id page defines** property. Select **Apply!**. You enter the symbol's identification number in the property sheet for each page you use to define the symbol's circuitry.

Once you have set a definition page for a symbol, you can display that page from the top-level functional diagram.

- To display a symbol's definition page:

  1) Select the symbol.
  2) Press the **EXPAND** key.

- Use standard schematic design procedures to draw the circuitry for the symbol on the definition page. Use the following special procedures:

  1) Use connectivity symbols to represent the pins of the symbol you are defining. If you are defining the signals in the bus that connects to the pin, use a connectivity symbol for each signal in the bus. Enter the pin name, a dash, and then the signal's number within the bus as the connectivity symbol's symbol designator. For example, figures 2.43 and 2.44 show the **CPU** symbol shown in figure 2.44, and the property sheet for a connectivity symbol representing the pin named "C"; it represents signal one in the **SYS-CLOCKS** bus that connects that pin to the **I/O** and **CLOCK** symbols.

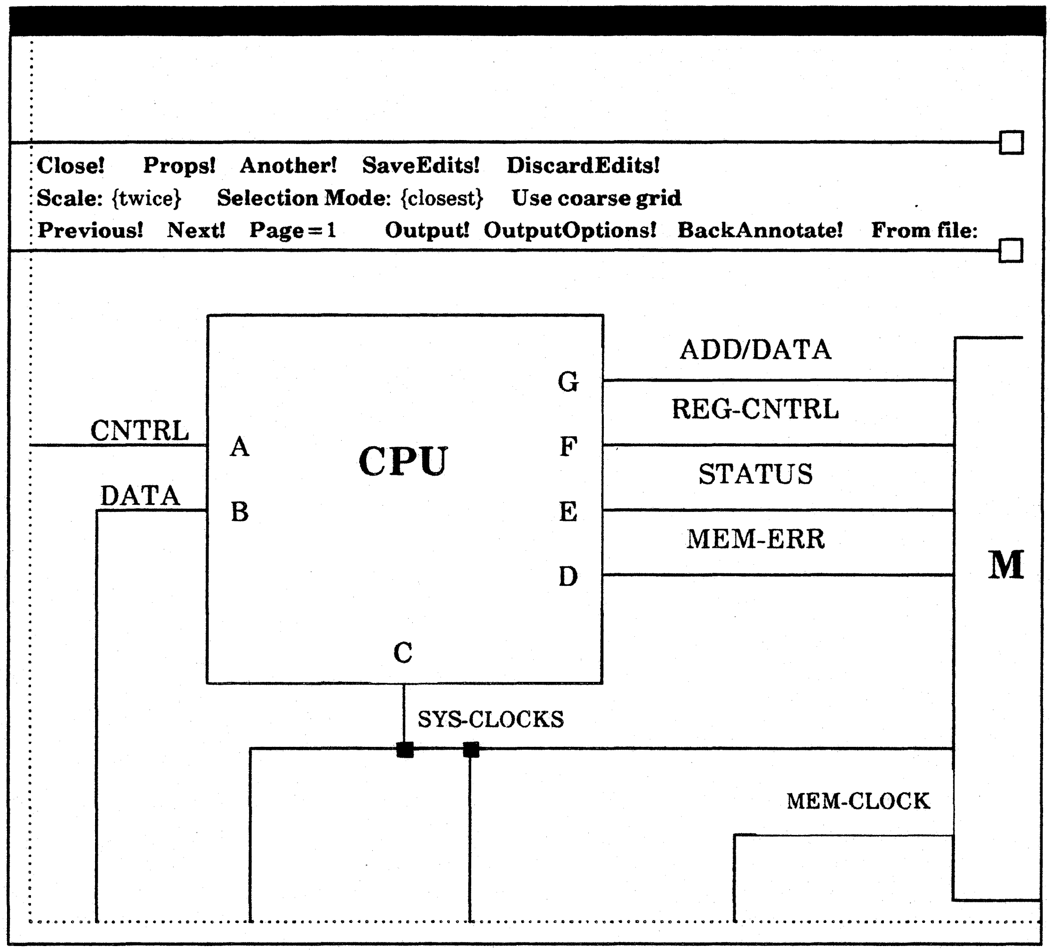

Close! Props! Another! SaveEdits! DiscardEdits!
Scale: {twice} Selection Mode: {closest} Use coarse grid
Previous! Next! Page=1 Output! OutputOptions! BackAnnotate! From file:
CPU
A
B
C
D
E
F
G
CNTRL
DATA
ADD/DATA
REG-CNTRL
STATUS
MEM-ERR
SYS-CLOCKS
MEM-CLOCK
M

Figure 2.43

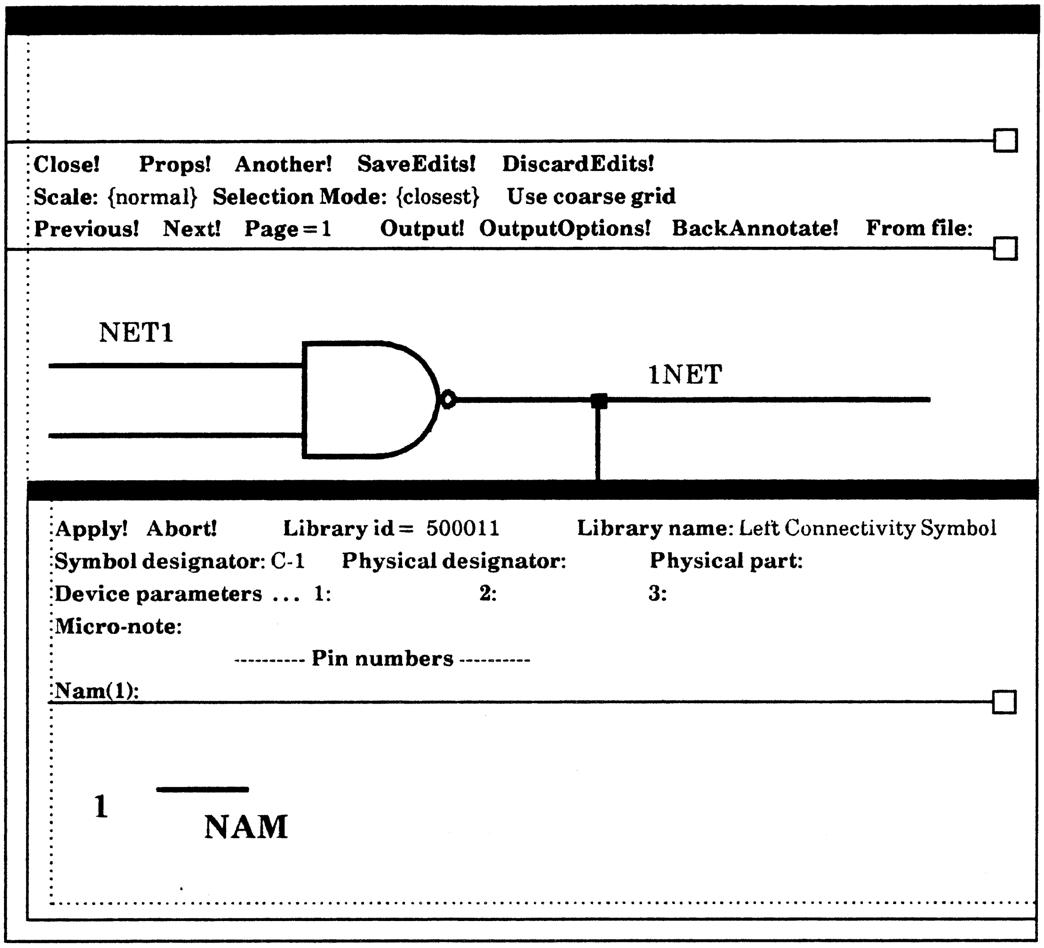


Figure 2.44

If you intend to pass the bus to a lower level of hierarchy, use only one connectivity symbol to represent the pin in the symbol you are defining. Enter the just the pin name as the connectivity symbol's symbol designator.

2) Nets that connect to connectivity symbols representing pins must be complete on one schematic page. If you want the expanded signal name to appear on the schematic, you must name the net. The name you use is not important; the system overwrites it with the expanded name.

2.7.3 Expanding the Hierarchy

When you've defined all hierarchical symbols in the schematic, you instruct the system to *expand* the hierarchy. The system copies the top-level functional diagram at the end of your design, and then expands the hierarchy, copying symbol definition pages, and removing defined symbols from the copied top-level diagram. It automatically names nets representing busses with the proper bus names from the functional diagram and signal numbers from the definition page.

- To expand the hierarchy:

1) Complete the necessary symbol definitions.
2) Select **SaveEdits!** in the schematic command subwindow.
3) Select **Expand Hierarchy!** in the command subwindow.

Symbols you did not define remain in the copied top-level diagram. If you want to define such a symbol, select the **DiscardEdits!** command in the command subwindow. The schematic returns to its state before you expanded the hierarchy. Select **EditSchematic!** and create the symbol definition.

2.8 Output

You can output a schematic in one of two ways. You can plot the schematic on a Versatec plotter, or you can write the schematic to magnetic tape in COMp80 format and use the tape to plot the schematic on microfiche with computer-output microfilm (COM) equipment.

2.8.1 The Output Options Window

The Output Options window is a property sheet associated with the schematic command subwindow. Properties in the Output Options window act as parameters for the **Output!** command in the command subwindow. Figure 2.45 shows an Output Options window.

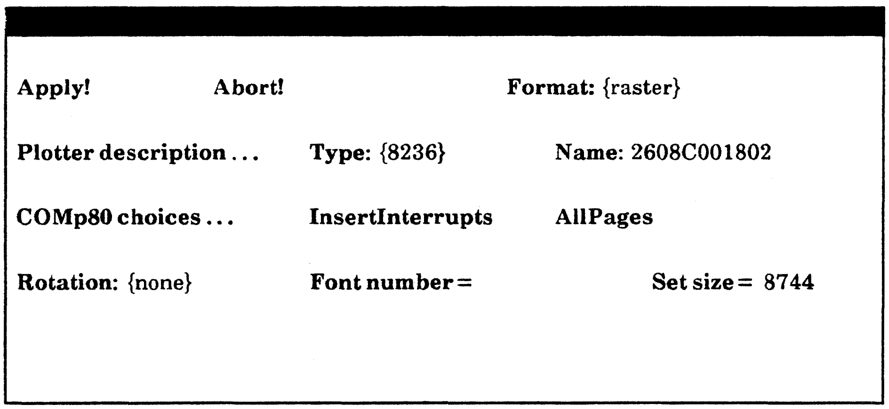


Figure 2.45

- To display the Output Options window:

  1) Select the **OutputOptions!** command in the schematic's command subwindow.

The value of the **Format** property controls the type of output. To plot a schematic, set the **Format** value to **raster**. To create a COMp80 file, set the value to **COMp80**. **Plotter description** properties tell the system on which plotter to plot the schematic. **COMp80 choices** give the system information about the particular format necessary for your COM equipment. Both **Plotter description** properties and **COMp80 choices** have default values that match the equipment in your system.

2.8.2 Plotting the Schematic

- To plot a schematic page on an electrostatic plotter:

1) Display the page you want to plot in the schematic subwindow.
2) Select the scale at which you want to plot the schematic. The scale must be less than or equal to normal.
3) Set the **Display Options** in the schematic window property sheet, so that the system displays the symbol properties you want to plot.
4) Select **OutputOptions!** in the command subwindow. The **Format** value must be **raster**. Enter the correct **Plotter description** and **Name**. The name of the plotter is the hexadecimal address of the output server to which it is connected.
5) Select **Apply!**
6) Select **Output!** in the schematic's command subwindow.

2.8.3 Translating the Schematic to COMp80 Format

You may want to store schematics on microfiche instead of keeping paper copies of each design. Expert 1000 provides interfaces to computer-output microfilm (COM) equipment using COMp80 format. You can use either an FR80, COMp80, or COMp80/2 Universal Page Setter/COM Graphics Recorder to make microfiche plots of Expert 1000 schematics. Expert automatically scales drawings to industry standards for microfiche.

1) If you want to output just one page of the schematic, display that page in the schematic window. Set the **Display Options** in the schematic window property sheet, so that the system displays the symbol properties you want.
2) Select **OutputOptions!** Select the COMp80 options you want to use, entering the appropriate parameters for your COM system.

   InsertInterrupts is a boolean property. If you turn it on, Expert inserts a level 9 interrupt between schematic pages in the COMp80 file. The operator of the COMp80 system can instruct the system to stop and send a message when it encounters a level 9 interrupt, or set the system to ignore level 9 interrupts.

   AllPages is also a boolean property. When you turn it on, it tells the system to write all pages of the schematic to the COMp80 file. If you turn it off, the system writes only the page you are displaying to the COMp80 file.

   Because COM systems have cameras mounted differently, you may need to specify an angle of rotation for your system. **Rotation** options are **none**, **rotate90**, **rotate180**, and **rotate270**. **None** instructs the system not to rotate the image. **Rotate90** rotates the image 90 degrees counter-clockwise, **Rotate180** rotates the image 180 degrees, and **Rotate270** rotates the image 270 degrees. Ask your System Administrator for the proper rotation for your system. He or she can get that information from a Versatec Application Analyst.

Some COM systems have a set of numbered fonts. You can assign one of these fonts to the text in the schematic. Use the **Font** parameter to assign a font number. If you assign an invalid number, the COM system uses its default font.

Each COM system has a certain resolution, measured in *scope points*. Enter the number of scope points per inch as the value of the **Set size** parameter.

3) Select **Apply!**
4) Select the **Output!** command in the schematic subwindow. The system creates a local file in COMp80 format. If you selected **All pages**, it names the file with the documentation set name, a period, and the word "COMp80," for example: **Demo.COMp80**. If **All pages** was false, the system adds a dash and the page number after the documentation set name, for example: **Demo-1.COMp80**.

- Use the Tape Tool to write the COMp80 file to a magnetic tape.

1) Find the **Tape Tool** window. If it is tiny, make it active.
2) Set the **Type** parameter to COMp80.
3) Enter the name of the COMp80 file as the **Source**. If you want to write several files to the same tape, type each file name. Separate file names by a space.
4) Select **Store!**

Use the magnetic tape as input to your COM system.

2.9 Saving and Closing the Schematic

Closing a schematic is similar to placing a hard copy of the schematic into a file folder. When you *close* the schematic, you store it in its documentation set.

Before you can close a schematic, you must either save the changes that you made since you last saved it, or discard those changes. Select either **SaveEdits!** or **DiscardEdits!** in the command subwindow. Notice that the commands in the command subwindow change. The schematic returns to its uneditable state. You must enable editing again before you can make any changes to the schematic.

- Once you have either saved or discarded your edits, you can *close* the schematic. There are two ways to close a schematic.

1) Select **Close!** in the schematic's command subwindow.

Or

1) Select any character in the word "**Schematic**" in the documentation set window.
2) Select **Close** in the Document Operations Menu.

2.10 Recreating Edits

You may decide after discarding your edits, and closing the schematic, that you wanted to save your edits. You might discard edits accidentally, or loose edits because of a workstation failure. The **Recreate Edits** command in the Document Operations Menu lets you replay the changes you made to the schematic before you last saved or discarded those edits. You can recreate edits one at a time, or recreate all remaining edits at once. At any point you may choose to save or discard the edits.

- To recreate the last schematic edits you discarded or lost:

1) Select any character in the word "**Schematic**" in the documentation set window.
2) Select **Recreate Edits** in the Document Operations Menu, or press the key marked **AGAIN**. The system displays the schematic window. The schematic appears as it did before you began your last editing session (figure 2.46).

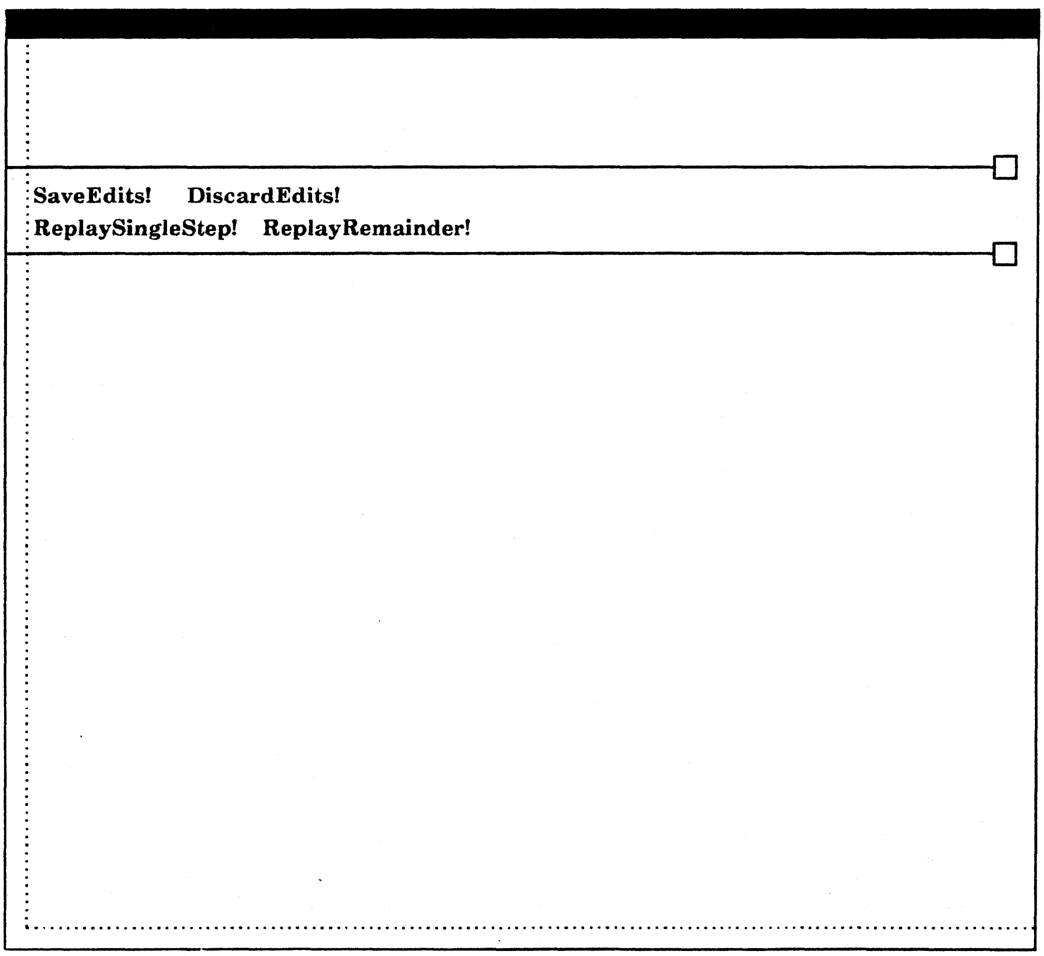


Figure 2.46

3) To replay a single edit, select **ReplaySingleStep!** in the command subwindow. Continue replaying steps until you have replayed all the steps you want to save. If you want to save all of your edits, select **ReplayRemainder!**. You can also replay several steps and then select **ReplayRemainder!** to replay the remaining steps.
4) Select **SaveEdits!** to save the edits you replayed. Select **DiscardEdits!** to discard the edits you replayed.
5) Select **Close!** to close the schematic window.

2.11 Editing Schematic Formats

Usually you have several empty documentation sets with different formats on your display desktop. Frequently these are standard formats for your organization. You can edit schematic formats, for example if you need a special format for a design, or if you are setting up formats for your organization. To edit a schematic format, you must log in to the system with the organization password.

- To edit a schematic format:

1) Open the schematic.
2) Select the **EditSchematic!** command in the command subwindow.
3) Display page one in the schematic subwindow.
4) Select **Previous!**. The system displays schematic page 65534. Use this page for editing the format for the first page of the schematic. Select **Previous!** again. The system displays page 65533. Use this page to edit formats for the rest of the pages in the schematic.
5) Select **Props!** to open the schematic window property sheet. In the property sheet is a list of fonts, numbered from one to 16 (figure 2.47). Each font has a family and a height property. These fonts correspond to text fonts in the symbol library. For instance, the pin names and numbers may be assigned font one, device parameters may be assigned font three, and symbol designators may be assigned font five. By assigning family and height properties to a certain font, you instruct the system to display text assigned to that font with those properties. If pin names are assigned font one, and you assign font one to family Sans Serif with a height of eight, the system displays all pin names as 8-point Sans Serif. At the bottom of the list of fonts are family and height properties for net names.

Apply! **Abort!** **Coarse Grid Unit:** {1/2 inch}
Display options: {thisView}
Designators: {none} **Pin names:** {monoliths} **Pin numbers** **Physical Name** **Random Text**
Device parameters ... 1 2 3 **Show grid ticks**
Drawing Size (inches) ... **Height:** 23.00 **Width:** 35.00

Font # 1	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 2	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 3	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 4	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 5	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 6	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 7	**Family:** {SansSerif}	**Height(hundredths):** {12}
Font # 8	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 9	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 10	**0Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 11	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 12	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 13	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 14	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 15	**Family:** {SansSerif}	**Height(hundredths):** {8}
Font # 16	**Family:** {SansSerif}	**Height(hundredths):** {8}
Net Name	**Family:** {SansSerif}	**Height(hundredths):** {12}

Figure 2.47

6) Measure the size of the schematic format you want to create. The system lets you create schematics of any format size. Enter the width and height in the schematic window property sheet. The width and height properties prevent users from placing symbols, lines, and text outside of the format boundries. Close the property sheet.
7) Edit the format by creating, and editing lines and text.
8) Turn on the **Use Coarse Grid** command in the schematic property sheet when you want the source and destination markers to be unconstrained by the grid marks in the schematic subwindow. Use this command when you want to draw lines that have endpoints not on grid points, or to add text at off-grid locations.

- To draw lines on the format pages:

1) Mark the endpoints of a line with the source and destination markers.
2) Select **Create Line** in the Editing Schematic Menu.
3) Select the line and press the **PROP'S** key. Assign a style and width to the line. One line width unit is one-four-hundredth of an inch.

- To add text:

  1) Change the selection mode to **text**.
  2) Set the source marker at the location where you want to add the text.
  3) Type the text.
  4) Press the **PROP'S** key to display the text property sheet. Assign a family and height to the text.

Use the editing keys, **DELETE**, **MOVE**, **COPY**, and **PASTE**, to edit lines and text. When you have finished editing the format on both the title page and the continuation page, save your edits and close the schematic.

3. The Net List

After you close a schematic, the system can create several lists to help you analyze the schematic. You use the *net list* window to display and print these reports and to interface to simulation programs and pc board design systems.

- To open the net list window:

1) Select any character in the words "**Net List**" in the documentation set window.
2) Select **Open** in the Document Operations Menu, or, with the cursor inside the documentation set window, press the key marked **OPEN**.

The net list window has a system message subwindow, a command subwindow, and a third subwindow in which it displays listings. See figure 3.1.

