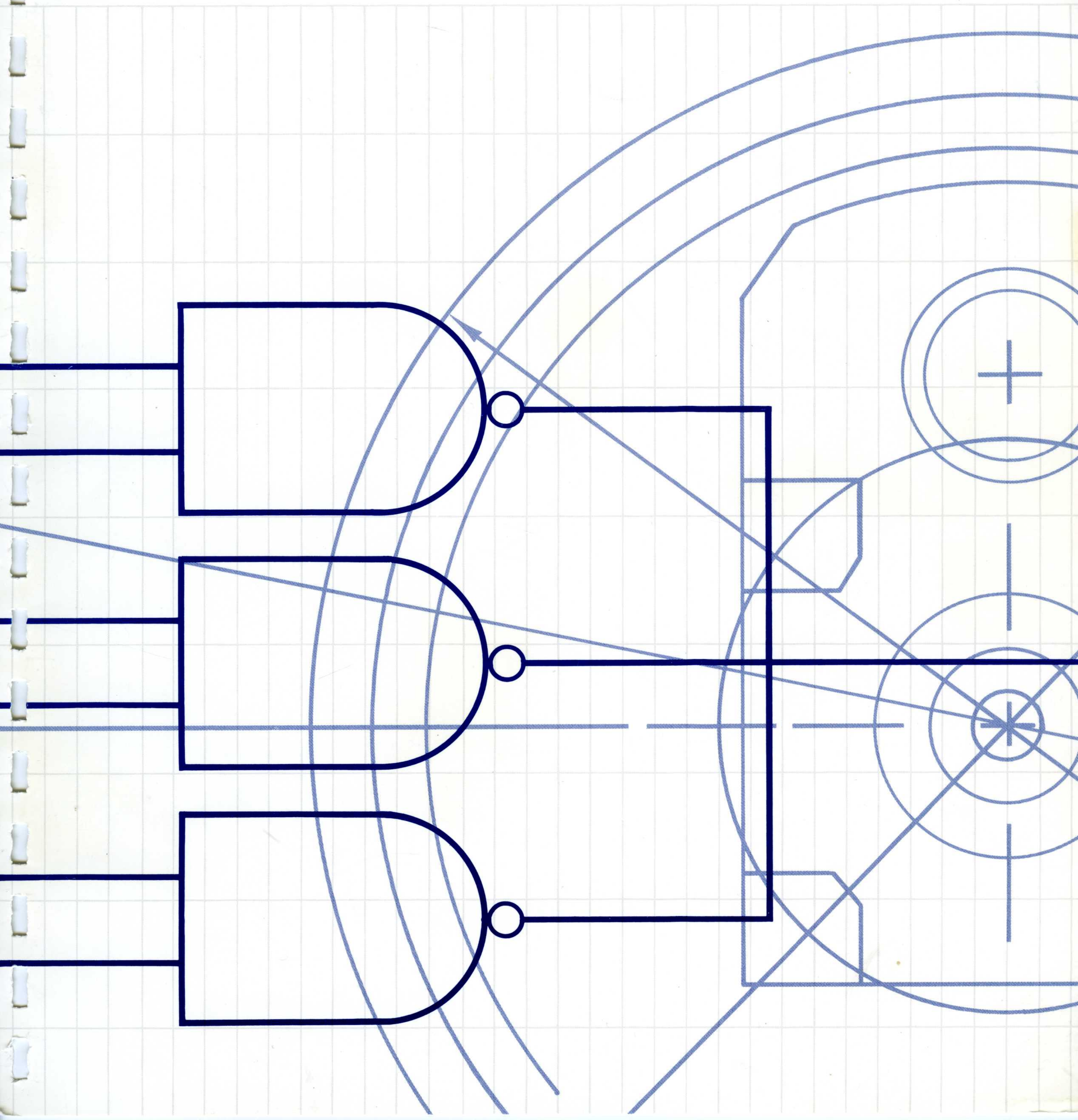
VERSATEC
A XEROX COMPANY

Expert Product Description



Expert Product Description

All text in this document was prepared with the Xerox 8010 Information System. Illustrations are scaled reproductions from the Expert workstation display.

Document Number 0-10B March 1984 Printed in USA

Expert Product Description

Table Of Contents

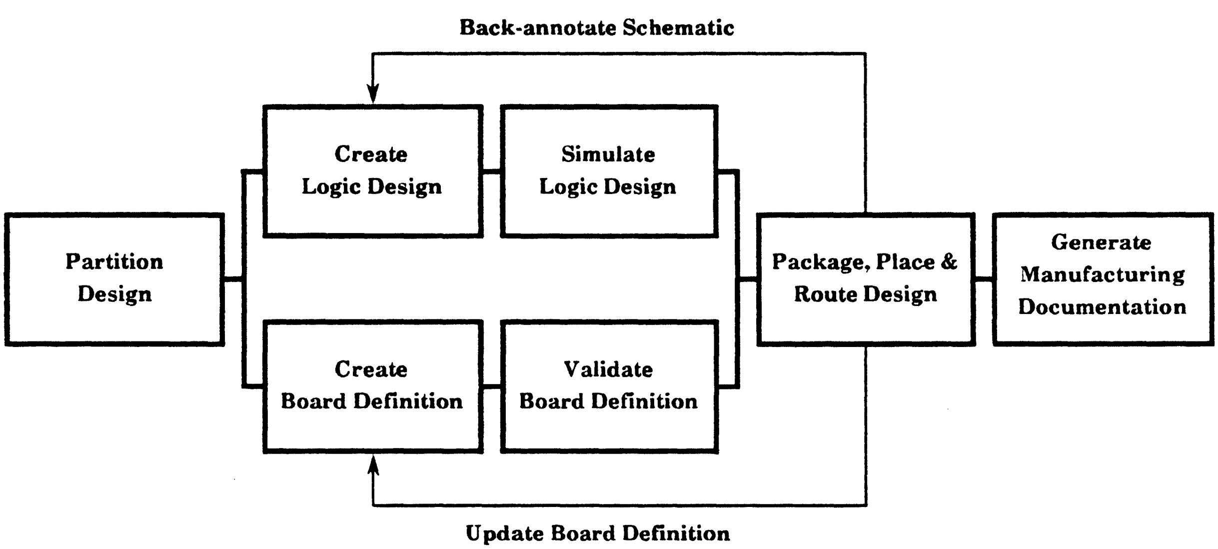
1. Expert Overview

The Versatec Expert system is an integrated set of engineering productivity tools. Xerox and Versatec, its wholly owned subsidiary, pooled their talents to produce Expert, the state of the art in computer-aided engineering systems. Expert combines Xerox's powerful 8010 professional workstation and Ethernet local-area network with Versatec-developed engineering application software. The system's unique user interface, based on 12 years of Xerox research, makes it easy to learn and easy to use.

Expert consists of four software modules: Expert Schematics, Expert Logic Simulator, Expert Placement and Routing, and Expert Drafting. Together, they form an integrated set of application tools for system design and printed circuit board layout.

Expert Applications

All Expert software modules are used to design printed circuit boards from schematic capture to photoplotting. The engineer uses the schematic created with the Expert Schematics Package and verified with the Expert Logic Simulator together with the board drawing and electronic parts created with the Expert Drafting module to complete placement and routing of the board. The diagram below illustrates how engineers and designers use Expert's software packages to produce a pc board.



The Expert Design Cycle

Expert Schematics

An electronics engineer uses the Expert Schematics package to design a logic diagram for the electronic product. Expert's hierarchical data base lets the engineer begin a design with a functional block diagram. He or she then designs the circuitry for each block in that functional diagram. The system automatically connects the functional units and creates a net list describing the complete design. Expert's common data base insures that the final pc board meets all engineering specifications. When the board design is complete, the engineer can back-annotate the schematic with information describing the board's physical implementation.

Expert Logic Simulator

For verifying a design created with the schematics package, Expert provides the engineer with a powerful logic simulator. The event-driven simulator's features include several functional primitives, and high-level design languages for adding primitives and for describing inputs and clocking schemes. The simulator supports MOS, TTL, and ECL technology and provides a rich set of features for control and debugging.

Expert Placement and Routing

The pc board designer uses the verified schematic, along with the Expert Placement and Routing package to design the board layout.

Designers can use manual or automatic methods to place parts on the pc board. There are several ways that the user can edit part placement when he or she sees ways to improve the design.

Expert has a two-phase routing system. The first phase, called global routing, assigns connections to pre-defined routing channels. The second phase, called etch routing, determines the exact position of etch traces within those channels. The designer can use both manual and automatic routing methods to complete an individual board. Because the Expert routing system is based on routing channels and manufacturing rules, the system routes traces with angles at multiples of 45 degrees, routing the maximum number etch connections possible within the available space.

Expert Drafting

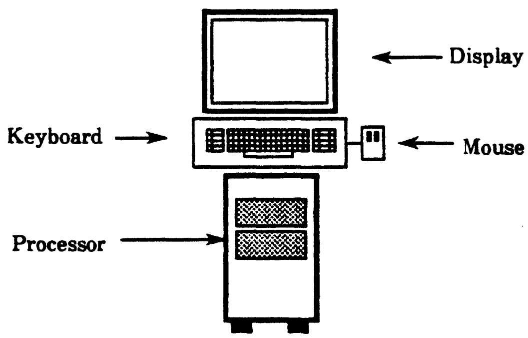
Expert's drafting package is a design tool for the mechanical engineer, designer, and drafter. It is a system for drafting which includes special features for printed circuit board definition and parts libraries.

The engineer uses special multi-page drawing windows and a menu-driven command structure to create geometry and text, just as he or she might use paper and pencil. From these basic items, the user issues special editing commands that cut, corner, trim, and dissect trajectories. Other editing commands duplicate, drag, rotate, and mirror geometry within a drawing window. Global drawing management commands carry out filing, dimensioning, output, and drawing creation functions.

The drafting module accelerates the design cycle thanks to special features dedicated to creating printed circuit board drawings, photoplot drawings, electronic parts libraries, and standard mechanical parts. An Expert user can update these complex objects to meet changing product requirements far more quickly and accurately than the engineer using manual methods.

The Expert Workstation

The Expert workstation consists of a processor, a high-resolution, bit-mapped graphics display, a keyboard, and a pointing device called a mouse.



The Expert Workstation

The processor is based on 2901 bit-slice technology. It addresses 16 megabytes of virtual memory, and has 768 kilobytes (or optionally 1.5 megabytes) of physical memory, a 42-megabyte, local Winchester disk, and a 1.2-megabyte floppy disk drive. In addition, it has a connection to the 10-megabit-per-second Ethernet. The processor executes approximately 1 million instructions per second, and stores design information in a 32-bit data base that provides adequate resolution for all state-of-the-art printed circuit board designs. The system's virtual memory lets the engineer create large design data bases without imposing limits such as maximum number of schematic pages, maximum net name length, and maximum number of connections.

The display screen is the Expert working area. The resolution of the display is 808 pixels vertically by 1,024 pixels horizontally, and it is refreshed at 38 frames per second to avoid flicker.

The keyboard contains 24 special function keys as well as standard typewriter keys. The function keys perform frequently-used commands, and the alpha-numeric keys let the user easily enter and edit text for annotations or reports.

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. The engineer uses the mouse to point to objects on the display, and uses the mouse buttons to mark, or select, objects.

The Expert Network

At each Expert installation, an Ethernet local area network connects Expert workstations to *servers* and to each other. Network servers are specially-designed processors that let users share network resources and peripherals. Like the Expert workstation, network servers were developed by Xerox, a pioneer in local area network technology.

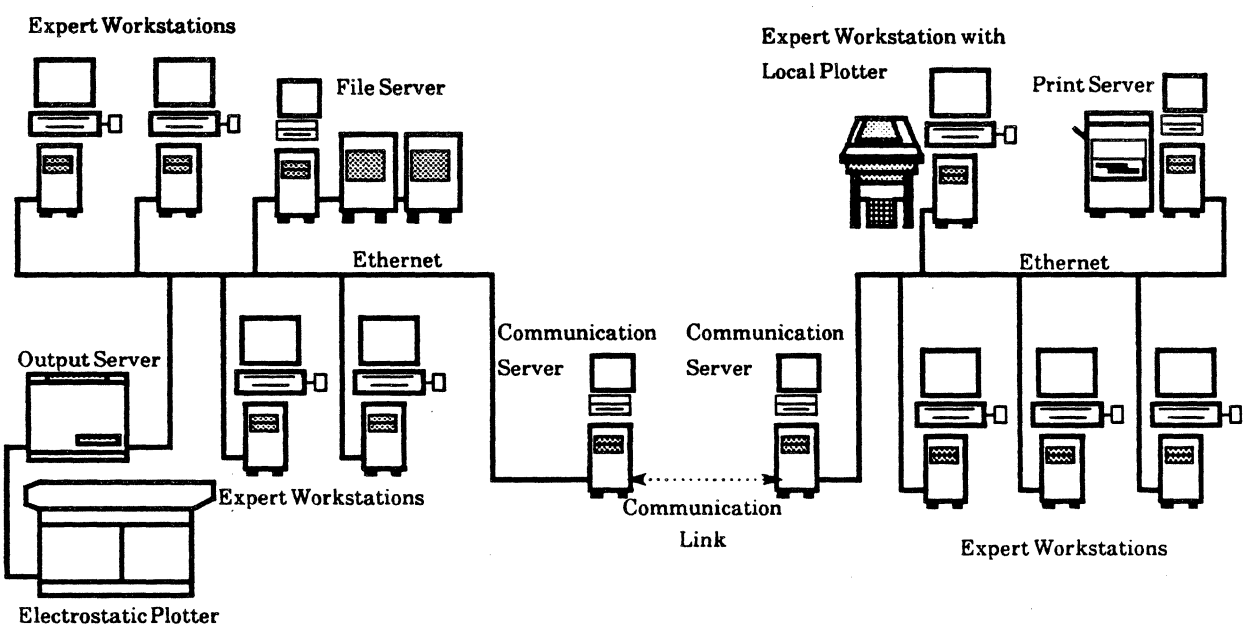
A File Server lets users store design data in a central location. A group of users working on a project can share design information stored on the File Server, while preventing other users from altering that information.

Users can output designs in several ways. An Output Server, developed by Versatec, lets users plot drawings with an electrostatic plotter and optionally store design data on magnetic tapes. A Print Server gives network users access to a high-quality electronic printer. In addition to network printing resources, electrostatic and pen plotters can be connected locally to any workstation.

A Communication Server lets users on remote networks communicate with each other and share network resources. Ethernet transmits data between workstations and servers at 10 megabits per second.

Ethernet lets each installation build a custom Expert network. A single network can connect as many as 1024 servers and workstations. Networks can be joined together to create an *internetwork*. The diagram below illustrates two networks joined to form an internetwork.

Ethernet makes an Expert system easy to expand as the demand for resources at an installation grows. Simply add a workstation or server to an existing network, or connect two or more networks with Communication Servers to form an internetwork. Adding workstations to a network does not affect the performance of existing workstations.



An Expert System Configuration Showing Internetwork Communication

Creating a Custom Expert Network

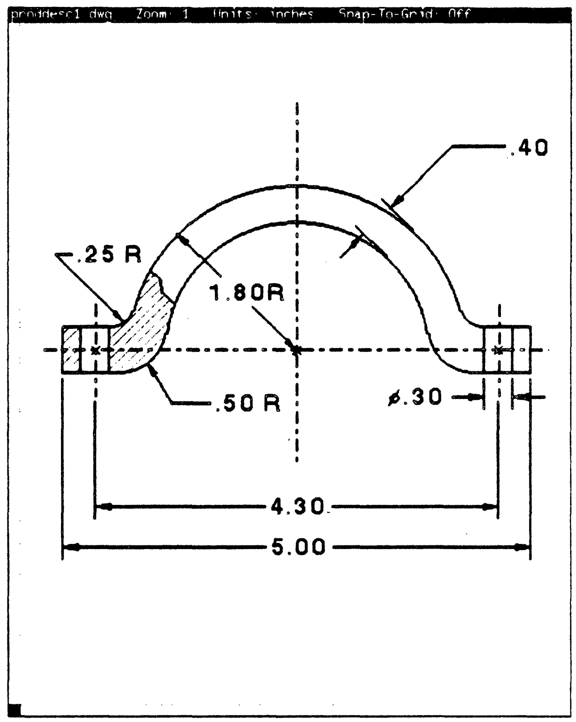
Ethernet and the Expert software modules let each organization build a custom network that fits the unique needs of that organization. As an organization grows, it is easy to expand its Expert network by adding new software modules to existing workstations, or adding workstations and servers to the network.

For example, a large installation might purchase the schematic design and logic simulation modules for workstations in the engineering department. The pc board design department would purchase Expert workstations outfitted with the schematic design, printed circuit board layout, and drafting modules. Both groups would share design information and network services via Ethernet.

A smaller site might have fewer workstations shared by engineers and pc board designers. That site might purchase all four modules with each of its workstations. As the installation grows, it can easily add workstations and servers to its network. Expert's range of applications, powerful engineering workstation, and flexible networked environment let each installation create a custom system that grows with the installation.

2. The Expert User Interface

The Expert user interface is based on innovative research conducted at Xerox's Palo Alto Research Center. Although the user interface for each software package is tailored to its application, all Expert software packages share fundamental user interface features such as windows, command menus, and command accelerators. This chapter presents the basic techniques for using Expert, later chapters describe each software package in more detail.



A Drawing Window

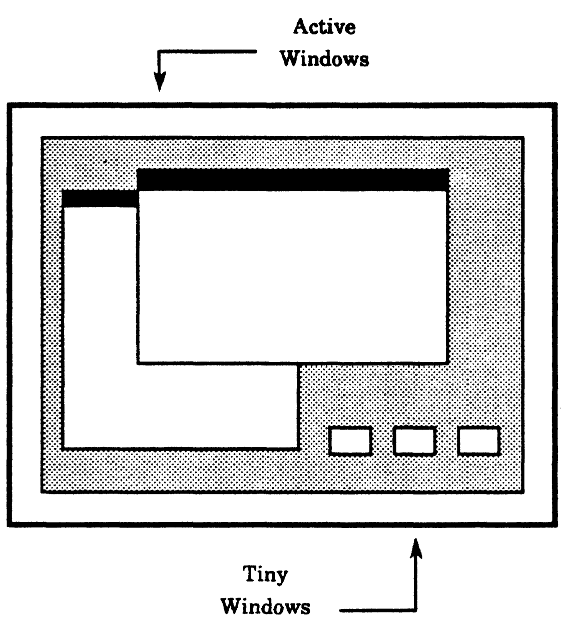
Windows

Expert users interact with the system through *windows* on the display. A window is a rectangular region on the display framed by a narrow black border. The name of each window appears in the black *name frame* at the top of the window. Windows act like pieces of paper on the user's electronic drawing table. One window can cover another window, obscuring all or part of the lower window, just like two sheets of paper on the drawing table.

Different types of windows perform different functions. For instance, a mechanical designer uses a *drawing window* to draft. A *command window* contains drafting commands. An electronics engineer designs and edits a schematic inside a *schematic window*. The printed circuit board designer uses a *placement and routing window* to produce a pc board layout.

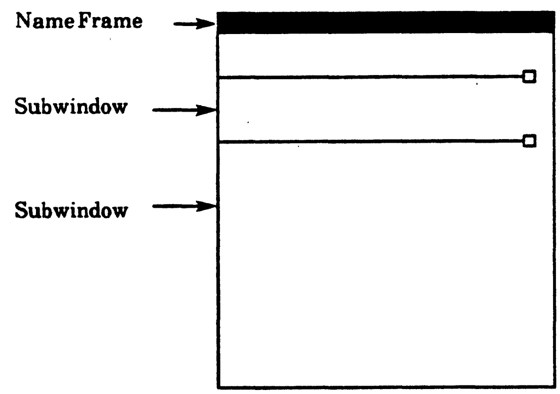
The user can display any combination of windows on the display at the same time. This ability gives the Expert user quick access to different applications and easy interaction between applications. With multiple windows on the display, an Expert user can view all aspects of a design simultaneously. For example, a pc board designer can simultaneously display a schematic, a pc board layout, and the board definition drawing for the design on which he or she is working.

The user can easily change the size and shape of windows on the display. When the user is not working with a window, he or she can make it *tiny* on the display as he or she would put a paper aside on a desk. A tiny window is a one-inch by one-half-inch representation of the window that contains the name of the window.



Windows on the Expert Display

Horizontal lines ending in a small box divide some windows into *subwindows*. Each subwindow represents a different function of the window. Subwindows can contain more information than their size lets the user see. The user can scroll vertically and horizontally through subwindows.



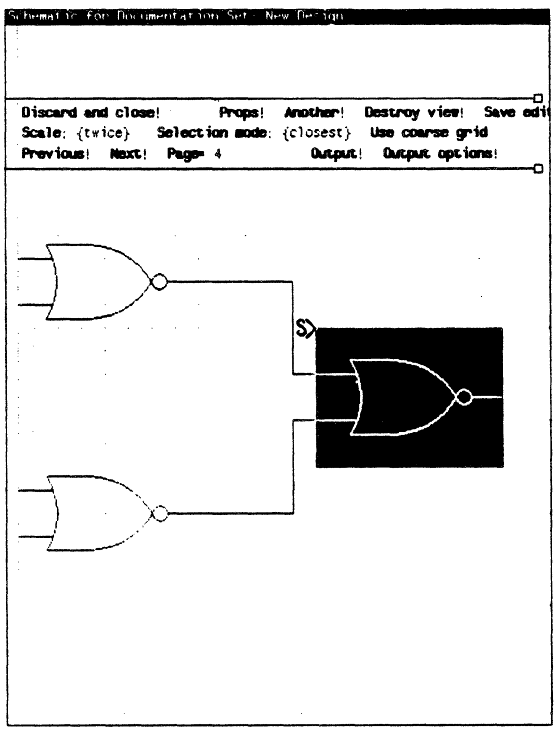
A Window's Name Frame and Subwindows

Some Expert windows have *command subwindows*. Command subwindows contain lists of commands. Instead of memorizing and typing commands, the user issues a command by simply pointing the cursor at the command, and pressing the left mouse button.

Selecting Objects

Moving the mouse on a flat surface, for example a desktop, moves a *cursor* on the display. The engineer uses the cursor as a pencil to *select* objects inside windows, for example a logic symbol inside a schematic window, and to issue commands to the system. To select an object, the engineer moves the mouse until the cursor points to that object and presses the left mouse button. The system highlights selected objects.

Once an object is selected, the user can manipulate that object. For example, to delete an object, the user first selects the object, and then presses the **DELETE** key.

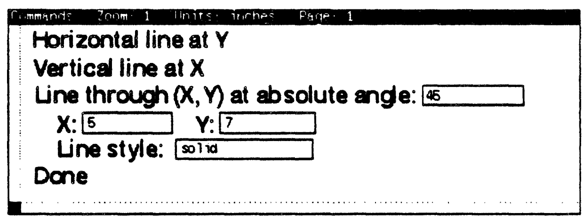


A Selected Object inside a Schematic Window

Selecting Commands

Command windows and command subwindows list commands available to the user. To issue a command, the user selects that command.

Some commands have *parameter fields* associated with them. If the command requires additional information, the user enters that information in the command's parameter fields before selecting the command. Most drafting commands, for example, require numeric or text parameters, for instance the radius of a circle, or the amount by which to zoom a drawing. The user enters these parameters in the appropriate fields in the command window. In general, the user selects the field and types the appropriate text. Some fields have a menu that contains a list of fixed parameter options. The user sets a parameter by selecting an option from the menu. Other parameters act as switches that the user can turn on or off.



Parameter Fields inside a Command Window

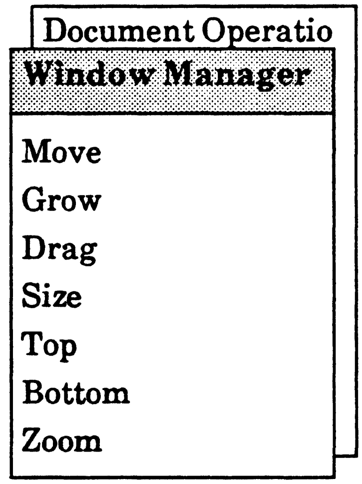
Dynamic Menus

Another feature of the Expert user environment is the *dynamic* or *pop-up menu.* A dynamic menu is a list of commands or options usually hidden from the user. The user displays a dynamic menu only when he or she wants to execute a command; when the user doesn't need the menu, it doesn't clutter the working area.

Dynamic menus make it possible for even casual users to become proficient. To display dynamic menus, the user presses both mouse buttons simultaneously. The cursor's position inside a window or a subwindow tells the system which functions are valid, and the system displays only menus that are relevant to those functions. For example, if the cursor is in a schematic window, the system displays only commands for creating and editing the schematic.

When there is more than one menu associated with the subwindow, the menus appear stacked on top of each other. Menu names inside name frames describe the functions of each menu. To view the commands in a particular menu, the user selects that menu's name from the stack.

One dynamic menu, the Window Manager Menu, is associated with every Expert window. The Window Manager Menu contains commands that alter an entire window. Window Manager commands let the user "shuffle" windows on the display as he or she shuffles papers on a desk. The user can move the window on the display by selecting the **Move** command. **Grow** and **Drag** let the user change the size of the window. **Size** makes the window active if it is tiny, or tiny if it is active. **Top** puts the window on top of other windows on the display, and **Bottom** puts it beneath them. **Zoom** causes the window to take up the entire display or returns it to its normal size from this state.