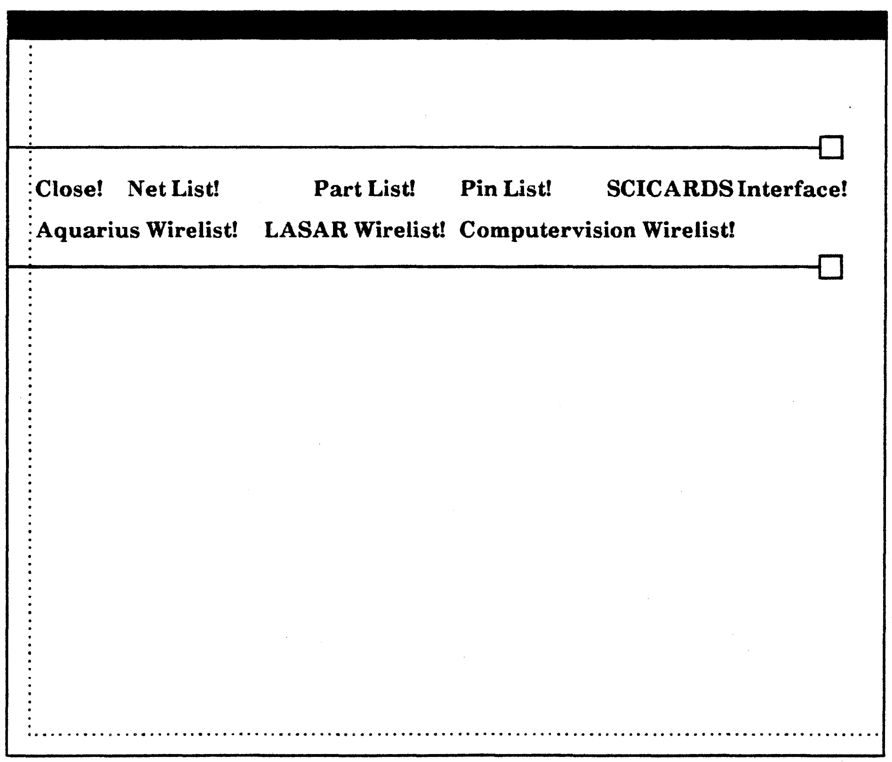


Figure 3.1

3.1 The Net List

- To display a net list:

1) Open the net list window.
2) Select the **Net List!** command in the command subwindow.

The system lists nets alphabetically by name, along with symbol designators, part designators, and pin numbers. It shows you the total number of nets and the number of points assigned to nets. Figure 3.2 shows a sample net list.

Close! Net List! Part List! Pin List! SCICARDS Interface!
Aquarius Wirelist! LASAR Wirelist! Computervision Wirelist!

```
*  NET LIST FOR DOCUMENTATION SET: Demo

*  Extra long signal names will be flagged with a ~ to indicate truncation on
*   output

*  () = System generated numbers used as a substitution for unspecified
*        symbol designators.

*  [] = User specified symbol designators and pin names used as a
*  substitution for unspecified part designators and pin numbers.

*  # = Appended to pin names that occur in schematic with a negating bar.

*  SIGNAL LISTING
FF00                              (539)-[Nam] U9-[Q]
FF01                              (538)-[Nam] U9-[Q#]
FF03                              (537)-[Nam] U9-[Q]
FF03                              (536)-[Nam] U9-[Q#]
INC                               U10-5 P1-14
IND                               U11-1 U11-10 U11-4 P1-15
NN281                             U9-[K] U9-[K] U11-3
NN282                             U9-[J] U11-8
NN284                             U11-2 U11-9 U11-15 U10-6
NN287                             U9-[J] U11-6
*  Number of Signal Nets ................ 10
*  Number of Points Assigned to Nets .....25
*  END OF LISTING
```

Figure 3.2

3.2 The Part List

- To display a part list:

  1) Open the net list window.
  2) Select the **Part List!** command in the command subwindow.

The part list shows you the symbol designator and physical part name for each symbol, its device parameters if applicable, and the total number of symbols in the design. Figure 3.3 shows a part list.

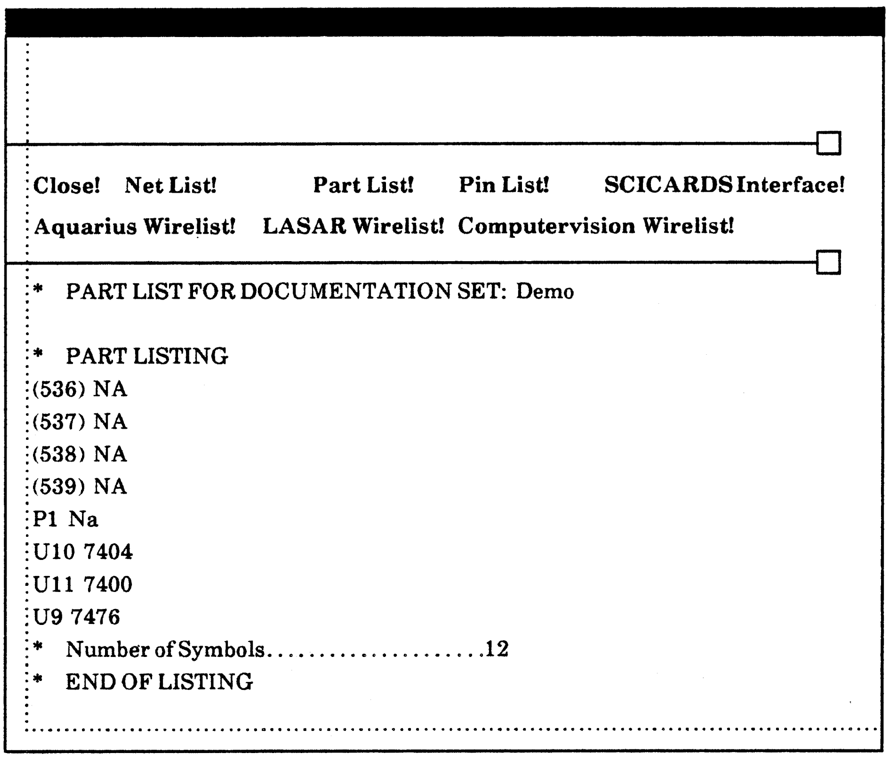


Figure 3.3

3.3 The Pin List

- To display a pin list:

  1) Open the net list window.
  2) Select the **Pin List!** command in the command subwindow.

In the pin list, each pin connected to a net has an entry that includes the symbol designator, pin type, part reference designator (physical designator), pin number, pin name, physical part name (generic), device parameters (value, wattage, and tolerance), schematic page number, and pin list line number. See figure 3.4.

If a pin list entry exceeds the amount of space allocated to its field, the system truncates it and marks it with a tilde (~) to show you the entry is truncated.

Close! Net List! Part List! Pin List! SCICARDS Interface!

Aquarius Wirelist! LASAR Wirelist! Computervision Wirelist!

```
*  PIN LIST FOR DOCUMENTATION SET: Demo
*  # = Appended to pin names that occur in schematic with a negating bar.
*  ~ = Appended to a field when truncation occured during output
*Signal Name (system generated begins with NN)
*!                  Symbol Designator (() = system generated)
*!                  !  Pin Type (i = input, o = output, g = ground, p = power,
*!                  !  u = unspecified, d = duplex)
*!                  !  !      Part Reference Designator
*!                  !  !      !  Pin Number
*!                  !  !      !  !  Pin Name
*!                  !  !      !  !  !    Generic, value, wattage, tolerance
*!                  !  !      !  !  !    !                     Page
*!                  !  !      !  !  !    !                     !  Line
*!                  !  !      !  !  !    !                     !  !
*!                  !  !      !  !  !    !                     !  !
*!                  !  !      !  !  !    !                     !  !
*!                  !  !      !  !  !    !                     !  !
*!                  !  !      !  !  !    !                     !  !
*!                  !  !      !  !  !    !                     !  !

FF00      (539)  u    ?   ?  Nam                    1   1
FF00      (541)  o   U9   ?  Q     7476,,,          1   2
FF01      (538)  u    ?   ?  Nam   ,,,              1   3
FF01      (541)  o   U9   ?  Q#    7476,,,          1   4
FF02      (537)  u    ?   ?  Nam   ,,,              1   5
FF02      (540)  o   U9   ?  Q     7476,,,          1   6
FF03      (536)  u    ?   ?  Nam   ,,,              1   7
FF03      (540)  o   U9   ?  Q#    7476,,,          1   8
INC       (542)  i  U10   5  in    7404,,,          1   9
INC       (546)  u   P1  14  Nam   ,,,              1  10
IND       (543)  i  U11   1  In    7400,,,          1  11
IND       (544)  i  U11  10  In    7400,,,          1  12
IND       (545)  i  U11   4  In    7400,,,          1  13
IND       (547)  u   P1  15  Nam   ,,,              1  14
NN281     (541)  i   U9   ?  K     7476,,,          1  15
NN281     (540)  i   U9   ?  K     7476,,,          1  16
NN281     (543)  o  U11   3  Out   7400,,,          1  17
NN282     (540)  i   U9   ?  J     7476,,,          1  18
NN282     (544)  o  U11   8  Out   7400,,,          1  19
NN284     (543)  i  U11   2  In    7400,,,          1  20
NN284     (544)  i  U11   9  In    7400,,,          1  21
NN284     (545)  i  U11   5  In    7400,,,          1  22
NN284     (542)  o  U10   6  out   7404,,,          1  23
NN287     (541)  i   U9   ?  J     7476,,,          1  24
NN287     (545)  o  U11   6  Out   7400,,,          1  25
```

Figure 3.4

3.4 Printing the Contents of the Net List Window

- To print the contents of the net list window:

1) Open the net list window. Display the list or lists you want to print. For example, if you want to print only the net list, then display only the net list. To print only the part list, display only the part list. To print just the pin list, display just the pin list. To print the net list and part list, display both. Close the net list window.
2) Open the File Tool.
3) In the File Tool parameter subwindow, enter the name of your output server as the **Host**.
4) Enter the name of the net list's documentation set followed by the characters ".nl" as the **Source**.
5) Enter the word "ASCII" as the **Destination**. See figure 3.5.
6) Select **Store!** in the command subwindow.

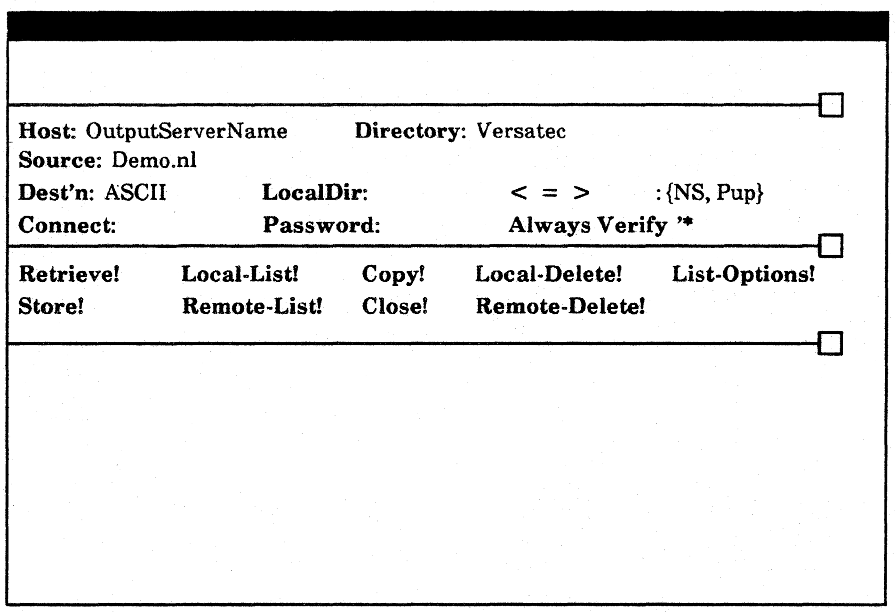


Figure 3.5

3.5 Interfacing to SCICARDS™

Design services departments with the SCICARDS printed circuit board design system can design printed circuit board layouts from the schematics you create with Expert 1000. You design the schematic, convert it to SCICARDS format, and write it to a magnetic tape on an output server. The magnetic tape serves as input to the SCICARDS system.

- The SCICARDS system must meet the following requirements:

1) SCICARDS version 82.01.
2) VAX/VMS version V3.1 or Prime version 18.3

3.5.1 Designing a Schematic to Interface with the SCICARDS System

- To design a schematic to interface with the SCICARDS system:

1) Check that your symbol library is compatible with the SCICARDS library. Expert symbol library pin names must match the SCICARDS part library pin names for all SCICARDS functional parts.
2) In symbol property sheets, enter pin numbers for all "fixed pin devices." SCICARDS defines fixed pin devices as devices that perform a single function. If you do not enter all pin numbers, Expert passes pin names but is unable to back-annotate pin numbers.
3) Enter SCICARDS part names as physical name values. If no comparable SCICARDS part name exists, enter the SCICARDS device type and shape (separate the two with a space).
4) Enter the SCICARDS part designator as the physical designator for all fixed-pin parts. If you do not enter the part designator, Expert uses its symbol designator as the SCICARDS part designator. If you do not enter the symbol designator, Expert generates a number preceded by "ND" (for example: ND330) and uses it as the SCICARDS part designator. Do not rename SCICARDS part designators; Expert does not back-annotate physical designators from SCICARDS part designators you rename.
5) Enter symbol designators for all SCICARDS *functional parts*. SCICARDS uses the Expert symbol designator as its function designator. If you do not enter a symbol designator, Expert generates a number preceded by "ND" (for example: ND220), and uses it as the SCICARDS function designator. If you entered a SCICARDS part designator as the Expert physical designator, Expert assigns the function to the first available slot in the part. Expert leaves the function free and swapable for SCICARDS users. Do not enter pin numbers; if you do SCICARDS cannot swap functions and pins. Expert assigns nodes to the designated pin numbers and fixes that function.
6) Save the schematic.

SCICARDS is a registered trademark of Scientific Calculations Inc., Fishers, New York.

3.5.2 Converting the Schematic to SCICARDS Format

- To convert a net list to SCICARDS format:

1) Open the net list window. If it is already open and you have listed a net list, part list, or pin list in it, close the net list and reopen it.
2) Select the **SciCards Interface!** command in the command subwindow. The system displays the net list in SCICARDS format (figure 3.6). Do not select any other commands.
3) Close the net list window. Write the net list to magnetic tape before reopening the window.

Close! Net List! Part List! Pin List! SCICARDS Interface!
Aquarius Wirelist! LASAR Wirelist! Computervision Wirelist!

```
--SCICARDS Interface FOR DOCUMENTATION SET: Demo

PARTS
7400               U11
7404               U10
7476               U9
EOS
FUNCTIONS
7476               ND541      ND540
ASSIGN ND541 U9
ASSIGN ND540 U9
EOS
NET LIST
NODE   280 $
    ND539-Nam  ND541      Q
NODE   279 $
    ND538-Nam  Nd541     QN
NODE   278 $
    ND537-Nam  ND540      Q
NODE   277 $
    ND536-Nam  ND540     QN
NODE   286 $
U10           5          P1-14
NODE   285 $
U11           1  U11        10  U11          4        P1-15
NODE   281 $
ND541         K  ND540       K  U11          3
NODE   282 $
ND540         J  U11         8
NODE   284 $
U11           2  U11         9  U11          5  U11        6
NODE   287 $
ND541         J  U11         6
EOS
NAMED NODES
   280 FF00
   279 FF01
   278 FF02
   277 FF03
   286 INC
   285 IND
   281 NN281
   282 NN282
   284 NN284
   287 NN287
EOS
```

Figure 3.6

- Before writing the net list to magnetic tape, Prime users must replace carriage returns created by Expert with line feed characters. To edit the net list:

1) Create an empty file window. See Book 1 section 3.5.
2) Move the cursor into the third subwindow of the file window and press the left mouse button.
3) Type the name of the net list's documentation set, followed by a period, followed by the letters ".nl." See figure 3.7.

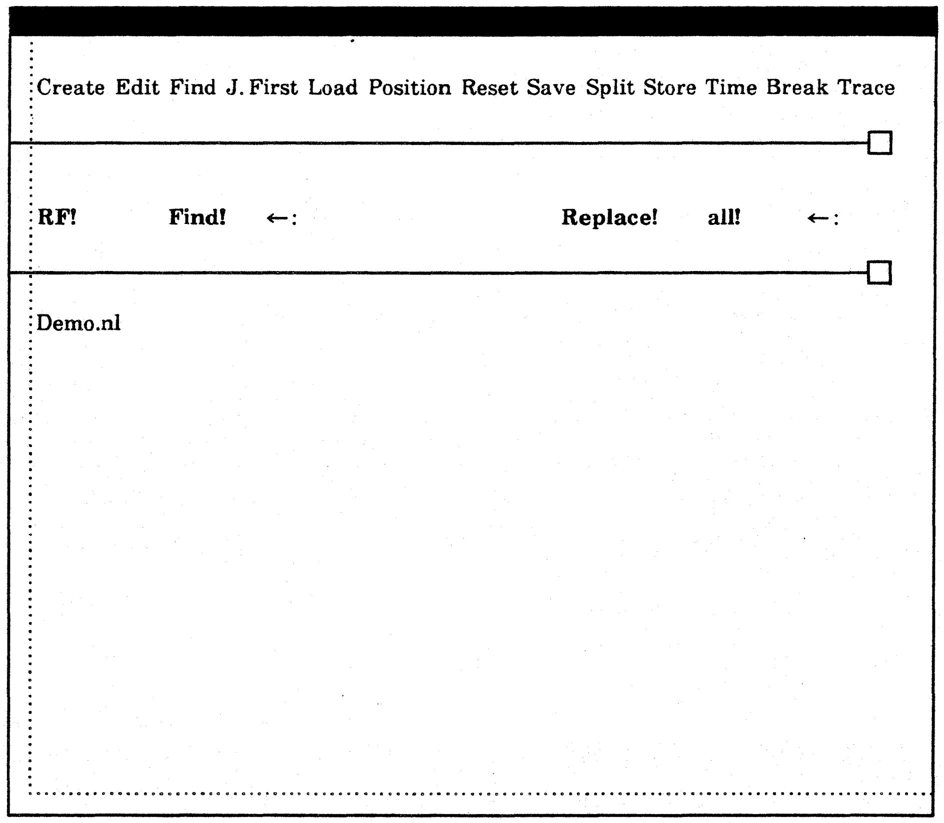


Figure 3.7

6) Select the letters you typed.
7) Load the SCICARDS interface list. Select **Load** in the file window's first subwindow. The system displays the net list in the third subwindow in SCICARDS interface format.
8) Select **Edit** in the first subwindow.
9) Select the colon following the **Find!** command in the second subwindow. Leave the cursor inside that subwindow. Type a carriage return; press the carriage return key.
10) Scroll back to the top of the subwindow. Select the colon after the **all!** command. Leave the cursor inside the subwindow. Type **PROP'S** and **J** simultaneously.
11) Select the **all!** command.
12) Save the file. Select the **Save** command in the first subwindow.

13) Select the **Destroy** command in the first subwindow to remove the window from the display.

3.5.3 Writing the Net List to Magnetic Tape

- To write the net list to magnetic tape:

1) Make the Tape Tool window active.
2) Change the **Tape Type** to **unlabeled**.
3) Enter the name of the output server to which you want to send the net list as the **Host**.
3) Enter the name of the net list's documentation set, followed by a period, followed by the letters "nl" as the **Source**.
4) Select any character in the word "**Dest'n**." Type the word "magtape," followed by a period, followed by the magnetic tape file number, followed by a period, followed by the record length. Figure 3.8 shows the tape tool with a sample **Host, Source**, and **Dest'n** entered.

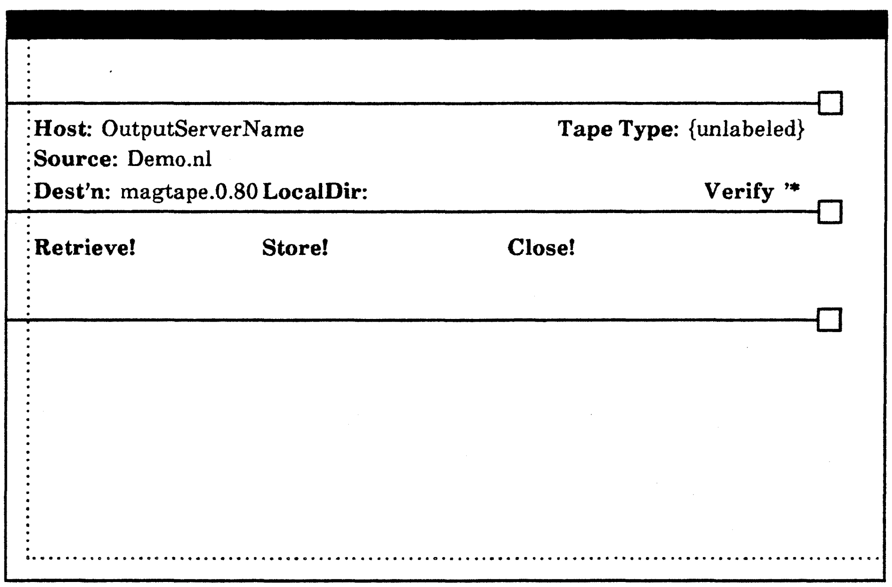


Figure 3.8

5) Select **Store!** in the Tape Tool command subwindow.

3.5.4 Designing the PC Board Layout with SCICARDS

- To design a printed circuit board with the SCICARDS system from an Expert 1000 schematic, SCICARDS users must follow several procedures.

  1) Users must rename SCICARDS connectors to match Expert connector names and numbers. Connectors must have an Expert symbol type of **Connector**.
  2) The user must check that there is a sufficient number of parts for all assignable functions. Use the SETUP COUNT command. Add additional parts to the SCICARDS subfile as required.
  3) Before reading the Expert file into the SCICARDS system, users must rename connectors to Expert designators and pin numbers. CARD, GRID, etc. should exist.

- To read the Expert magnetic tape:

  1) Using VAX/VMS, read the tape as /FOR/DENS=1600 (foreign density=1600 BPI), and create a VMS file. The following paragraph shows you the commands you use to instruct the VAX to read the magnetic tape. Prompts that the system displays are in regular type. Commands you enter are in bold type. Enter a carriage return at the end of each line. Replace the word **filename** with the name you want to use for your design file.

     $ **ALL MT:**
     $ **MOU MT: /FOR/DENS=1600**
     $ **COPY MT: filename.DAT**

     When using Prime, read the tape with MAGNET, and create a PRIMOS file. The following paragraph shows you the commands to use. Prompts that the system displays are in regular type. Commands you enter are in bold type. Enter a carriage return at the end of each line. Replace the word **filename** with the name you want to use for your design file.

     OK, **AS MT1**
     OK, **MAGNET**
     OPTION: **READ**
     MTU#= **1**
     MT FILE=**1**
     LOGICAL RECORD SIZE= **80**
     BLOCKING FACTOR= **1**
     ASCII, BCD, BINARY, OR EBCDIC? **ASCII**
     OUTPUT FILE: **filename**

  2) Complete the procedures above before reading the file to SCICARDS.
  3) Use the INPUT CR command to read the VMS or PRIMOS file into SCICARDS.

3.5.5 Back-Annotating from SCICARDS

To back-annotate the schematic with Expert, the SCICARDS user must first write the file to magnetic tape. He or she uses the OUTPUT CP command to create a VMS or PRIMOS file of LIST ALL. The VAX user writes the file to magnetic tape with VMS answering "yes" to the carriage control question, and entering the commands in the paragraph below.

$ **ALL MT:**
$ **RUN UTIL: TAPECRE**

The Prime user uses the TOVAX command at the SC command level as in the example below.

OK, **PROFILE**
SC COMMAND: **TOVAX filename**

- The Expert user uses the Tape Tool to read the magnetic tape.

1) Make the Tape Tool window active. Set the **Tape Type** to **unlabeled**.
2) Enter the name of your output server as the host.
3) Enter "MAGTAPE.*filenumber*" as the source. The file number is the number of the file you want to retrieve from the magnetic tape.
4) Enter a file name of your choice as the destination.

- To back-annotate the schematic.

1) Open the schematic.
2) Enter the file name you entered as the destination in the Tape Tool after the words "**FromFile:**" in the schematic command subwindow.
3) Select **BackAnnotate!**

3.6 Interfacing to Computervision™

You can use schematics created with Expert 1000 as input to a Computervision printed circuit board design system. The Computervision interface requires you to build your symbol library and schematic within certain limitations. When you complete the schematic, you convert it to Computervision format and write it to a magnetic tape. Give the tape to Computervision users to complete the board design. Expert 1000 currently does not support back-annotation from the Computervision system.

3.6.1 Building a Symbol Library to Interface with Computervision

- If you plan to use a Computervision system to design pc boards from Expert 1000 schematics, you must build your library with the following requirements:

    1) Symbol pin names must be less than or equal to 10 characters.

3.6.2 Building a Schematic to Interface with Computervision

Computervision must know symbol designators, physical part names, physical designators, physical pin numbers, and net names, and each has a maximum length. If you do not enter these properties, Expert automatically assigns default values.

- Use the following guidelines to create a schematic for use with a Computervision system:

    1) Expert automatically generates symbol designators for interfacing with Computervision. If you enter a symbol designator the system ignores it.
    2) Net names cannot exceed 20 characters. If you do not enter a net name, the system automatically generates a name such as "NN305." The prefix "NN" tells the Computervision user that the name is an Expert default name.
    3) Physical part names must be no longer than 12 characters. If you do not enter a physical part name, Expert assigns the symbol function from the Expert symbol library.
    4) Physical designators must be no longer than 7 characters. You enter the physical designator for each symbol or else Expert marks the symbol as an unassigned symbol awaiting assignment by the Computervision user.
    5) Physical pin numbers must be less than or equal to 10 characters. If you do not assign physical pin numbers, they are translated to Computervision with unassigned status.
    6) Device-specific parameters must be no longer than 4 characters. DEV1 must be value, DEV2 must be wattage or voltage, and DEV3 must be tolerance.

3.6.3 Creating a Computervision Wire List

- To convert the net list to Computervision format:

1) Open the net list window.
2) Select the **Computervision Wirelist!** command. The system displays the wire list in Computervision format. Figure 3.9 shows a Computervision wire list.
3) Select the **Close!** command to close the net list window.

Close! **Net List!** **Part List!** **Pin List!** **SCICARDS Interface!**

Aquarius Wirelist! **LASAR Wirelist!** **Computervision Wirelist!**

1536	FF03	CONNECTOR	?	Nam	Nam	
1537	FF02	CONNECTOR	?	Nam	Nam	
1538	FF01	CONNECTOR	?	Nam	Nam	
1539	FF00	CONNECTOR	?	Nam	Nam	4
1547	IND	P1	P1	Nam	15	
1546	INC	P1	P1	Nam	14	2
1545	IND	7400	U11	In	4	
1545	NN284	7400	U11	In	5	
1545	NN287	7400	U11	Out	6	
1544	IND	7400	U11	In	10	
1544	NN282	7400	U11	Out	8	
1544	NN284	7400	U11	In	9	
1543	IND	7400	U11	In	1	
1543	NN281	7400	U11	Out	3	
1543	NN284	7400	U11	In	2	
1542	INC	7404	U10	in	5	
1542	NN284	7404	U10	out	6	1
1541	FF00	7476	U9	Q	Q	
1541	FF01	7476	U9	Q#	Q#	
1541	NN281	7476	U9	K	K	
1541	NN287	7476	U9	J	J	
1540	FF02	7476	U9	Q	Q	
1540	FF03	7476	U9	Q#	Q#	
1540	NN281	7476	U9	K	K	
1540	NN282	7476	U9	J	J	2

Figure 3.9

3.6.4 Writing the Computervision Wire List to Magnetic Tape

- Use the Tape Tool to write the Computervision wire list to a magnetic tape.

  1) Activate the Tape Tool window.
  2) Set the **Tape Type** to **unlabeled**.
  3) Enter the name of the documentation set, followed by a period and the letters "nl" as the **Source**.
  4) Select **Store!**

3.7 Interfacing to LASAR

You can format the net list for input to LASAR, and output it to magnetic tape with the output server. Use that tape as input to the LASAR program running on a mainframe computer.

3.7.1 LASAR Map

Before you can build schematics to interface with LASAR, you must have a special text file named "LASARMap.$Teton" stored on your local disk. The LASAR Map file maps Expert symbol descriptions to LASAR terminology.

The LASAR Map file consists of several *sections*. Each section describes a symbol. Each section has a name. The section name is either the symbol's Expert library function or physical part name from the symbol's property sheet in the schematic. The section name is enclosed in brackets. For example:

    [74LS14]

Each section contains *key:item* pairs that map the Expert terminology to LASAR terminology. Each pair is on a separate line in the file. One pair maps the the section name to the LASAR component code. The *key* for this pair is "LASAR Alias." The *item* is the LASAR component code. For example:

    LASAR Alias: AM25LS14

The remaining pairs form a *pin table*. The pin table maps either Expert pin names or pin numbers to LASAR pin codes. The first pair in the pin table tells the system whether the mapping is by Expert pin name or pin number. The key for this pair is "Pin Table." The item is "Names" if you are describing the following pins by their Expert library names, or "Numbers" if you are describing the pins by their Expert library pin numbers. For example:

    Pin Table: Numbers

"Names" is the default case; if you omit the "Pin Table" pair, the system assumes that you are describing pins with their Expert names. The two options exist because different pins can have the same name, but pin numbers are unique.

The remaining pairs map individual Expert pins to the LASAR pin code. The Expert pin name or number is the key, and the corresponding LASAR pin code is the item. For example:

    A:1

Separate LASAR Map sections with an empty line. Figure 3.10 shows a LASAR Map section that describes Expert symbol 74LS14.

[74LS14]	The section name is the Expert library name.
LASARAlias: AM25LS14	The LASAR component code for Expert symbol 74LS14.
Pin Table: Names	Pin table listing is by Expert names.
A:1 B:2 Y:3	Expert pin A is equivalent to LASAR pin 1. Pin B is equivalent to LASAR pin 2. Pin Y is equivalent to LASAR pin 3.

Figure 3.10

The LASAR Map file must have one section named "Global Definitions" that describes sections associated with input/output type functions. For example, you must define connectors and test points in this section so that Expert can interpret them correctly when generating the LASAR wire list.

The Global Definitions section maps function types to sections. Figure 3.11 shows a Global Definitions section.

	[Global Definitions]	
The type of function →	I/O: CONNECTOR	← LASAR Map section associated with the function

Figure 3.11

Creating a LASAR Map File

Although individual users can create their own LASAR Map files, it is more efficient to have one person, perhaps the system librarian, create a single LASAR Map file that contains LASAR descriptions of all symbols in your installation's library. You can store that file on a central file server. Users can retreive it with the File Tool.

- To create a LASAR Map file:

1) Compile the information you need from the Expert symbol library, and from the LASAR library.
2) Create an empty file window. See Book 1 section 3.5.
3) Use text manipulation techniques to enter each section in the file window. Use carriage returns to create new lines. Leave an empty line between sections.
4) Create a second file window. In its text subwindow type the word "LASARMap.$Teton" Do not type the quotation marks. Select the letters you typed. Select **Store** in the LASAR Map file window.

Editing the LASAR Map File

If you are responsible for updating the LASAR Map file, you might want to add a section to the file each time you add a symbol to your library. You then store the new LASAR Map on a central file server, and notify other users.

- To edit the LASAR Map file:

1) Create an empty file window.
2) In its text subwindow type the word "LASARMap.$Teton" Do not type the quotation marks. Select the letters you typed, and select **Load** in the empty file window's command subwindow or File Window Menu. The system loads the LASAR Map file into the text subwindow.
3) Use the text editing techniques described in Book 1, section 3.5 to edit the file.
4) Save your changes by selecting **Save** in either the command subwindow or File Window Menu.

3.7.2 Building a Schematic to Interface with LASAR

- To build a schematic to interface with LASAR follow these rules:

1) All symbol properties and net names must be three characters long. The first character must be alphabetic, and the second two numeric. For example:

   C55

   The simplest way to construct a schematic to interface with LASAR, is to construct it without entering any symbol properties (except for connectors) or net names. The system automatically assigns LASAR codes.
2) You must enter physical designators and pin numbers for all connectors. Physical Designators must also be three characters long. The first character must be alphabetic, and the second two numeric.
3) The maximum number of symbols in the schematic must be less than 2,600.

3.7.3 Converting the Net List to LASAR Format

- To convert the net list to LASAR format:

  1) Open the net list window.
  2) Select the **LASAR Wirelist!** command. The system displays the wire list in LASAR format. It marks truncated fields with a tilde (~). Figure 3.? shows a LASAR wire list.

```
Close!   Net List!          Part List!     Pin List!     SCICARDS Interface!
Aquarius Wirelist!   LASAR Wirelist!  Computervision Wirelist!

$ALEC
PRINT
MODEL DEBUG
WIRELIST TERADYNE
$EOD
Demo   /CREATED ON 24-May-83 14:44:39
7400   AAA     SN7400D
7404   AAB     7404
7476   AAC     SN7476D
CIRCUIT
P0 P2T1,f41P3    /FF00
P0 P3T1,f41P4    /FF01
P0 P4T1,f40P3    /FF02
P0 P5T1,f40P4    /FF03
P1 P1T14,f42P1   /INC
P1 P1T15,f43P1,f44P1,f45P1     /IND
   f41P2,f40P2,f43P3 /NN281
   f40P1,f44P3    /NN282
   f43P1,f44P1,f45P1,f42P2     /NN284
   f41P1,f45P3    /NN287
DEVICE
f43,f44,f45=7400
f42=7404
f40,f41=7476
$EOD
```

Figure 3.12

3) Check the wire list for errors. Errors occur when the LASAR description of a pin is not in the LASAR Map. The system flags errors with the word "ERROR," and a description of the error. If there are errors, close the net

list window, and edit the LASAR Map file. Then repeat steps 1 through 3.

4) When the system displays the wire list without errors, select the **Close!** command to close the net list window.

3.7.4 Writing the LASAR Wire List to Magnetic Tape

- Use the Tape Tool to write the LASAR wire list to a magnetic tape.

1) Activate the Tape Tool window.
2) Set the **Tape Type** to **unlabeled**.
3) Enter the name of the documentation set, followed by a period and the letters "nl" as the **Source**.
4) Select **Store!**

4. Index

VERSATEC
A XEROX COMPANY

Expert 1000 User's Guide

The Library Editor

This document was prepared with the Xerox 8010 Information System.

Document Number 1-40B May 1983 Printed in USA

Table of Contents

1. Introduction

The Expert 1000 library is a data base for storing descriptions of the logic symbols used in creating schematics. The library contains information such as the identification number, name, logic function, electrical characteristics, and graphic representation of each symbol.

The *Library Editor* is your tool for adding, removing and changing library entries. This book tells you how to use the Library Editor. Section one introduces you to the Library Editor window, library entry types, and library identification numbers. Section two explains all commands for creating logic symbols; it ends with an example that shows how to use the commands to create a NAND gate. Section three explains library maintenance tasks, and section four is an index.

A special organization password helps your organization to maintain a standard library. You may *write* symbols to the library and thus change the library, only if you log in to the system with the organization password. If you do not, you may only read library entries; you may look at entries but you cannot change them in the library. Your System Administrator has the organization password, and decides who may change the library.

1.1 Opening The Library Editor

The Library Editor is stored in the *Inactive Menu.*

- To **open** the Library Editor:

1) Move the cursor to an empty (grey) area on the display.
2) Bring up the Inactive Menu.
3) Select **Library Editor**.

Initially, the Library Editor window has two subwindows. The first is a system message subwindow, and the second is a command subwindow. See figure 1.1.

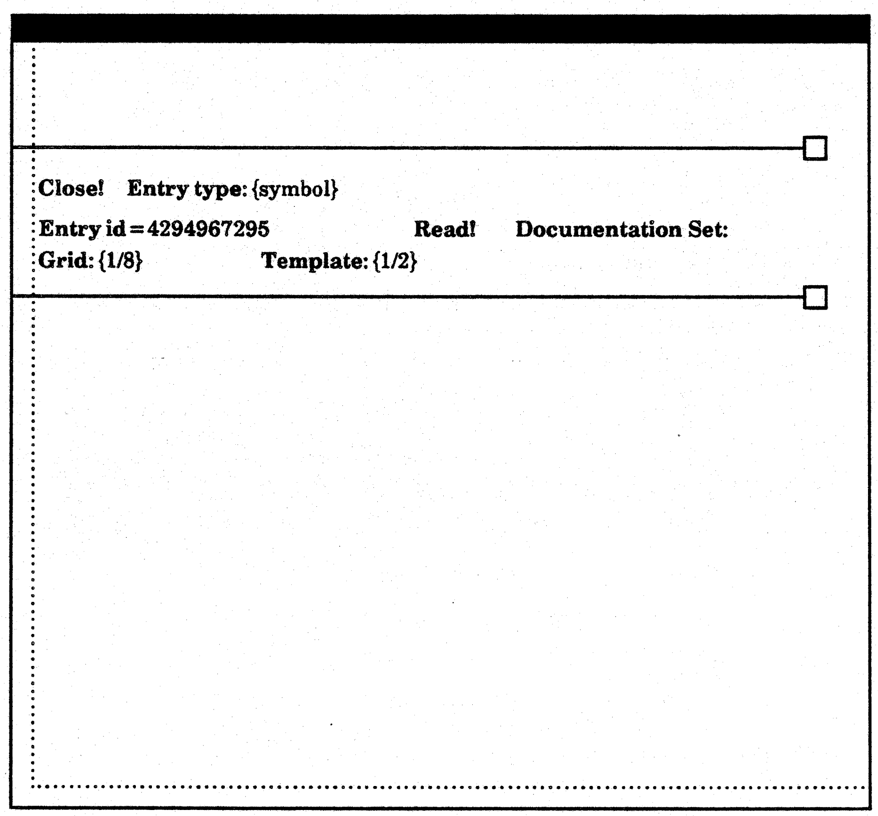


Figure 1.1

1.2 Entry Types

Logic symbols are one *type* of library entry. There is another entry type, called *library*, that lets you perform library maintenance tasks such as removing entries from the library and converting libraries to new formats (when you receive a new release of Expert 1000). When you first open the Library Editor, the entry type is set to **symbol**. To perform library maintenance tasks, change the entry type to **library**.

- To change the entry type:

1) Point the cursor at the words "**Entry type**" in the command subwindow.
2) Bring up the Entry Type Menu.
3) Select an entry type from the menu. Notice that the system displays the new type in the command subwindow.

Try changing the entry type. Notice that several of the commands inside the command subwindow change when you change the entry type. Two commands always remain: **Close!** and **Entry id**.

1.3 The Entry Identification Number

All library entries have a unique identification number. The **Entry id** field in the command subwindow lets you specify a certain entry. You specify an entry identification number when you want to look at a particular symbol, for example to change it.

Versatec assigned a certain range of library identification numbers to your organization. Use the numbers in this range when creating new library entries. If you do not know what these numbers are, ask your System Administrator.

- Use the text editing techniques described in Book 1 to change the entry identification number:

1) Select any character in the words "**Entry id**" with the right mouse button. The system deletes the entry identification number.
2) Type the new number.

1.4 Closing the Library Editor

The **Close!** command closes the Library Editor. When you issue the **Close!** command, the system removes the Library Editor window from the display and stores the Library Editor in the Inactive Menu.

2. Creating Logic Symbols

To create a logic symbol, you set the symbol's *grid* and *template* size, then create a graphic description of the symbol from several symbol items in the Library Editor's third subwindow. Next you enter symbol characteristics in the command subwindow, and then *write* the symbol to the library.

To add new a logic symbol to the library, you must first delete the entry identification number, set the proper grid and template sizes, and *read* the symbol from the library. Since the symbol identification number is blank, the system displays a blank grid on which you design the symbol. When you've created the graphic description of the symbol, and set its characteristics, you write the symbol to the library.

If you want to alter an existing symbol, you enter that symbol's identification number in the entry identification number field, and read the symbol from the library. You use the symbol drawing and editing techniques to alter the appearance of the symbol and change any of the symbol characteristics, and write the symbol back to the library.

If you want to create a new symbol by copying several of the attributes of an already-existing symbol, you first read the original symbol, and edit it You then change the entry identification number, name, and function of the new symbol and write that symbol to the library. Now both symbols are part of the library.

2.1 Setting the Grid and Template Size

In addition to its entry identification number, a logic symbol also has a grid and template size. The symbol's grid size should match the grid size of the schematics in which you will use the symbol. All symbols in a schematic should have the same template size. The system draws circles for inverting pin leads proportional to the template size. The only time you can change a symbol's grid or template size is before you create the symbol.

- To change the grid size:

  1) Place the cursor over the word "**Grid**" in the command subwindow.
  2) Activate the Grid Menu.
  3) Select a grid size from the options in the menu. Grid size options are: **one-fourth**, **one-fifth**, **one-eighth**, **one-tenth**, and **one-twentieth.**

- **To change the template:**

  1) **Place the cursor over the word "Template" in the command subwindow.**
  2) **Bring up the Template Menu.**
  3) **Select a template from the options in the menu. Template options are: one, three-fourths, one-half, three-eighths, and one-fourth.**

Expert 1000 release 1.0 supports only two grid and template combinations: one-tenth grid and three-quarter template, and one-eighth grid and one-half template. The standard library you received with the system uses one-eighth grid and one-half template.

2.2 Displaying Text Fonts

When you create a symbol, you assign font numbers to the text items in its definition. You assign properties (family and height) to each font when you create a schematic format. Font properties control the appearance of the text when you display the schematic and when you plot it. (See Book 2, section 2.11.)

Unless you instruct the Library Editor otherwise, the system applies default font properties to text items inside the Library Editor window. The Library Editor's **Documentation Set:** command lets you instruct the Library Editor to display text items as they would actually appear in a particular schematic. For instance if you assign font 3 to a pin name, the Library Editor looks at the schematic you specify to see with which properties to display font 3. It then displays the pin name with those properties.

- To specify font properties from a particular schematic:

  1) Enter the name of the schematic's documentation set after the words "**Documentation Set:**" in the Library Editor's command subwindow.

2.3 Reading Symbols

Before you can alter a symbol you must transfer a copy of that symbol from the library to the Library Editor window; this is called *reading* the symbol from the library. To read a symbol, you must know its identification number. If you don't know the symbol's identification number, set the entry type to **library** and use the **List!** command to list all entries (see section 3.1), or use the *Abbreviation Map* to list symbols and their identification numbers (see section 2.9.1).

- To read a symbol from the library:

  1) Replace the entry identification number with the symbol's identification number.
  2) Set the grid and template sizes.
  3) Select the **Read!** command.

The system displays the symbol in a third subwindow, and adds several commands to the command subwindow. See figure 2.1.

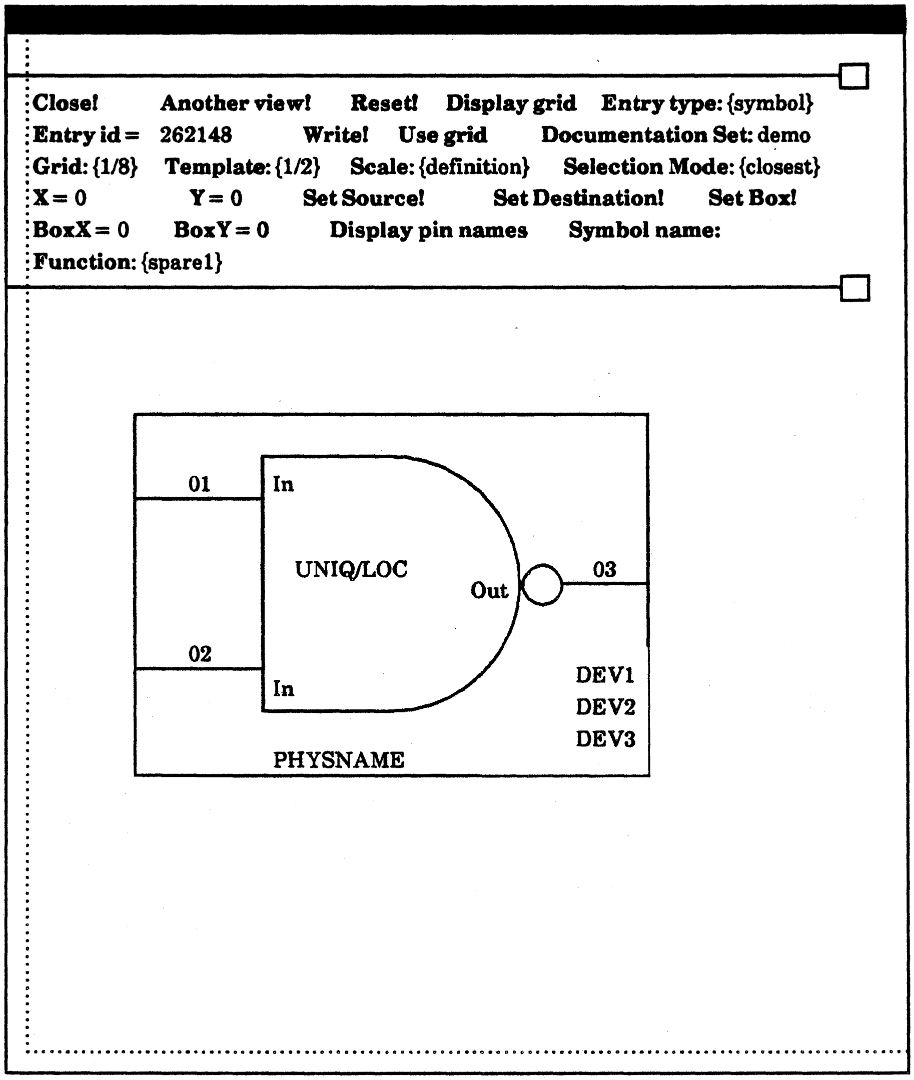

Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {spare1}
01
In
UNIQ/LOC
Out
03
02
In
DEV1
DEV2
DEV3
PHYSNAME

Figure 2.1

If you want to create a new symbol, you read an empty grid on which to create the symbol.

- **To read an empty grid:**

1) Clear the entry identification number using text editing techniques.
2) Set the grid and template sizes.
3) Select the **Read!** command.

The system displays an empty subwindow in which to draw the new symbol. Figure 2.2 shows an empty subwindow.