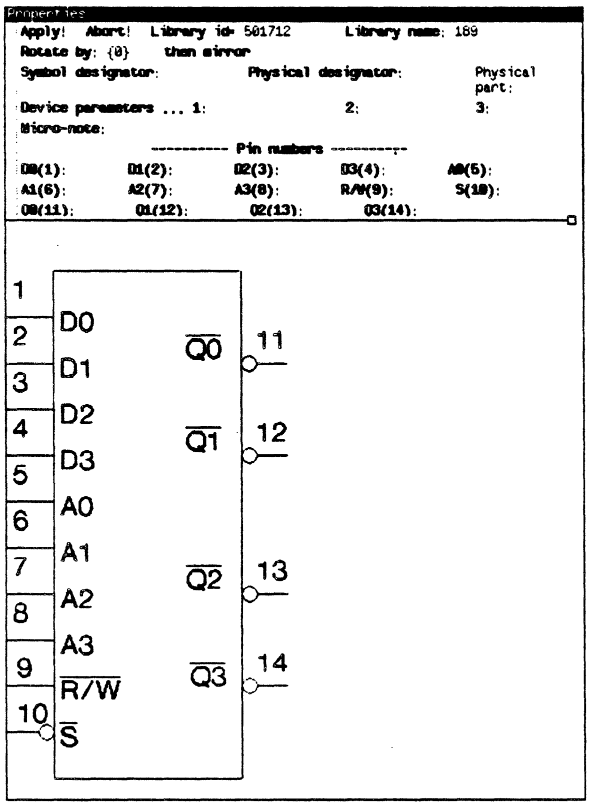
The Window Manager Menu at the Top of a Menu Stack

Accelerators

Expert's user interface features two styles of command entry. Dynamic menus make using Expert easy for the novice or casual user. For more experienced users, the system provides command *accelerators*. Command accelerators are "short-cut" methods for issuing commands. Accelerators are often special keys or special combinations of the cursor's position and the mouse buttons. Function keys act as accelerators for the most frequently-used commands. For instance, to delete a line inside a schematic or drawing window, the user selects the line and presses the **DELETE** key.

Property Sheets

Some windows and objects inside windows have special *properties*. For example, a logic symbol's properties include its symbol designator, physical designator, physical part name, and physical pin numbers. The user can view and change properties in a special type of command window known as a *property sheet*. The user displays a logic symbol property sheet, like the one in the illustration below, by selecting the logic symbol inside a schematic window and pressing the **PROP'S** function key.



A Logic Symbol Property Sheet

3. Expert Schematics

The Expert Schematics package is the electronics engineer's tool for logic design. With it, the engineer can create a schematic diagram of an electronic product. Expert's hierarchical data base lets the engineer partition a design into functional blocks, design the logic for each block, and then combine the functional blocks into a complete schematic.

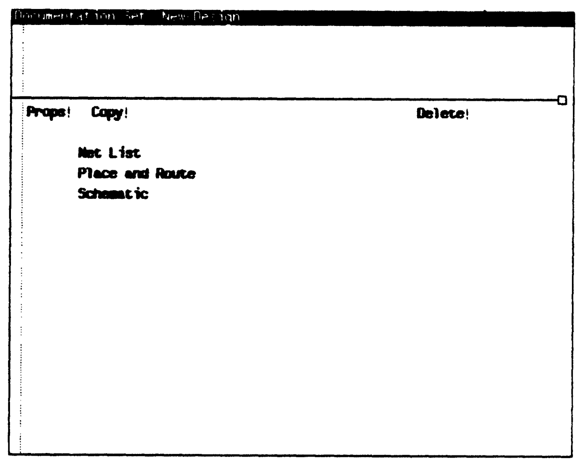
Expert removes most of the limits imposed by more traditional computer-aided design methods. The engineer can use any size drawing format, and a schematic contains as many pages as the engineer needs. An engineer can work with several schematics at once, or share the design of a particular schematic with engineers at other workstations.

The Expert Schematics package works in unison with the Expert Logic Simulator and Expert Placement and Routing package to produce a printed circuit board from the schematic diagram. After completing a schematic, the engineer uses the Expert Logic Simulator to verify the design. If the simulator finds errors, the engineer edits the schematic to correct them and runs the simulation again. The engineer can display both the schematic and the simulator while verifying and correcting the design.

Once the simulator proves the accuracy of a design, a pc board designer uses the schematic along with the Expert Placement and Routing package to design the pc board layout. The system insures that the board layout conforms to the schematic. When the board design is complete, the engineer back-annotates the schematic in order to record information about its physical implementation.

The Documentation Set Window

There is a *documentation set* window for every Expert schematic, its net list, and printed circuit board layout. A documentation set window is like a folder that contains documentation for a design. By storing all the data for a design in a central location, the documentation set window helps users manage design data. Users access a design's schematic, net list, and pc board layout through that design's documentation set window. The figure below illustrates a sample documentation set window.



A Documentation Set Window

An engineer's first step when beginning a new design is to create a new documentation set for that design. He or she copies a standard blank documentation set, and then renames the new documentation set.

The Schematic Window

An engineer creates a schematic, or logic diagram, for a design in a *schematic window*. He or she displays a design's schematic window by selecting the word **Schematic** in the design's documentation set window and pressing the **OPEN** key. If the engineer needs to work on more than one schematic at a time, he or she simply locates the documentation sets containing the schematics, and opens the schematics. The user can perform design operations in any open schematic.

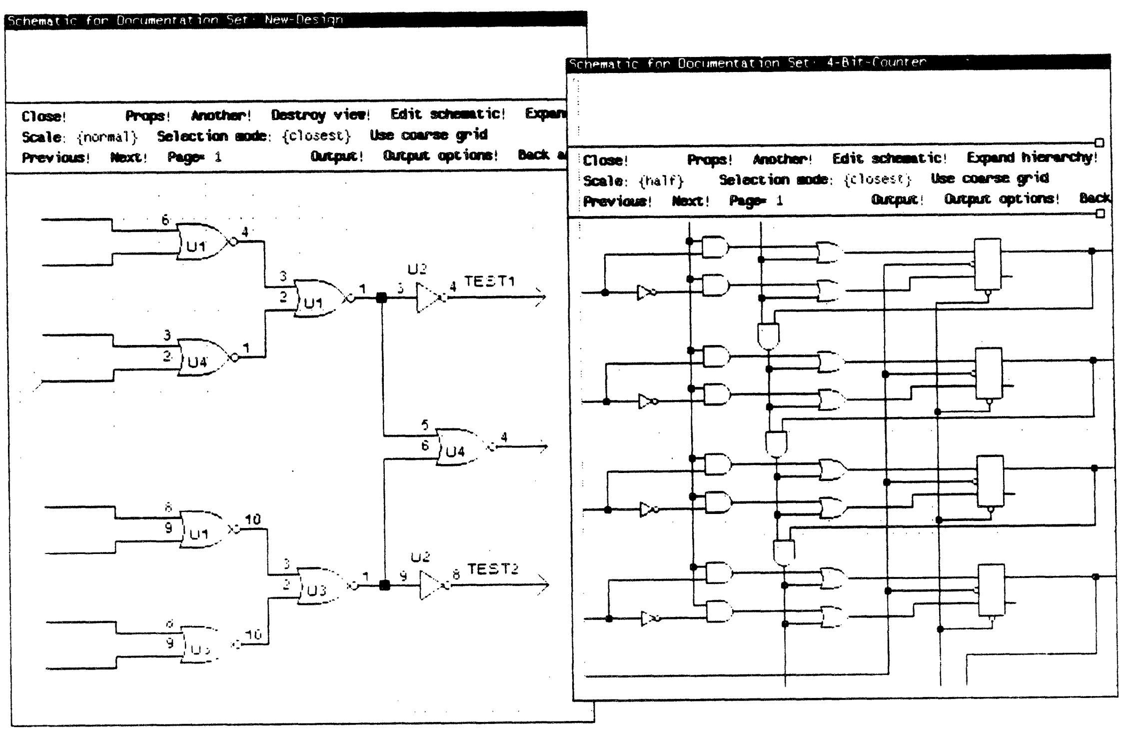
The schematic window is divided into three subwindows. The first, the system message subwindow, gives the user feedback from the system. The second subwindow contains schematic window commands, and the third subwindow displays the schematic. The engineer uses commands in the Editing Schematic Menu, associated with the third subwindow, to create and edit the schematic.

The schematic can be any size. Initially, the schematic subwindow displays as much of the schematic as its size permits. The user can scroll the subwindow to display other portions of the schematic, or change the window's size and shape to display more or less of the schematic. In addition, the user can scale the schematic to make it appear smaller or larger.

The schematic window contains as many pages as the user needs. The system makes multiple-page work quick and convenient.

An engineer can display as many views of a schematic window as he or she desires. This useful feature allows the engineer to view several pages of the schematic at once, to view different parts of the same schematic page, or to view parts of the schematic at different scales. The engineer can independently scale, scroll, and page different views forward and backward. The user can make changes to the schematic in any view, and all views display those changes.

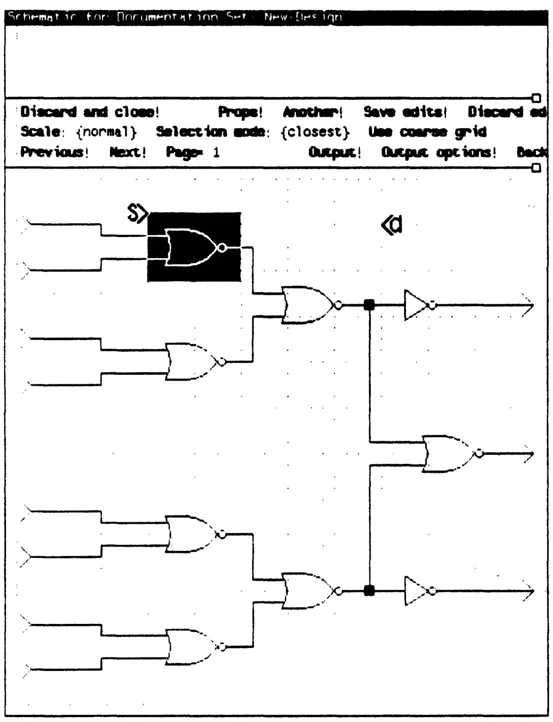
The **Another!** command instructs the system to display a duplicate of the window. The user discards a window by selecting the **Destroy view!** command in the command subwindow.



Two Schematic Windows

The system does not let the user destroy the last window displaying the schematic.

Within the schematic subwindow, the engineer uses the mouse buttons to mark source and destination locations for moving and copying objects. The user selects a point inside the subwindow with the left mouse button to mark the source (s>). The system highlights the closest object. The user selects a point with the right mouse button to mark the destination (<d). For example, the user moves a logic symbol by selecting it, setting the destination marker at the desired target location, and then pressing the **MOVE** key.

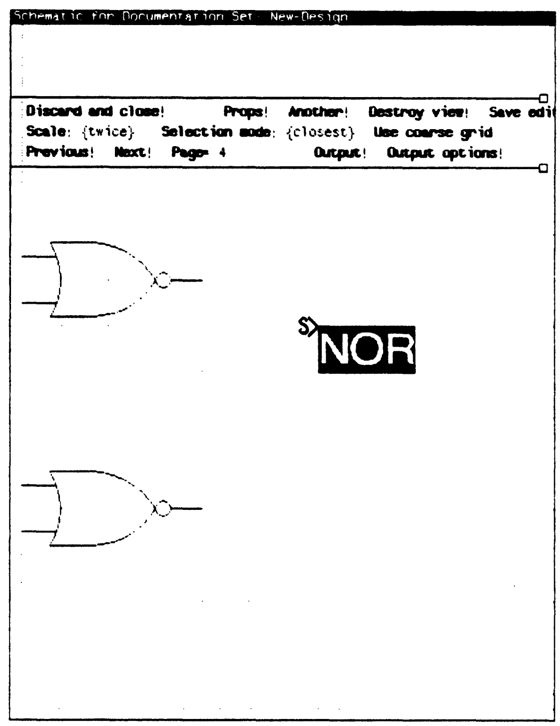


Source and Destination Markers

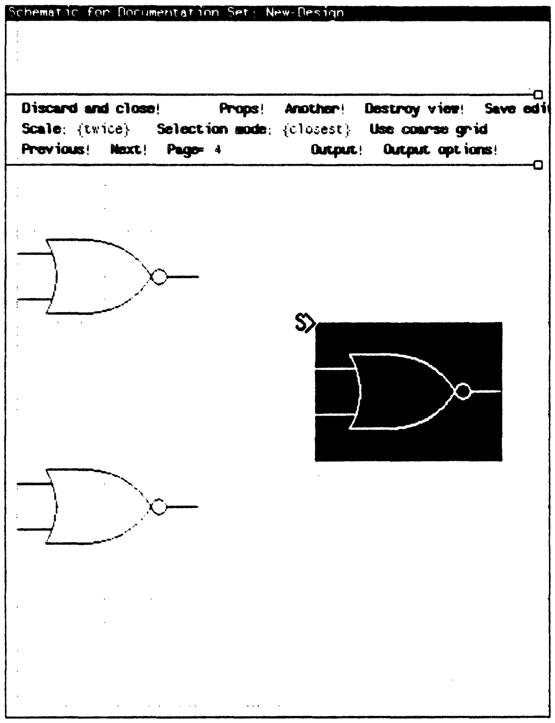
Placing Logic Symbols

The system stores logic symbol descriptions in a central library. Each symbol has a graphic description as well as an identification number and a name. The system librarian uses a special window to give each logic symbol one or more *nicknames*. These easily-remembered nicknames speed the creation of commonly-used logic symbols.

To place a logic symbol in the schematic, the user selects the position for the logic symbol on the grid by marking that point with the source marker. The user types the logic symbol's nickname or identification number (the system selects the text), and then issues the dynamic menu command **Make into a symbol**. The system replaces the selected text with the corresponding logic symbol.



The Source Marker and Symbol Name



The Source Marker and Logic Symbol

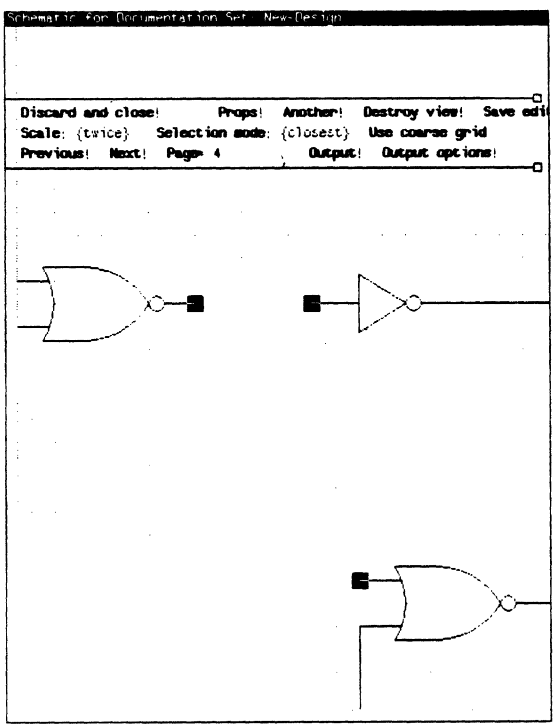
The user enters symbol characteristics in the symbol's property sheet. A logic symbol's properties include its symbol designator, physical designator, physical part name, device parameters, and physical pin numbers. If rotated and mirrored versions of the symbol exist in the symbol library, the engineer can use the property sheet to rotate and mirror the symbol. The user can set default values for the device parameters of discrete devices so that he or she does not have to enter parameters for each device individually.

When the engineer back-annotates the schematic from the pc board created with Expert, SCICARDS, or Computervision, the system updates logic symbol properties to reflect each symbol's physical implementation on the pc board.

Connecting Logic Symbols

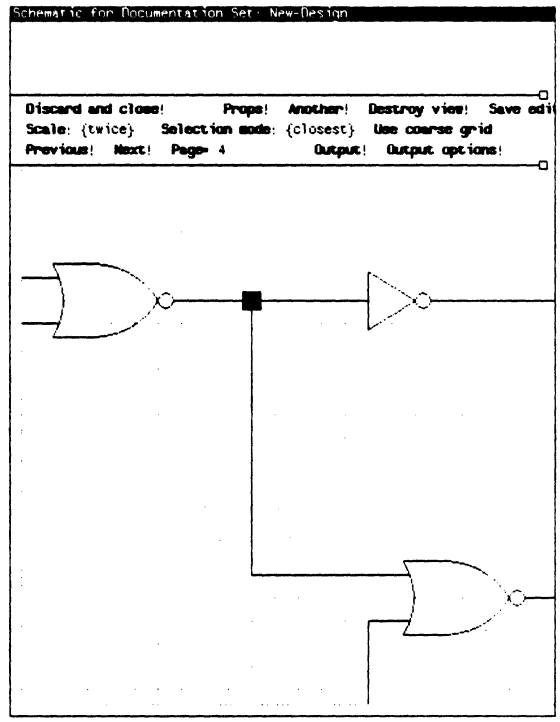
The engineer connects logic symbols with *nets*. There are two ways to connect logic symbols on the same schematic page.

Using the first method, the user selects the dynamic menu command **Create net**, and then selects connection points on logic symbols. Once the user has selected all the endpoints for the net, the user displays the Create Net Menu and instructs the system to draw the net. The system determines a reasonable path between the connection points, runs the connection line segments along that path, and marks junctions with solder dots.



Marked Endpoints of a Net

Using the second method, the user can draw the exact path of a net. The user selects **Create connection path** in the Editing Schematic Menu, marks the first endpoint of the connection, and draws the path to the second endpoint. When he or she completes the connection, the user displays the Create Connection Path Menu, and selects **Apply** to confirm the connection path. The system places solder dots where the connection meets existing nets.



The Completed Net

The user can edit the path of a net by drawing a new path from one point on the existing net to another, or by deleting a connection path and its junctions and drawing a new connection.

To make a connection between logic symbols on different pages of the schematic, the user can place two instances of a special logic symbol known as an off-page connector, one on each page of the schematic. On each page, the user creates a net connecting each off-page connector to the logic symbols on the same page. The user brings up the property sheet for each of the nets, and gives the nets the same name.

Annotating the Schematic

The user annotates the schematic by drawing lines and adding text. The user may also choose to display properties such as net names, symbol names, pin names and numbers, physical designators, and symbol designators as annotation.

Display options in the schematic window's property sheet let the user control which symbol properties the system displays on the schematic. For instance, to create a logical schematic, the user displays symbol designators. To create a physical schematic, the user might display physical designators, pin numbers, physical names, and pin names.

Clustering Objects

Often a schematic consists of several functional units, such as device drivers or microprocessor memory subsystems. When several objects are related, the engineer can group them into a *cluster*. The user can then manipulate the cluster as if it were a single object. For instance, the user could group the circuitry for a device driver into a cluster, then move or copy it to another place in the schematic, or even copy it to another schematic.

Editing a Schematic

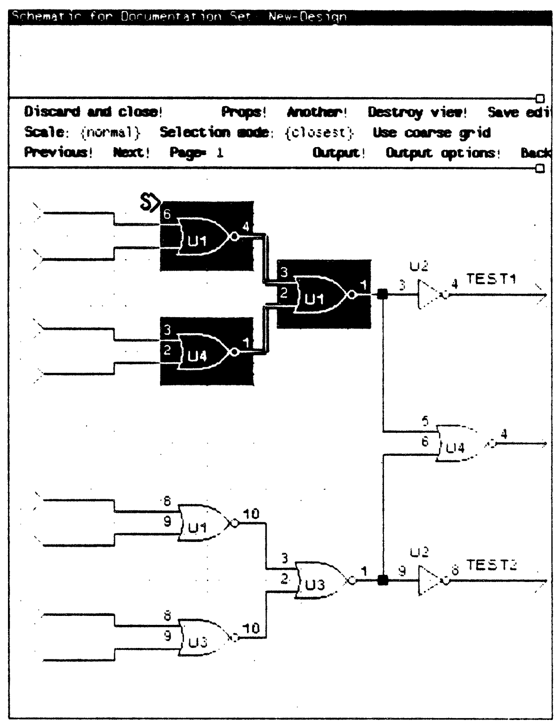
The user can edit a schematic at any point in the design process.

To delete an object, the user selects the object, and presses the **DELETE** function key. If he or she accidentally deletes an object, the user can press the **SAME** key to *paste* it back into the schematic.

To move or copy an object, the user selects the object, marks the destination to which to move or copy the object, and then presses the **MOVE** or **COPY** function key. The engineer can copy objects from one schematic to another with the same method; the engineer displays both schematics, selects the object to copy in the first schematic, marks the destination location in the second schematic, and presses the **COPY** key.

The user could also select several objects, then press **DELETE**, **MOVE**, or **COPY** to delete, move, or copy all of those objects at once. The user can select a cluster, and delete, move, or copy it as if it were a single object. He or she can also select objects that are part of the cluster individually and move, copy, or delete them, and remove them from the cluster.

The system deletes, moves, or copies any objects fully associated with the selected object. For example, if the user instructs the system to move a logic symbol, the system *stretches* the nets associated with the symbol so that they still connect to that symbol. If the user instructs the system to copy the symbol, the system copies only the symbol. If a user instructs the system to copy several symbols, the system copies any nets connecting the symbols as well.



A Selected Cluster

Hierarchical Designs

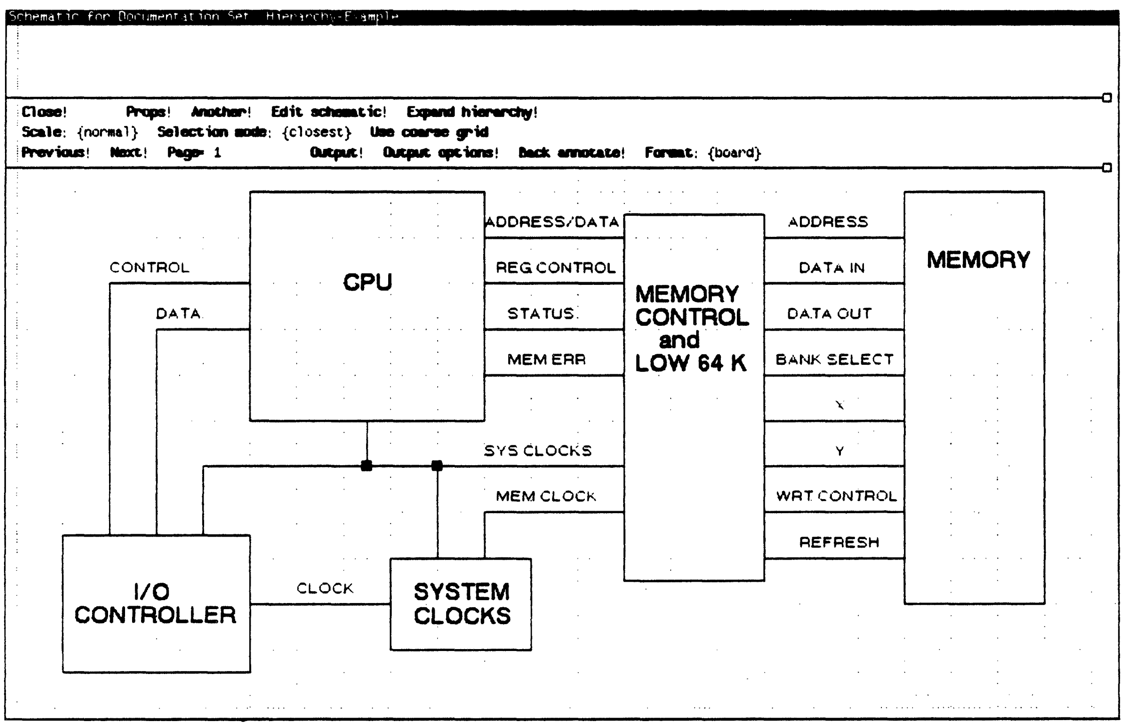
When an engineer thinks about a design, he or she often divides the design into several functional blocks, and typically draws a block diagram sketching the functional units and their relationship to each other. The engineer then considers the logic for each functional unit before drawing a schematic diagram of the whole design.

The Expert Schematics package manages design data in a hierarchical data base that parallels the way an engineer thinks about a design. Using hierarchy, the engineer can design a schematic as a series of descending blocks.

The top level of a hierarchical design is a block diagram representing the entire design. The engineer draws the block diagram as he or she would create any schematic using symbols to mark the blocks, and connecting blocks with nets. The completed block diagram, as shown in the illustration below, shows the connections between different functional blocks.

To define the circuitry for one of the functional blocks, the user displays a blank schematic page and assigns that page the corresponding block's library identification number in the page's property sheet. The engineer uses standard methods to create a schematic for the functional block. Off-page connectors mark the connections between elements in this functional block and other functional blocks.

The engineer defines each additional functional block in the same manner. The net list for the completed schematic is a listing of all the connections within functional blocks as well as connections between blocks.



A High-Level Block Diagram

Printing a Schematic

To review a schematic, the engineer can print it with a Versatec electrostatic plotter, a Hewlett-Packard pen plotter, or a CalComp pen plotter. In addition, engineers can create data files describing the schematic in COMp80 format for input to computer output microfilm equipment.

Back Annotation

After a pc board designer completes the pc board layout for a schematic, the engineer can back-annotate the schematic with information about the physical implementation of the pc board. The system supports back-annotation from printed circuit board designs created with SCICARDS and Computervision systems as well as from those created with Expert.

Design Iteration

Often engineering designs are re-worked or modified. Prototype fabrication might result in valuable opportunities to improve a product's quality or increase it's efficiency. After several years on the market, customer response might indicate ways to make a product more functional, and new technologies offer opportunities to decrease product costs and increase reliability. Expert gives its users the ability to incorporate such changes easily and efficiently.

4. The Net List

Every Expert schematic has a net list window associated with it. In the net list window, the system displays reports that aid the engineer in analyzing the schematic and aid the pc board designer in designing the board layout. These reports are:

- Net list
- Part list
- Pin list

The net list is an ASCII file. Users can output an Expert net list in machine-readable format to drive other design automation packages such as simulation, test generation and pc board design tools.