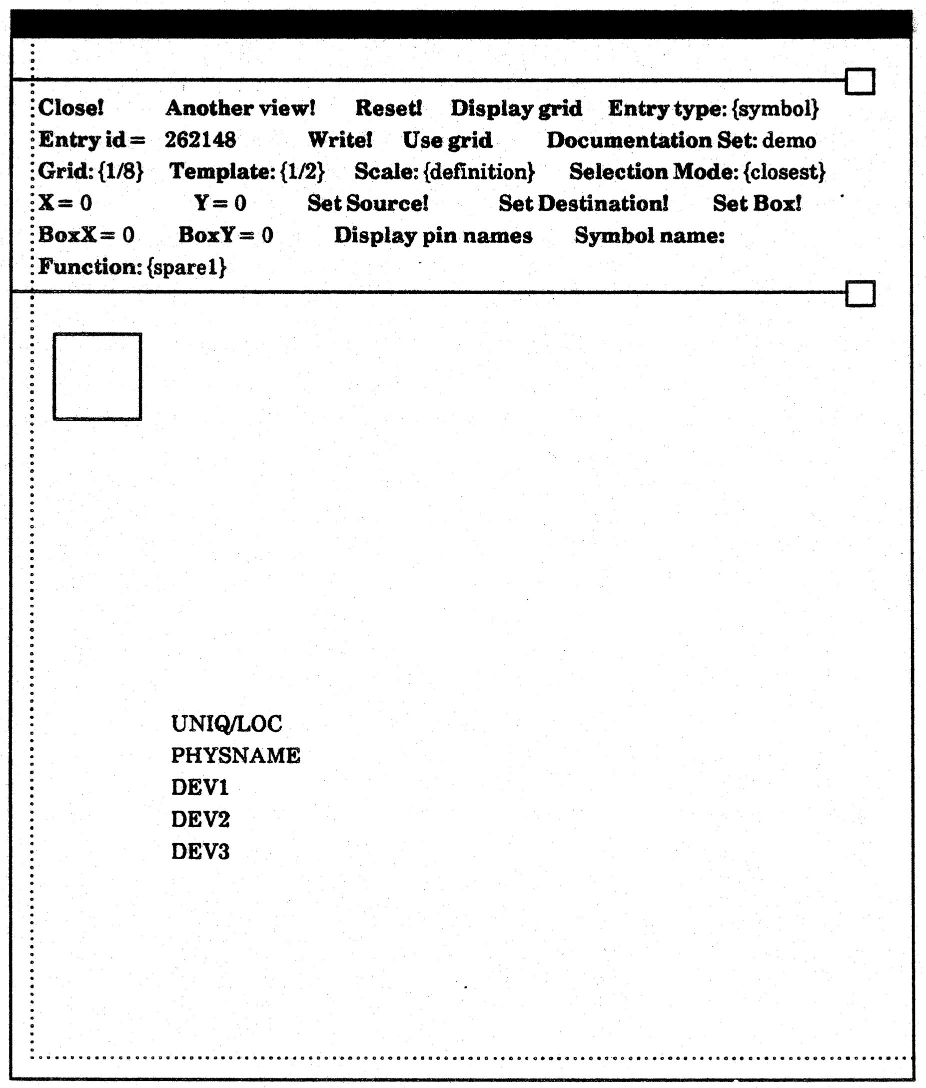


Figure 2.2

2.4 Window Manipulation

2.4.1 Creating Duplicate Views of the Library Editor Window

You can display as many views of the Library Editor window as you like. For instance, you might want to create two Library Editor windows so that you can see a symbol at two different scales simultaneously. If you make changes to the symbol in one window, you see those changes in both windows.

- To display a duplicate of the Library Editor window:

  1) Select the **Another view!** command in the command subwindow.

Notice that the name of the first window is "Library Editor Master View"; the name of the duplicate window is "Library Editor Another View." The duplicate windows do not have as many commands in their command subwindows. You can remove the duplicate views from the display, but you cannot remove the Master View.

- To remove a duplicate window from the display:

  1) Select **Destroy view!** in that window's command subwindow.

2.4.2 Displaying the Grid

The words "**Display grid**" in the Library Editor's command subwindow act as a toggle switch. When they are video-inverted, the system displays a grid in the third subwindow. When the words are not video-inverted the system does not display the grid.

- To turn the grid on:

  1) While the grid is turned off, and the words "**Display grid**" are not video-inverted, point the cursor at any character in the words "**Display grid**," and press the left mouse button.

- To turn the grid off:

  1) While the grid is on, point the cursor at any character in the words "**Display grid**," and press the left mouse button.

2.4.3 Changing the Scale

When the system first displays the grid, it displays it at a magnified scale called *definition* scale. Definition is the scale you use to define the physical appearance of the symbol. If you like, you can change the scale to eighth, fourth, half, normal, twice, or four times. Turn off the grid before making the scale smaller than normal. In practice, you may want to create symbols at definition scale, then look at them at normal scale before writing them to the library. A good way to do this is to display a second view of the symbol, and change the scale in that view to normal.

- To change the scale:

1) Turn off the grid if you plan to set the scale smaller than normal.
2) Point the cursor at the word "**Scale**" in the Library Editor command subwindow.
3) Bring up the Scale Menu.
4) Select a scale value from the options: **eighth, fourth, half, normal, twice, four,** or **definition**.

Figure 2.3 shows two views of the Library Editor window. One view is at definition scale with the grid on; the other is at four times normal scale with the grid off.

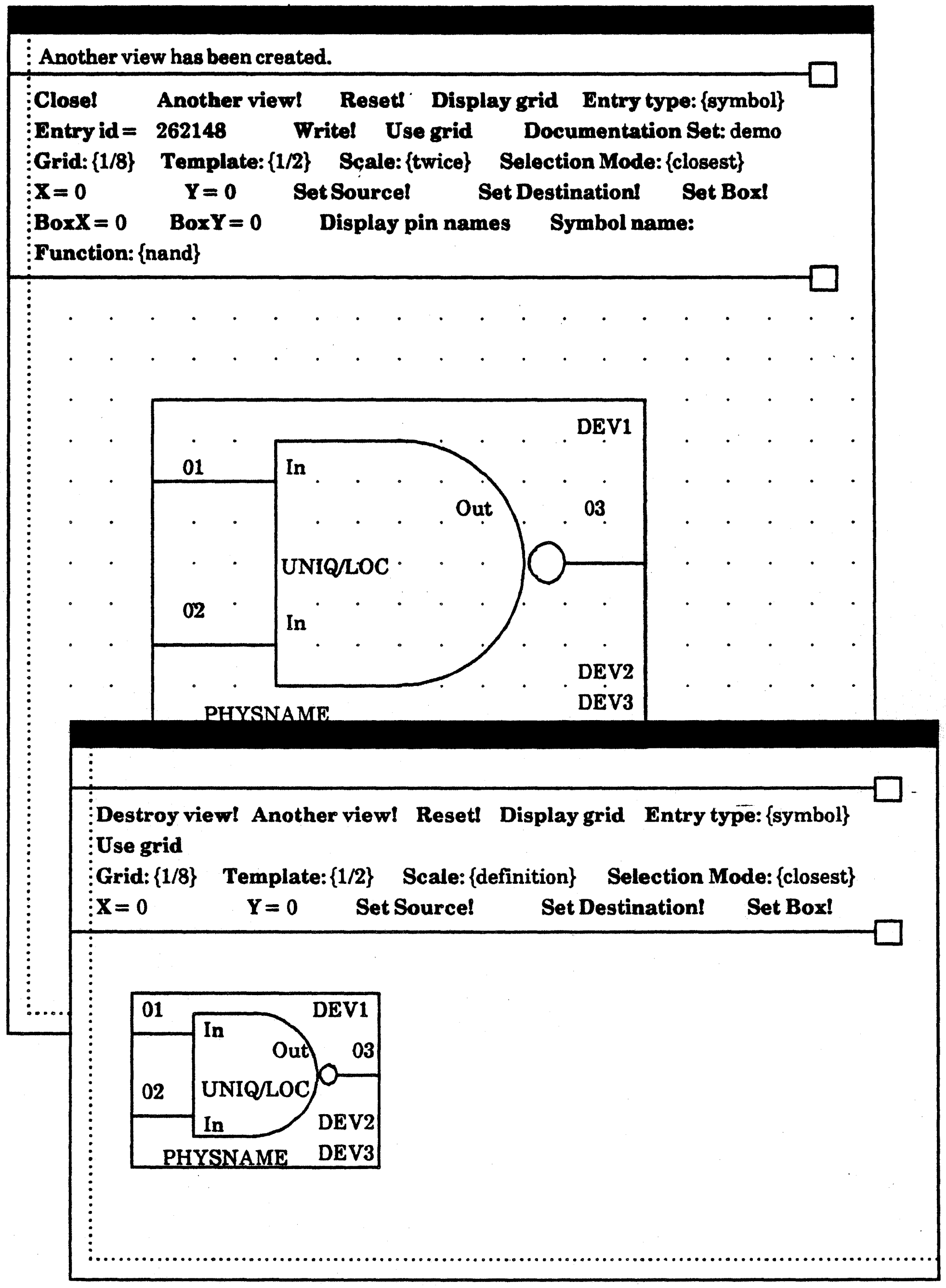

Another view has been created.
Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {twice} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {nand}
DEV1
01
In
Out
03
UNIQ/LOC
02
In
DEV2
DEV3
PHYSNAME
Destroy view! Another view! Reset! Display grid Entry type: {symbol}
Use grid
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
01
DEV1
In
Out
03
02
UNIQ/LOC
In
DEV2
PHYSNAME
DEV3

Figure 2.3

2.5 Selection

2.5.1 The Source and Destination Markers

You select grid points at which to place logic symbol items with source and destination markers. Figure 2.4 shows source and destination markers.

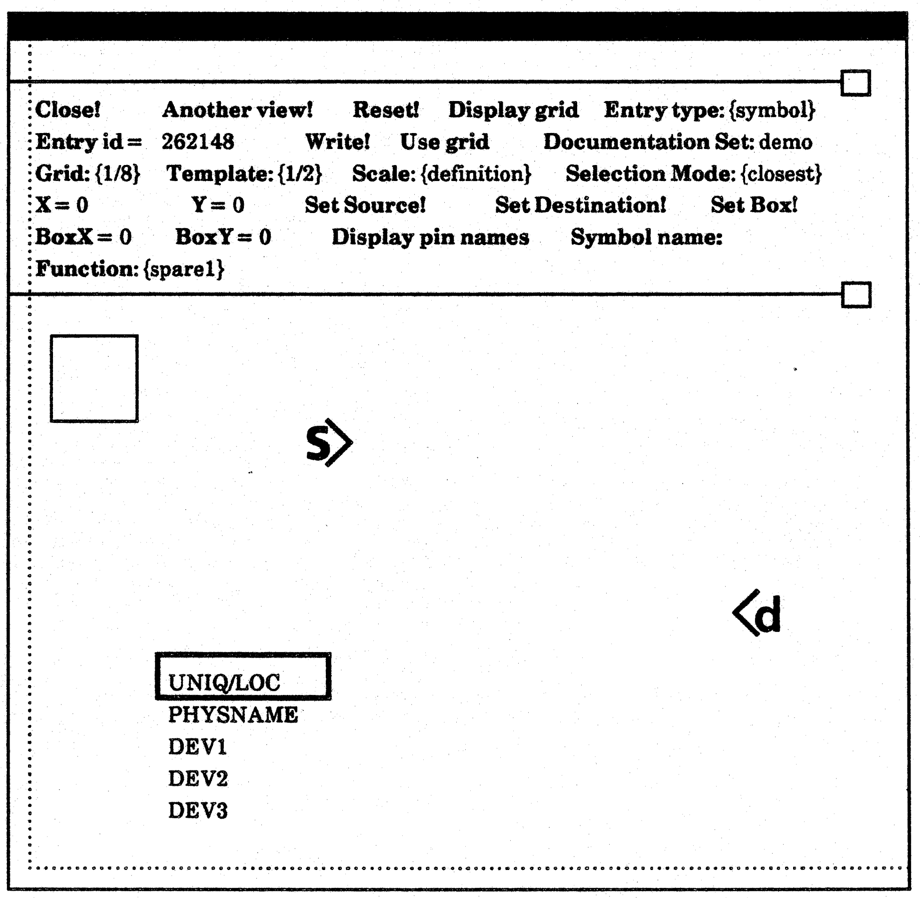


Figure 2.4

There are two ways to select source and destination markers inside the third subwindow. Using the first method, you point the cursor at a point in the subwindow and press the left mouse button to set the source marker. Point the cursor to a point in the subwindow and press the right mouse button to set the destination.

The second method lets you place the source and destination markers at specific points in the subwindow. The values of "X=" and "Y=" in the command subwindow show the source marker's coordinates on the grid. Notice that, as you select different points on the grid, the values of X and Y change. You can change values of X and Y, and then instruct the system to move the source or destination marker to the point corresponding to those X and Y coordinates. Figure 2.5 shows the subwindow's coordinate system.

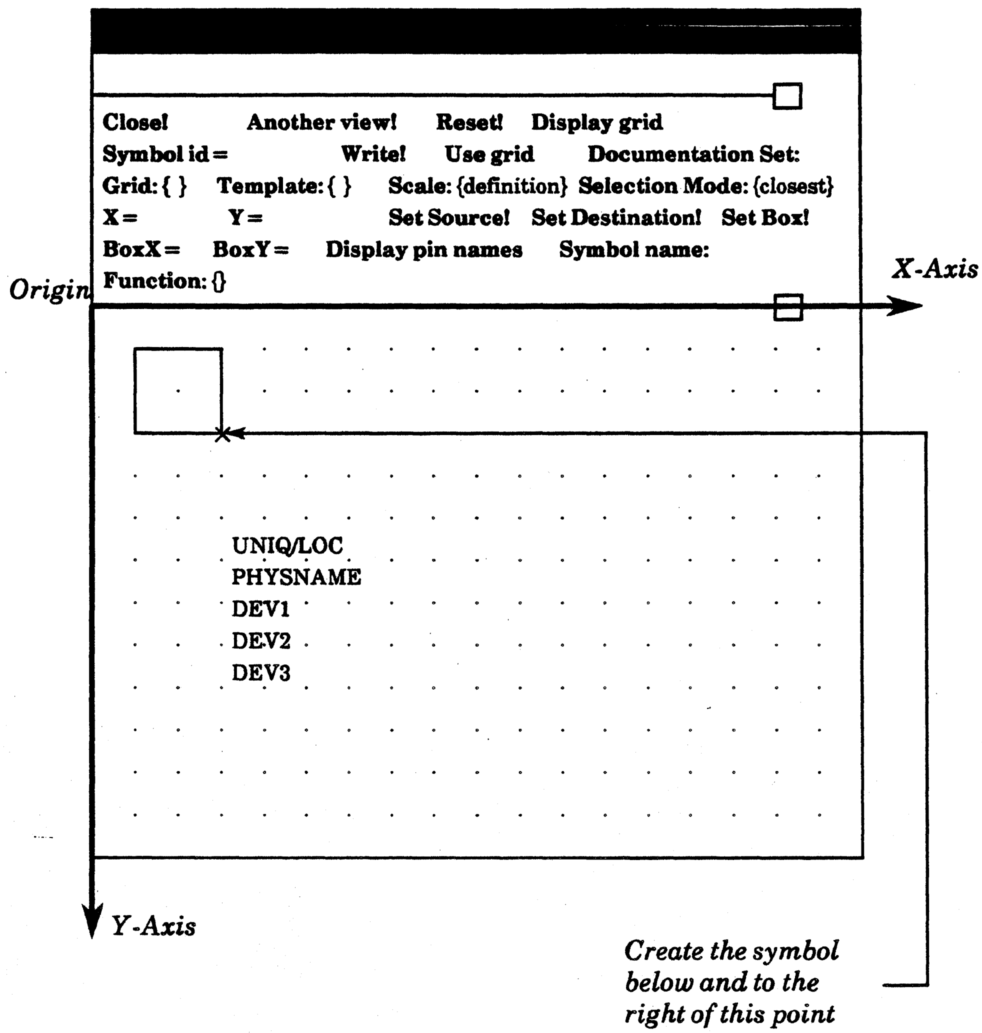


Figure 2.5

- To move the source marker to the point defined by the X and Y coordinates:

1) Select the character "X." The system selects the number following the "X."
2) Press **DELETE** to erase the number.
3) Type the x-coordinate of the grid point you want to select.
4) Select the character "Y." The system selects the number following the "Y."
5) Press **DELETE** to erase the number.
6) Type the y-coordinate of the grid point you want to select.
7) Select the **Set Source!** command in the command subwindow.

- To move the destination marker to the point defined by the X and Y coordinates:

1) Select the character "**X.**" The system selects the number following the "**X.**"
2) Press **DELETE** to erase the number.
3) Type the x-coordinate of the grid point you want to select.
4) Select the character "**Y.**" The system selects the number following the "**Y.**"
5) Press **DELETE** to erase the number.
6) Type the y-coordinate of the grid point you want to select.
7) Select **Set Destination!**

2.5.2 Using the Grid

You draw symbols more accurately when you align all of the items that make up the symbol with grid points. The **Use grid** command in the command subwindow restricts you to selecting only grid points in the third subwindow. The command acts as a toggle switch. As long as it is selected (video-inverted) the grid feature is on. Select the commands a second time to turn off the grid feature.

2.5.3 Selection Mode

Each time you select a grid point with the source marker, the system automatically highlights the closest symbol item. The selection mode is set to **closest**.

You can change the selection mode so that you select just one type of item, for example pin names. You often place pin names close to other symbol items (pin leads, pin numbers, perhaps an arc or a line). When you want to move a pin name, it is easier to select it if you change the selection mode to **pin name**.

- To change the selection mode:

1) Point the cursor at any character in the words "**Selection mode:**" in the command subwindow.
2) Activate the Selection Mode Menu. Choose a selection mode value from the options in that menu. Selection mode options are: **closest, arc, line, pin lead, text, logic uniqueness text, physical name text, devl text, dev2 text, dev3 text, pin name text, pin number text, random text**. The system displays the option you selected inside the brackets following the words "**Selection mode:**."

2.5.4 Selecting Several Items at Once

You may want to select several symbol items at once, for instance to move a group of items while maintaining their relationship to each other. There are two ways to select several items at once. You can select individual items, or select all items within a rectangular region.

- To select several symbol items individually:

1) Hold down the **SHIFT** key, and select each item with left mouse button. The system highlights all of the items that you select.

- To select all items in a rectangular area:

1) Mark the upper-left corner of the area with the source marker.
2) Mark the lower-right corner of the area with the destination marker.
3) Bring up the Symbol Menu (associated with the grid subwindow), and select **Selection By Box**.

2.6 Drawing the Symbol

You create the graphic description of the symbol from several graphic items as well as from several text items. Symbol descriptions consist of AND arcs, OR arcs, pin leads, lines, random text, special text markers, and a bounding box that surrounds the symbol. You create these items with commands in the Symbol Menu. The Symbol Menu is associated with the Library Editor's third subwindow.

2.6.1 Drawing an AND Arc

- To create an AND arc:

1) Mark the endpoints of the arc with the source and destination markers. The system draws the arc counter-clockwise from the source marker to the destination marker. The distance between the source and destination markers must be centered at a grid point. See figure 2.6.
2) Select **Create AND Arc** in the Symbol Menu. (If the distance between the source and destination markers is not centered at a grid point, the system moves the source and destination markers so that it is. Select **Create AND Arc** once more). The system draws the AND arc. See figure 2.7.

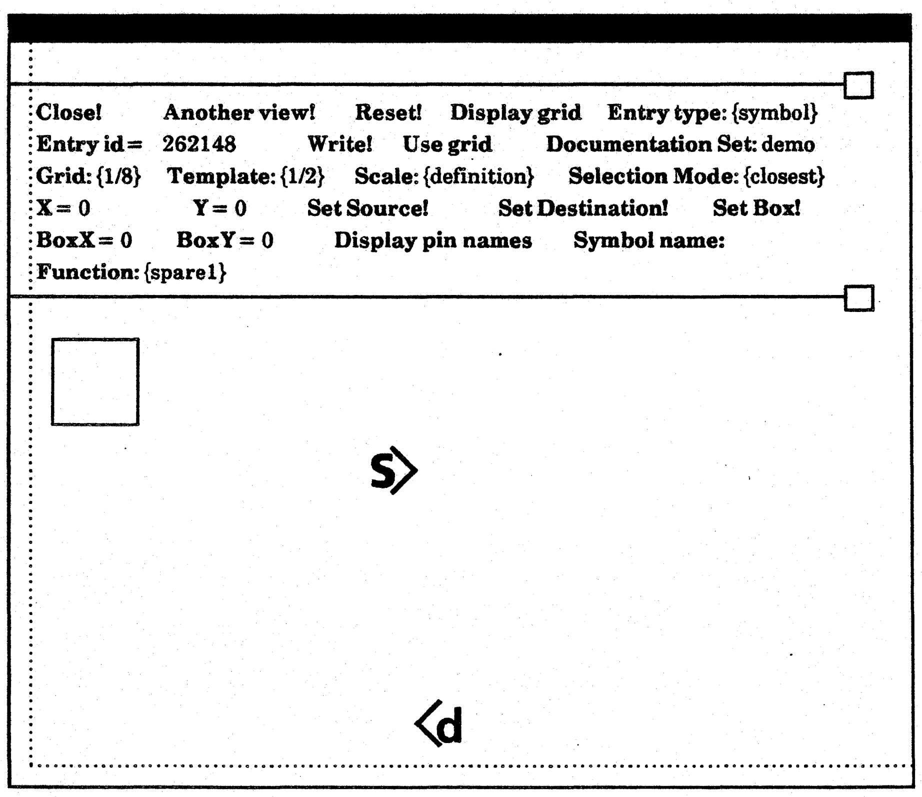


Figure 2.6

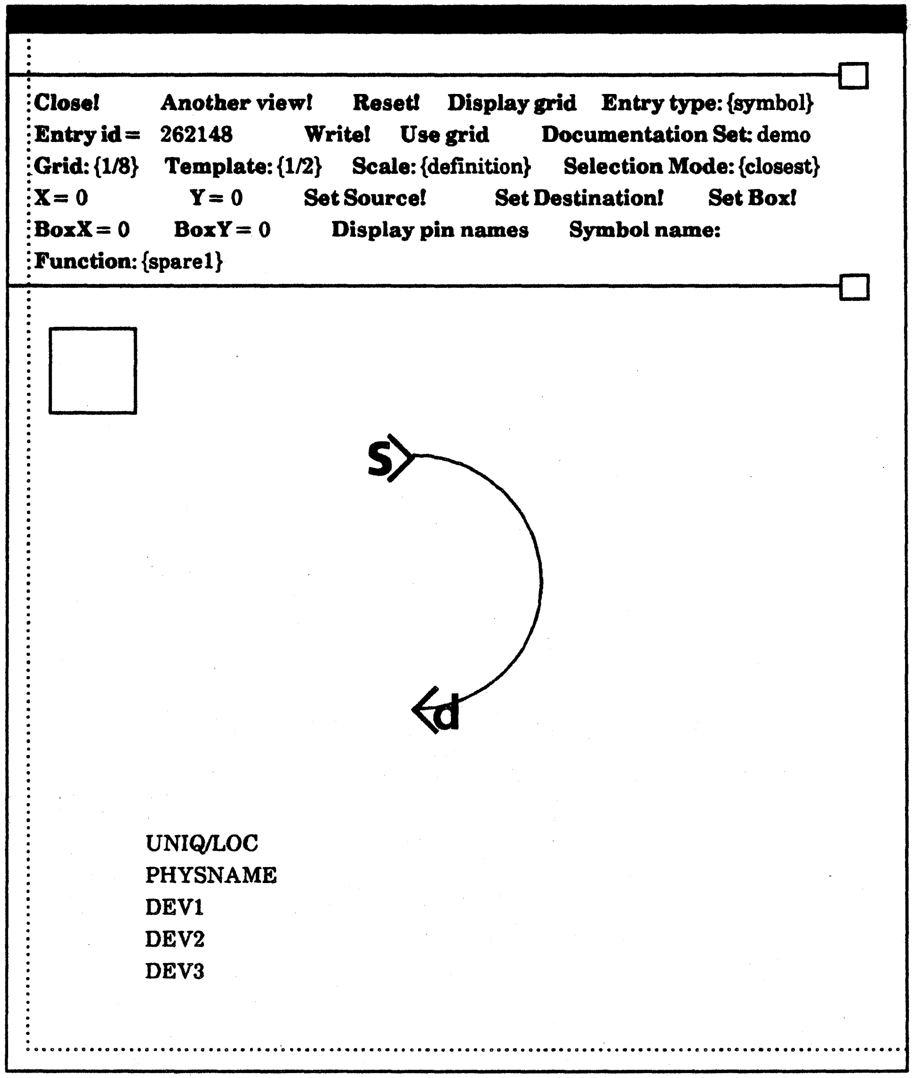

Close!
Another view!
Reset!
Display grid
Entry type: {symbol}
Entry id = 262148
Write!
Use grid
Documentation Set: demo
Grid: {1/8}
Template: {1/2}
Scale: {definition}
Selection Mode: {closest}
X = 0
Y = 0
Set Source!
Set Destination!
Set Box!
BoxX = 0
BoxY = 0
Display pin names
Symbol name:
Function: {spare1}
s
d
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.7

2.6.2 Drawing an OR Arc

- **To create an OR arc:**

1) Mark the endpoints of the arc with the source and destination markers. The system draws the arc counter-clockwise from the source marker to the destination marker. The source and destination markers must be centered at a grid point. See figure 2.8.
2) Select **Create OR Arc** in the Symbol Menu. The system draws the OR arc. See figure 2.9.

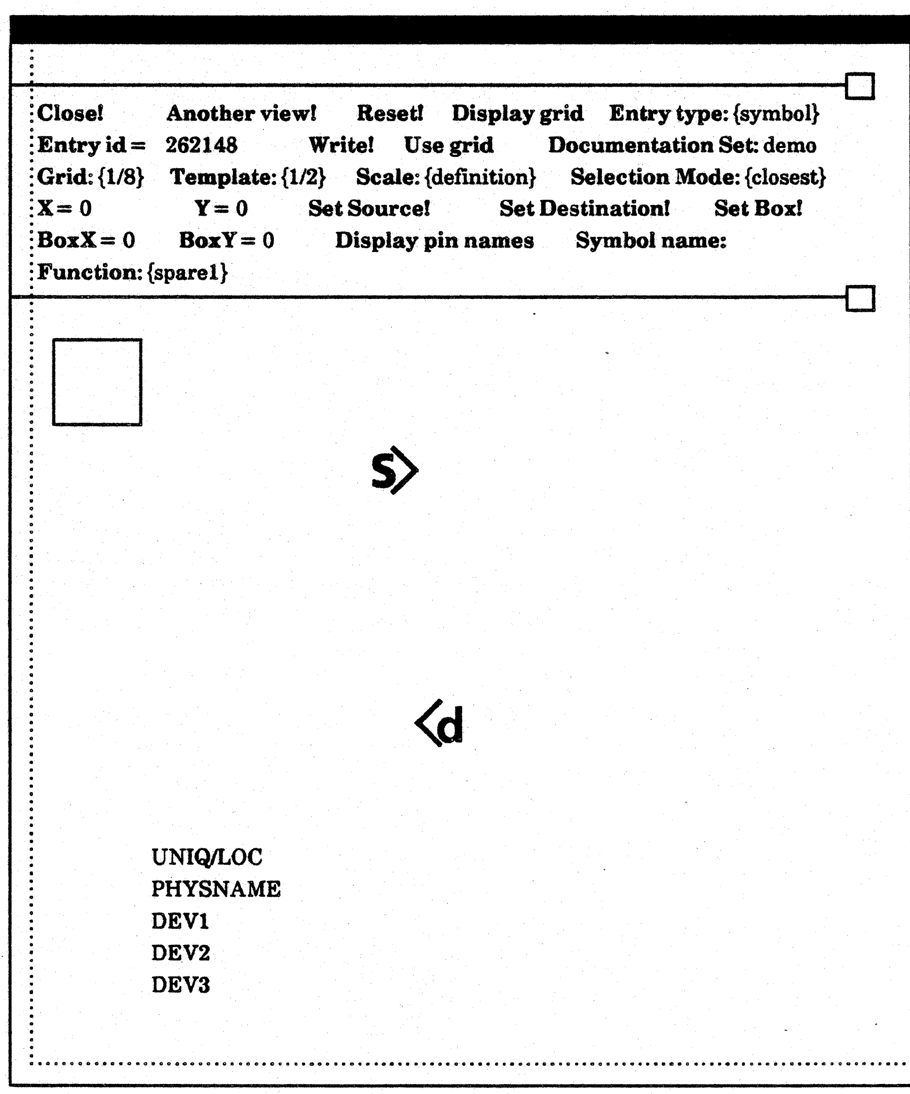


Figure 2.8

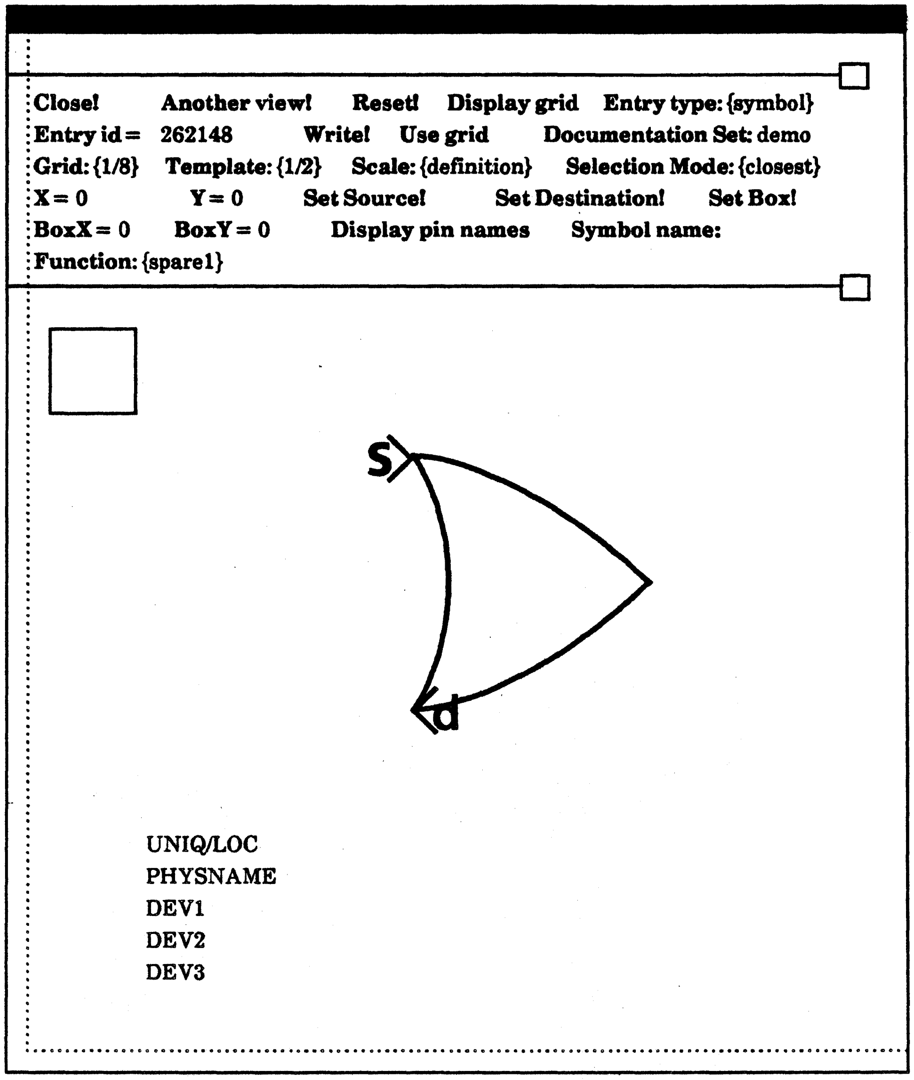

Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {spare1}
s
d
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.9

Selecting an OR Arc

An OR arc is really three separate arcs as shown in figure 2.10. Once you have created an OR arc you can select and edit the three arcs individually.

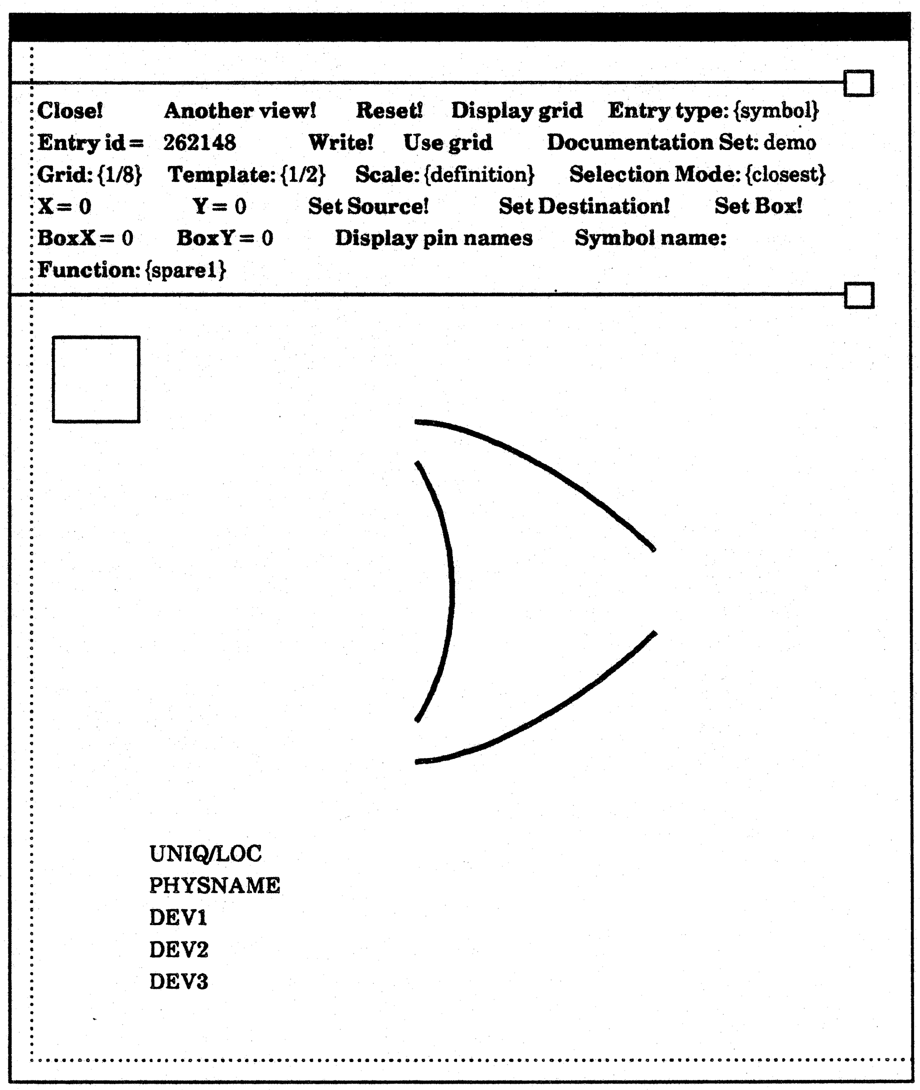


Figure 2.10

2.6.3 Drawing a Line

- To create a line:

1) **Mark the line's endpoints with the source and destination markers.**
2) **Select Create Line in the Symbol Menu. If you want the line to be orthogonal (horizontal or vertical), select Create Orthogonal Line.**

Figure 2.11 shows a nonorthogonal line. Figure 2.12 shows an orthogonal line.

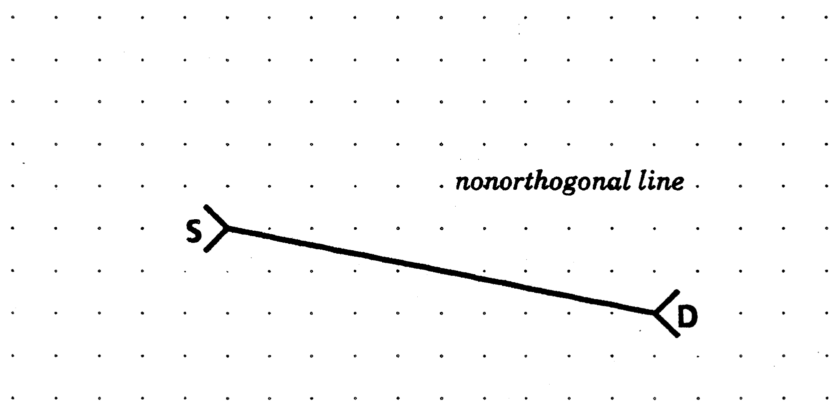


Figure 2.11

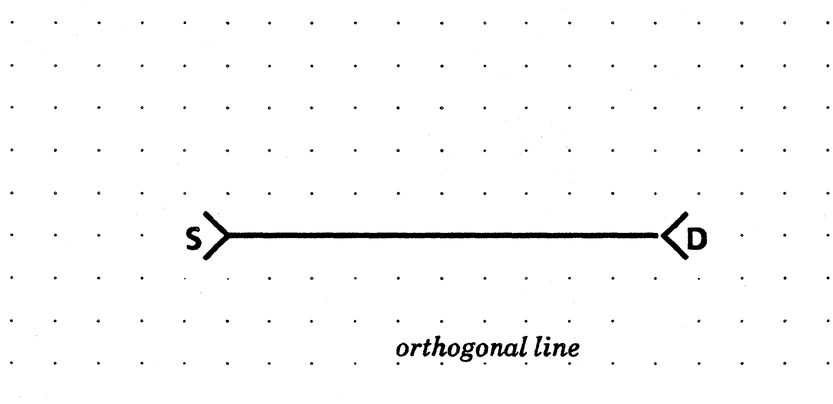


Figure 2.12

Line Properties

The system shows you the coordinates of a line's endpoints in the line's property sheet. The values of **fromX** and **fromY** are the x- and y-coordinates of the line's first endpoint, the point that you marked with the source marker. The values of **toX** and **toY** are the x- and y-coordinates of the second endpoint, the point that you marked with the destination marker.

You can change the line's length and its position on the grid by changing the coordinates of its endpoints in the property sheet. When you apply the line properties, the system changes the line.

2.6.4 Drawing Pin Leads

- To create a pin lead:

1) Mark the endpoints of the lead with the source and destination markers. The source marker marks the pin's *connection point*. The connection point is the end of the pin lead that connects to a net in a schematic. The source marker must be on a grid point. See figure 2.13.
2) Select **Create Pin Lead** in the Symbol Menu.

The system draws the pin lead with the pin number and pin name. See figure 2.14. To change the position of the number or name select it and move it.

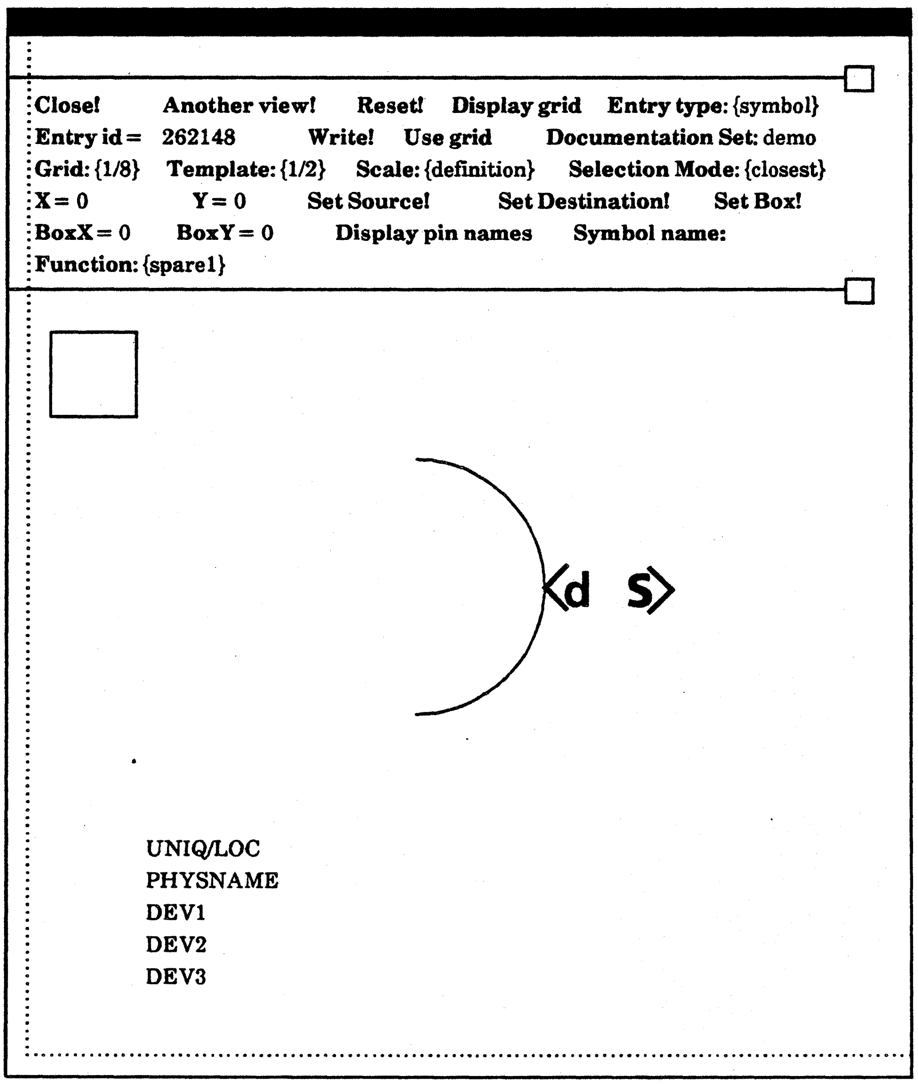


Figure 2.13

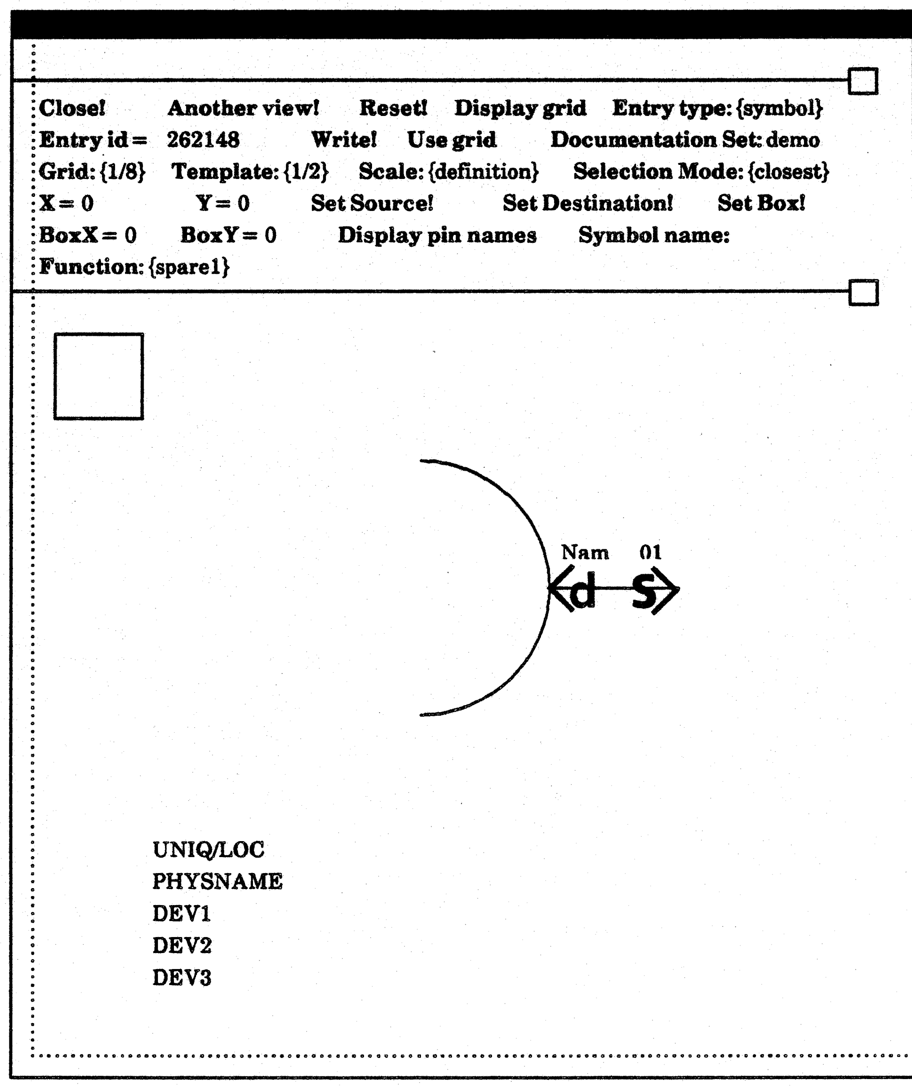

Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {spare1}
Nam 01
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.14

Pin Lead Properties

Pin lead properties define several of the symbol's electrical characteristics. For instance, you change the pin's name and number in the property sheet,

- To display the pin lead's property sheet.

1) Select the pin lead.
2) Press the **PROP'S** key.

Figure 2.15 shows a pin lead property sheet.

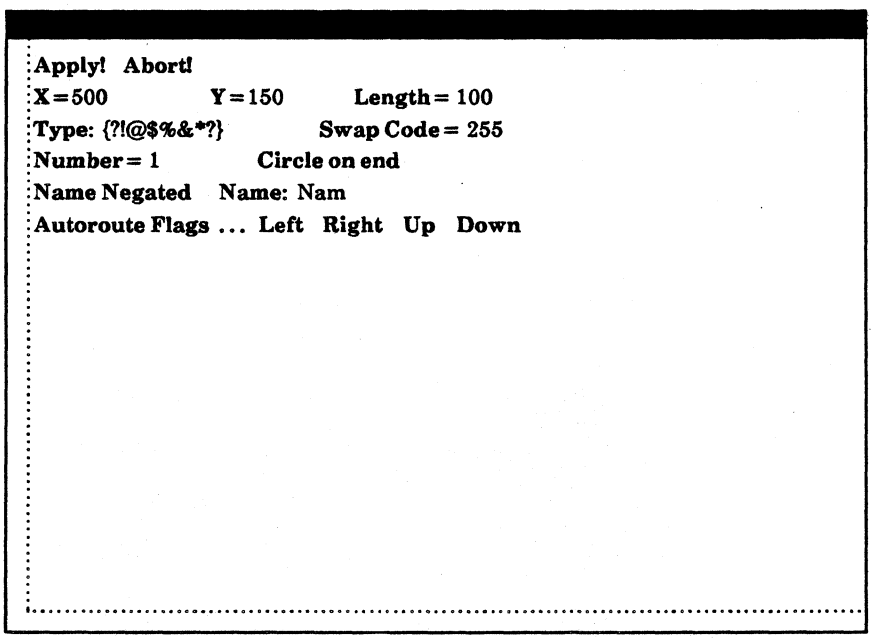


Figure 2.15

Connection Point Coordinates
The values of the **X** and **Y** numeric properties in the pin lead's property sheet are the coordinates of the pin lead's connection point. You can move the pin lead on the grid by changing these properties, and applying the property sheet.

Length
The value of the **Length** property is the length of the pin lead on the grid. You can change the length of the pin lead by changing the value of the length property, and applying the property sheet.

Type
You must assign each pin lead a type. Notice that when you first open the property sheet, after creating the pin lead, the type value is nonsensical. This is to show you that you have not assigned a type to that pin.

- To assign a pin type:

1) Point the cursor at the word "**Type**" in the pin lead property sheet.
2) Bring up the Type Menu.
3) Select a type from the options in the menu. The type options are: **unspecified, input, output, duplex, power**, and **ground**. The system displays the new type in the property sheet.

Swap Code
Some symbols have equivalent pins. For instance, in an integrated circuit containing several functions, two or more pins might perform the same function. Equivalent pins have the same pin type and *swap code*. Legal swap code values range from 0 to 255.

- To assign a swap code:

1) Point the cursor at the words "**Swap Code.**"
2) Press the left mouse button. The system places a flashing caret after the equal sign.
3) Type the swap code number.

Number
The system automatically numbers pins. You can change the pin numbering by changing the pin number in the pin lead's property sheet. All pins must have a number.

Circle on End

You negate a pin lead by turning on the boolean property "Circle on end." Once you apply the properties, the system draws a circle on the end of the pin to mark it as negated. The output pin of the NAND gate in figure 2.16 is negated.