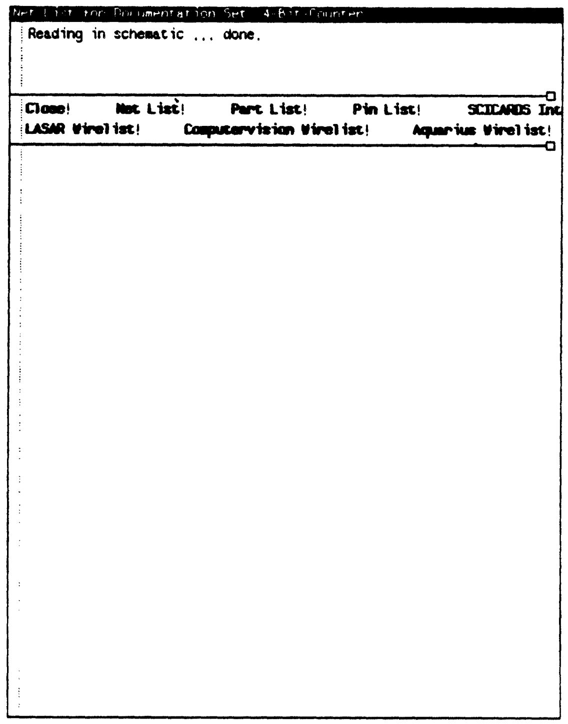
With the net list window, an Expert user can translate Expert design data to the proper format for particular engineering test and verification systems and pc board design systems. Expert supports interfaces to:

- LASAR automatic test generation system
- SCICARDS printed circuit board design system
- Computervision CADDS III and CADDS IV printed circuit board design systems

The engineer can print listings with a Versatec V-80 Printer/Plotter, a Xerox 8040 series electronic printer, or with a Diablo 630 printer. He or she can write data files to magnetic tape and use those tapes as input to the other system, or send files directly to a Digital Equipment Corporation VAX computer connected to the local network.

The Net List Window

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



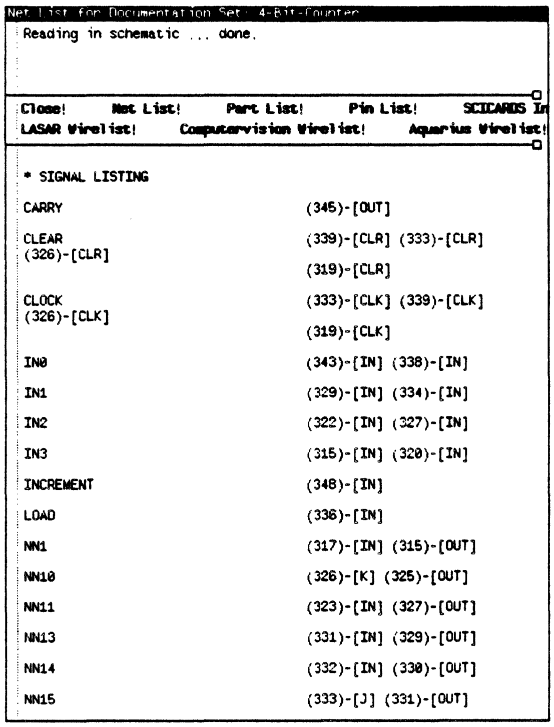
The Net List Window

The Net List

In the net list report, the system lists nets alphabetically by name along with symbol designators, part designators, and pin numbers. It shows the total number of nets and the number of points assigned to nets.

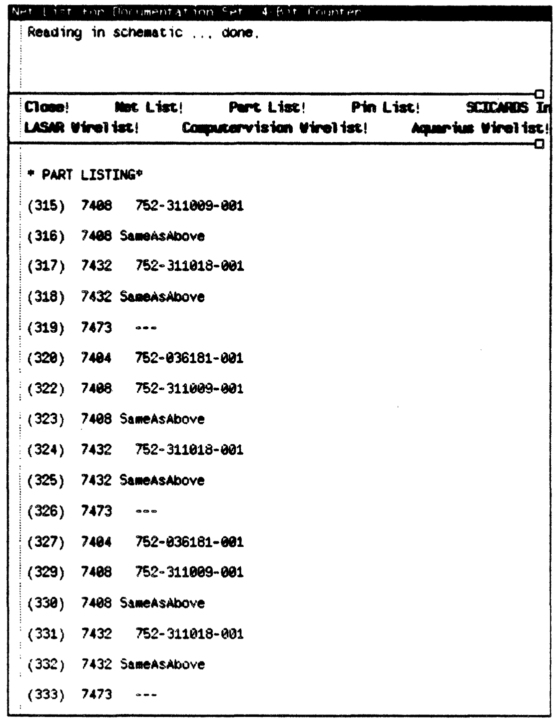
The Part List

The part list contains a component usage list for the schematic. The part list displays the symbol designator and physical part name for each symbol, as well as the total number of symbols in the schematic.



```
Net List for Documentation Set: 4-Bit-Counter
Reading in schematic ... done.

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

* SIGNAL LISTING
CARRY                    (345)-[OUT]
CLEAR                    (339)-[CLR] (333)-[CLR]
(326)-[CLR]
                         (319)-[CLR]
CLOCK                    (333)-[CLK] (339)-[CLK]
(326)-[CLK]
                         (319)-[CLK]
IN0                      (343)-[IN] (338)-[IN]
IN1                      (329)-[IN] (334)-[IN]
IN2                      (322)-[IN] (327)-[IN]
IN3                      (315)-[IN] (320)-[IN]
INCREMENT                (348)-[IN]
LOAD                     (336)-[IN]
NN1                      (317)-[IN] (315)-[OUT]
NN10                     (326)-[K] (325)-[OUT]
NN11                     (323)-[IN] (327)-[OUT]
NN13                     (331)-[IN] (329)-[OUT]
NN14                     (332)-[IN] (330)-[OUT]
NN15                     (333)-[J] (331)-[OUT]
```

A Net List

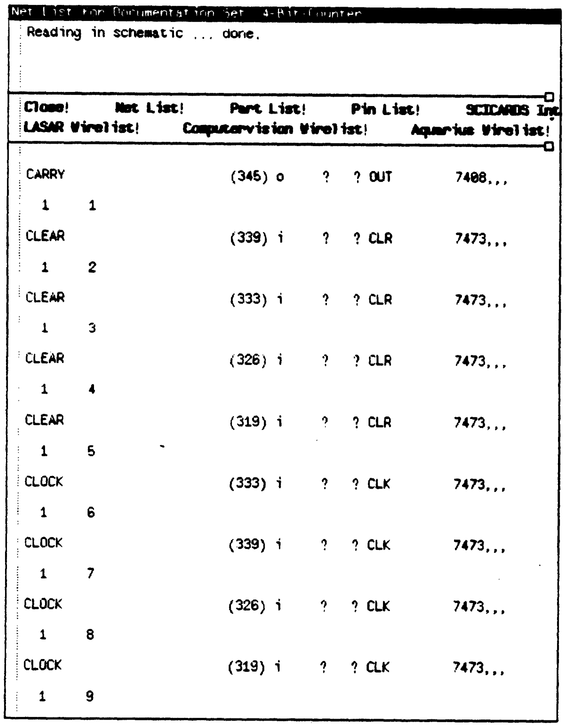


```
Net List for Documentation Set: 4-Bit-Counter
Reading in schematic ... done.

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

* PART LISTING*
(315)  7408   752-311009-001
(316)  7408 SameAsAbove
(317)  7432   752-311018-001
(318)  7432 SameAsAbove
(319)  7473   ---
(320)  7404   752-036181-001
(322)  7408   752-311009-001
(323)  7408 SameAsAbove
(324)  7432   752-311018-001
(325)  7432 SameAsAbove
(326)  7473   ---
(327)  7404   752-036181-001
(329)  7408   752-311009-001
(330)  7408 SameAsAbove
(331)  7432   752-311018-001
(332)  7432 SameAsAbove
(333)  7473   ---
```

A Part List

The Pin List

The pin list lists each net by a signal name generated by the system. For each signal it lists the symbol designator, pin type, part reference designator, pin number, pin name, physical part name, device parameters, schematic page number, and pin list line number.



A Pin List

Interface to LASAR

The user can format the net list for input to LASAR, and output it to magnetic tape with the Output Server. He or she then loads that magnetic tape onto the computer running the LASAR program. This system of test data generation allows the engineer to off-load lengthy, noninteractive processing from the workstation, and leaves the engineer free to begin new designs or reports.

Interface to SCICARDS

Design engineers can use Expert to build schematics at their own workstations and then give the schematic data to pc board designers using the SCICARDS system.

The engineer instructs the system to translate the net list to SCICARDS format and to write the net list to a magnetic tape on the Output Server. That tape serves as input for the SCICARDS system.

When the pc board layout is complete, the SCICARDS user writes the layout data to magnetic tape and returns it to the design engineer. Expert reads the data and back-annotates the schematic with information about the physical implementation of the pc board.

Interface to Computervision

The engineer can also format the net list for use with a Computervision CADDS III or CADDS IV printed circuit board design system. Again, the engineer writes the formatted file to a magnetic tape on the Output Server. Printed circuit board designers then use that tape to complete the design on the Computervision system. Once the pc board design is complete, Expert users can back-annotate their schematic with a magnetic tape from the Computervision system.

5. Expert Logic Simulator

The Expert Interactive Logic Simulator lets an engineer verify the functionality of designs created with the Expert Schematics software. The simulator is event-driven and supports a range of primitives from gates to random-access memories, read-only memories, and programmed logic arrays. Users can add primitives by describing functions in a high-level design language. The simulator fully supports MOS, TTL, and ECL technology, and provides a rich set of features for control and debugging.

The simulator derives the model of the logic network directly from the Expert schematic. The engineer supplies logic states for the design's inputs during simulation by building a stimulus file with the simulator's Interface Processing Language or by interactively defining inputs and clocking schemes as the simulation progresses. The simulator supplies the engineer with several data-derivation features, including the ability to give a group of signals a common name, to supply non-uniform timing for input values, and to divide data into *test steps*.

The simulator's set of functional primitives includes such items as register, queue, memory, counter, and serializer. The computational functions include such items as adder, comparator, shifter, and programmed logic array. The steering functions include multiplexer, demultiplexer, encoder, and driver.

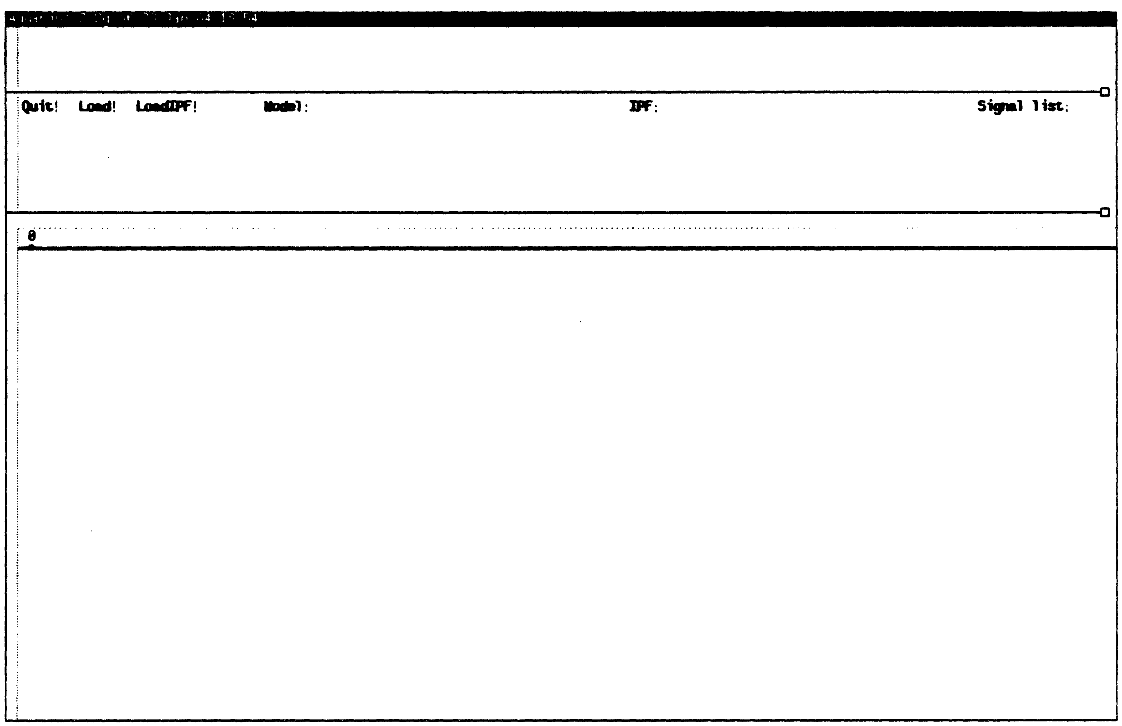
The simulation process uses nine-value logic. At any point a signal may hold one of nine values formed by a combination of *level* and *strength*. A signal's level may be low, high, or unknown, while its strength may be driven, resistive, or floating.

The engineer can select one of three methods for determining the rise and fall characteristics of logic elements. When the engineer selects *unit delay*, the simulator assumes that each logic element propagates changes at the same rate, with negligible rise and fall times. When the engineer selects *unit transitions*, the simulator assumes each logic element propagates changes at the same rate, with rise and fall times of approximately the same duration as the propagation time between two elements. This mode catches tight-loop race conditions that might otherwise go unnoticed. If the user specifies *time independent* mode, the simulator assumes that each logic element propagates changes at extremely high speeds, with comparatively slow rise and fall times. In this mode, the simulator catches all race conditions.

Because Expert's schematic design and logic simulation software run at the same workstation, the engineer can easily alternate between simulating a design and editing the schematic to correct design faults. While the simulator is running, the engineer can use the workstation for other tasks such as designing another schematic or analyzing the net list or part list for the design he or she is simulating.

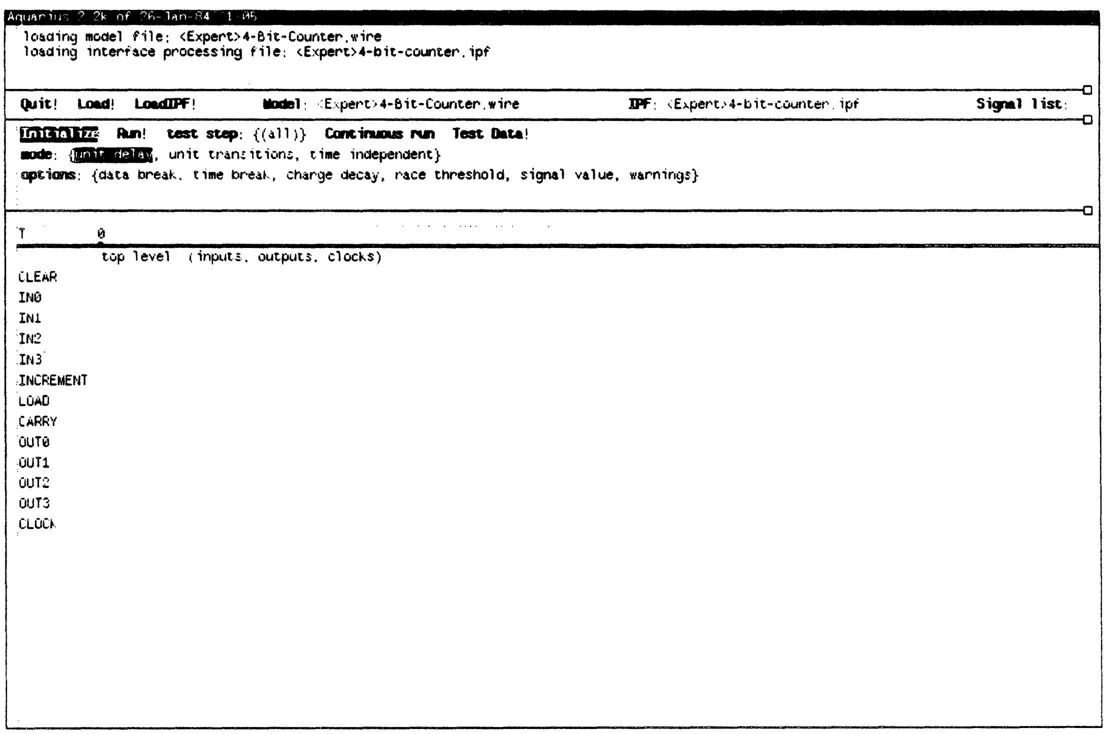
The Logic Simulator Window

Like other Expert engineering tools, the logic simulator interacts with the user through a window on the workstation display. The logic simulator window has three subwindows. The system displays status and error messages and other information in the *message subwindow*. The user provides parameters and issues commands in the *command subwindow*, and the system displays values of selected signals for the user to examine in the *display subwindow*. The following illustration shows the logic simulator window when the user first activates it.



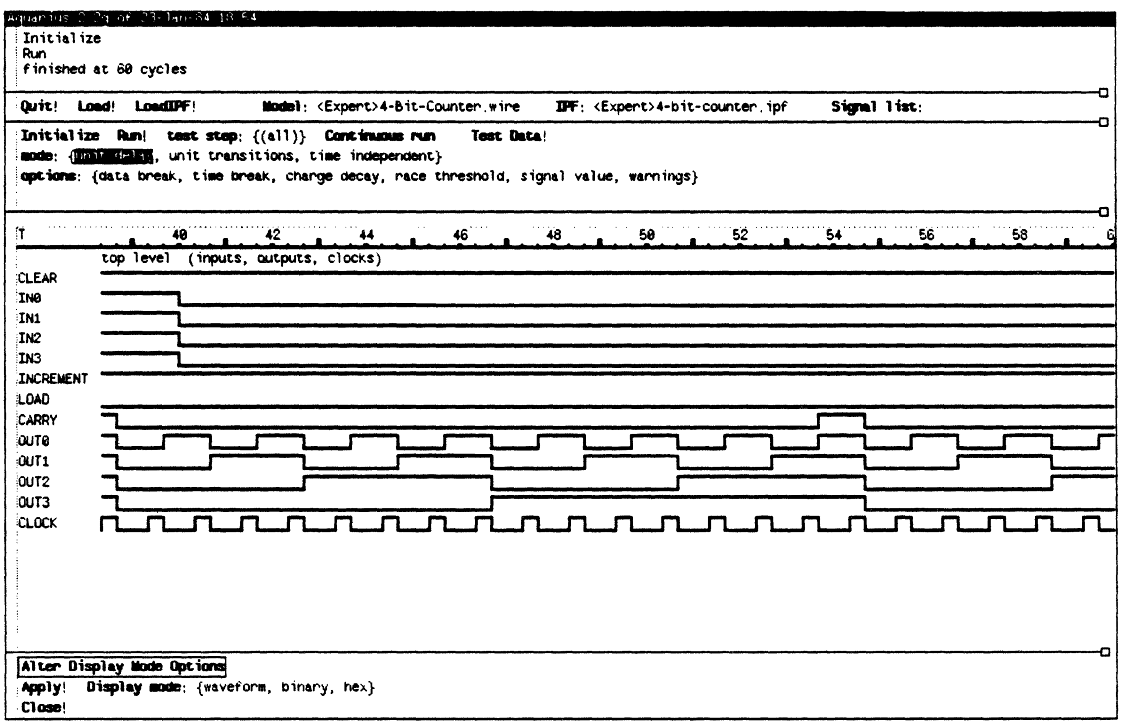
The Logic Simulator Window

Commands and options in the command subwindow change as the engineer progresses through the simulation. At any time, the simulator only displays commands appropriate for its current state. For instance, in the preceding illustration the command subwindow contains commands for loading a logic model and interface processing files (**Load!** and **LoadIPF!**) and closing the simulator window (**Quit!**). The engineer loads a logic model by entering the model name and selecting **Load!** Once the simulator loads the model, the command subwindow displays commands and options for running the simulation as in the following illustration.



The Simulator Window after Loading

In addition to selecting options and commands in the command subwindow, the engineer interacts with the simulator through dynamic menus and *form subwindows*. Form subwindows contain options and parameters that modify a command. For instance, when the user wants to alter the display format, he or she activates the Signal Ops Menu associated with the display subwindow. The user selects the **Alter Display Mode** command, and the system displays a form subwindow containing display format options. The user selects a display format then selects the **Apply!** command. The system displays signal values in the new format.



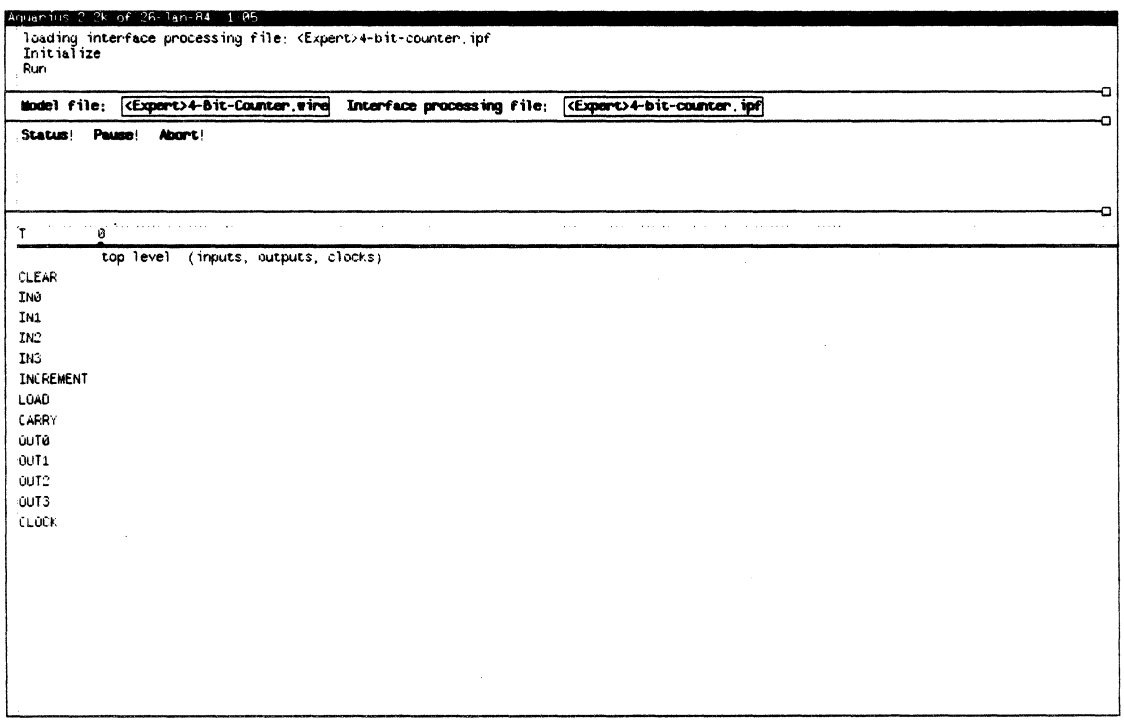
A Form Subwindow

Running the Simulation

There are two input files for every simulation: a *logic model* and an *interface processing file*. The *logic model* describes the engineer's logic design in a language that the simulator understands. The engineer creates the logic model by using the net list window to translate the schematic into the simulator's format. The *interface processing file* describes logic states for the design's inputs and timing information. The engineer uses the special Interface Processing Language to create an interface processing file.

To start the simulation, the engineer loads the logic model and the interface processing file and selects **Run!** While the simulation is running, the engineer can move the cursor to other windows, and work with other tools. For example, the engineer might display the schematic for the logic model or edit a different schematic. The following illustration shows the simulator window during the simulation run.

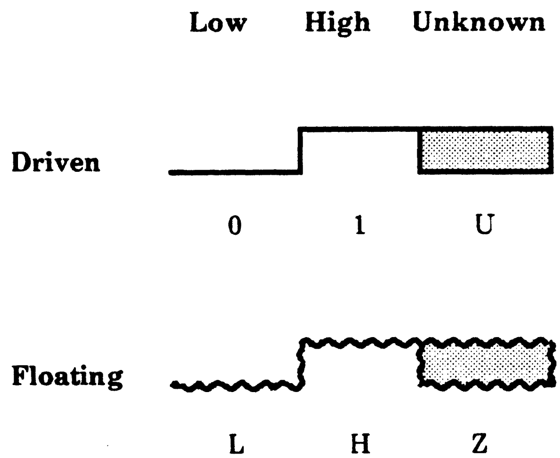
While the simulation is running, the command subwindow displays three commands: **Status!**, **Abort!**, and **Pause!** The engineer can select the **Status!** command to learn how far the simulation has progressed. If the engineer wants to end the simulation run before it is finished, he or she selects the **Abort!** command. The engineer uses the **Pause!** command to interrupt the simulation when he or she wants to resume it later.