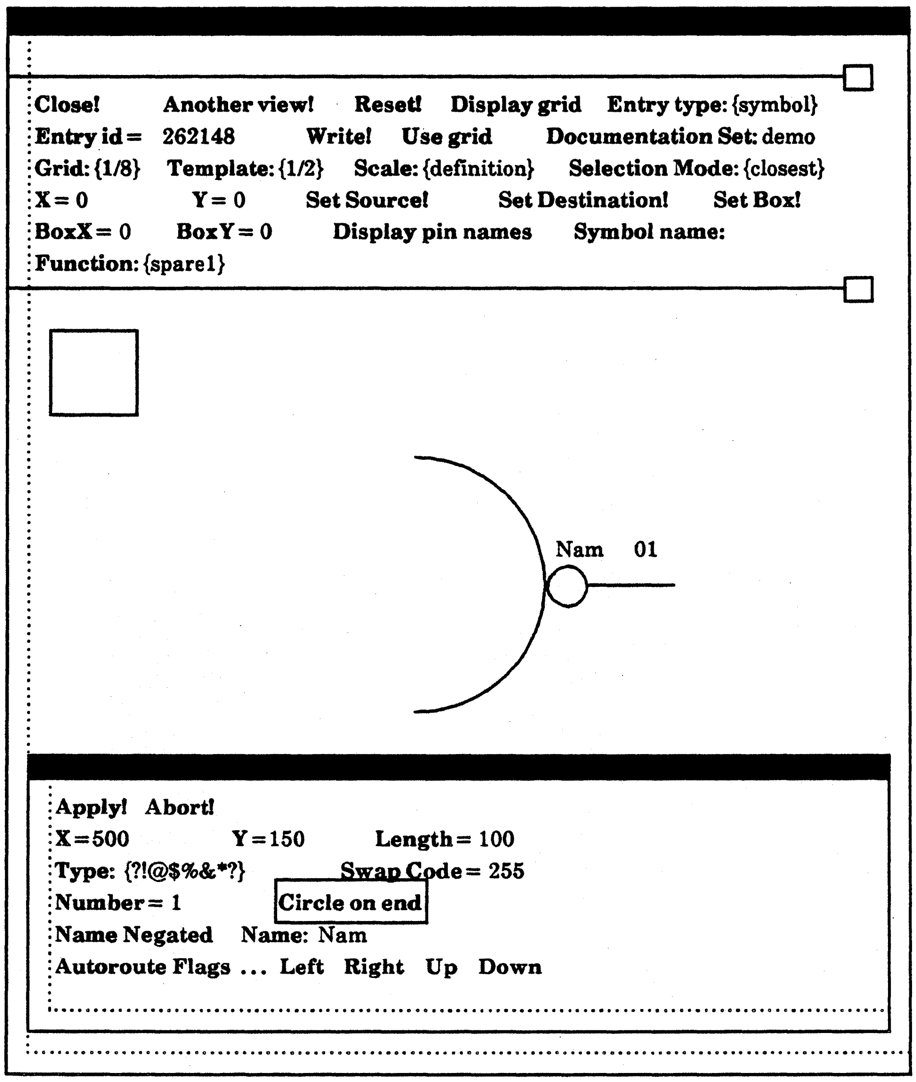


Figure 2.16

Name
When the system draws a pin lead, it marks the location of the pin name text with the letters "nam."

- To change the pin lead name:

1) Select any character in the word "Name." The system selects the word "nam."
2) Press the **DELETE** key.
3) Type the new name.

Name Negated
To negate a pin lead's name, turn on the boolean property "**Name Negated.**" After you apply the properties, the system draws a bar above the pin lead's name. Figure 2.17 shows a pin lead with its name negated.

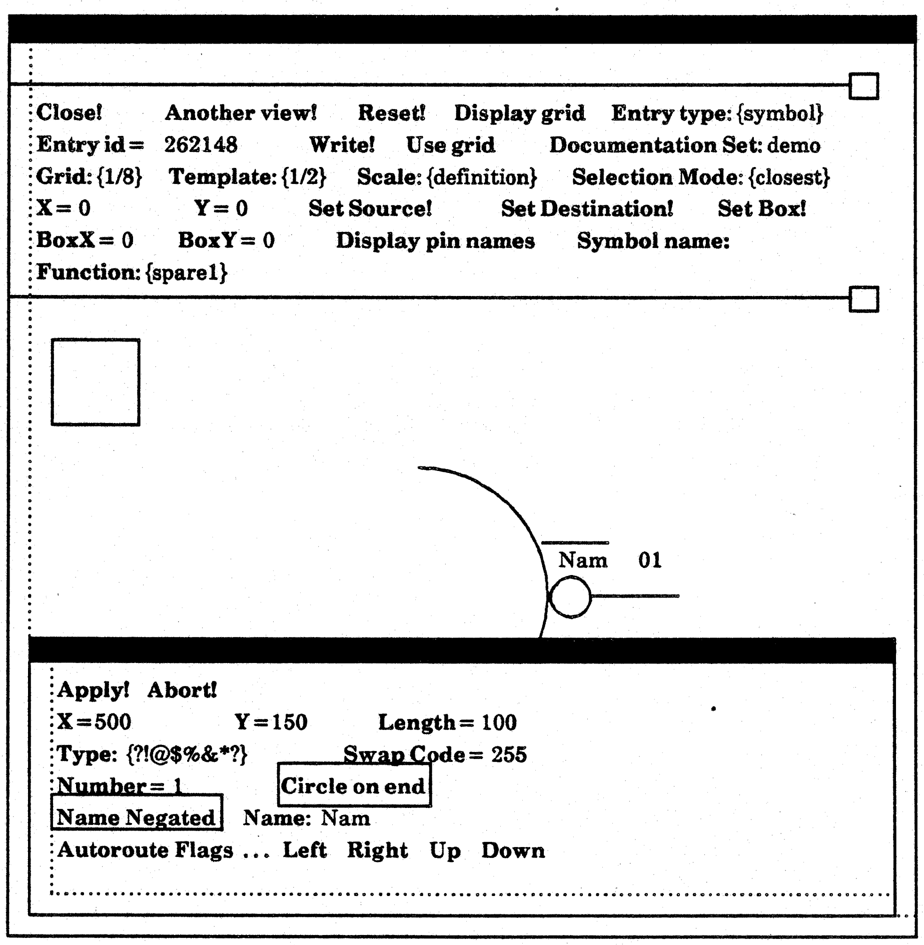


Figure 2.17

Autoroute Flags
Autoroute flags tell the system in which direction to draw connections between symbols on the schematic. You set the autoroute flags in each pin lead's property sheet. You must set the autoroute flags before you can apply the property sheet.

In figure 2.18, the autoroute flags for pin 01 are **Left** and **Up**. A connection that came from below would cross pin 02; one that came from the right would pass through the symbol. The autoroute flags for pin 02 are **Left** and **Down**. The autoroute flags for pin 03 are **Up, Down,** and **Right**.

- To set an autoroute flag:

1) Select **Left, Right, Up,** or **Down**. While the flag is inverted, it is set. Select it again to "unset" it.

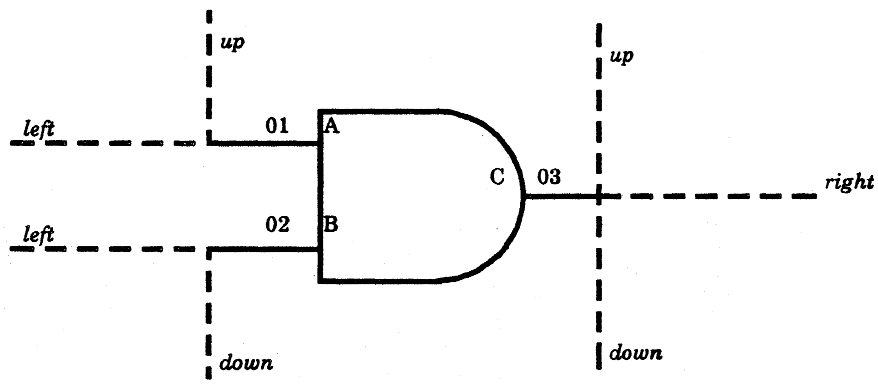


Figure 2.18

2.6.5 Text

You can add random text to the symbol description. For example, you might add text to describe pin functions.

- To add text:

1) Select the grid point at which you want to place the text.
2) Type the text.
3) Select **Finish Text Creation** in the Symbol Menu.

Text Properties

- To display the text's property sheet:

1) Select the text.
2) Press the **PROP'S** key.

Figure 2.19 shows a text property sheet.

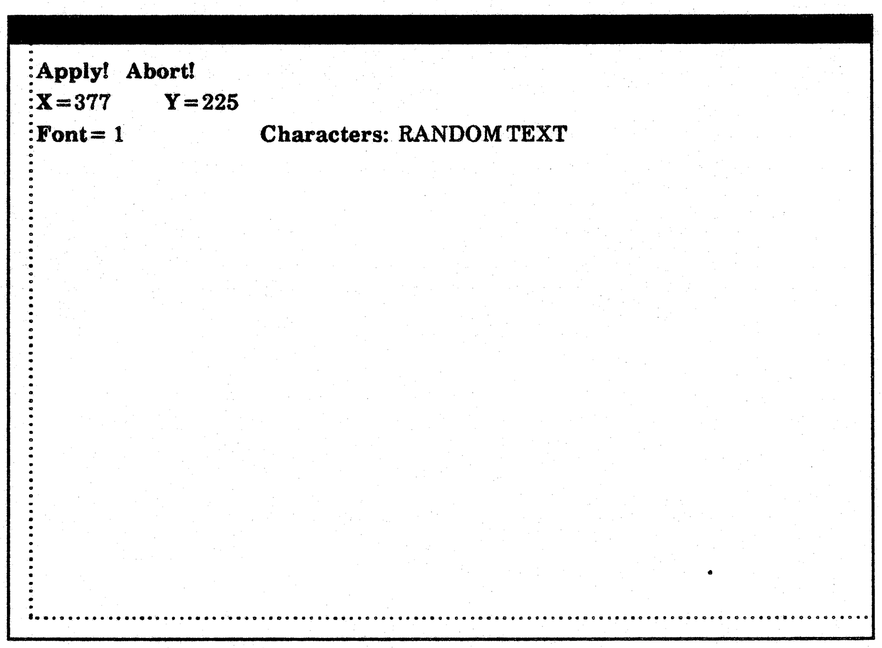


Figure 2.19

Text Coordinates
The values of the numeric properties **X** and **Y** in the text's property sheet show the text's coordinates on the grid. You cannot change these numbers.

Characters
The text property sheet also shows you the characters in the associated body of text.

- To change the characters:

1) Use the text editing techniques to change the text following the colon.
2) Apply the properties. The system changes the text to match that in the property sheet.

Font
You must assign all text items a font number. Font numbers range from 1 to 16. A format designer assigns font number descriptions on the format page of each schematic. Section 2.7.5 describes fonts in more detail.

2.6.6 The Bounding Box

When you select a logic symbol in the schematic, the system video-inverts a rectangular area surrounding the symbol as well as the symbol itself. You use the box in the upper-left corner of the subwindow to define that selection box. The box's upper-left corner is anchored at the point (40,40) on the grid. You set the size of the box by selecting the lower-right corner. Before you write the symbol to the library, its graphic description must be inside the bounding box.

- To set the size of the bounding box:

1) Mark the point at which you want the lower-right corner of the selection box with the source marker. The corner of the box must be on a grid point.
2) Select the **Set box!** command in the command subwindow. The system stretches the box to the corner you marked.

- You can also set the exact coordinates for the lower-right corner of the box.

1) Type the x-coordinate after the equal sign in the **BoxX** field in the command subwindow.
2) Type the y-coordinate after the equal sign in the **BoxY** field in the command subwindow.
3) Select the **Set box!** command in the command subwindow. All of the symbol items should be within the box.

2.6.7 Symbol Property Location Markers

You see five lines of text in the Library Editor's third subwindow. Each marks the location of a symbol property when the system displays it on the schematic. The first line reads, "UNIQ/LOC"; it is the *logic uniqueness text*. It marks the relationship of the symbol's symbol and physical designators to the other parts of the symbol. When you instruct the system to display symbol or physical designators on the schematic, it displays each symbol's designators at the location marked by the logic uniqueness text.

The second line of text reads, "PHYSNAME." It is the symbol's *physical name text*; it marks the location at which the system places the symbol's physical name, when instructed to show that name in the schematic.

The remaining three lines, "DEV1," "DEV2," and "DEV3," show the location of device parameters when displayed in the schematic.

You cannot delete these location markers from the symbol subwindow. Move each word so that it is inside the bounding box, and its location marks the point at which you want the system to display the symbol property it represents on the schematic.

- To move a location marker:

1) Select the marker.
2) Mark the point to which you want to move it with the destination marker.
3) Press the **MOVE** key.

2.7 Editing Symbol Items

Once you have created symbol items you can use the editing functions to change the physical appearance of the symbol. You can *delete* symbol items, *paste* them back into the symbol design, *move* items to different points on the grid, and *copy* items.

In general, you select the item that you want to delete, move, or copy, mark the destination for a move or a copy, and press the function key. If you want to perform an editing function with several symbol items, for instance to delete several parts of the symbol, you select all the items and then press **DELETE**.

It is easier to move or copy items to grid points if you select the item and select the **Move Source to End** command in the Symbol Menu before moving or copying the item. The **Move Source to End** command moves the source marker to the end of the item, usually a grid point. The **Move Destination to End** command moves the destination to the closest end of the closest item. Use the **Move Destination to End** command when you want to move or copy one item to the endpoint of another. Turn on the **Use Grid** command in the command subwindow to assure that symbols items are on grid points.

2.7.1 Delete

- To delete a symbol item:

1) Select the symbol item.
2) Press the function key marked **DELETE**.

You cannot delete the physical name text, logic uniqueness text, and device parameters; they are a necessary part of the symbol. You cannot delete pin number or pin name text without deleting the associated pin lead. When you delete a pin lead, the system also deletes both the pin number and pin name text.

- To delete several symbol items:

1) Use the **SHIFT** key or the **Selection By Box** command in the Symbol Menu to select all the items you want to delete.
2) Press **DELETE**.

2.7.2 Paste

- To *paste* the last item you deleted back into the symbol design:

1) Press the key marked **SAME**.

2.7.3 Move

- To move a symbol item:

1) Select the symbol item.
2) Mark the point to which you want to move it with the destination marker. The system moves the source marker to the destination location, and moves the item so that it retains its relationship with the source marker.
3) Press the function key marked **MOVE**.

The system moves the source marker to the destination and the destination marker to the source marker's position before the move so that you can easily move the symbol back to its former location.

- To move several items:

1) Select all the items you want to move.
2) Mark the point to which you want to move them with the destination marker.
2) Press **MOVE**.

2.7.4 Copy

- To copy a symbol item:

1) Select the symbol item.
2) Mark the point to which you want to place the copy with the destination marker. The system moves the source marker to the destination location, and copies the item so that it retains its relationship with the source marker.
3) Press the function key marked **COPY**.

- To copy several items at once:

1) Select all the items you want to copy.
2) Mark the point to which you want to copy them with the destination marker.
2) Press **COPY**.

2.8 Symbol Characteristics

In addition to its graphic description, the symbol has several characteristics that describe it. You define these characteristics in the command subwindow and in property sheets.

2.8.1 Entry Identification Number

If you are creating a new symbol, you must enter the entry identification number for that symbol. Change the entry identification number in the command subwindow.

- To change the entry identification number:

1) Point the cursor at the words "**Entry id =**" in the command subwindow. Press the right mouse button. The system deletes the number following the words "**Entry id =**."
2) Type the new number.

2.8.2 Displaying Pin Names

You can stipulate when the system displays the symbol's pin names on the schematic. An engineer has three options for displaying symbol pin names in the schematic. He or she may display pin names for all symbols, for none, or for *monoliths*. By turning on the **Display pin names** command in the Library Editor command subwindow, you can instruct the system to treat the symbol as a monolith.

- To label the symbol as a monolith:

1) Point the cursor at the words "**Display pin names**" in the command subwindow.
2) Press the left mouse button. The system video-inverts the words.

2.8.3 Symbol Name

Each symbol must have a name that corresponds to its identification number. That name should be unique.

- To name the symbol if the name is blank:

1) Point the cursor at the words "**Symbol Name.**"
2) Press the left mouse button.
3) The system marks a type-in point with a caret after the words "**Symbol Name.**" Type the name.

- If you want to change the symbol name:

1) Point the cursor at the words "**Symbol Name.**"
2) Press the right mouse button. The system deletes the entire name.
3) Type the new name.

2.8.4 Symbol Function

You must also assign the symbol a logic function.

- To change the symbol's function:

1) Position the cursor over the word "**Function:**" in the command subwindow.
2) Bring up the Function Menu.
3) Select a function from the options in the menu. The function options are: **and, connectivity symbol, inverter, JK flip-flop, memory, nand, nor, or, physical connector, xor, spare 1, spare 2, spare 3, spare 4, capacitor, coil, diode, relay, resistor,** and **transistor**.

Naming Connecting Nets

When you label a symbol as a connectivity symbol, the system gives you the option of automatically naming the nets that connect to that symbol. The system displays the **Name for connecting nets** command in the command subwindow. If you choose to enter a name for the nets that connect to the symbol, the system automatically gives that name to connecting nets . Power and ground symbols, for instance, are connectivity symbols that automatically name the nets that connect to them.

- To automatically name connecting nets:

1) Point the cursor at the words "**Name for connecting nets**" in the ~~command subwindow. Press the left mouse button. The system~~ marks the type-in point with a caret following "**Name for connecting nets.**"
2) Type the name.

2.8.5 Assigning Fonts to Text Items

You assign a font number to each text item, including symbol property location markers, pin names, pin numbers, and random text. Font numbers range from 1 to 16. A format designer assigns each font number a family and height in each schematic. For example, if you want the system to display pin number text the same as pin name text, assign the same font number to all pin names and pin numbers.

A new symbol has default font settings. The default for the logic uniqueness text is font 2. The default for physical name text is font 3. The default for device parameter text is font 4. Pin numbers default to font 5, pin names to font 6, and random text to font 7.

- To assign a font number to a text item:

1) Select the text item.
2) Press the **PROP'S** key to display the text item's property sheet.
3) Delete the current font value and replace it with the value you want to assign.
4) Select **Apply!**.

2.9 Writing the Symbol to the Library

Once you have completed the symbol, you must instruct the system to write it to the library.

- To write a symbol to the library:

1) Select **Write!** in the command subwindow.

2.10 The Abbreviation Map

After writing a symbol to the library, you enter the symbol's identification number and its nicknames in the Abbreviation Map.

- To open the Abbreviation Map window:

1) Move the cursor to an empty area on the display.
2) Bring up the Inactive Menu.
3) Select **Abbreviation Map**.

The Abbreviation Map window has a system message subwindow, a command subwindow, and a third subwindow in which it lists symbol identification numbers and nicknames. Figure 2.20 shows the Abbreviation Map window.

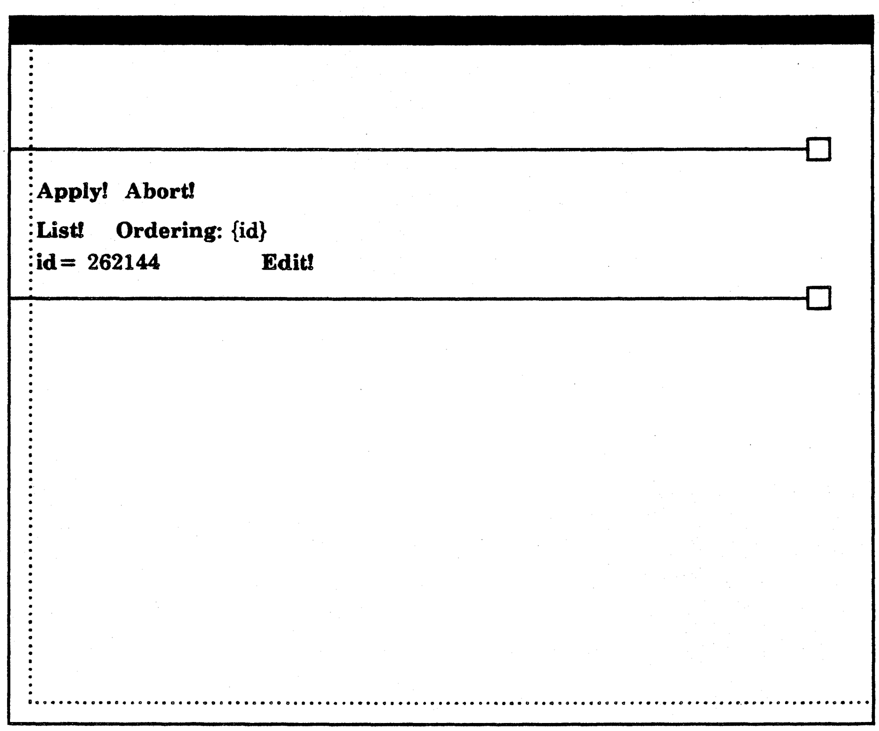


Figure 2.20

2.10.1 Listing Abbreviation Map Entries

You can list all entries in the Abbreviation Map before adding a symbol to ensure that you don't duplicate Abbreviation Map nicknames.

- To display a list of symbols in the abbreviation map in the order of their identification numbers:

1) Select the **List!** command in the command subwindow.

Figure 2.21 shows Abreviation Map entries listed by identification numbers.

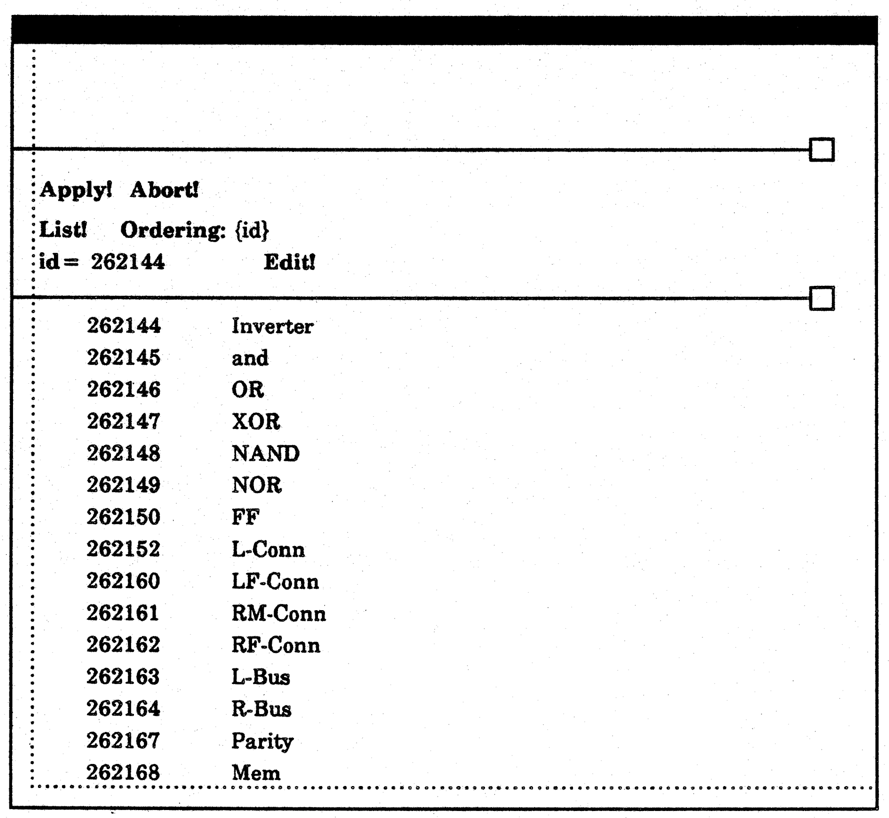


Figure 2.21

- To list symbols alphabetically by nicknames:

1) Point the cursor at the word "**Ordering**" in the command subwindow. Activate the Ordering Menu.
2) Select **nickname**.
3) If you want to list only the symbol with a particular nickname, type that nickname after the words "**NicknameListKey**."
4) Select the **List!** command.

Figure 2.22 shows Abreviation Map entries listed alphabetically.

Apply! Abort!

List! Ordering: {nickname} Nickname ListKey:

id= 262144 Edit!

+12V	500147
+5V	500146
-12V	500144
-5V	500143
1-OF-16-SELECTOR/MULTIPLEXER	500068
1-OF-8-SELECTOR/MULTIPLEXER	500067
1024-BIT-ROM	500105
2-TO-1-SELECTOR/MUX-INVERT	500074
2-TO-1-SELECTOR/MUX-NON-INVERT	500072
2-TO-4-LINE DECODER	500057
2114	500130
2116	500156
2118	500129
2147	500128
2164	500127

Figure 2.22

2.10.2 Adding a Symbol to the Abbreviation Map

To add the symbol to the Abbreviation Map:

1) Delete the identification number following the word "id' in the command subwindow.
2) Type the identification number of the symbol you are adding.
3) Select the **Edit!** command in the command subwindow. The system replaces the commands in the command subwindow with Abbreviation Map editing commands.
4) Type the symbol nicknames after the word "**Nickname.**" Separate nicknames by spaces.
5) Select **ApplyEdit!** (Select **AbortEdit!** to forget the new nicknames.) The system replaces the editing commands in the Abbreviation Map subwindow with listing commands.

2.10.3 Closing the Abbreviation Map

- To close the Abbreviation Map:

1) Select **Close!** in the Abbreviation Map's command subwindow.

2.11 Resetting the Library Editor

You *reset* the Library Editor to remove a particular symbol from the Library Editor.

- To reset the Library Editor:

1) Select the **Reset!** command in the command subwindow.

The reset command removes the third subwindow from the Library Editor window. If you haven't written a symbol to the library, selecting **Reset!** destroys the symbol.

2.11 An Example: Building a NAND Gate

To build a NAND gate, you open the Library Editor window, set the entry type to **symbol**, set the grid and template sizes for the symbol, delete the entry identification number, and select the **Read!** command. The system displays a third subwindow containing a grid on which you draw the NAND gate. On the grid, the system displays the selection box, and markers for the symbol designator, physical designator, and device parameters. See figure 2.23.

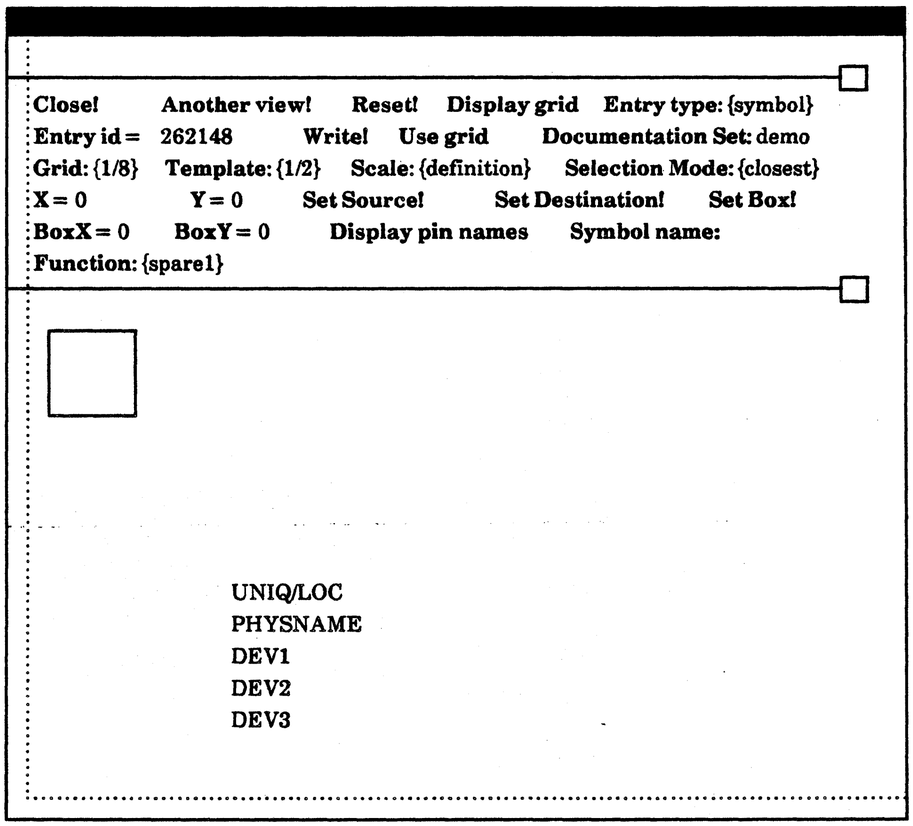


Figure 2.23

You might first define the physical appearance of the NAND gate. You begin by creating an AND arc. You mark the arc's endpoints with the source and destination markers (figure 2.24), and select the dynamic menu command, **Create AND Arc** (figure 2.25).

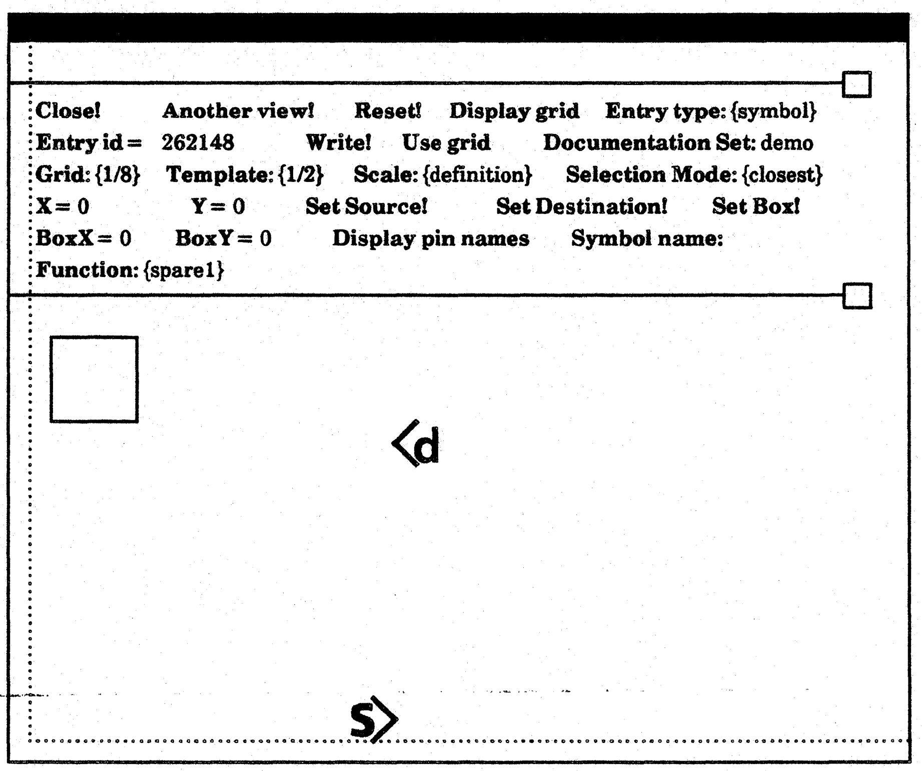


Figure 2.24

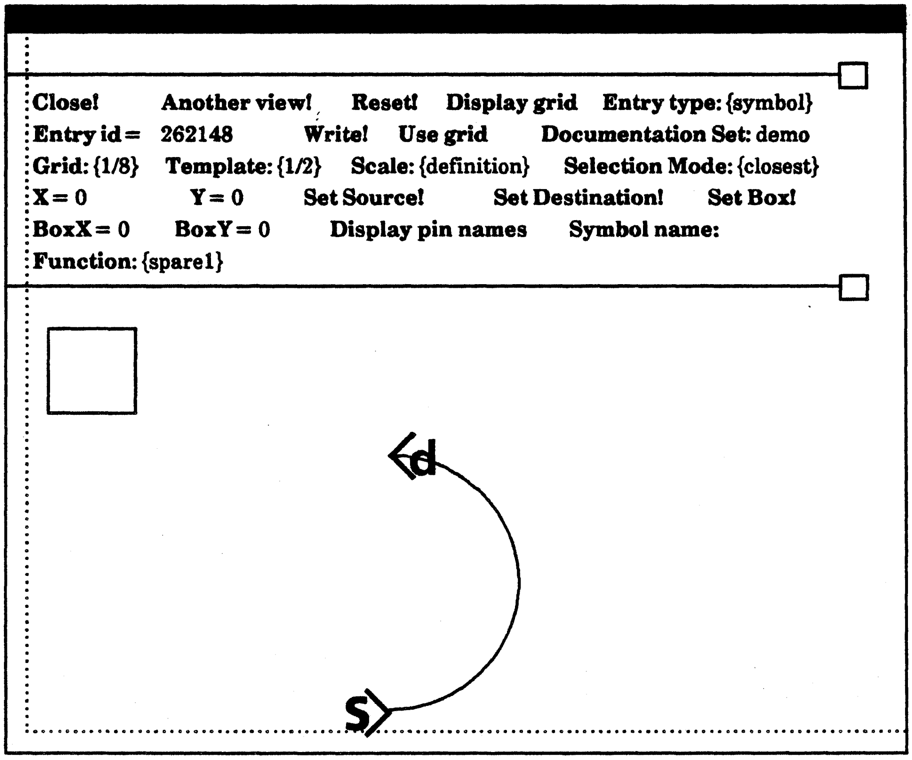

Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {spare1}
d
s

Figure 2.25

Next you create two lines extending the arc to the left. The source marker marks one endpoint of the first line. You move the destination marker to the other endpoint (figure 2.26), and select **Create Orthogonal Line** in the dynamic menu (figure 2.26). Figure 2.27 shows the line.

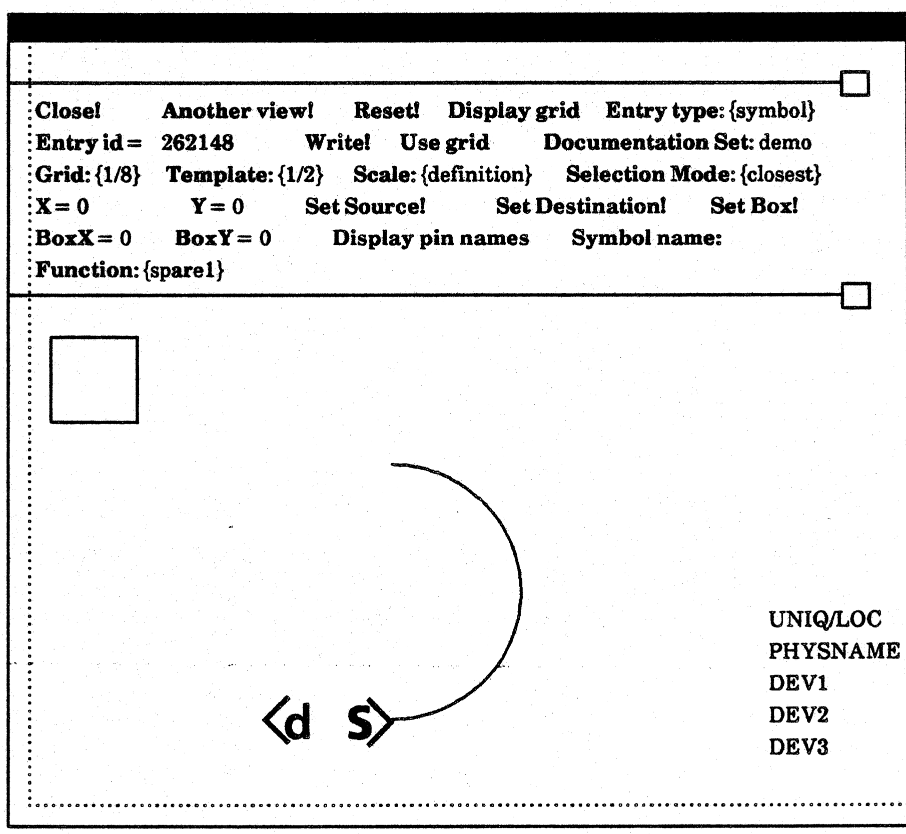


Figure 2.26

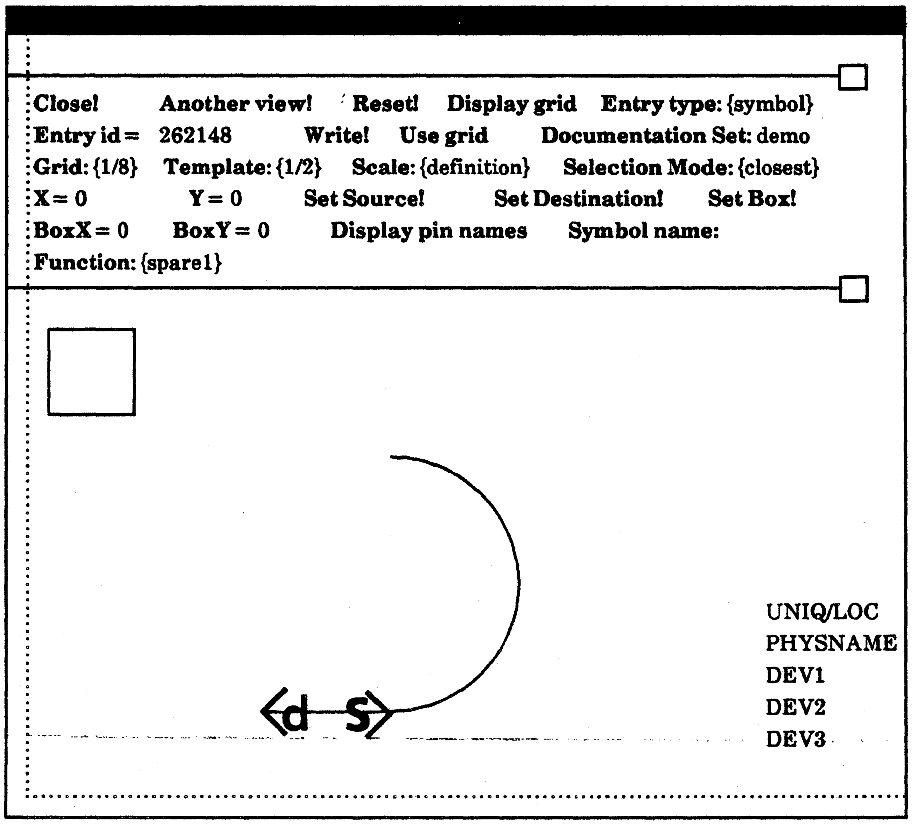

Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {spare1}
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.27

You move the destination marker to the other endpoint (figure 2.28) of the arc, and copy the first line (figure 2.29).

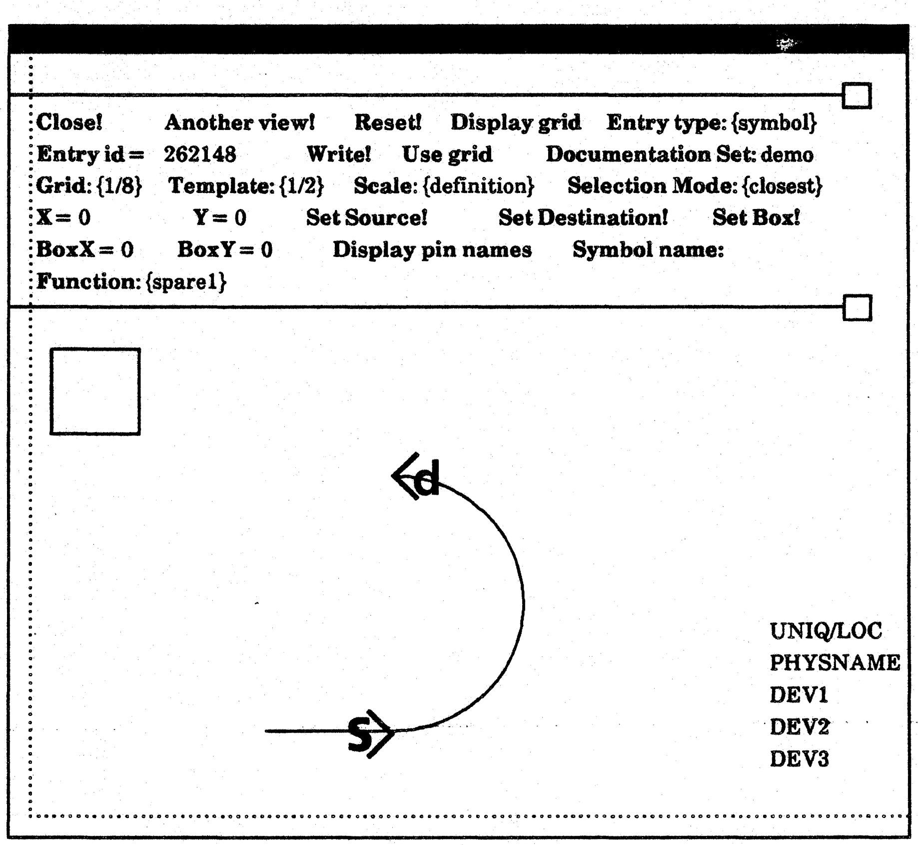


Figure 2.28

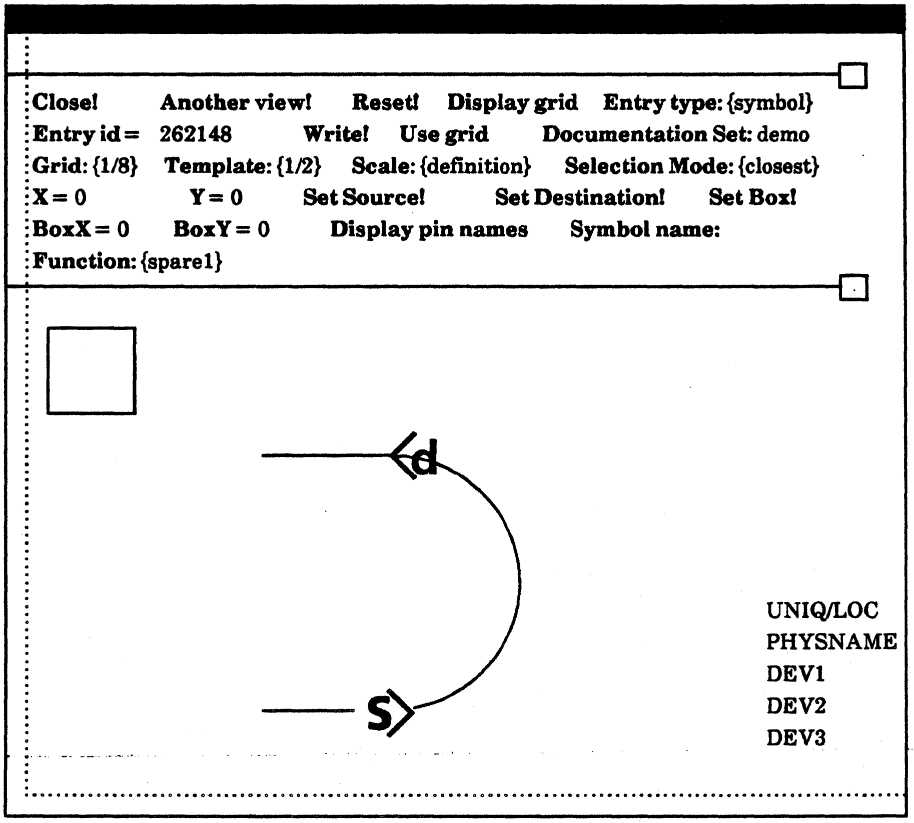

Close!
Another view!
Reset!
Display grid
Entry type: {symbol}
Entry id = 262148
Write!
Use grid
Documentation Set: demo
Grid: {1/8}
Template: {1/2}
Scale: {definition}
Selection Mode: {closest}
X = 0
Y = 0
Set Source!
Set Destination!
Set Box!
BoxX = 0
BoxY = 0
Display pin names
Symbol name:
Function: {spare1}
d
s
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.29

Next you draw a line between the endpoints of the two lines. You move the source marker to one endpoint, move the destination marker to the other, and select **Create Orthogonal Line** (figure 2.30). Figure 2.31 shows the completed line.

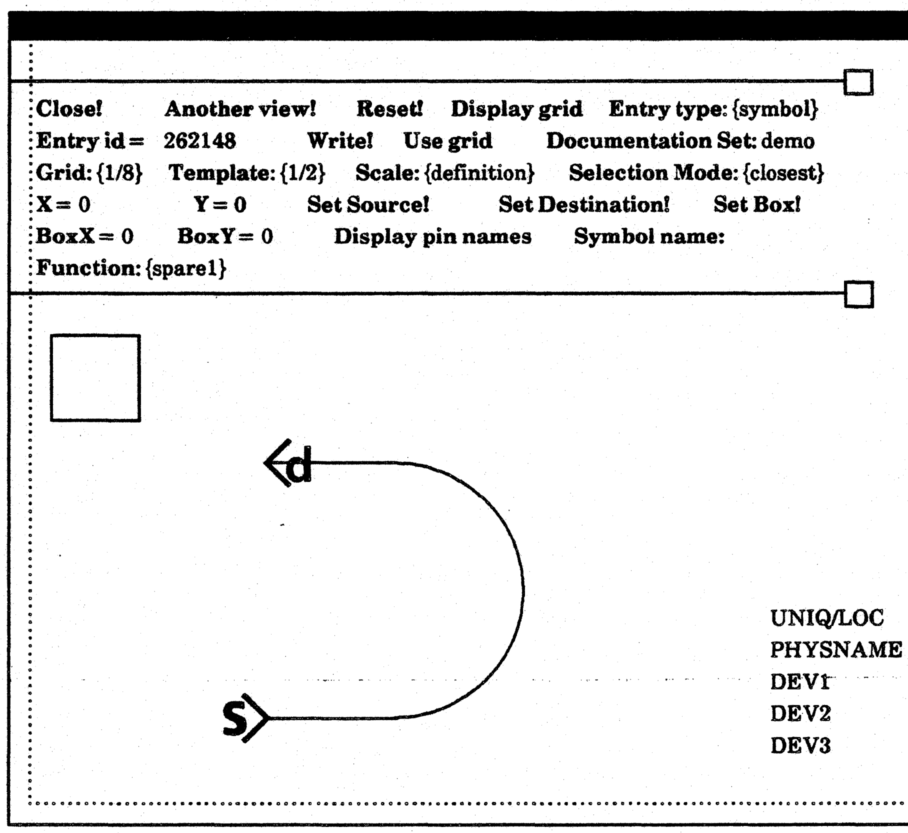


Figure 2.30

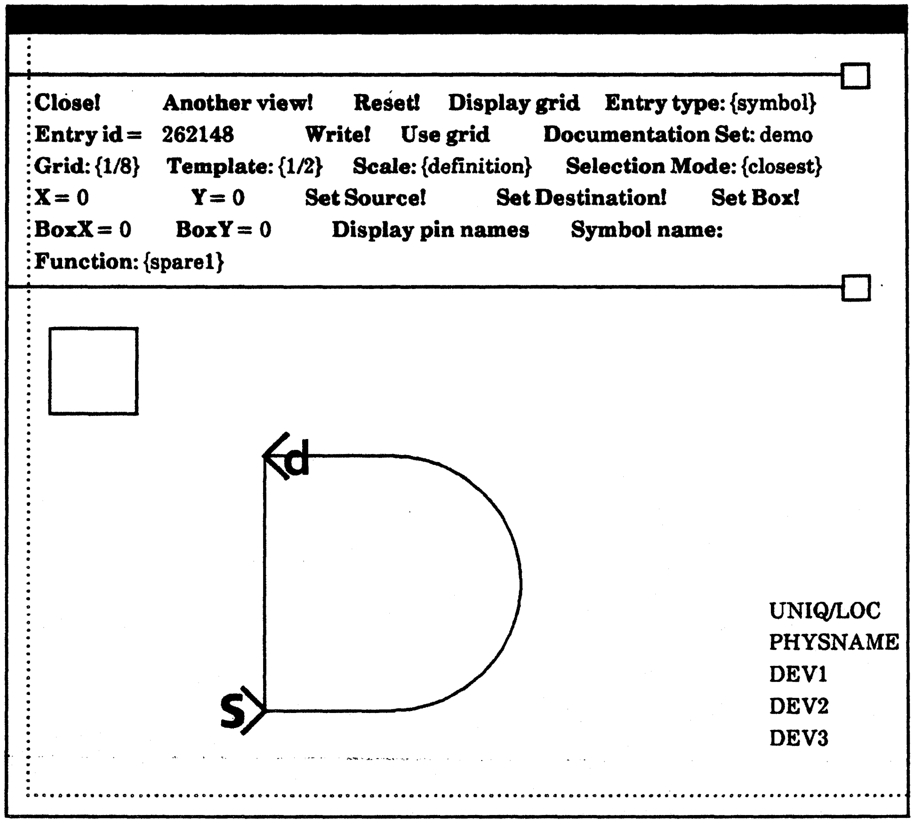

Close!
Another view!
Reset!
Display grid
Entry type: {symbol}
Entry id = 262148
Write!
Use grid
Documentation Set: demo
Grid: {1/8}
Template: {1/2}
Scale: {definition}
Selection Mode: {closest}
X = 0
Y = 0
Set Source!
Set Destination!
Set Box!
BoxX = 0
BoxY = 0
Display pin names
Symbol name:
Function: {spare1}
d
s
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.31

You draw the pin leads by marking the endpoints of a lead with the source and destination markers, and selecting **Create Pin Lead** (figure 2.32). Figures 2.33, 2.34, and 2.35 show the NAND gate's pin leads.