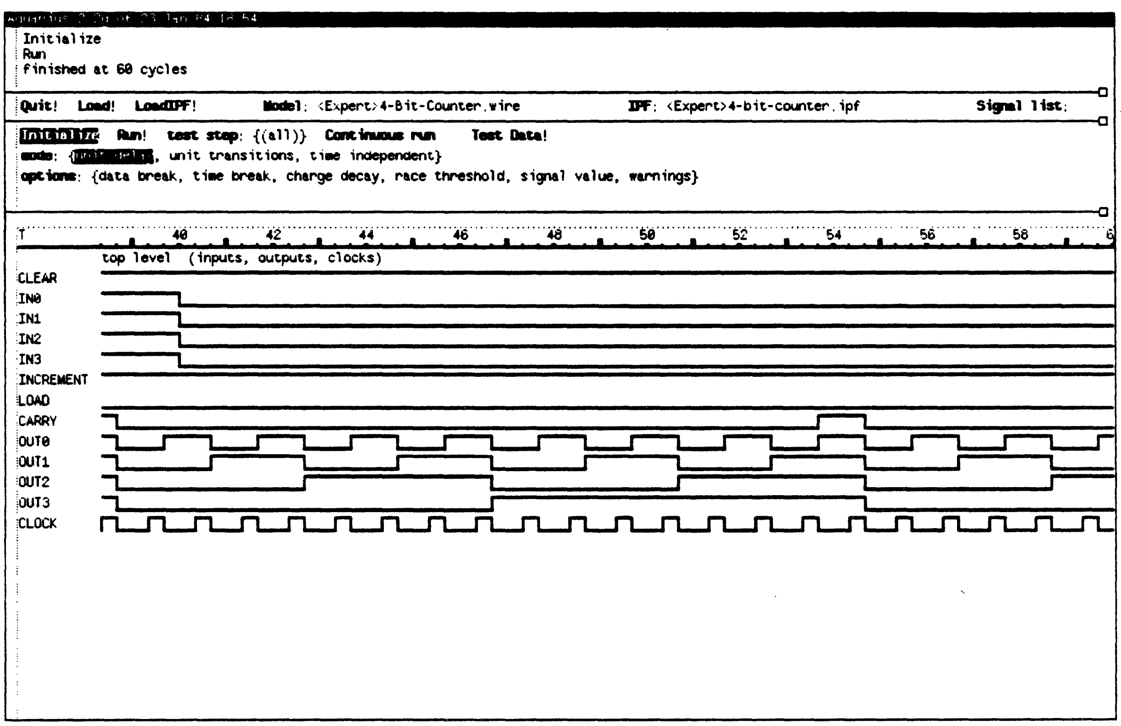
The Simulator Window During a Simulation

The Display

When the logic simulator finishes running or when the engineer asks for status, waveform displays for the signal values appear in the display subwindow. The simulator shows the values in effect at the end of each clock cycle for the entire cycle. Wavy lines denote floating values, and horizontal lines indicate changes in level from one cycle to the next. A time line across the top of the display subwindow shows the relationship between cycles and waveforms. The engineer sees the latest clock cycle and as many previous cycles as will fit in the window. The engineer can scroll the display subwindow to see hidden cycles.



Character and Waveform Representations

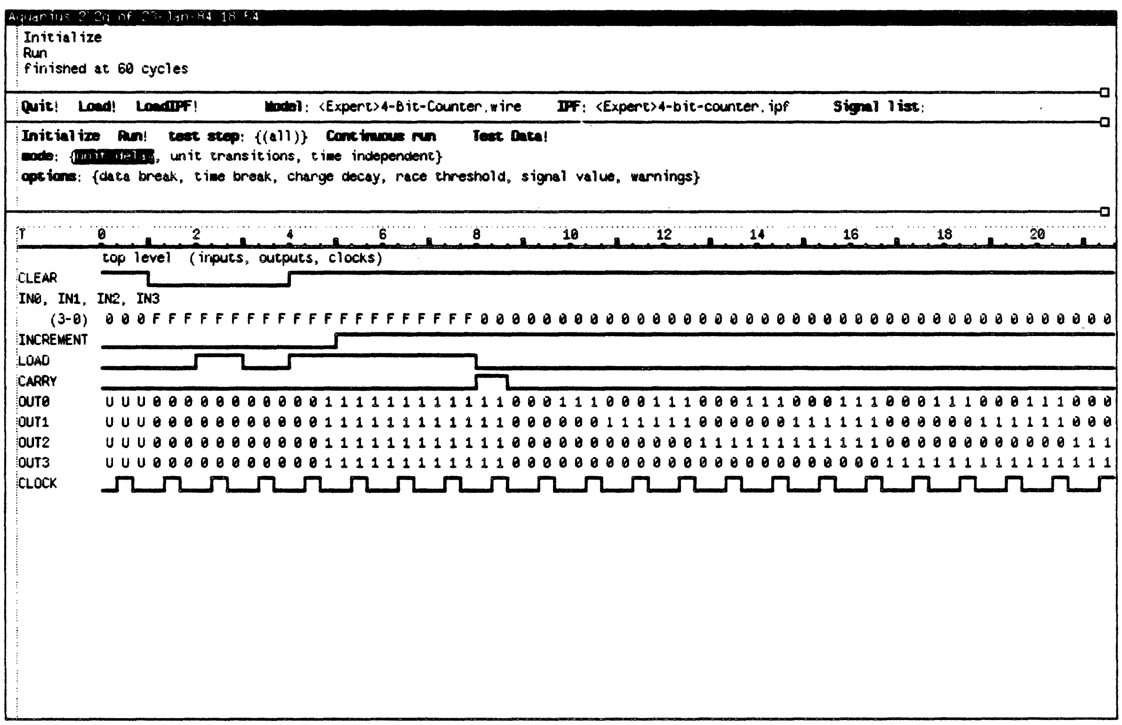


Waveform Display

The simulator gives the engineer several ways to tailor the display to his or her needs. These display options let individual users view simulated values in the manner most convenient to each user. For instance, an engineer may display the results of the simulation in binary or hexadecimal characters as well as in waveform format.

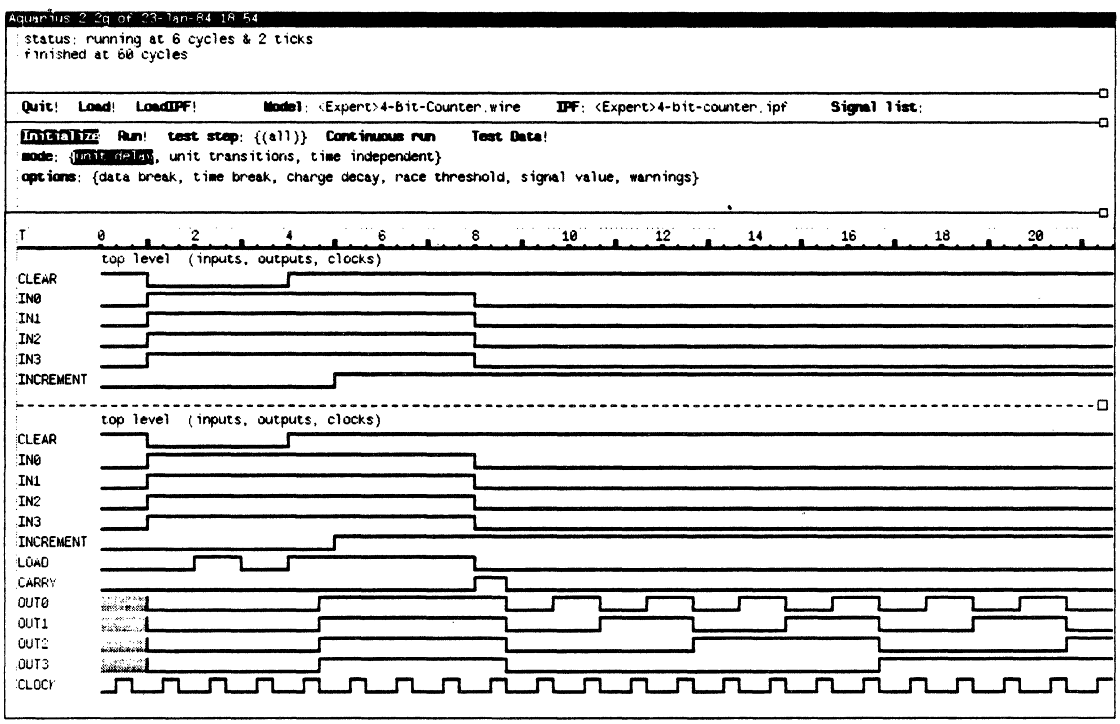
There are various methods of displaying signal frequency. The simulator normally displays signal values every clock tick. The engineer may select units of test vectors, cycles, ticks, or phases. The engineer specifies how many cycles (vectors, ticks, and phases) one interval contains, and has the option of displaying selected ticks within each clock cycle. The system numbers ticks from one to the total number of clock ticks per cycle. The engineer can select any or all ticks for display.

Sometimes it is difficult to line up waveforms with clock cycle marks and signal names. The simulator provides the engineer with crosshairs for examining signal values.



Binary, Hexadecimal, and Waveform Display

The user may split the display subwindow into two or more horizontal partitions, each with its own vertical scrolling. This feature lets the engineer view separate parts of the signal list at the same time. Dashed lines, with a small box on the right end, mark the boundaries between split sections. The illustration below shows a split display subwindow.



A Split Display Subwindow

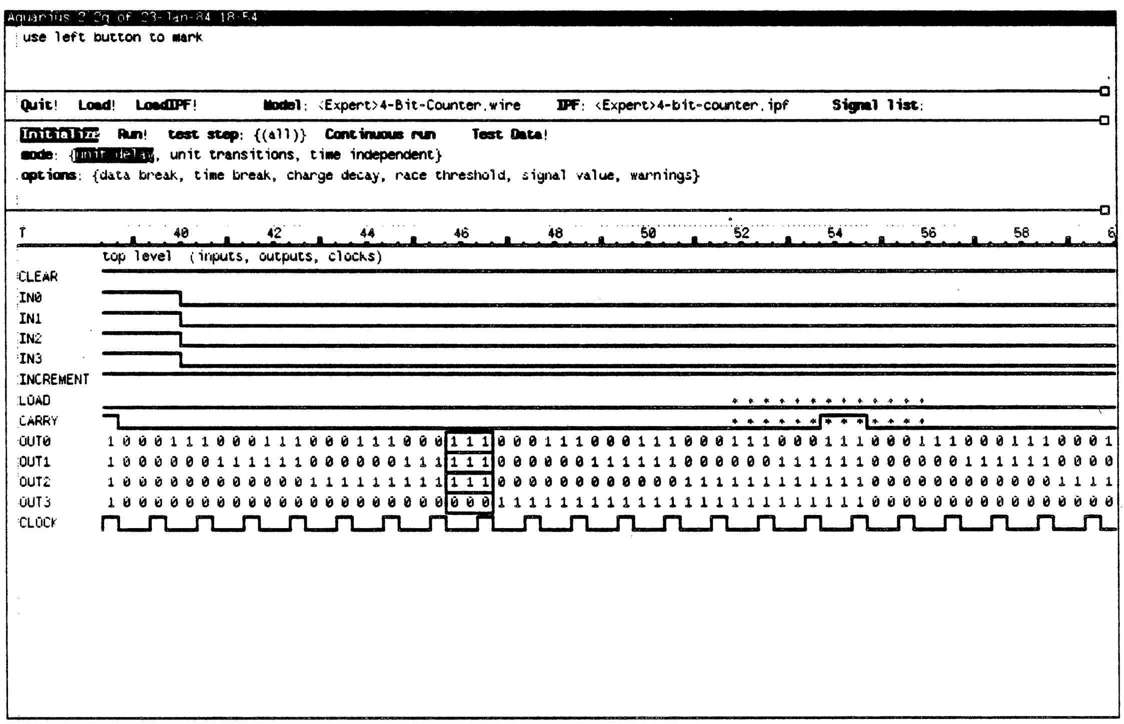
The user can mark any value or range of values on the display to distinguish them from surrounding values. The simulator indicates marked values by asterisks within waveforms or boxes around binary or hexadecimal characters. The illustration below shows several marked values.

Debugging Aids

The simulator gives the engineer a rich set of commands to use for "debugging" the logic model. Debugging options let the engineer place breakpoints within the simulation, examine signal values, change the internal controls of the simulator, and instruct the simulator to issue warnings when certain events occur. In addition, the user can add, delete, or reorder individual signals or groups of signals at any time.

The engineer has two types of breakpoints to use in debugging the logic model. A *data break* lets the engineer monitor a signal for changes in its value. The engineer can set the break to occur for any change from one of the nine simulated values to another, from floating to driven or vice-versa, or from known to unknown or vice-versa. A *time break* lets the engineer pause the simulation at a particular clock value.

Because the simulator is event-driven, changes introduced by input vectors or clocks propagate through the model until all signals reach stable values. When race conditions exist, however, the model will never settle to a stable state. For this reason, the simulator has a maximum number of iterations that it attempts before deciding that race conditions exist. Normally that number is 200. The user may change the limit to any number other than zero.



Marked Values

When the simulation pauses, the engineer can instruct the simulator to display the current level and strength of a particular signal. In addition, he or she can simulate individual faults by freezing a signal at a particular value throughout the simulation.

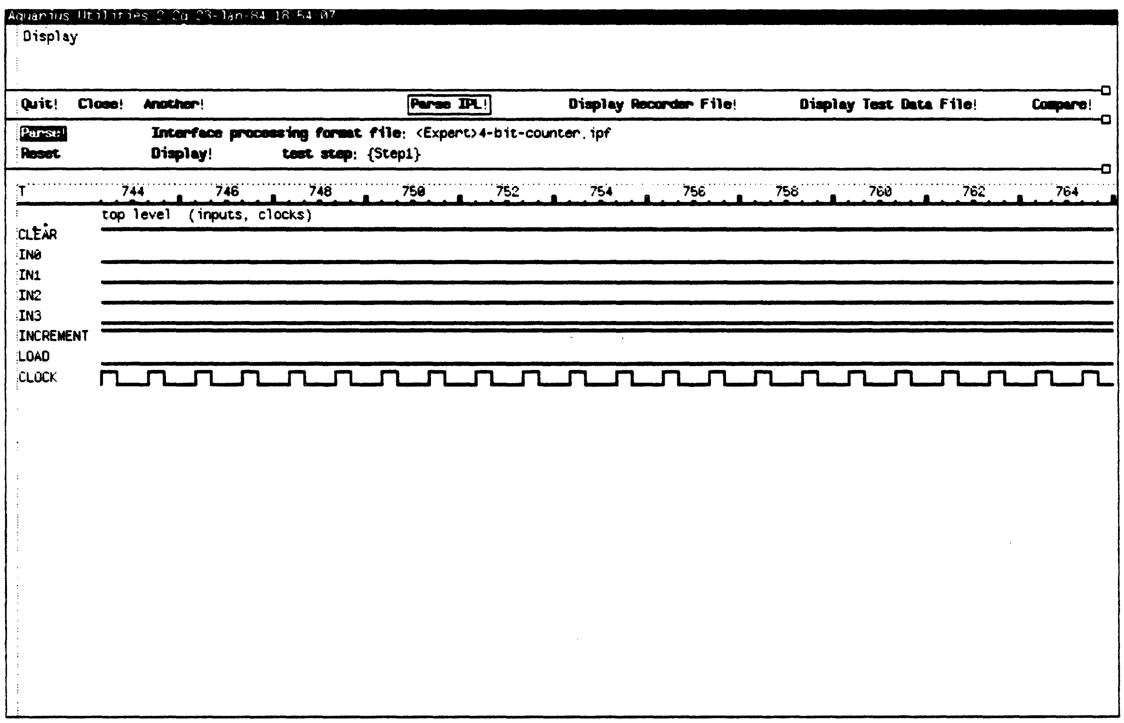
Depending on the circumstances of the simulation, certain events may be of particular interest to the engineer. The engineer can set *warnings* for such events. If the events occur during the simulation run, the simulator pauses and places a message in its message subwindow. The simulator provides three warning options: *race condition, charge decay* and *output triggering*. When the engineer turns on the race condition option, the system issues a warning whenever it detects a race condition. When the engineer turns on the charge decay option, the simulator issues a warning whenever one or more floating (or undriven) charges dissipate (the engineer can set the time taken for a floating charge to dissipate). When the engineer turns on the output triggering option, the simulator issues a warning whenever feedback through a memory element's output connections (for example a latch or a flip-flop) changes its value.

The Simulator Utility Tool

Expert provides the user with a utility tool that aids the user in running simulations.

With the simulator utility tool, a user can preprocess an interface processing file to display the input pattern vectors it generates. If the simulator utility tool finds errors in the interface processing file, it displays both a list of errors and the interface processing file for the user to edit.

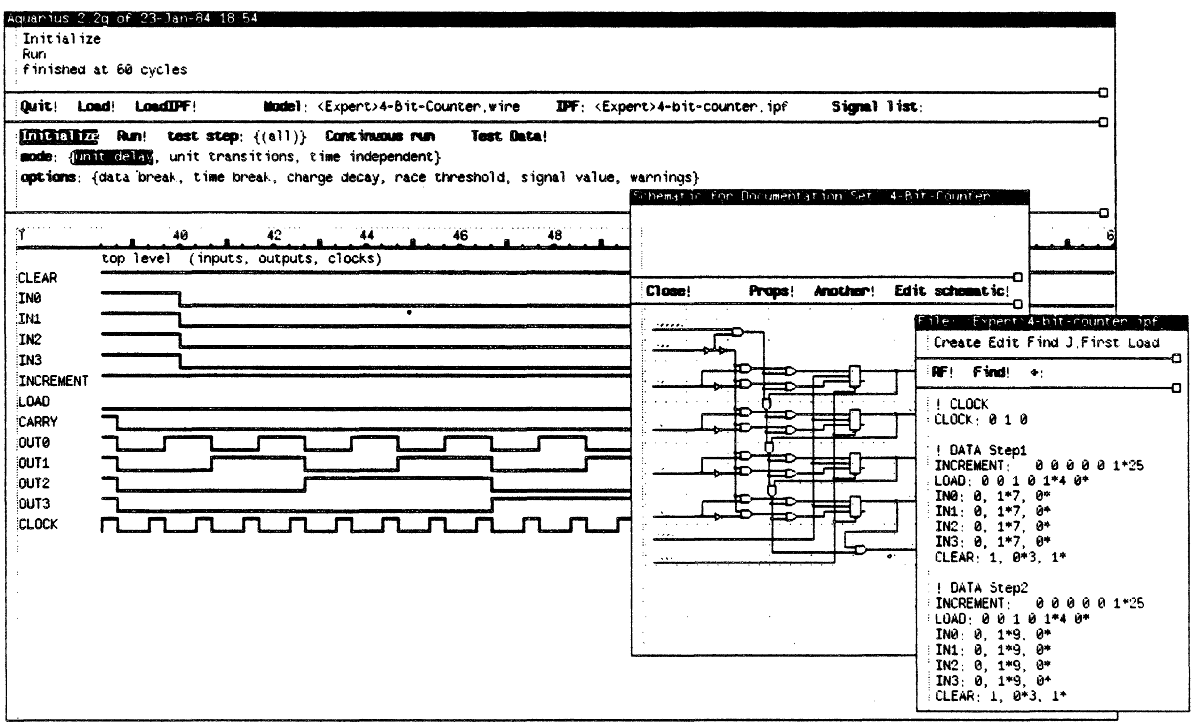
The simulator utility tool also compares predicted and simulated test data in order to detect flaws in a logic model, or compares the results of two simulations. It displays any discrepancy between simulated test data and predicted output or between the output of the two simulations for easy reference.



The Simulator Utility Tool

Logic Simulation with the Engineering Workstation

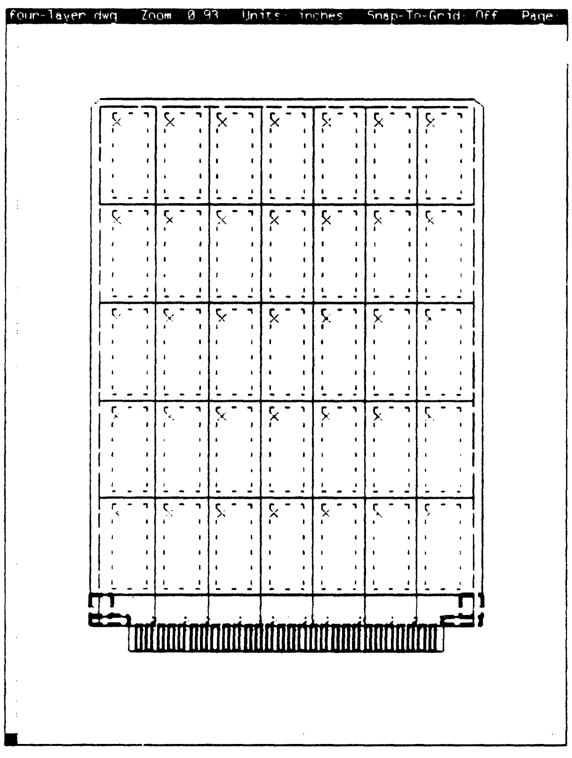
With schematic design and simulation tools at the same workstation, it's easy to alternate between testing and editing a design. To correct design faults found with the simulator, the engineer simply opens the schematic window and edits the schematic. The figure below shows the results of the simulation on the engineer's display, a schematic window, and the interface processing file.



Windows on the Engineer's Display

6. PC Board Definition

While an engineer is designing a schematic and verifying his or her design, a packaging engineer uses Expert's drafting package to create a board drawing for the design. The engineer edits data tables and constructs lines, curves, and rectangles to represent the outline, placement area, routing channels, and restricted areas required for any board.



A Sample Board Outline

By creating this geometry using the eleven line styles dedicated to pc board definition, the engineer tells the system to give specific electronic meanings to each item. The figure below shows a typical board drawing as it appears on the workstation display.

Data Tables

The pc board drawing format supplied by Versatec contains nine *data tables* used to define each board. Users can edit these tables at any time before placing parts on the board. These tables include a Layer Description Table, an Aperture Table, a Silkscreen Aperture Table, a Clearance Table, a Trace Table, and four Pad Stack Tables for part pins and vias.

While some entries vary from board to board at a given installation, most tables remain the same after the user has created a number of boards. The system saves all table entries with the drawing.

The *Layer Description Table* defines each board layer for boards from two to sixteen layers. Each layer may be either a power, ground, signal vertical, or signal horizontal layer.

The *Aperture Table* defines the photoplotting characteristics of up to 255 separate apertures while the *Silkscreen Aperture Table* defines the aperture number used for photoplotting silkscreens.

The *Clearance Table* lists the nominal and minimum clearances in mils between any two pins, vias, traces, and areas.

The *Trace Table* defines characteristics of each trace type.

Finally, the four *Pad Stack Tables* describe the characteristics of apertures when pad stacks for parts and for vias either connect or do not connect on particular board layers.

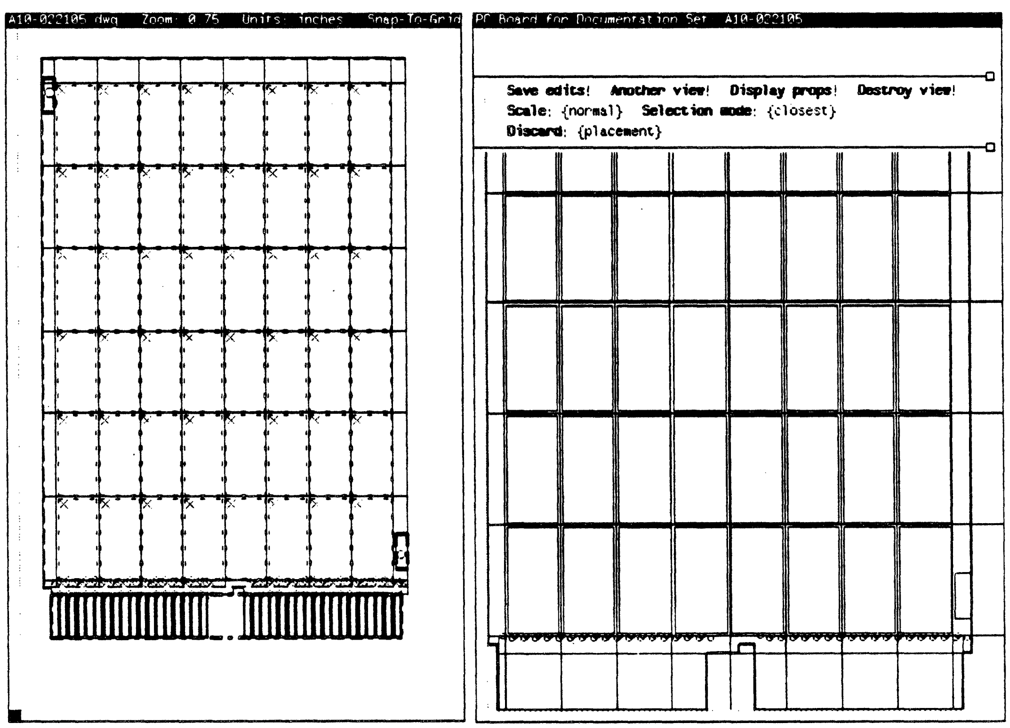
Board Validation

The **Board-Design** command window contains commands for validating and updating board drawings and tables. The engineer uses some of these commands to define the fingers on the board's edge connectors.

The **Validate board definition** command causes Expert to check the board drawing for completeness and for adherance to basic design rules. Expert lists design rule violations in the message window and, in most cases, highlights offending objects on the display.

Once a drawing meets all design rules, the system creates a place and route file in the board's documentation set. At this point, the pc board designer can begin to place parts on the board.

Should the pc board designer choose to modify the board drawing during placement or routing, he or she updates it with a single command. Similarly, the packaging engineer can send changes made in the drawing to the board's place and route file.



A Valid Board Drawing and its Place and Route Window

7. Expert Placement and Routing

After an electronics engineer designs a schematic with the Expert Schematics package and a packaging engineer has used the Expert Drafting package to create the design's board drawing, a pc board designer uses Expert's Placement and Routing package to place parts on the board and route the connections between parts. The system lets the designer do these tasks in the way that he or she finds most efficient. A user may place and route a single net, several nets, or a complete board at once.

The user can choose to place parts either interactively or automatically. Interactive placement gives the user the flexibility to tailor the design to particular needs. After the designer has interactively placed key parts, the system's automatic placement feature can quickly complete the placement. In either case, the system assures that the final pc board conforms to the schematic.

The system's two-phase routing system parallels the way a designer thinks about designing a printed circuit board. During the first phase, called *global routing*, the system assigns connections, or nets, to pre-defined channels. Global Routing sketches the approximate direction, but not the exact path, of a connection. The system chooses the best path based on a maximum channel density set by the user.

Once the global router has sketched the flow of connections, the user can edit the global routes, modify the placement, or move on to *etch routing*. During etch routing, the system defines the actual etch trace for each global route. The designer can manually etch route critical connections, or automatically etch route all global routes at once. In addition, the designer can easily edit complete etch routes.

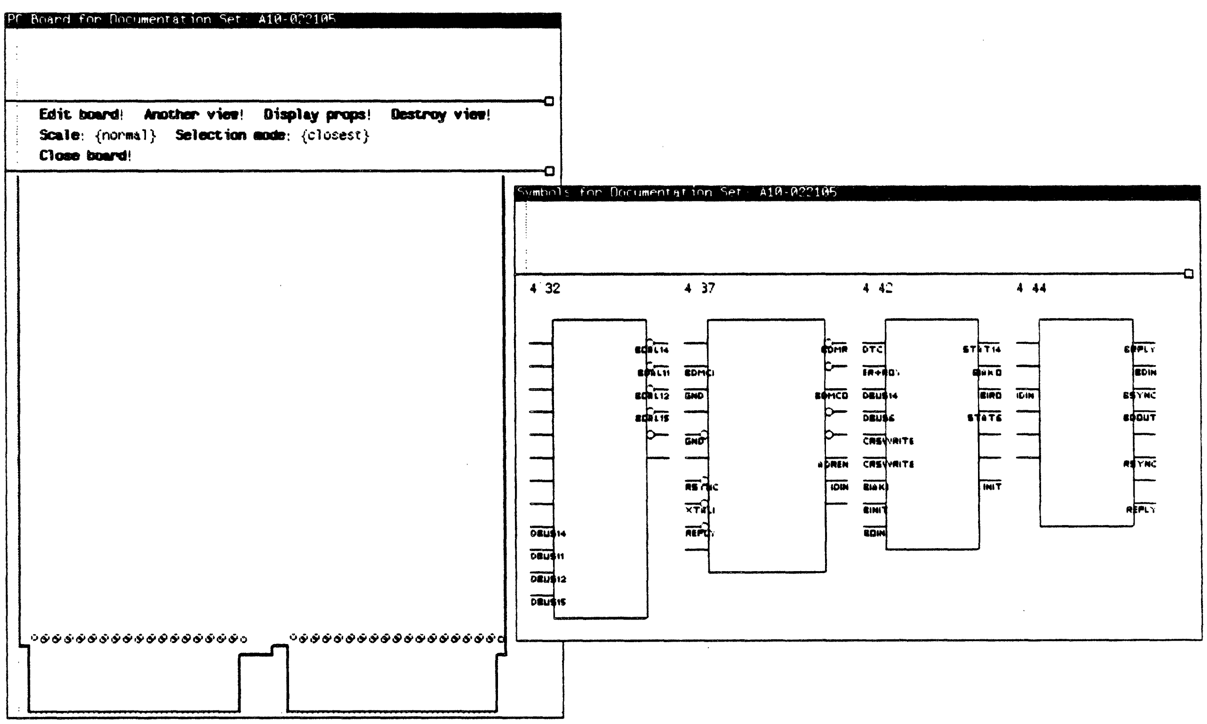
Unlike many other automated pc board design tools, Expert's etch router is not based on a grid, but on manufacturing rules. The system routes traces at multiples of 45°, making more effective use of the space available on the board. The system routes digital and mixed digital and analog boards of up to 16 layers with up to 350 equivalent integrated circuits.

Placement and Routing Windows

A pc board designer places parts and routes connections in a *pc board window*. A *symbol window* displays the logic symbols that the designer assigns to parts on the pc board. To display pc board and symbol windows for a design, the pc board designer opens that design's documentation set, selects **Place and Route**, and presses the **OPEN** key.

The symbol window has two subwindows, a message subwindow and a symbol subwindow that displays logic symbols from the schematic.

The pc board window has three subwindows. The first subwindow displays messages from the system. The second contains commands for the pc board window. The third displays the pc board as specified by the packaging engineer.

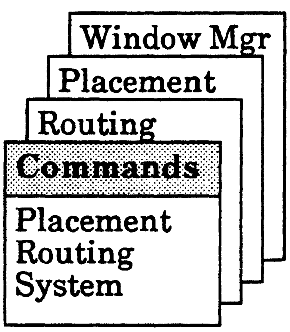


PC Board and Symbol Windows

Users can display as many views of the pc board window as they want. Two pc board windows might show the same pc board at different scales or display different routing layers.

The pc board subwindow has four dynamic menus associated with it. The Window Manager Menu controls the pc board window's size and location on the display. The Placement Menu contains commands for placing parts on the pc board. The Routing Menu contains routing commands. The Commands Menu controls which set of commands the system displays in the command subwindow.

The designer uses the Commands Menu to display one of three sets of commands in the command subwindow. Each set of commands applies to a different design task. Placement commands control part packaging and placement. Routing commands affect global and etch routing. System commands include commands for saving or discarding changes to the board layout and commands that close the pc board window.



The Four Menus in the PC Board Subwindow

Display and Routing Properties

The pc board window has two sets of properties: *display properties* and *routing properties.*

Display properties control which objects the system displays in the pc board subwindow and how it displays them. For instance, the user may choose whether or not to display channel boundaries, location borders, and the board outline. He or she uses the display property sheet to tell the system which layers to display, and what line style to use to display etch routes on each layer. Display properties apply only to the window to which the user assigns them; they could be different for two windows displaying the same design. For example, the user might create two views of the pc board window. One might display internal layers; the other could display external layers.

Routing properties control global and etch routing. With the routing property sheet, the user can control the maximum channel density during global routing and the layers that the system routes during automatic etch routing. The user can instruct the system to begin etch routing on either internal or external layers, and tell the system to stop routing after a particular etch routing phase.

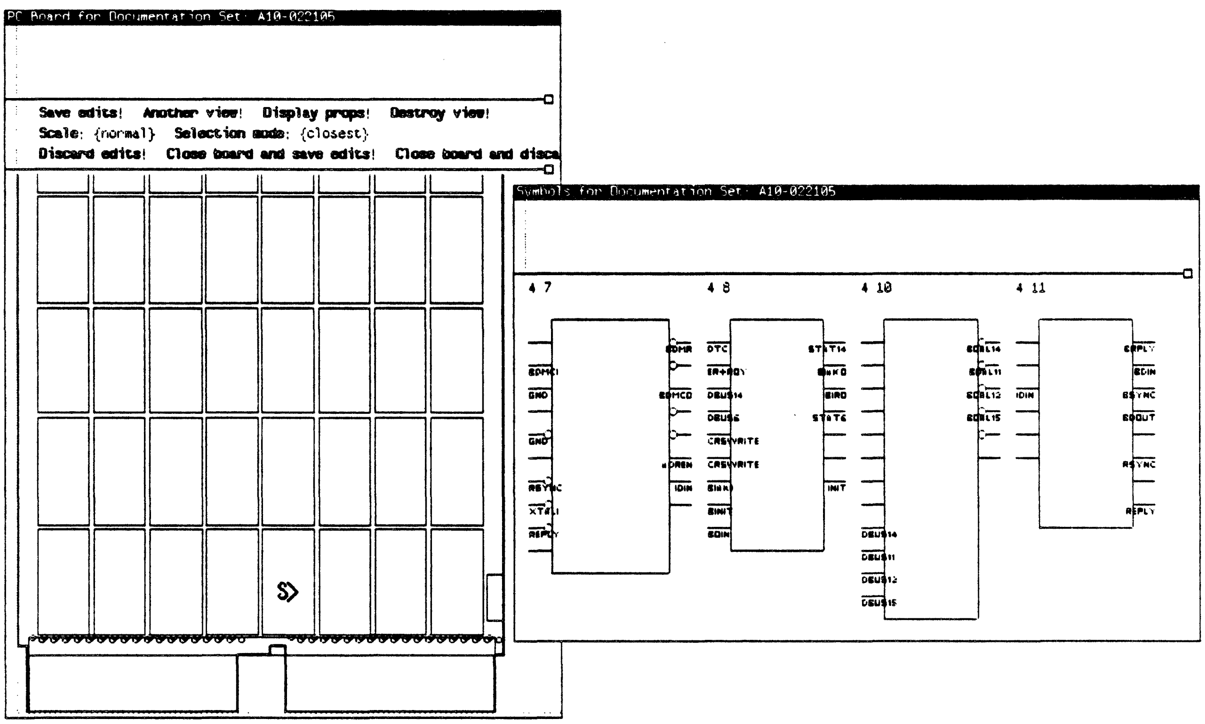
Placing Parts on the PC Board

The system supports four methods of pc board packaging and placement. The first three methods are interactive; the user exercises complete control over the design. The fourth method is fully automatic. The designer can switch from one method to another using any combination of the four methods.

Using the first method, the designer selects a board location and instructs the system to display symbol choices. The system scans the schematic and pc board, and displays the most appropriate choices in the symbol window. If the designer has already placed a part at the location, the system displays only the symbols that make sense to place inside that part.

To aid the user in choosing a symbol, the system displays the number of pins on the symbol that would connect to pins already placed on the board, and the estimated rectilinear length of those connections. The user can see the symbol's relationship to others by instructing the system to display the symbol in the schematic, or by displaying all symbols connected to the symbol in the symbol window.

The user selects a symbol and instructs the system to assign it to the location. The system automatically assigns the symbol to the corresponding part and places the part at the location. If it finds more than one corresponding part, the system asks the user to select a part from the choices.

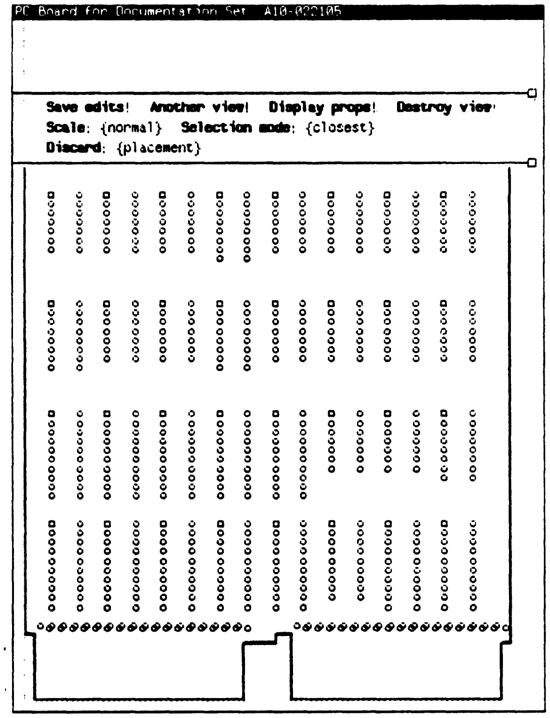


Symbol Choices inside the Symbol Window

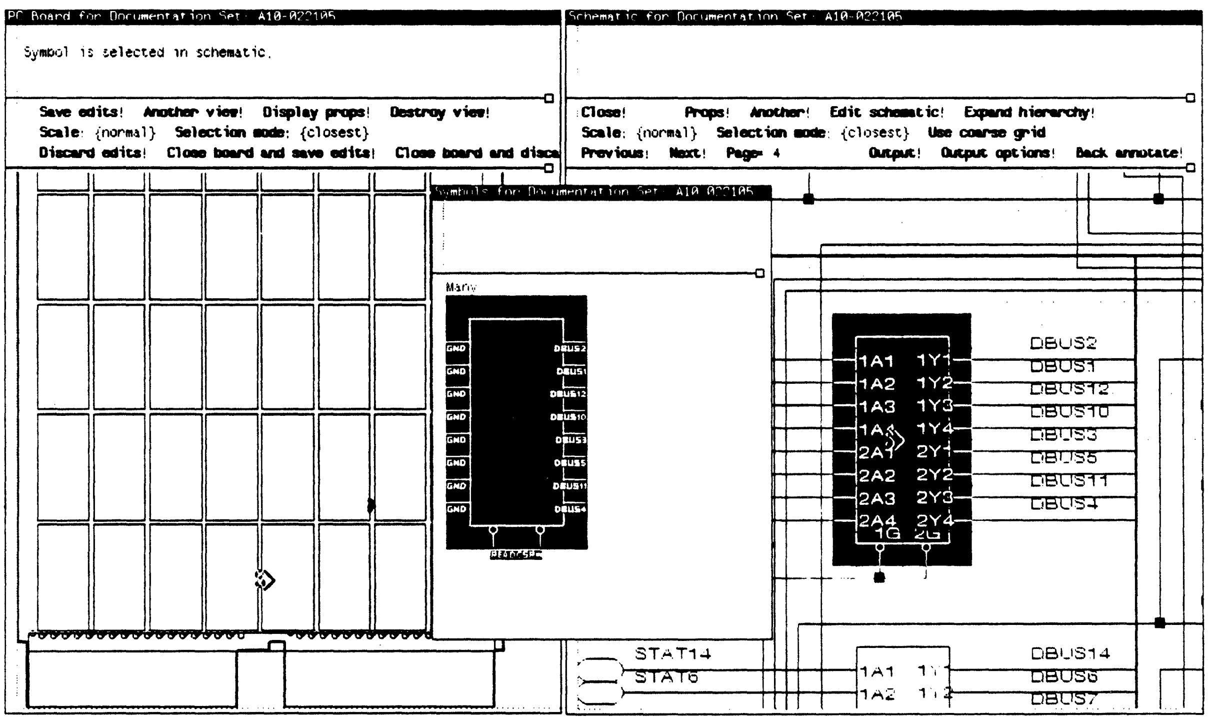
Using the second method, the designer selects a board location, selects a symbol in the schematic, and instructs the system to place the symbol at the selected location.

With the third method, the designer can place a discrete device at any point on the board. The designer selects the logic symbol for the device in either the symbol or schematic window, marks a destination location on the pc board, and instructs the system to place the symbol at the destination. When choosing where to place discrete devices, the designer can view connections between devices by selecting a pin and asking the system to highlight all pins connected to it.

The fourth method is completely automatic. After using interactive methods to place critical parts, the designer can use the system's automatic placement to quickly place the remaining parts.



Placed Parts



Cross-Reference from PC Board to Schematic

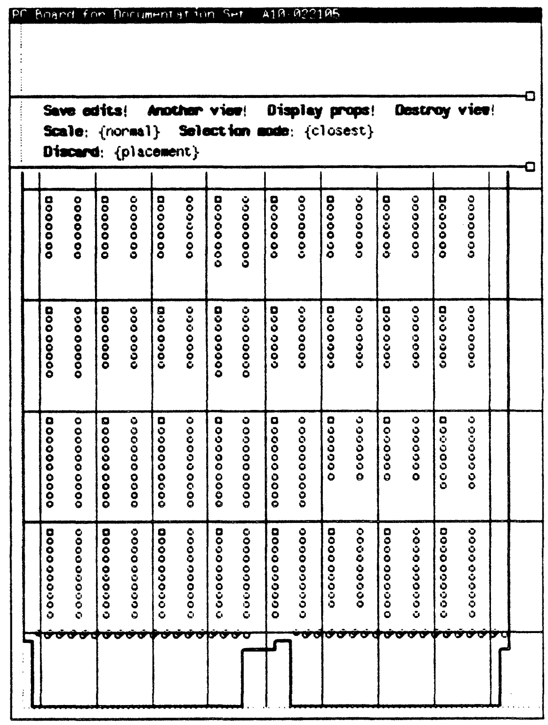
Editing Part Placement

A few simple functions give the designer the ability to change the board design. If he or she sees an opportunity to improve the placement, the user can remove symbols assigned to a particular part, swap equivalent symbols within a part, swap equivalent pins, move parts to different board locations, and delete parts from locations.

In addition, each board location has a property sheet defining its size and orientation. The user can change these properties by editing the property sheet.

Global Routing

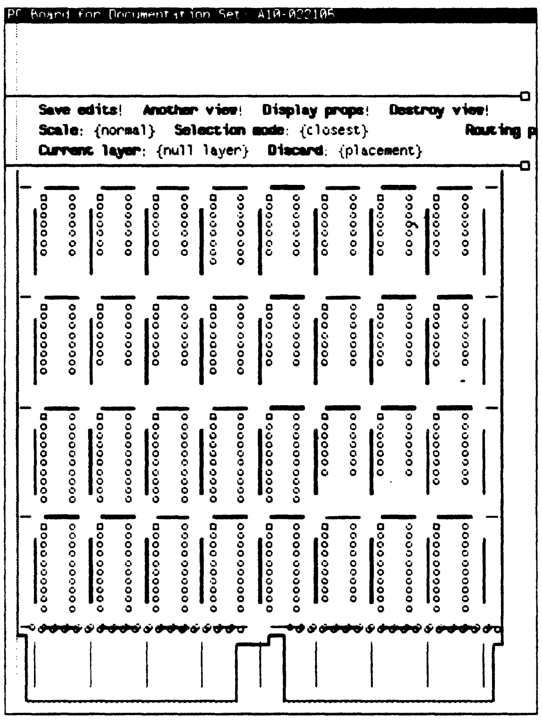
During the global routing phase, the system sketches approximate paths of connections by assigning connections to predefined routing channels. The user controls global routing by setting a maximum limit for channel density and by moving, copying, and deleting channel borders. The illustration below shows routing channels in the pc board subwindow.



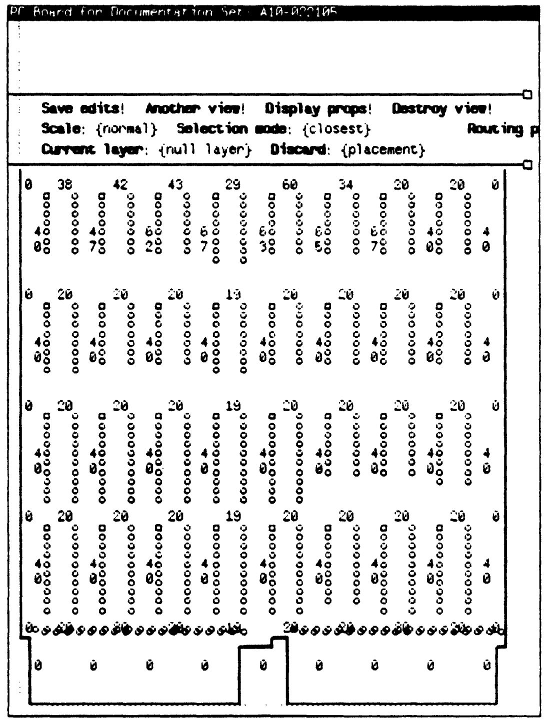
Routing Channels

The user can instruct the system to display the density of each channel in the pc board subwindow. The system displays channel density in one of two ways: graphically, by showing a bar between channel regions (the bar width increases as the channel approaches 100% utilization), and numerically by showing the exact percentage of utilization.

Before global routing, the user sets the maximum channel density in the routing property sheet, and specifies how density is displayed with the display property sheet.



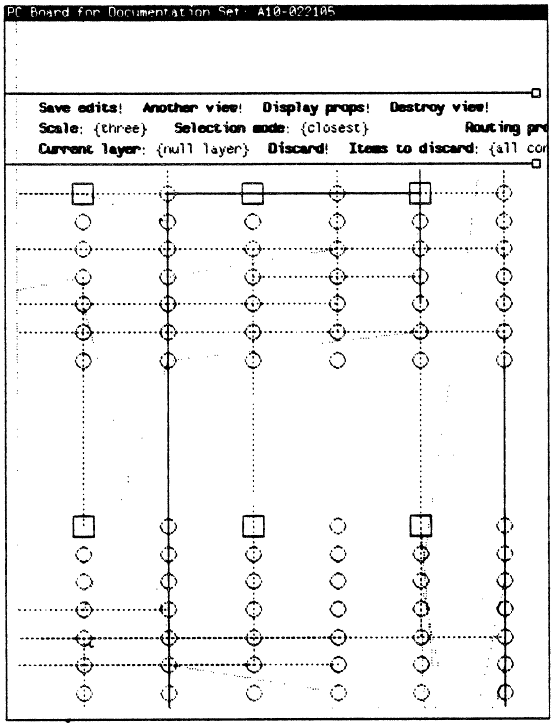
Graphic Density Display



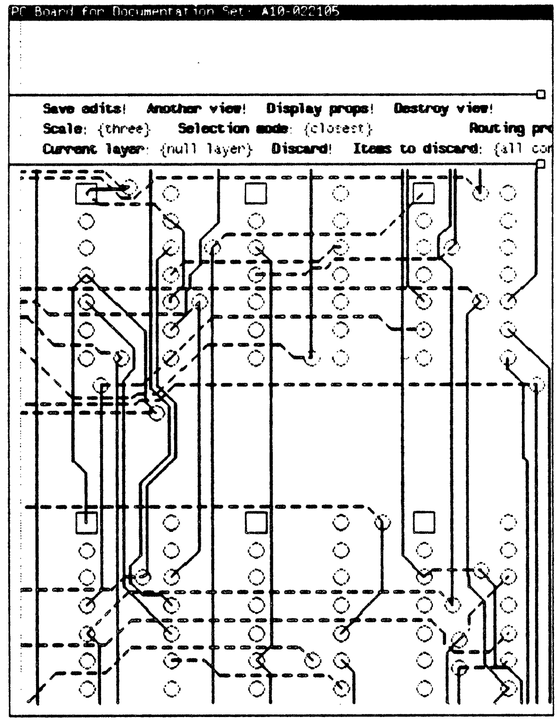
Numeric Density Display

Whenever the user has placed all the pins on a net, the system considers that net *placed*. At any time, the user can instruct the system to route placed nets. The system chooses the best path through the channels for each net, displays the path taken, and updates the density display.

If the system cannot globally route a net within the density requirements set by the user, it searches for an alternate path. If it cannot find an alternate route for the net, it marks that net as incomplete. The user can manually route incomplete nets, or change the density requirements and reroute.



Globally-routed Nets



Etch-routed Nets

Etch Routing

Etch routing determines the actual etch pattern and the location of vias for each global route. The user can manually route individual routes, or instruct the system to automatically route all complete global routes.

Automatic Etch Routing

The system gives the user several ways to control automatic etch routing. With the routing property sheet, the designer can specify that the system route only certain layers, limit the number of etch bends in the routing channels, and stop routing after a particular phase.

The designer controls which layers the system routes and on which layers it begins routing. The designer chooses to route either all layers or any pair of vertical and horizontal layers, and also specifies whether the system begins by routing internal or external layers.

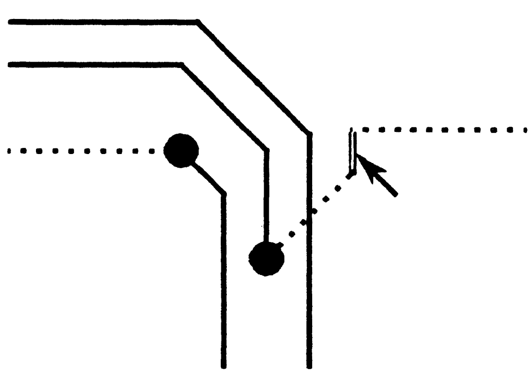
The designer can control the number of etch bends within routing channels by setting a maximum angle for bends within horizontal and vertical channels. When the user allows a greater number of bends within a channel, it is easier for the system to route connections within that channel, but it is more likely that routes will interfere with others. With fewer bends, routes are less likely to interfere with each other, but it becomes more difficult for the system to route all connections. The system relies on the designer's expertise to choose the optimum situation for each board.

During automatic etch routing, the system routes connections in three phases; first it routes connections between pins, second it routes connections between pins and existing vias, and finally it routes *crossovers* for which it must choose via placement. The user controls automatic routing by instructing the system to stop routing after any one of the three phases. For example, when the designer wants to choose locations for new vias, he or she instructs the system to stop routing after the second phase, and then manually routes any incomplete connections.

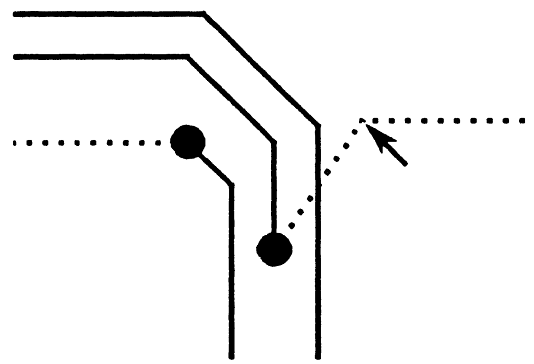
Manual Etch Routing and Editing

The designer can use manual routing techniques to route critical connections before automatically routing, to edit etch routes drawn by the system, or to route connections left incomplete after automatic etch routing. To manually route a connection, the user first selects the connection, and selects the **Edit etch** command. The designer then draws the new etch route, segment by segment, between old endpoints or between any two other points on the connection.

If the user sees ways to improve an etch segment, he or she can use any of four special commands for editing etch. The **Drag** command moves the etch segment in a direction that is at a right angle to its direction. The **Stretch** command lengthens an etch segment. The **Notch** command creates a trapezoidal detour around another object, for example a via. The **Drop** command moves an etch segment to different layers, creating or removing vias when appropriate. The following illustrations show an example of dragging an etch segment.



The designer selects the etch segment, and issues the Drag command.



The designer moves the cursor to mark the new endpoint of the segment.

Interface to Manufacturing

Once a board layout is complete, Expert increases manufacturing productivity by producing data tapes to drive Gerber photoplotters. Before photoplotting, users can review checkplot drawings created by electrostatic or pen plotters.

The Expert Drafting **Board-Design** command window contains commands for generating checkplot and photoplot drawings of a completed pc board. Selecting the **Checkplot** command converts the board design into a hardcopy plot. Standard checkplot data tables let the user determine the appearance of all holes, pads, traces, silkscreens, and etch incompletes on the checkplot.

A single command, **Photoplot**, generates the various photoplot drawings required for manufacturing a completed board. The user can edit these drawings to include additional annotation, for example, or to make other special changes to the the artwork. The user writes the edited photoplot drawings to magnetic tape for transfer to a Gerber photoplotter.