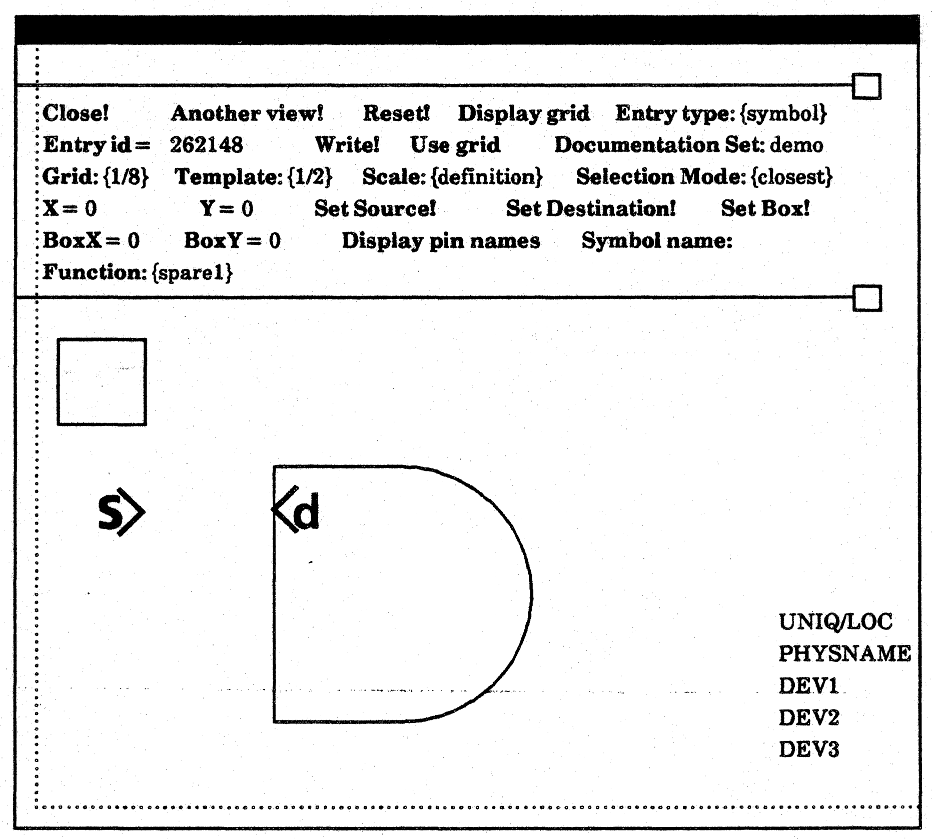


Figure 2.32

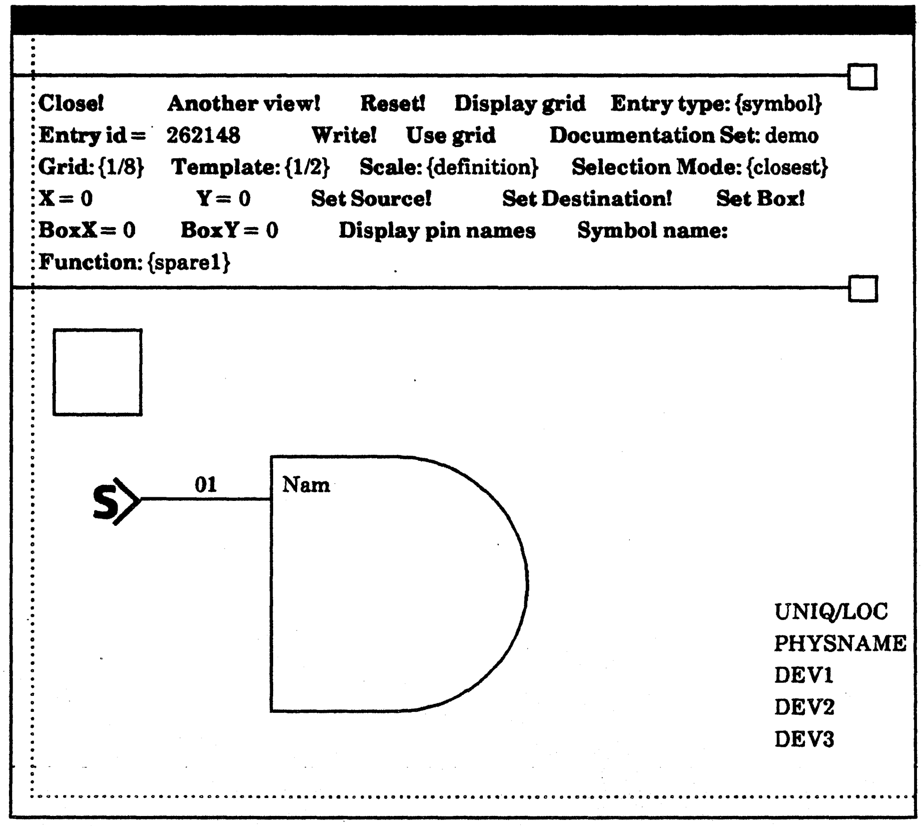

Close! Another view! Reset! Display grid Entry type: {symbol}
Entry id = 262148 Write! Use grid Documentation Set: demo
Grid: {1/8} Template: {1/2} Scale: {definition} Selection Mode: {closest}
X = 0 Y = 0 Set Source! Set Destination! Set Box!
BoxX = 0 BoxY = 0 Display pin names Symbol name:
Function: {spare1}
01
Nam
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.33

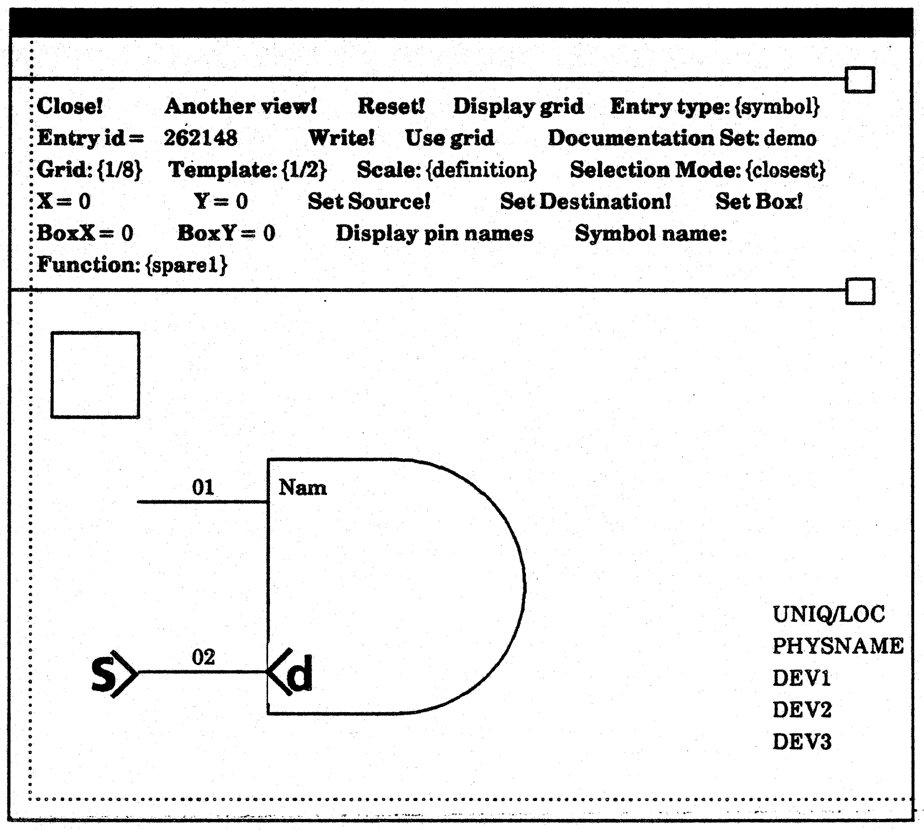

Close!
Another view!
Reset!
Display grid
Entry type: {symbol}
Entry id = 262148
Write!
Use grid
Documentation Set: demo
Grid: {1/8}
Template: {1/2}
Scale: {definition}
Selection Mode: {closest}
X = 0
Y = 0
Set Source!
Set Destination!
Set Box!
BoxX = 0
BoxY = 0
Display pin names
Symbol name:
Function: {spare1}
01
Nam
02
UNIQ/LOC
PHYSNAME
DEV1
DEV2
DEV3

Figure 2.34

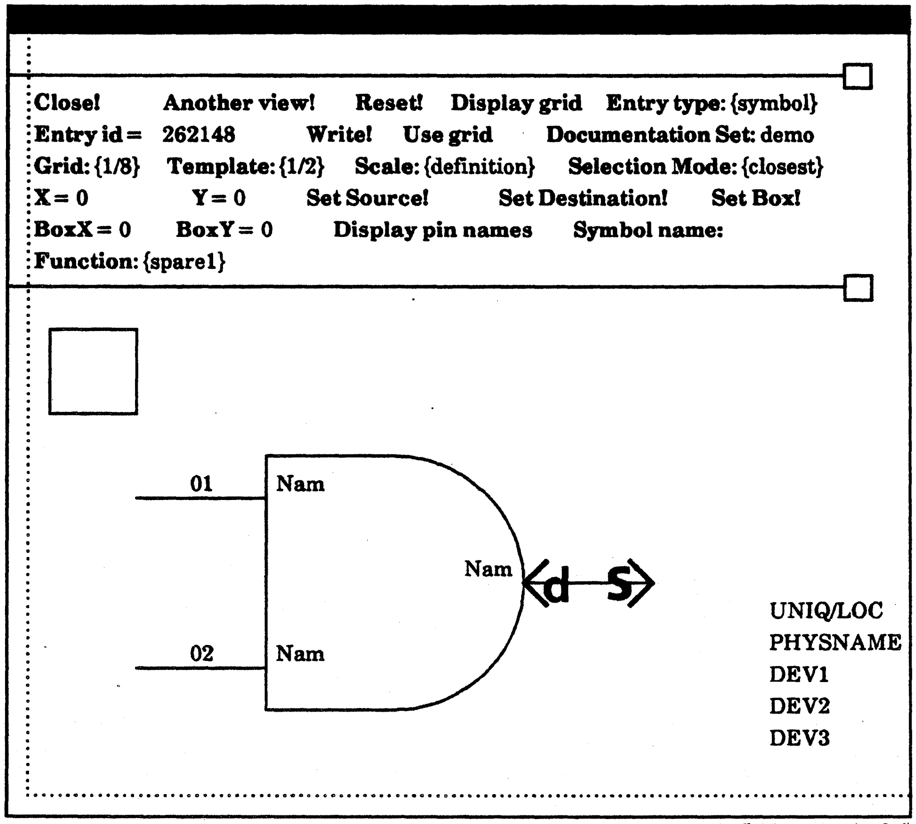


Figure 2.35

In each pin lead's property sheet you select the pin lead's type and assign the pin a name, number, swap code, and autoroute flags for drawing connections. You also use the property sheet to negate the NAND gate's output pin.

Finally, you set the size of the selection box, and the placement of the physical name text, logic uniqueness text, and device description text. You assign different fonts to text items in text property sheets. See figure 2.36.

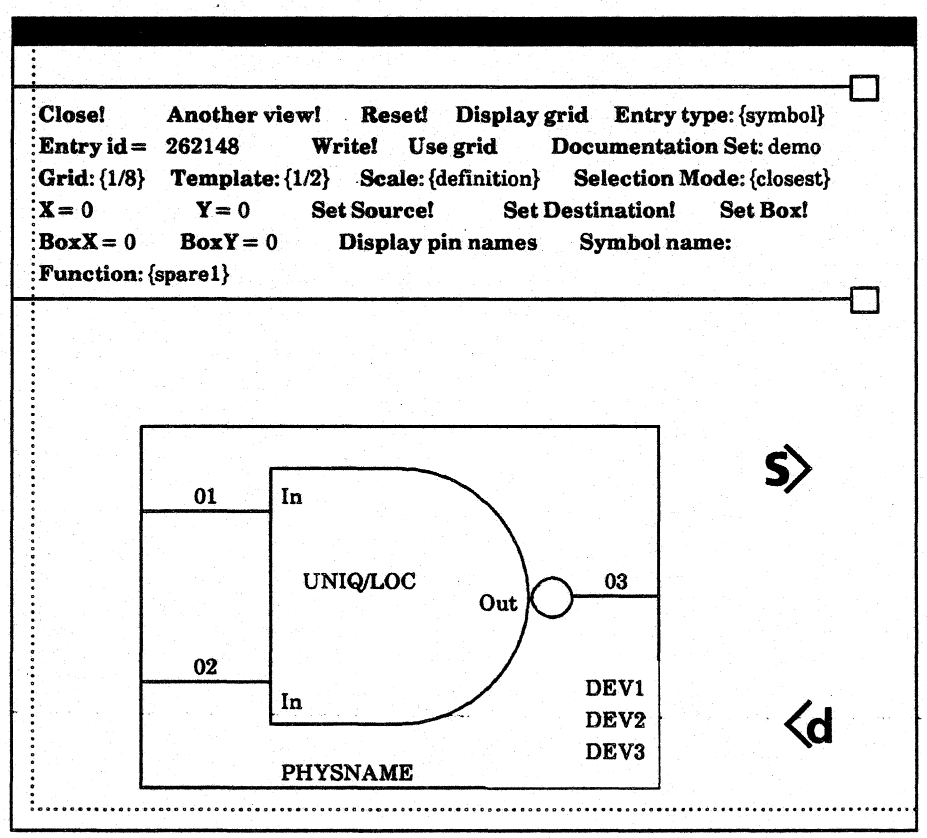


Figure 2.36

You give the symbol a unique identification number and name by filling in the identification number and name fields in the command subwindow. You assign a logic function to the symbol, and choose when the system should display the symbol's pin names on the schematic. Figure 2.37 shows the symbol's name and function.

When the symbol is complete, you write it to the library by selecting the **Write!** command. You then add the symbol and its nickname to the Abbreviation Map.

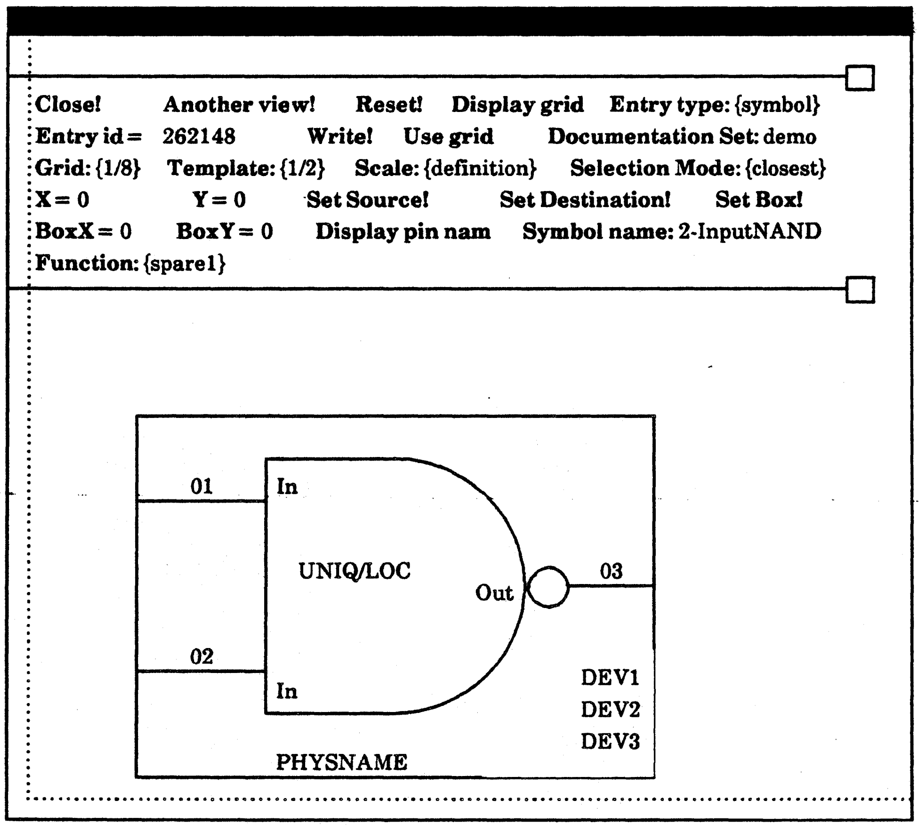


Figure 2.37

3. Library Maintenance

Change the entry type to **library** in the command subwindow to perform library maintenance tasks. These tasks include listing library entries, removing entries from the library, compressing the library, and converting the library to a new format. Figure 3.1 shows Library Editor commands when the entry type is **library**.

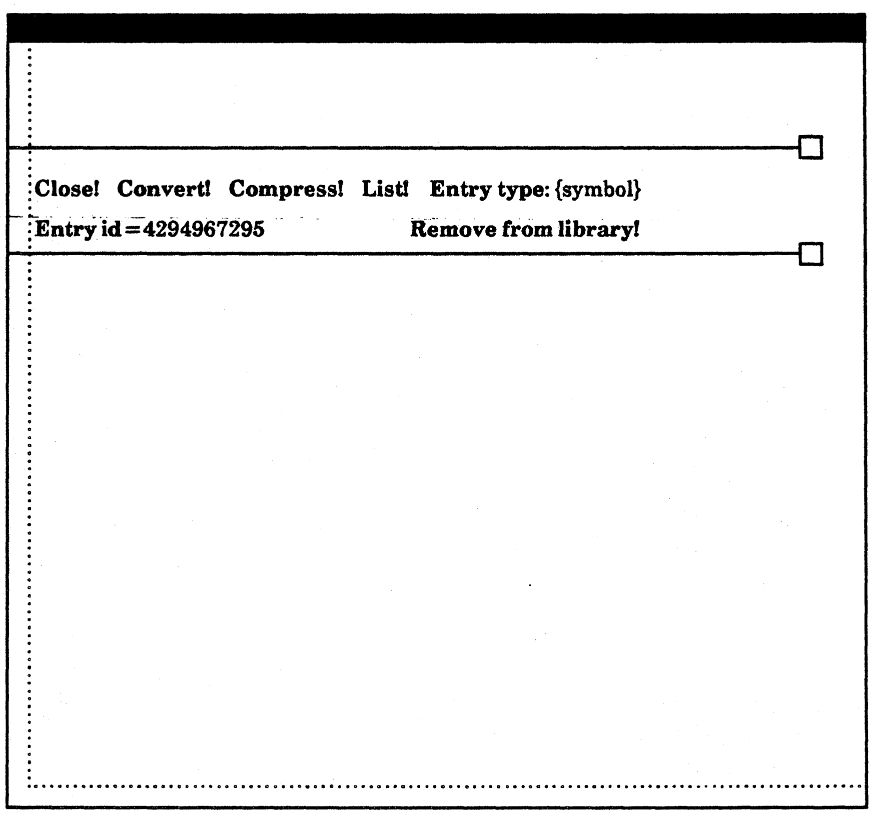


Figure 3.1

3.1 Listing Library Entries

- To list all entries in the library:

1) **Select the List! command. The system lists library entries, by their identification number, type, and name, in the Library Editor's third subwindow.**

Figure 3.2 shows library entries listed in the Library Editor's third subwindow.

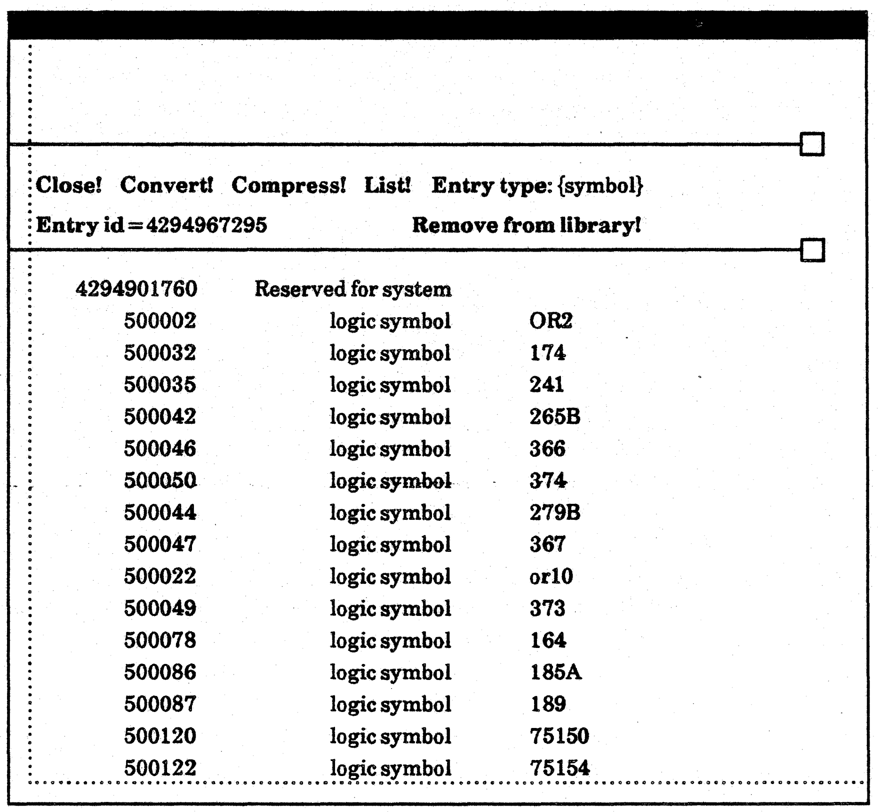


Figure 3.2

3.2 Removing an Entry from the Library

- To remove an entry from the library:

1) Enter the entry identification number after the words "**Entry id =**" in the command subwindow.
2) Select the **Remove from library!** command.

3.3 Compressing the Library

You *compress* the library after removing several entries in order to make room for more entries.

- To compress the library:

  1) Select the **Compress!** command.

3.4 Converting a Library

Each time Versatec issues a new release of the Expert 1000 software you may have to *convert* your library. Converting your library updates it so it is compatible with the new software.

- To convert the library:

  1) Select the **Convert!** command in the command subwindow.

4. Index