In addition to checkplots and photoplots, the system creates manufacturing drawings and machine-readable documentation packages. A user can create different documentation packages for manufacturing, field service, and archiving.

8. Expert Drafting

The Expert drafting package is a personal tool designed to aid mechanical engineers, designers, and drafters in creating drawings. In unison with other Expert software, it also provides special features for the design of printed circuit boards.

Its user interface makes using the Expert drafting package as easy as drawing manually. A user completes a drawing by first creating a number of basic geometric shapes. From these shapes, the user constructs more complex geometrical objects such as filled areas and standard parts. The system records and displays a description of each object created and computes dimensions automatically. Mistakes are easy to correct and design changes are easy to incorporate.

Parts libraries accelerate the design cycle. Instead of redrawing complex mechanical or electrical parts, the user can copy a part from a library to a new location in another drawing.

A user typically chooses to send finished drawings to one of several output or filing devices. These devices include plotters, File Servers, magnetic tape drives, the floppy disk drive, or VAX host computers. Expert also converts drawings to and from IGES (Initial Graphics Exchange Specification) format for exchange with other computer-aided design systems.

With its human-engineered design tools, powerful engineering workstation, and Ethernet communications network, the Expert drafting package provides a cost-effective way to increase engineering productivity.

Drafting User Interface

An engineer using the Expert drafting package interacts with the system through *drawing windows*, *command windows* and their *pop-up menus*, the *mouse*, and the *keyboard*.

Drafting Windows

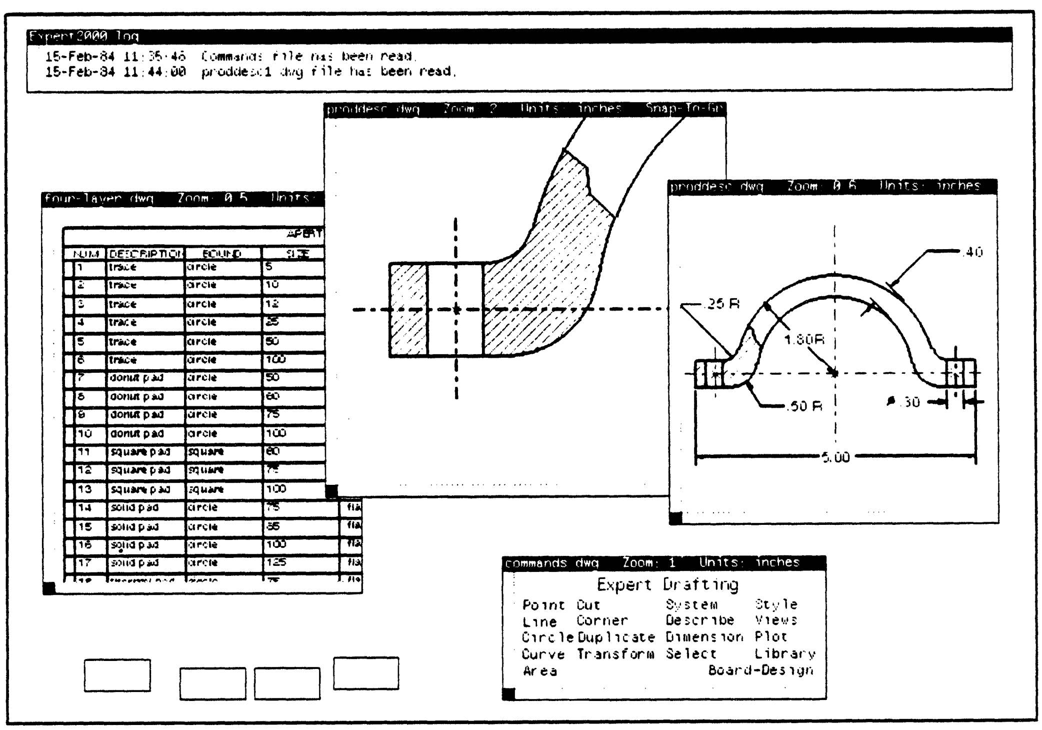
The engineer creates and edits drawings inside drawing windows. Drawing windows are rectangular areas on the display framed by narrow, black borders. The engineer may have any number of drawing windows on the display at one time.

Some of these, for example, may show the same drawing at different magnification or *zoom* factors. Others may display different pages of the same drawing, while still others may show views of different drawings. The black name frame at the top of each drawing window contains the drawing's name, its zoom factor, the linear units, the status of the snap-to-grid feature, and (for multi-page drawings) a page number.

An engineer can magnify, scroll, and change drawing window pages to show only the portion on which he or she is currently working. The engineer also determines the dimension style for each drawing. Linear unit types are: mils, inches, and feet, millimeters, centimeters, and meters. Angular unit types are: degrees, degrees and minutes, degrees, minutes, and seconds, radians, and slope.

The Expert message window displays a record of most drafting activities. This short and narrow window normally spans the top of the display to show only a few lines of system feedback at a time. The engineer, however, can change its shape or make it display more or less information. The system reports normal user actions with a simple message in the message window. It reports failed actions and errors with a message, a screen flash, and an optional beep.

The command window contains all drafting, electronic part library, and board outline commands and their associated parameters.



Drawing Windows on the Display

Selection and Deselection

Selection procedures single out commands and the objects on which to perform them. To *select* a geometric object, the user points the cursor at the object and presses the left mouse button. Selected objects blink. To *deselect* the object the user points to the object and presses the right mouse button. The object stops blinking.

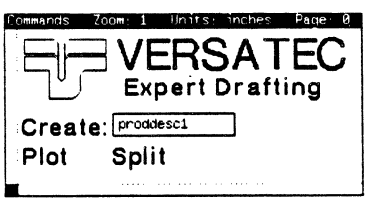
The user can select any number of objects inside a single drawing. Selecting an object in another drawing deselects all objects in the first.

Other selection commands let the user select and deselect all of the objects in a window, select objects within a rectangular area, select contiguous objects, and join several objects together into a *cluster*.

Command Input

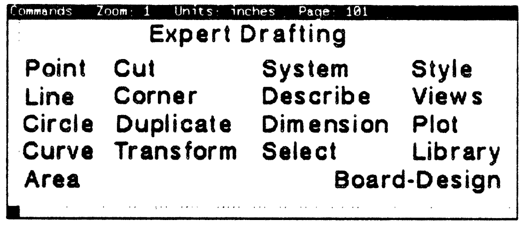
In general, the user issues commands to the system in the drafting command window. The command options shown inside the window change when the user changes the current selection or when the user selects a command requiring further definition. The command window also adjusts its size when command options change to give users the largest possible drawing area.

The illustration below shows the *initial top-level* command window that appears at the start of each session.



Initial Top-level Command Window

The system displays a second, *standard top-level* window after the user creates at least one drawing window. Selecting a command from this window causes the system to display other command windows for creating geometry, editing a drawing, or performing filing and output actions.

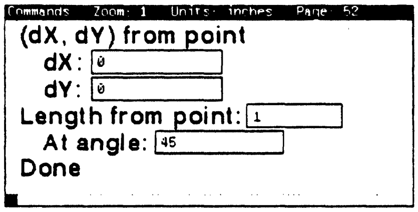


Standard Top-level Command Window

The **Done** command in every subordinate window tells the system to return to the standard top-level window.

Command options are selection-sensitive, that is, they depend at any moment on the kind and number of items currently selected in a drawing window. For example, selecting the **Line** command when two points are selected causes the system to display the **Line between points** command. If the user selects a single point, the system displays a different set of command options for creating a line relative to the selected point.

Many actions require the user to enter additional parameters like the radius of a circle or the style of a line. The user enters these parameters in command window *fields*. Fields are marked by narrow rectangles. In most cases, the user positions the cursor inside the field, presses the left mouse button to delete any existing text in the field, and then types the new entry. Other parameters are selected from dynamic menus.

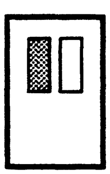


Point Command Window with Parameter Fields

To ease the entry of certain commonly-used parameters, certain fields have dynamic menus associated with them. The menu displays the options for that particular text field. For example, the menu associated with a **Line style:** field contains an alphabetical listing of all twenty-eight available line styles.

Some commands require the user to qualify an action by selecting an area or item after issuing the command. In these cases, the cursor changes to the shape of the mouse to show that the system is waiting for the user to confirm the area or item. The system at the same time displays precise instructions in the message window. To complete the command, the user moves the cursor to the area or item and presses the left mouse button. Pressing the right mouse button cancels the command.

For example, to create a line parallel to an existing line, the engineer issues the commands and then selects to one side or another of the selected line. The system then draws the new line parallel to the selection in the indicated area.



The Mouse-shaped Cursor

Command Accelerators

The drafting package offers a number of command accelerators. Pop-up menus act as accelerators, duplicating the commands in the command window. The user can issue a command with a dynamic menu without moving the cursor from the drawing window to the command window. He or she simply brings up the menu and selects the command from the list of options. This method of issuing commands is most useful for commands without special parameters, or when the user wants to repeat the same command using the same parameters.

Three sets of special function keys act as accelerators for many frequently-used actions. These keys perform the following types of functions:

Pressing one of the special *direct function keys* lets the user bypass the command selection process entirely to complete an action. To delete a line, for example, the user selects the line and then presses the **DELETE** key; no actual command window need be involved.

Fast function keys make it possible to perform the same action repeatedly without reselecting a command from the command window. For example, the user can obtain descriptions for any number of objects by pressing and holding the **DESCRIBE** key and selecting the objects from the drawing window.

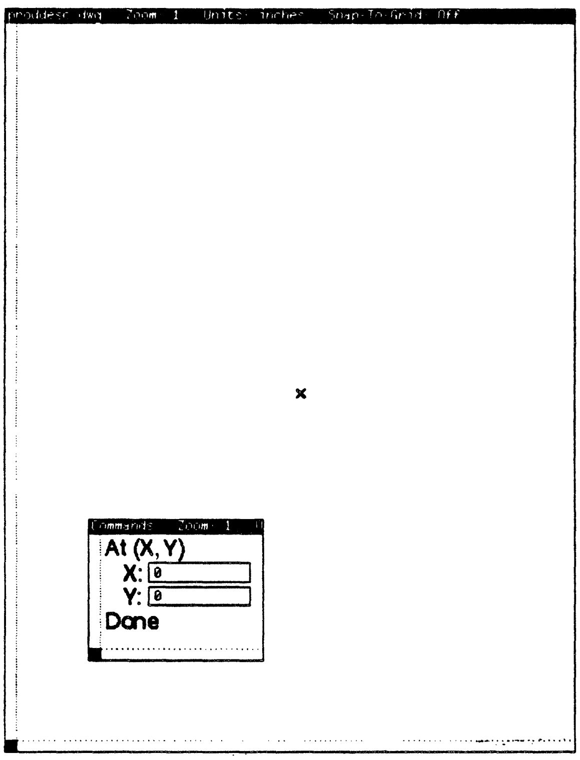
Page-switching keys let the user move from one command window to another without passing through the top-level window. By pressing the **ARC** key after creating a point, for instance, the user moves immediately from one of the **Point** command windows into the **Arc** command window without returning first to the top-level window.

Geometry Creation

All geometry creation and editing occurs inside drawing windows. Versatec supplies a number of drawing windows with formats for drafting, pc board outlines, and checkplots. The engineer typically copies one of these standard formats, renames it, and then begins to create geometry.

Creating Points

Expert lets the engineer place points at specific coordinates, relative to existing objects, or at user-defined grid locations. For example, he or she can create a point by typing in values corresponding to a point's X- and Y-coordinates and selecting the **At (X,Y)** command from the **Point** command window. In the following illustration, the engineer has created a point at (0,0).



Point Creation

Other commands let engineers place points on existing geometry, for instance, at a circle's center point or at the intersection points of a number of trajectories.

A special accelerator key makes the creation of freehand points both on and off the grid a quick and simple matter. With the snap-to-grid feature on, the engineer first presses and holds the **POINT** key. A point appears at the nearest grid mark wherever the engineer presses the left mouse button until he or she releases the key. With the snap-to-grid feature turned off, the same procedures place the point at the exact cursor location.

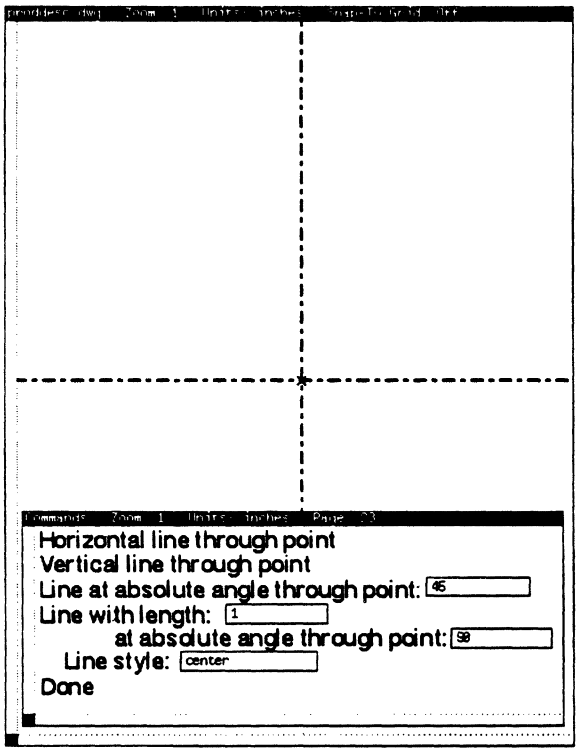
Creating Lines

The engineer can create and edit a variety of trajectories from coordinates or relative to existing geometry. Trajectory types include lines, circles, rectangles, arcs, ellipses, and conics. The engineer assigns a line style to each trajectory as he or she creates it.

Trajectories have a *line style* governing their thickness and dash pattern. The available *line styles* include solid, center, hidden, witness, medium, thick, dimension, outline, invisible, eight user-defined styles, and eleven styles reserved for creating pc board outlines.

To draw lines through the point just created, the engineer first selects the point and then selects **Line** from the top-level command window. The new command window shows options for creating vertical, horizontal, and diagonal lines through the point.

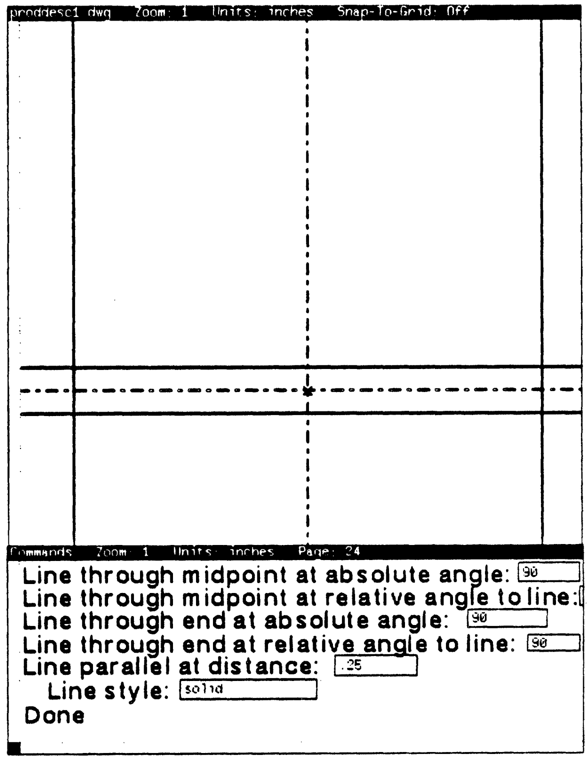
In the example, the engineer creates vertical and horizontal centerlines through a point after setting the line style field to *center*.



Line Creation

The engineer can create lines parallel to an existing line with the **Parallel at a distance:** command. He or she specifies the distance separating the existing line and the desired parallel line before selecting the command. Because two lines at the same distance can lie parallel to the selection, the system asks the engineer to place the cursor in the area to either side of the selection. The new line now appears in that area at the specified distance.

In the example, the engineer creates two solid lines parallel to the vertical line at a separation of 2.5 inches, and two lines parallel to the horizontal centerline at 0.25 inches.



Parallel Line Creation

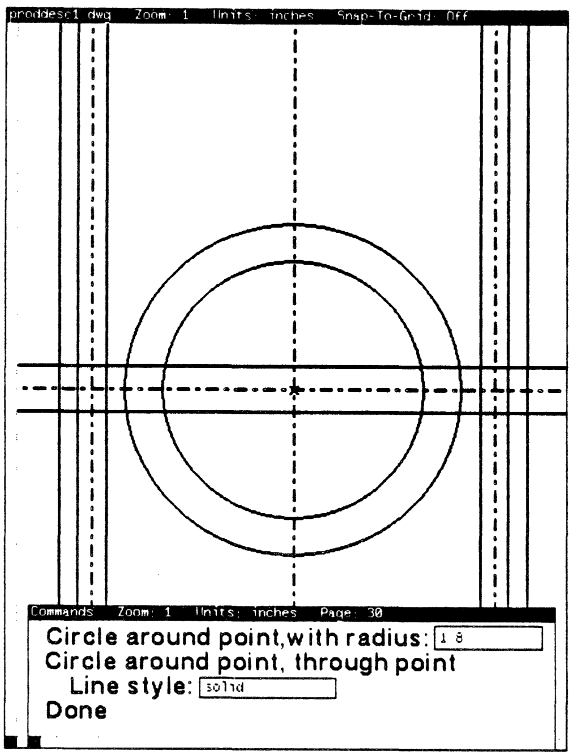
Additional line commands let the engineer construct angle bisectors for intersecting trajectories, draw lines between endpoints of selected lines, and create tangent and perpendicular lines relative to various selections.

Creating Circles

The procedures for creating circles resemble those for lines. The engineer typically specifies a center point by selecting an existing point or by typing the center's coordinates in the fields provided. After specifying the radius of the desired circle and the line style, the engineer then selects a circle command to create the circle.

Additional circle commands let the engineer create circles tangent to or between various selections. Where the system recognizes that more that one tangent circle is possible given the selection, the system prompts the user to select one of the circles to keep.

In the example below, the engineer creates the outer circle by selecting the same point and entering 1.8 in the radius field. To create the inner circle, the engineer reselects the center, changes the radius to 1.4, and issues the command.

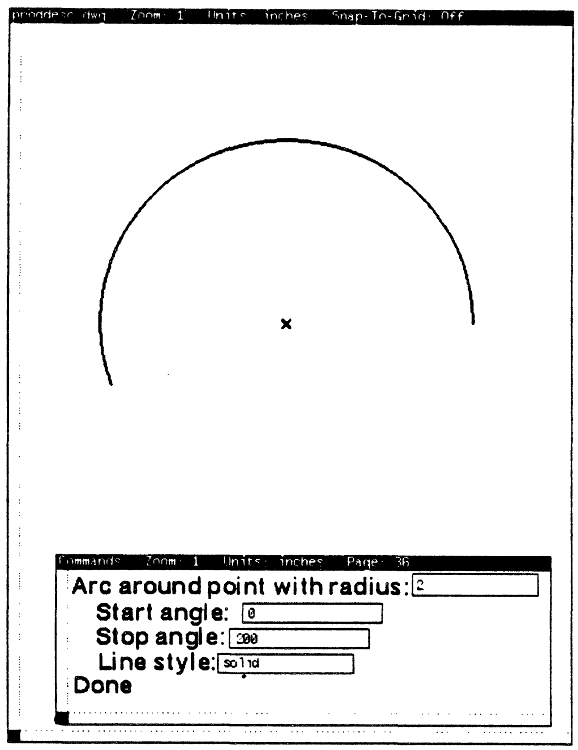


Circle Creation

Creating Curves

In addition to the points, lines, and circles described above, Expert gives engineers the ability to create arcs, ellipses, and conics (hyperbolas and parabolas) in a number of ways. Users can cut, corner, transform, duplicate and change the line style of a curve as they would any other trajectory. Access to these additional geometry creation commands is through the top-level command window.

Curve command windows, like those for other geometry creation modes, are selection-sensitive. Selecting a point, for example, while in the arc creation mode lets a user create an arc around that point. With a point and a trajectory selected instead, the commands let an engineer create an arc around the point and tangent to the trajectory. The figure below shows typical drawing and command windows for creating an arc around a selected point.

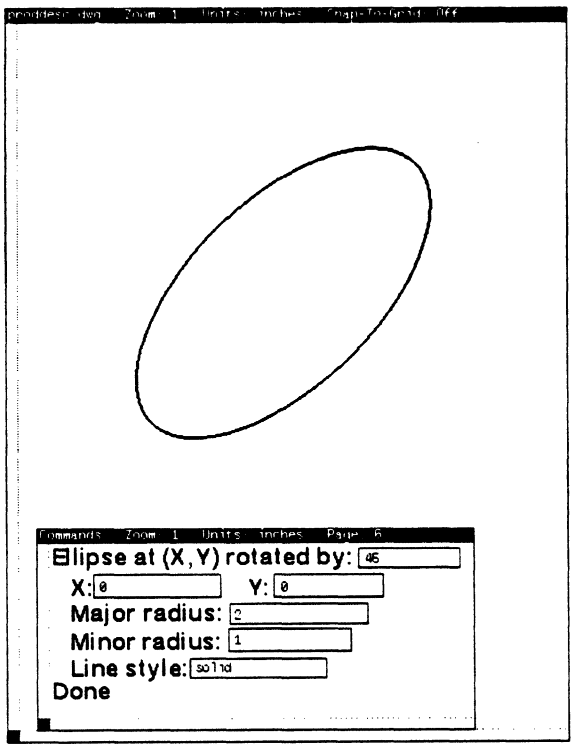


Arc Creation

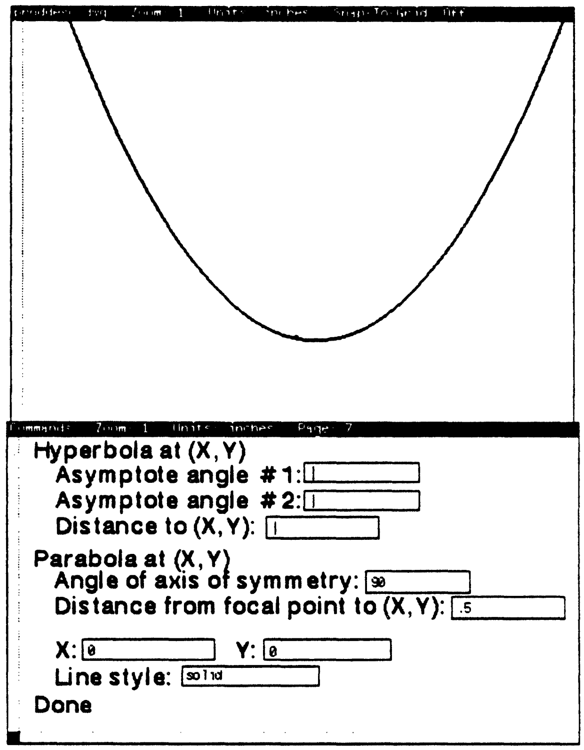
Arcs generally require the engineer to select or supply coordinates for a center point and choose values for the radius, line style, start angle, and stop angle. *Ellipses* require parameters for a center point, major and minor radius lengths, line style, and angle of rotation. The figure below shows drawing and command windows during the creation of a typical ellipse.

Hyperbolas require the engineer to choose parameters for center point coordinates, line style, two asymptote angles, and a slackness value (the distance from the center to the vertex of the curve). *Parabolas* require vertex coordinates, an angle describing the axis of symmetry, and a distance from the focal point to the vertex.

The figure below shows a typical parabola and the **Conic** command window.



Ellipse Creation



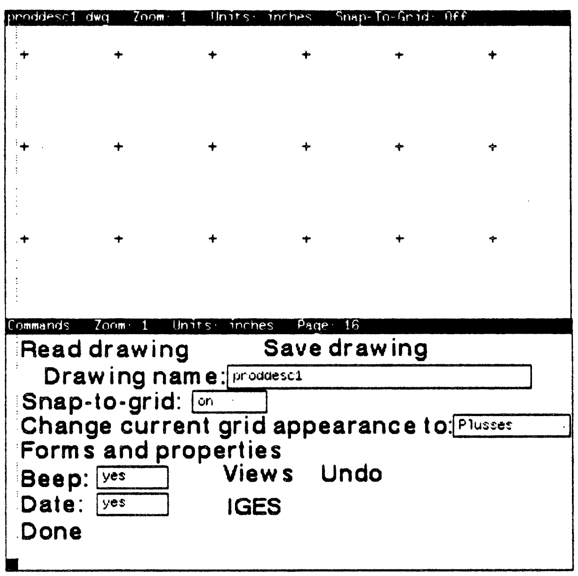
Conic Creation

Grids

Expert lets users define grids for precise placement and movement of objects inside a drawing. Expert's *snap-to-grid* feature lets users move objects or place freehand points at grid locations without specifying exact coordinates. The part and via grids created while defining a pc board ensure precise placement of the board's parts and vias.

Each grid has a name, a number of defining parameters, and an appearance. Each new drawing has a default grid, called the *normal* grid. The center of the normal grid is (0,0); it has a rotation of zero degrees. Each drawing likewise has a *current grid*. The current grid is the grid currently displayed.

Each grid has several possible appearances on the display: dots, plusses, dotted lines, solid lines, or invisible. The normal grid is invisible until the engineer changes its appearance in one of the **Grid** command windows. While visible on the display, grids do not appear in the plotted drawing. Similar procedures let the engineer delete, change, and describe the current grid in the message window.

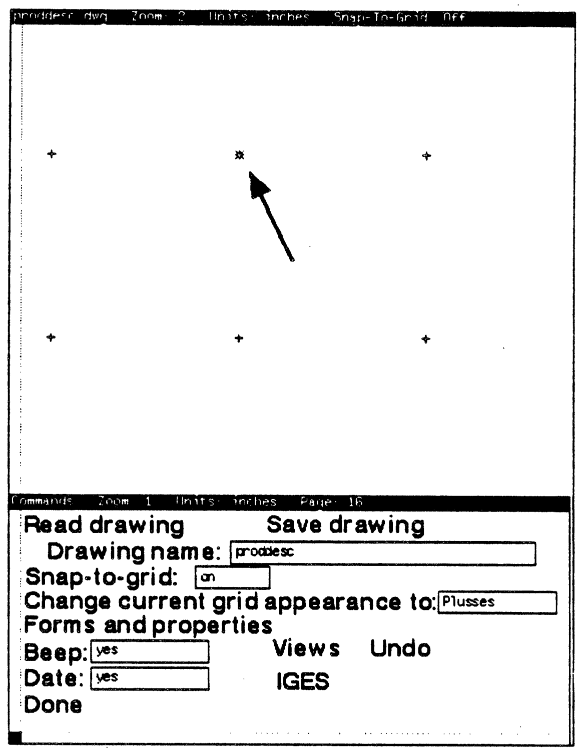


Grid Appearance and Commands

The user activates the snap-to-grid feature by first calling up the **System** command window. He or she then selects a grid appearance, sets the **Snap-to-grid** field to the *on* position, and selects the command. The drawing window's name frame indicates whether the snap-to-grid feature is on or off.

To create a new grid, the engineer selects **Create new grid** in the **Views** command window. The system displays a command window in which to enter the grid parameters. The parameters required to define the new grid differ depending on what drawing objects the engineer has selected.

When nothing is selected, the engineer enters the grid appearance, the X- and Y-coordinates of the origin, X-spacing, Y-spacing, and the grid name. Separate grid creation windows appear with selections of one and two points.



Point Creation with the Snap-to-grid Feature

Geometry Editing

Geometry editing commands transform drawing objects according to the user's instructions. They let users construct complex geometry from more basic shapes and correct drafting mistakes. Geometry editing command windows include commands for cutting trajectories, creating corners, replicating, moving, and mirroring selections, and for performing isometric transformations.

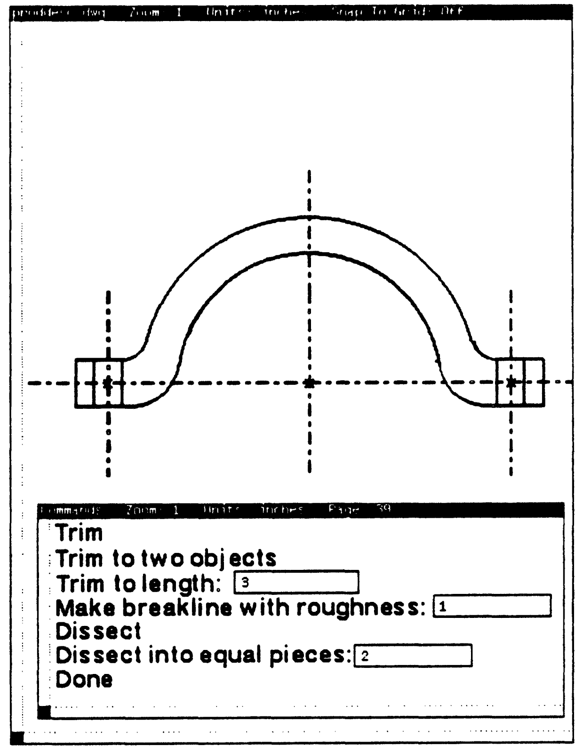
Cutting Trajectories

The commands located in the **Cut** command window let the user trim and dissect trajectories and create breaklines from existing segments. To display and issue commands from this window, the user selects **Cut** from the top-level window.

Users can issue one of three separate commands for trimming trajectories in a drawing window. The first of these, **Trim,** lets users trim a selection where it intersects another object. The user then indicates with a single press of the left mouse button which part of the trimmed trajectory to save and which to discard.

Selecting **Trim to two objects** lets the user trim a trajectory relative to two other objects. After issuing the command, the user indicates the objects to be used in trimming the trajectory. The system then trims the trajectory as indicated and keeps the portion between the intersection points nearest the selections.

The system also offers a way of trimming a segment to a specific length. Issuing the **Trim to length:** command shortens or lengthens a selection to a length indicated in the command's parameter field. The user enters the desired length at the keyboard and selects an endpoint from which to calculate the trim. The following illustration shows three vertical centerlines trimmed to various lengths.



Trimming to Length

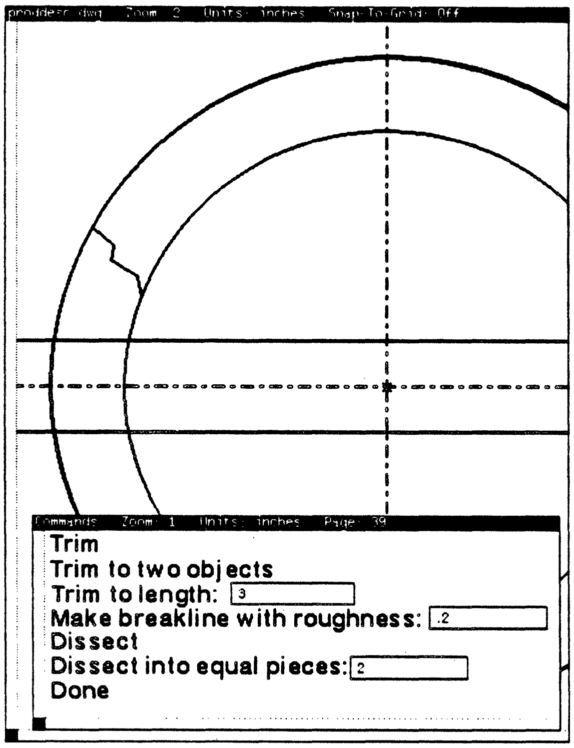
Dissecting a trajectory cuts it into shorter segments. Expert lets the user dissect trajectories where they intersect other objects or into any number of equal pieces. After a dissection, the user can then edit each resulting segment individually. Three dissection commands are located in the **Cut** command window.

The **Dissect** command dissects trajectories at their intersection with a single cutting object. The user selects the trajectories to dissect and, after issuing the **Dissect** command, uses the mouse to select a cutting object.

Issuing the **Self-dissect** comand for two or more intersecting trajectories dissects each trajectory at its intersection which every other. To divide a trajectory into several pieces of equal length, the user selects the trajectory, enters the number of pieces desired, and then selects **Dissect into equal pieces**.

Breaklines

The **Make breakline with roughness:** command transforms a selected trajectory into a breakline. The user-assigned roughness parameter governs the size of the individual lines that make up the breakline. When the system forms a breakline, it keeps a record of the original trajectory. Selecting **Undo breakline** restores the trajectory to its initial appearance.



A Breakline

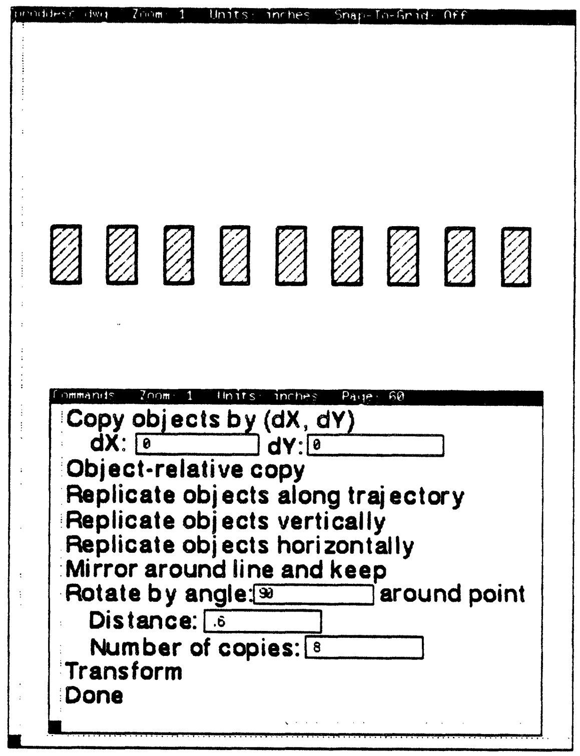
Duplication

Commands located in the **Duplicate** command window let the engineer copy, mirror, replicate, and rotate selected objects. When a copy is to be placed at precise coordinates, the user indicates the X- and Y-displacement and issues the **Copy objects by (dX, dY)** command.

The **Object-relative copy** command copies the selection relative to two reference objects. The user selects the "from" reference object and the "to" reference object after selecting the **Object-relative copy** command.

A user can likewise create multiple copies (or replicas) of a selection either vertically, horizontally, along a specified trajectory, or around a pivot point. The user specifies the number of replicas and distance between each, then issues one of the replication commands and, if necessary, indicates a reference point or trajectory.

The figure below shows the horizontal replication of a simple rectangle.

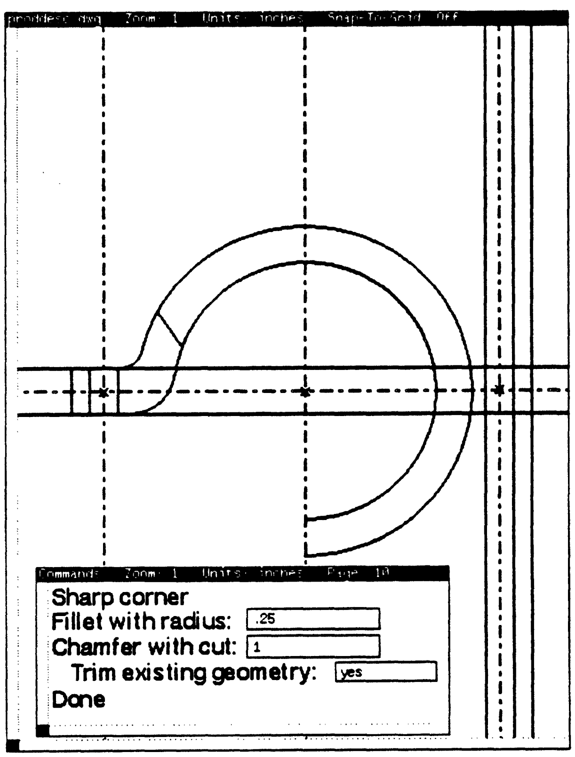


Horizontal Replication

Corners

Expert reserves a separate **Corner** command window for creating sharp, fillet, and chamfer corners. These commands let the user make a corner from any two non-parallel trajectories. When necessary, the system extends the trajectories to make the corner. The user can instruct the system to eliminate extraneous geometry when making a corner. In addition,. the special **CORNER** key lets the user make repeated corners of any type at an accelerated pace.

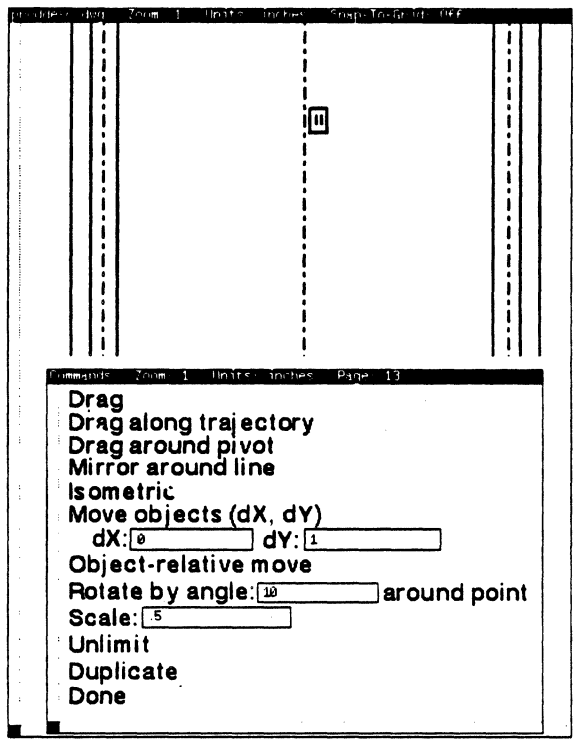
In the first example, the engineer has created fillet cuts with a 0.25-inch radius at the intersections of the circles and the horizontal lines. He or she then creates sharp corners at the intersections of the horizontal and vertical lines to give the results shown below.



Cornering

Transform

Selecting the top-level **Transform** command offers the engineer a choice from commands to move and transform objects inside the plane of the drawing. These transformations include rotating objects around a point, unlimiting bounded trajectories, mirroring across lines, and various isometric transformations.



Transformation: Mirroring

Drag commands let users move or "drag" selections through the drawing window along the cursor's path. While the user holds down the left mouse button, he or she slides the mouse to move the object on the display. The user releases the mouse button to fix the object at its current location. To drag the object along an existing trajectory or around a pivot point, the user selects the reference object and then slides the mouse after issuing either the **Drag along trajectory** or the **Drag around pivot** command. Expert also provides a **DRAG** key for quick freehand dragging.

An engineer can remove the bounds from geometric objects by selecting the **Unlimit** command. Unlimited lines span the length of the drawing format; unlimited arcs become circles.

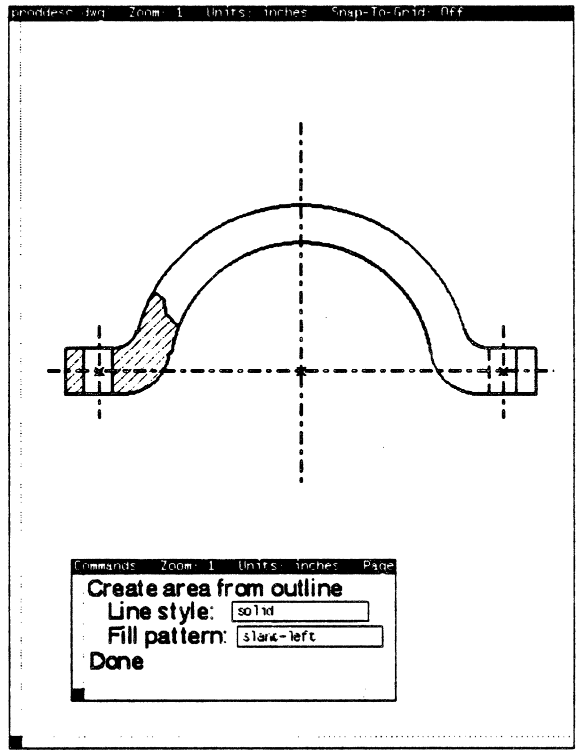
The **Mirror around line** command lets users transform an object into its own mirror image. The user selects the object, selects **Mirror around line**, and indicates the line around which to mirror the object.

Creating Areas

An *area* is a geometric region bounded by an arbitrary set of curves and lines. An area may have holes and self-intersecting edges, but it always represents a set of closed trajectories.

The engineer creates an area by first selecting its bounding curves and lines and then selecting the **Area** command in the top-level command menu. The user assigns a line style to the area's outline and a *fill pattern* for the area itself. Fill-pattern styles include solid, dark-grey, light-grey, slant-right, slant-left, cross-hatch, vertical, horizontal, and fifteen user-defined styles. After issuing the **Create area from outline** command, the system clusters the boundaries into a single item. The illustration below shows a sample area.

The user can destroy an area without destroying the trajectories that mark its edges. Expert removes the fill pattern, makes invisible bounding lines solid, and clusters the bounding trajectories.

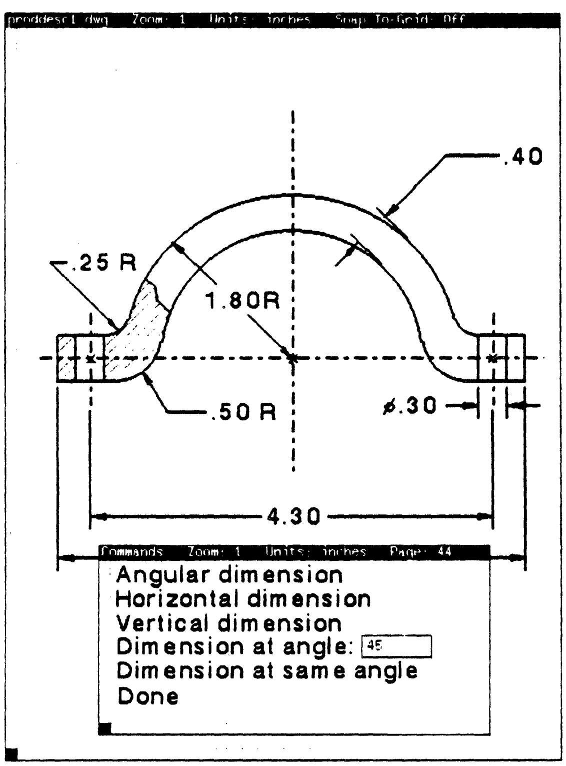


Area Creation

Dimensions and Text Notes

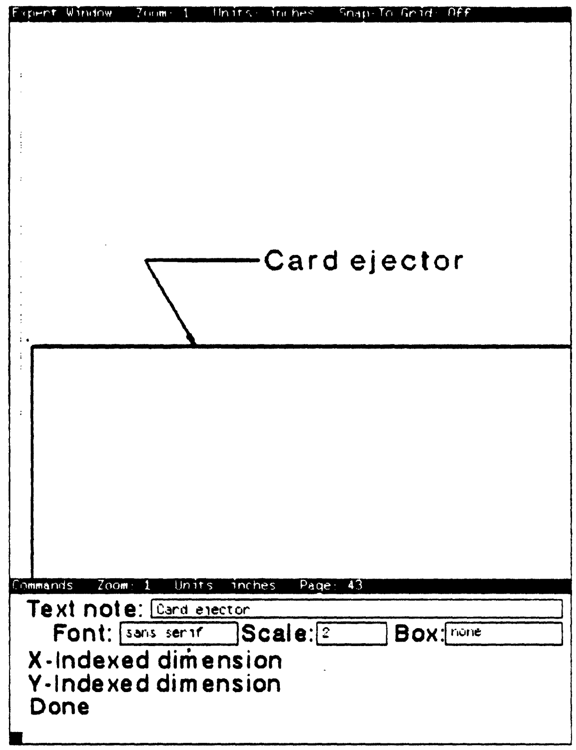
A drawing is not complete without dimensions, labels, reference notes, and text. Expert performs semi-automatic dimensioning according to the ANSI-Y14.5 standard. Semi-automatic dimensioning means that the engineer selects the items to be dimensioned and the dimension location. The system automatically computes the dimension value and displays it at the selected location. If the engineer so specifies, the system boxes or circles a text note. The system can display dimension text either at normal size for easy legibility or scaled according to its correct size relative to the geometry.

The engineer uses commands in the **Dimension** command window to dimension a drawing. As with geometry creation commands, the **Dimension** command options are selection-sensitive. For instance, when the engineer selects one object before selecting the **Dimension** command, the system shows only commands for displaying X- and Y-indexed dimensions and text notes.



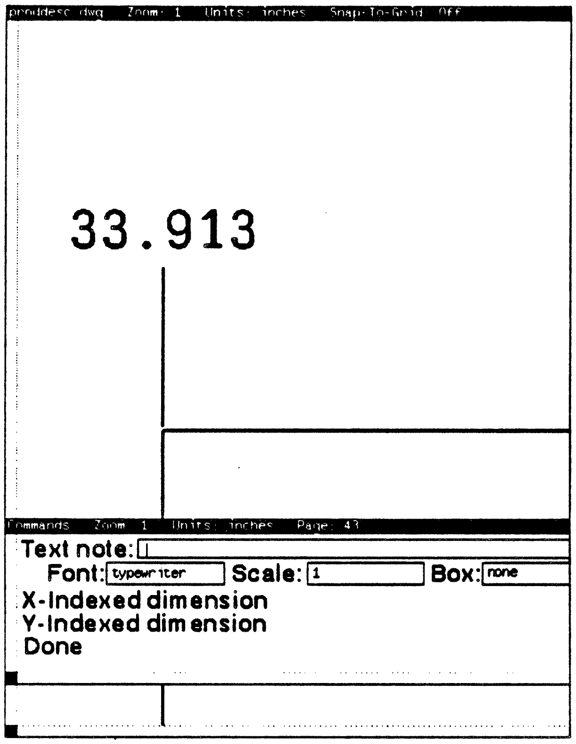
Dimension Commands

To make a text note on the drawing, the user enters the text in a special field and chooses a *scale, font,* and a *box style*. Font types are sans serif, typewriter, and vector. Box styles may be circle, square, circle-square, and diamond. After the user selects the **Text note:** command, the system asks the user to select a location for the note. A similar procedure lets the engineer draw an arrow from the note to the selected item.

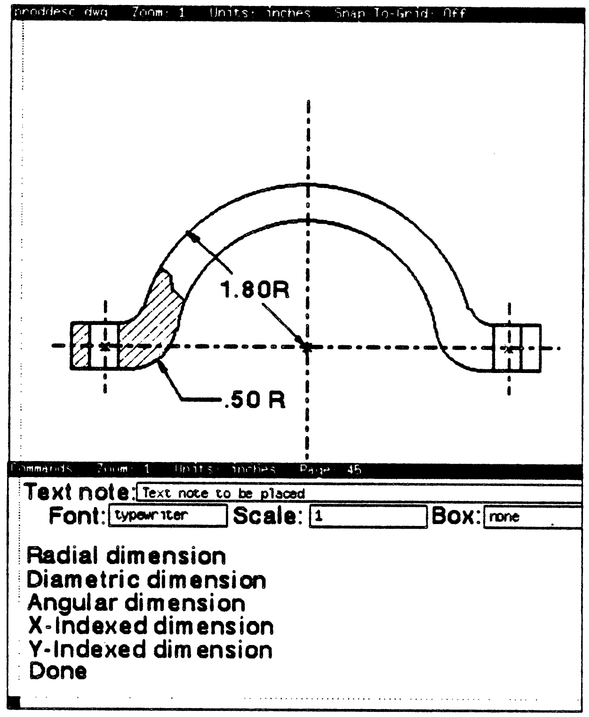


A Text Note

Indexed dimensions display the X- or Y-coordinate at a point on an object relative to a selected reference system. The user must select the object to dimension before selecting the **X-Indexed dimension** or **Y-Indexed dimension** command. After the user selects the command, he or she selects a point on the object, and the location at which to place the dimension text.



An Indexed Dimension



Radial Dimensions

When the user selects two objects, the system displays **Note** commands for dimensioning those objects. Those commands permit the user to display horizontal, vertical, and angular dimensions and to place dimensions at a specified angle.

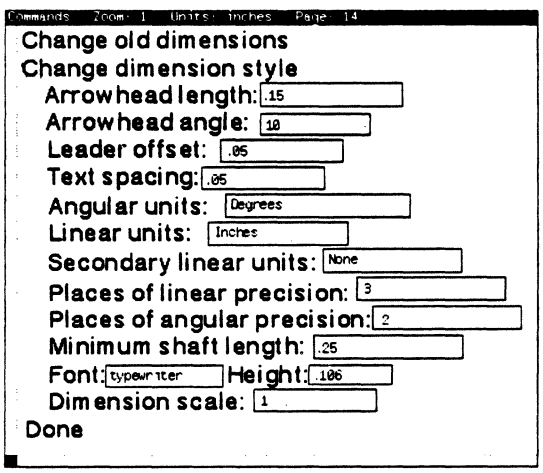
When the user selects an arc or a circle, the system displays commands for placing radial and diametric dimensions. The user selects the command and indicates the angle from the center at which to place the text. If the text does not fit inside the object, the system asks the user to indicate a location outside the geometry.

Dragging Dimensions

When a user drags a dimension from the extension lines, the ends of the lines closest to the dimensioned objects remain anchored while the dimension text retains its same position relative to the object. When the user drags dimension text, however, the extension lines stretch, while the ends of the lines closest to the dimensioned objects remain anchored.

Dimension Styles

The user defines how Expert creates dimensions at any moment by changing the *dimension style* parameters. These parameters set the shape of the dimension arrowhead, the offset of leader lines, the minimum length of shafts, the angular and linear units, and the degree of precision. *Linear units* may be mils, inches, feet, millimeters, centimeters, or meters. *Secondary linear units* may be mils, inches, feet, or none. *Angular units* may be degrees; degrees and minutes; degrees, minutes, and seconds; radians; or slope. The same command window also lets the user set the font, text height, and spacing of the dimension text. The user may alter these parameters and apply them to individual dimensions or to all dimensions in the drawing.



Dimension Style Commands

9. System Libraries

Versatec supplies a large standard library of logic symbols and pc board parts with every Expert system. However, particular installations may require parts built to their own specifications. Expert makes building logic symbols and parts easy. It applies the same user-friendly design to library maintenance tasks as it does to schematic creation, drafting, and pc board design.

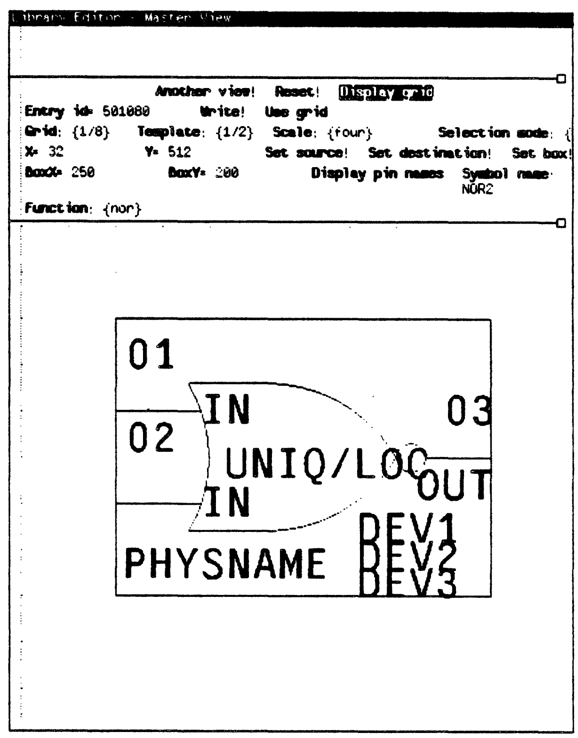
Expert lets each installation choose which users may change library entries. A small installation might have one user that serves as the system librarian. A larger installation might have a group of librarians. Expert's system of controlling library access helps an installation to maintain a standard library.

The Library Editor

The *Library Editor* is the system librarian's tool for creating and editing logic symbols. The system librarian also uses the Library Editor to perform library maintenance tasks such as converting the library to a new format, deleting entries, and merging entries from other libraries.

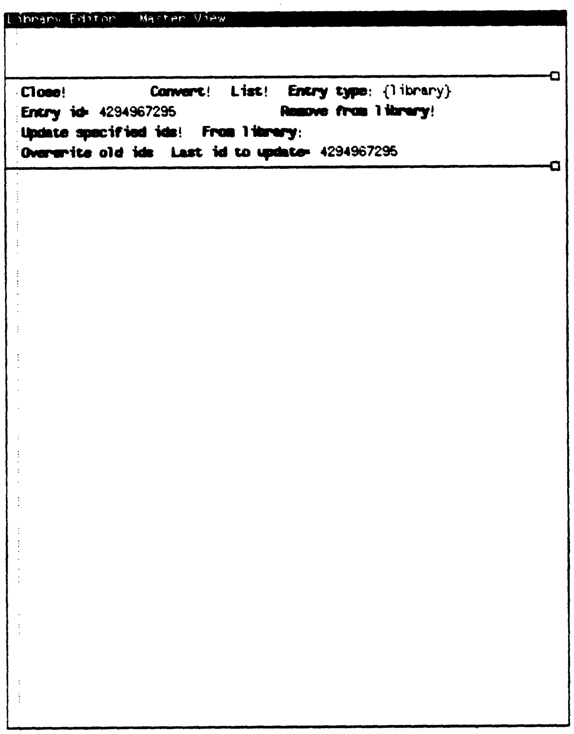
The Library Editor window contains a system message subwindow and a command subwindow. When the user first opens the Library Editor, the entry type is set to **symbol**. With the entry type set to **symbol**, the librarian can view logic symbol entries, edit symbol entries, or add logic symbols to the library.

The user creates logic symbols from lines, arcs, pin leads, and text. Each pin lead has a property sheet that describes its name, function, and number. Once the librarian has drawn the symbol, he or she enters its name, identification number, and function in the Library Editor's command subwindow, and *writes* it to the library.



A Logic Symbol inside the Library Editor Window

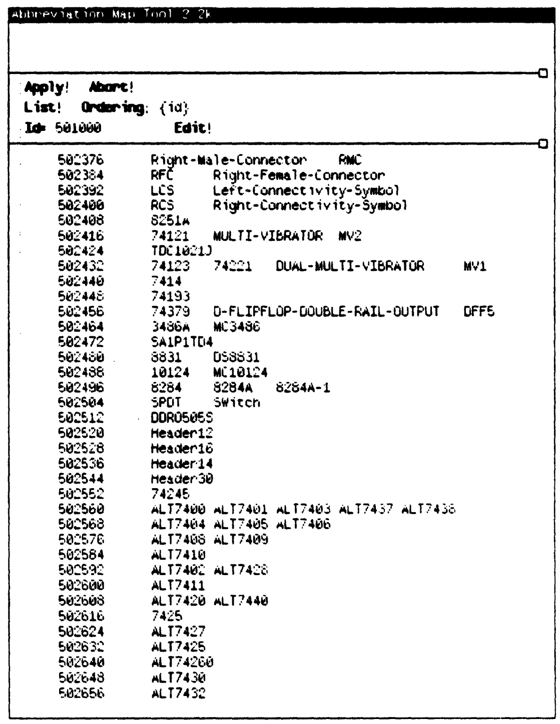
Changing the Library Editor entry type changes the commands in the Library Editor's command subwindow. For instance, selecting the **library** entry type displays library maintenance commands. With these commands, the user can remove entries from the library or convert the library to a new format when necessary.



Library Commands

The Abbreviation Map

After creating a library entry for a logic symbol, the system librarian enters that symbol into the Abbreviation Map. The Abbreviation Map is a list of all logic symbols and their nicknames. Engineers can use either a logic symbol's identification number or one of its nicknames to place it on a schematic. The illustration below shows an Abbreviation Map listing inside the Abbreviation Map window.

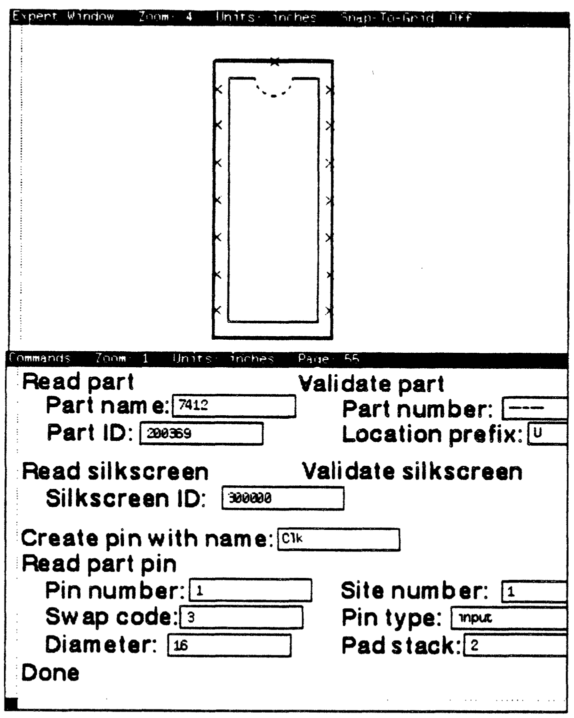


The Abbreviation Map Window

Creating a Part

A system librarian creates pc board parts in three stages with the Expert Drafting package. First, the librarian creates and validates the part's silkscreen. Second, the librarian creates and validates the part's individual pins, and then finally validates the part as a whole.

Each stage requires the librarian to enter parameters describing the unique characteristics of the part. He or she enters an ID number for each silkscreen, and enters a name, site number, pin number, swap code, pin type, diameter, and swap code for each pin. The system displays the data for each pin in a legible form. The illustration on this page shows a part named 7412 (triple 3-input positive-NAND gates with open-collector outputs).

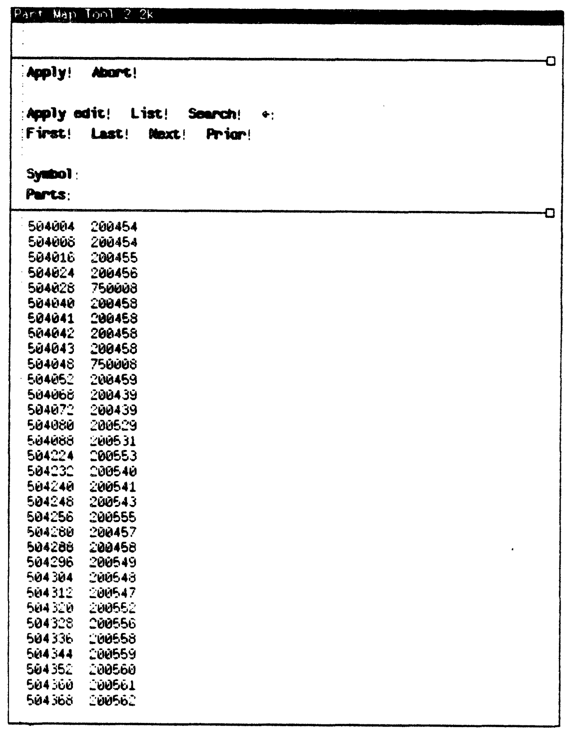


Electronic Part Creation

Validating a part requires the librarian to enter a part name, number, ID, and location prefix. The system saves these parameters in the network library to speed the editing of parts. An existing part can easily serve as a template for the creation of new parts.

After creating and validating a part, the system librarian enters it in the *Part Map* using the Part Map window. The librarian also specifies at that time which logic symbols map to a particular part.

The illustration below shows the Part Map window.



The Part Map Window

10. Customer Support

Versatec's dedicated customer support network supplies high-caliber training, maintenance, and service to all Expert users.

Every Expert installation receives the following product support services:

- Site Audit
- Management Overview
- Custom Drawing Formats, Report Forms, Checkplot Tables, and Photoplot Tables
- Standard Libraries
- Training
- Telephone Consultation

The Site Audit solves potential problems before they occur. A local Versatec Application Analyst visits the site, determines necessary supplies and vendors, reviews custom work requirements, and defines hardware and software support procedures.

The Application Analyst also conducts a Management Overview at each site. The Management Overview is a presentation previewing the installation procedure, discussing Versatec's commitment to its customers, explaining the Expert learning process, and answering any questions.

Before the system is installed, Versatec specialists create custom drawing formats, report forms, and photoplot tables for each site. Upon installation, each site receives a standard library consisting of pre-made logic symbols and pc board parts, or mechanical parts. The custom work and standard library allow Expert users to start work almost immediately.

Each installation also receives two training credits for each software module purchased. Because training begins immediately after the system is installed, users do not experience the frustration of learning by trial and error.

For continuing support, Versatec staffs its Santa Clara headquarters with Customer Support Specialists. Users phone a single 800-number for answers to questions about both hardware and software functionality. The Customer Support Center dispatches the trained technicians who support the Expert hardware in major cities across the country.

Productivity Services

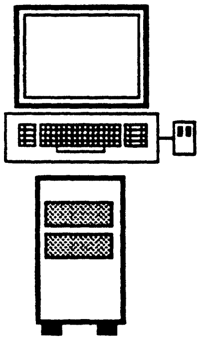
In addition to the standard product support services, Versatec also offers optional Productivity Services designed to enhance the productivity of an Expert installation. Productivity Services include:

- On-Site Consultation
- Training Service
- Custom Formats, Reports, and Forms
- Custom Library Service

On-Site Consultation assists an Expert installation in using the system more effectively. On-Site Consultation supplies the installation with the expertise of a qualified specialist, the Versatec Application Analyst. That same specialist also conducts Productivity Audits that result in a list of suggested improvements plus a detailed implementation plan for those improvements.

The Training Service provides courses that teach users the techniques and methods needed to use the system effectively and efficiently. The training courses consist of lectures and discussions, plus hands-on system usage.

Versatec will design custom formats, reports, and forms to ensure that a system meets the particular needs of its users. The Custom Library Service supplies individual Expert sites with logic symbols, pc board parts, or mechanical parts libraries built to their specification.



8014 Workstation

Expert design tools run on a Xerox 8014 graphics workstation. The workstation operates in a normal office environment, requiring no special power or air conditioning.

8014 Workstation Components:

- Xerox 8000 NS processor with 768 kilobytes or optional 1.5 megabytes of main memory and 24-bit virtual memory addressing
- Large-format, bit-mapped display
- Keyboard with mouse pointing device
- 42-megabyte capacity non-removable rigid disk storage
- 1.2-megabyte double-sided, double-density floppy disk drive
- Internal controller and cable for connection to 10-megabit-per-second Ethernet

Processor
- 2901 bit-slice technology
- Execution rate approximately 1 million instructions per second

Display
- Monochrome
- Bit map: 808 pixels (vertical) x 1024 pixels (horizontal)
  (72 pixels per inch of virtual image)
- 38 hertz refresh rate

Keyboard
- Standard ANSI layout
- 128 ASCII character set
- 24 function keys

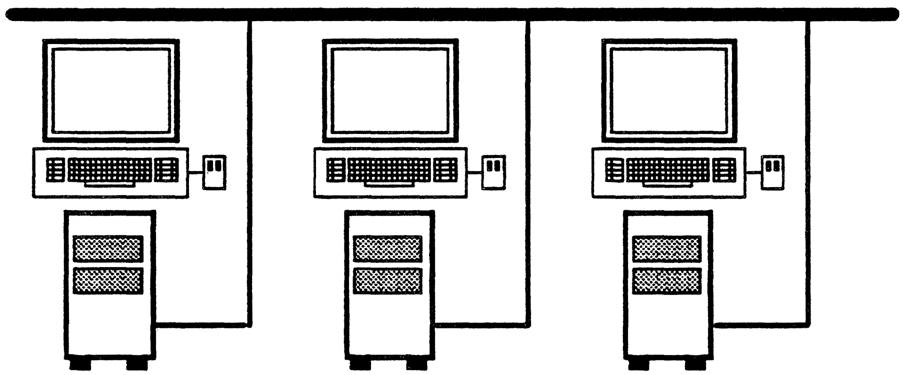
Pointing Device
- Two-button, hand-operated mouse

Specifications

Dimensions	Processor	Display/Keyboard/Mouse	
Height	25"	Display: 19"	Keyboard: 4"
Width	12"	Display: 17"	Keyboard: 20"
Depth	30"	Display: 15"	Keyboard: 9"
Weight	120 lbs	Display: 41 lbs	Keyboard: 8.7 lbs

Electrical Requirements		
Voltage	103-127 VAC	Included
Frequency	60 Hz	with
Current	7 Amps	processor

Operating Environment		
Temperature	50°-90°F	50°-90°F
Relative Humidity	15%-85%	15%-85%
Heat Dissipation	2048 BTU/Hr	410 BTU/Hr



Ethernet Network

The Ethernet local area network enables users at different workstations to share design data and resources.

Ethernet lets an Expert installation expand. Adding workstations to the network does not degrade the performance of existing workstations.

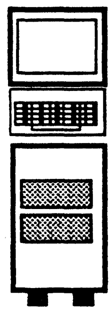
Ethernet Components:

- Coaxial cable (PVC or Teflon®)
- Coaxial connector
- Barrel connector
- Transceiver
- Drop cable
- Terminator

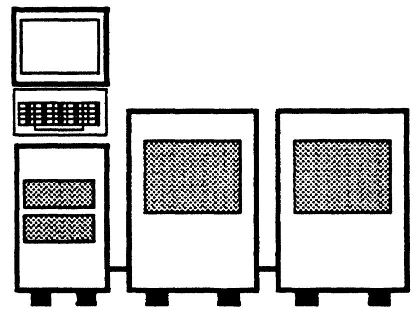
Specifications

50 Ohm Coaxial Cable	Jacket PVC or Jacket Teflon®
Cable Sections	75'
	230'
	385'
	500'
	1000'
Maximum Network Segment Length	1640'
Minimum Transceiver Spacing	8'2"
Maximum Repeaters in Data Path	2
Maximum Number of Transceivers (includes workstations, servers, and repeaters)	1024 per network
Drop Cable from Transceiver to Workstation, Server, or Repeater	15'
	30'
	60'
Data Transmission Rate	10 Megabits/second, baseband

8033 File Server



8037 File Server



8033 and 8037 File Servers

The File Server is a shared remote storage device. The File Server increases the amount of information a workstation can access without decreasing the local storage capabilities of that workstation.

File Server Components:

Model 8033

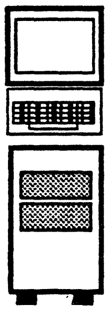
- Xerox 8000 NS processor with 512 kilobytes of memory and built-in Ethernet interface
- 1.2-megabyte floppy disk drive
- 42 megabyte capacity, non-removable rigid disk storage
- Administrator keyboard/display terminal
- RS232C communication kit (optional)

Model 8037

- Xerox 8000 NS processor with 512 kilobytes of memory and built-in Ethernet interface
- 1.2-megabyte floppy disk drive
- Removable disk storage modules. Dual 80-megabyte removable disk drives (one for on-line, one for backup) *OR* dual 300-megabyte removable disk drives (one for on-line, one for backup)
- Administrator keyboard/display terminal
- RS232C communication kit (optional)

Specifications

Dimensions	Server Terminal (All Models)	Processor and Disk Module (Model 8033)	Processor (Model 8037)	80-Megabyte Disk Module (Model 8037)	300-Megabyte Disk Module (Model 8037)
Height	14"	25"	25"	35"	36"
Width	16"	12"	12"	19.5"	19.5"
Depth	21"	30"	30"	33"	33"
Weight	32 lbs	120 lbs	105 lbs	320 lbs	480 lbs
Electrical Requirements					
Voltage	103-127 VAC	103-127 VAC	103-127 VAC	103-127 VAC	177-219 VAC
Frequency	60 Hz	60 Hz	60 Hz	60 Hz	60 Hz
Current (steady-state)	0.5 Amps	5.8 Amps	4.0 Amps	8 Amps	5 Amps
Operating Environment					
Temperature	50°-90°F	50°-90°F	50°-90°F	60°-100°F	60°-90°F
Relative Humidity	15%-85%	15%-80%	15%-80%	10%-80%	10%-80%
Heat Dissipation	200 BTU/Hr	2048 BTU/Hr	1366 BTU/Hr	2550 BTU/Hr	3500 BTU/Hr



8071 Communication Server

The 8071 Communication Server gives network users access to a variety of shared-resource communication services.

The Communication Server's **Internetwork Routing Service** provides communication between networks or connects a single, remote workstation to a network. The Internetwork Routing Service lets engineers on separate networks or at remote locations exchange information between offices. With the Internetwork Routing Service, sharing the design of a schematic with an engineer at a workstation across the country is as easy as sharing with an engineer across the room.

The **Clearinghouse Service** is a network resource directory. It keeps track of all the hardware on the network, all of the services executed on the hardware, and all of the people authorized to use the hardware and its services. The Clearinghouse Service software runs either on a Communication Server or File Server.

Communication Server Components:

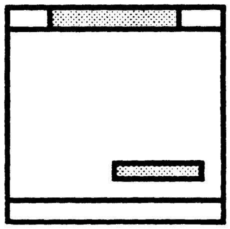
- Xerox 8000 NS processor with 512 kilobytes of memory and built-in Ethernet interface
- 1.2-megabyte floppy disk drive
- Built-in rigid disk storage--10 Megabytes
- RS232C communication port
- Administrator keyboard/display terminal

Specifications

Dimensions	Processor	Server Terminal
Height	25"	14"
Width	12"	16"
Depth	30"	21"
Weight	120 lbs	32 lbs

Electrical Requirements		
Voltage	103-127 VAC	103-127 VAC
Frequency	60 Hz	60 Hz
Current	5.8 Amps	0.5 Amps

Operating Environment		
Temperature	50°-90°F	50°-90°F
Relative Humidity	15%-85%	15%-85%
Heat Dissipation	2048 BTU/Hr	200 BTU/Hr



790 and 791 Output Servers

Output Servers let network users share the resources of output devices. The 790 Output Server allows workstations to communicate, via Ethernet, with a Versatec electrostatic plotter. The 791 Output Server allows workstations to communicate with one Versatec electrostatic plotter and a magnetic tape unit.

Multiple users can output design documentation on the electrostatic plotter. The 790 and 791 Output Servers accept plot data from the Ethernet, process it if necessary, and send it to the plotter in raster form.

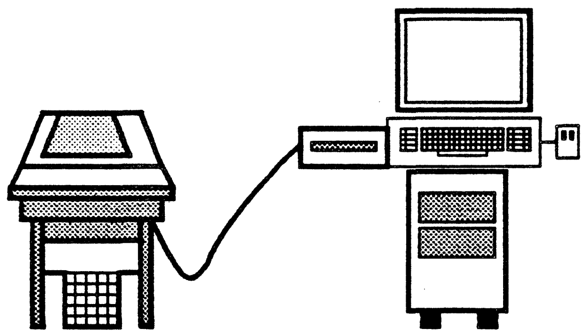
With the 791 Output Server, users can create magnetic tapes for driving off-line photoplotters, archiving files, and transferring data to other systems. The Output Server accepts data from the Ethernet and writes it to the file position specified by the user, or reads specified files from the tape and sends data to the requesting user.

Specifications

Dimensions	Model 790	Model 791
Height	6.75"	33.5"
Width	19"	29.5"
Depth	17"	23"
Weight	35 lbs	225 lbs

Electrical Requirements		
Voltage	103-127 VAC or 207-253 VAC	103-127 VAC or 207-253 VAC
Frequency	50/60 Hz	50/60 Hz
Current	1.3 or 0.7 Amps	5.5 or 2.7 Amps

Operating Environment		
Temperature	50°-90°F	50°-90°F
Relative Humidity	20%-85%, non-condensing	20%-80%, non-condensing
Heat Dissipation	495 BTU/Hr	2100 BTU/Hr



290 Personal Plotter Interface Unit

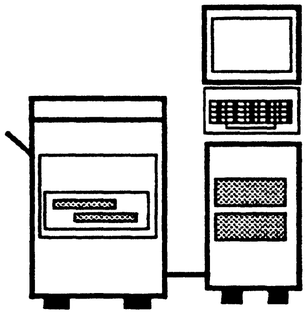
The 290 is a direct interface between an Expert workstation and a Versatec electrostatic plotter. The 290 enables a V-80 or any Versatec plotter to quickly produce local output of engineering reports and drawings.

Specifications

Dimensions	
Height	4"
Width	12"
Depth	15.5"
Weight	10 lbs

Electrical Requirements	
Voltage	130 VAC or 198-264 VAC
Frequency	50/60 Hz
Current	1.3 or 0.7 Amps

Operating Environment	
Temperature	50°-90°F
Relative Humidity	20%-85%, non-condensing
Heat Dissipation	237 BTU/Hr



8046 Print Server

The Print Server supplies network users with laser-image-generated printing at speeds of up to 12 pages per minute. With a 300 by 300 dot per square inch resolution, the Print Server produces high quality engineering reports.

The Print Server uses standard bond paper weights. It handles most printing operations automatically. The user simply sends the print request to the server.

8046 Print Server Components:

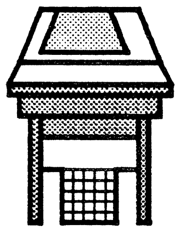
- Xerox 8000 NS processor with 512 kilobytes of memory and built-in Ethernet interface
- 1.2-megabyte floppy disk drive
- 42 megabytes of built-in rigid disk storage
- Administrator keyboard/display terminal
- Electronic printer

Specifications

Dimensions	Server Terminal	Processor	Electronic Printer
Height	14"	25"	36"
Width	16"	12"	22"
Depth	21"	30"	26"
Weight	32 lbs	120 lbs	262 lbs

Electrical Requirements			
Voltage	103-127 VAC	103-127 VAC	103-127 VAC
Frequency	60 Hz	60 Hz	60 Hz
Current	0.5 Amps	5.8 Amps	Without Heater: 2.4 Amps With Heater (Standby): 4.8 Amps With Heater (Running): 11.5 Amps

Operating Environment			
Temperature	50°-90°F	50°-90°F	50°-90°F
Relative Humidity	15%-85%	15%-85%	15%-85%
Heat Dissipation	200 BTU/Hr	2048 BTU/Hr	Without Heater: 900 BTU/Hr With Heater (Standby): 1800 BTU/Hr With Heater (Running): 4300 BTU/Hr



V-80 Printer/Plotter

The Versatec V-80 printer/plotter plots schematics, or pc board artwork drawings and prints engineering reports.

The V-80 can be used as a network resource or as a dedicated plotter for a single workstation. A 790 or 791 Output Server connects the V-80 to the Ethernet network for all network users to share. The 290 Personal Plotter Interface Unit connects the V-80 to a single workstation.

V-80 features include:

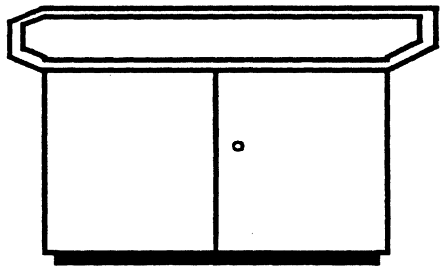
- High-speed printing, 1000 lines per minute
- High resolution, 200 points per inch
- Quiet operation
- Multifunctional printing and plotting
- Solid blacks, high contrast output
- Reliable operation
- Compact, lightweight

Specifications

Dimensions	
Height	10"
Width	24"
Depth	24"
Weight	75 lbs

Electrical Requirements	
Voltage	100/115/200/230 VAC
Frequency	50/60 Hz
Current	2.9/2.5/1.4/1.2 Amps

Operating Environment	
Temperature	32°-105°F
Relative Humidity	10%-95%, non-condensing
Heat Dissipation	973 BTU/Hr, average



8224F and 8236F Plotters

Versatec 24- and 36-inch electrostatic plotters supply the high-quality and high-resolution necessary for plotting engineering drawings. An Output Server lets all users share the resources of the Versatec plotter. A 290 Personal Plotter Interface Unit lets a single workstation send information to the plotter. Expert requires either an Output Server or Personal Plotter Interface Unit to operate the plotter.

8224F and 8236F plotter features include:

- High-quality, 200 points per inch images on a selection of electrographic films and papers
- Speed--plots up to 34 square feet per minute
- Switch selectable line enhancement--provides darker, bolder lines while maintaining resolution
- Accuracy--exclusive shaft encoder ensures 0.2% accuracy

Specifications

Dimensions	Model 8224F	Model 8236F
Height	40"	40"
Width	45"	63"
Depth		
Standard	34.5"	34.5"
With Winder	46"	46"
Weight	400 lbs	700 lbs

Electrical Requirements		
Voltage	100/115/200/230 VAC	100/115/200/230 VAC
Frequency	42 or 63 Hz	42 or 63 Hz
Current	16/14.5/8/7 Amps	16/14.5/8/7 Amps

Operating Environment		
Temperature	32°-105°F	32°-105°F
Relative Humidity	10%-98%, non-condensing	10%-98%, non-condensing
Heat Dissipation	4780 BTU/Hr	5462 BTU/Hr

VERSATEC
A XEROX COMPANY

2710 Walsh Avenue
Santa Clara, California 95051
Phone: (408) 988-2800
TWX: 910-338-0243
Telex: 334421

Printed in U.S.A.