

ACTIVE-CADTM

Real-Time Interactive CAE Tools

Schematic Editor

User's Guide

Seventh Edition

Revision 2



*Automated Logic Design Company, Inc.
3525 Old Conejo Rd. #111
Newbury Park, CA 91320
Phone (805) 499-6867
Fax (805) 498-7945*

Seventh Edition

Revision 2, January 15, 1996

COPYRIGHT

Copyright © 1985-1994 by ALDEC. All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language or computer language, in any form or by any means, electronic, mechanical, magnetic, optical, chemical, manual or otherwise, without the prior written permission of ALDEC, Newbury Park, CA 91320.

DISCLAIMER

ALDEC makes no representations or warranties with respect to the contents hereof and specifically disclaims any implied warranties of merchantability or fitness for any particular purpose. Further, ALDEC reserves the right to revise this publication and to make changes from time to time in the content hereof without obligation of ALDEC to notify any person of such revision or changes.

TRADEMARKS

ACTIVE-CAD, Active Schematic, SUSIE and SUSIE-CAD are trademarks of ALDEC, Inc.

ACCESS, ACTEL, ALTERA, AMD, MACH, MAX, NEC, EPSON, LASER JET, HP, DESIGN COMPUTATION, FUTURENET, OMATION, RACAL-REDAC, TANGO, CAPFAST, CADNETIX, APPLICATION BRAVO, XNF, LCA, XILINX, ORCAD, P-CAD, EDIF, P-DIF, Pentium, VIEWLOGIC, VIEWSIM, WINTEK, MENTOR, POST SCRIPT, CORELDRAW are trademarks or registered trademarks of their respective holders.

This edition applies to Release 2.0 of the software and higher. It is a reference guide for the ACTIVE-CAD and ACTIVE-FPGA products

ALDEC

Automated Logic Design Company, Inc.
3525 Old Conejo Rd., Suite 111
Newbury Park, CA 91320
Phone (805) 499-6867
BBS (805) 498-4086
Fax (805) 498-7945
Email: support@aldec.com

Preface

What is ACTIVE-CAD?

ACTIVE-CAD is an advanced design framework for Windows 95, Windows NT and the older Windows 3.1. Being a complete design solution, ACTIVE-CAD provides for front-to-end design solution:

☐ **Design Entry** tools

1. Schematic editor
2. State (machine) editor
3. HDL editor
4. Symbol editor

☐ **Design Verification** tools

1. Gate and system-level functional simulators
2. Gate and system-level timing simulators
3. Fault simulator
4. Timing analyzer
5. State machine graphical animator (*)
6. Behavioral 1076-IEEE VHDL simulator(*)
7. Virtual Hardware™

☐ **Logic Synthesis**

1. ABEL input files - CPLD and FPGA output files
2. VHDL input files - ASIC, CPLD and FPGA output files

☐ **Framework Support** tools

1. Design Manager
2. Project Manager
3. Library Manager

(*) these tools are currently under development

How ACTIVE-CAD differs from other tools?

ACTIVE-CAD is based on a patented incremental design technology which allows real time animation of designs. Its key features are:

- ❑ Real-time response to design changes; software simulates and animates the latest design changes without any compilation
- ❑ ACTIVE-CAD instantly simulates and animates design responses to new stimulus signals
- ❑ Schematics, state machines and HDL designs can directly interact with each other and with real hardware on an event-by-event basis
- ❑ Design paths and design sections can be instantly simulated or animated under selected set of signal conditions
- ❑ Any design section or signal path can be activated or deactivated from simulation with a click of a button to speed the design analysis process.
- ❑ ACTIVE-CAD shows how the current design changes have affected the past design behavior (reworks past simulation cycles)
- ❑ All tools have been developed by the same engineering company and they have consistent user interface. Reliable tool integration is a natural byproduct of the corporate-wide engineering effort.

What is ACTIVE-Schematic?

ACTIVE-CAD's schematic editor is called ACTIVE-Schematic. It is a real-time interactive design tool that is comprised of a schematic editor and a background simulator. Any device you draw instantly behaves like the real part, down to the last detail.

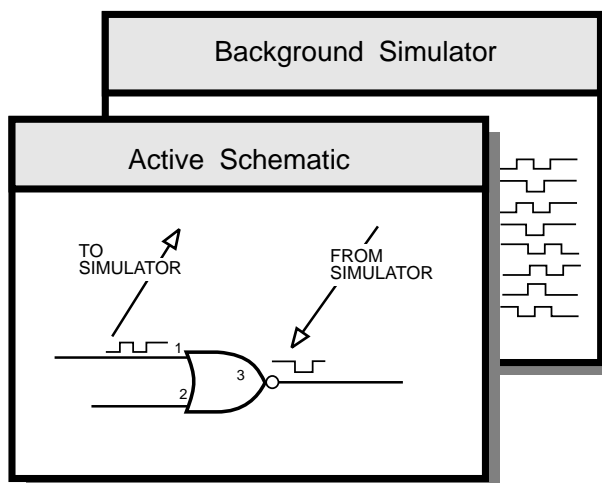


Figure 1. SUSIE Real-time Operation.

ACTIVE-CAD is based on the patented incremental design compilation technology that was first employed in *Standard Universal Simulator for Improved Engineering* (SUSIE). Because of the tradition, ALDEC still supports SUSIE-CAD product line which is a lower cost version of ACTIVE-CAD. Both products are identical except for the number of simulation cycles, selective simulation, test vector editor, and breakpoint editor. This manual can be used for learning both SUSIE-CAD and ACTIVE-CAD products.

- ❑ The schematic editor allows entry of ideas and designs. It has many powerful features that simplify the design process and gives instant access to other projects, files and tools that make your work easier and more effective.
- ❑ Finite state machine (FSM) editor allows to enter designs in a graphical form. The machine states and transitions may be described by ABEL and VHDL statements. The software automatically generates ABEL and VHDL code from the FSM graphs.
- ❑ HDL editor has provisions for entering designs in ABEL and VHDL design languages. The VHDL design source code is checked by a 1076 IEEE compatible VHDL compiler. The code generated by the FSM editor is automatically displayed in the HDL editor.
- ❑ The simulator screen allows creating test vectors, settings an optimal design analysis environment and effectively managing your design resources. Based on an incremental compiler, the simulator allows real-time editing of devices and test vectors while simulation is in progress.
- ❑ ACTIVE-CAD comes with Design Manager which automatically finds other EDA tools and dynamically creates a design flowchart with buttons that activate each application. Thus data transfer between all EDA applications is simple and easy.

ACTIVE-CAD is a true Windows application and forms a very powerful EDA framework. You can open and concurrently operate on multiple design views such as schematic sheets, IC models, waveform windows, etc.

ACTIVE-CAD Features

High Resolution Design Verification

ACTIVE-CAD analyzes each design with a 10 picosecond resolution and automatically reports all timing violations that occur on and off screen. Powerful error reporting facilities help locating all design problems quickly and effectively.

Real-Time Interactive Design Troubleshooting

Since ACTIVE-CAD is a real-time design development system, you can correct most design problems while simulation is in progress. For example, the Milestone feature allows performing “what-if” analysis by changing the design and test vectors and instantly comparing the new and the old design behavior.

Works Directly With Your Existing CAE Tools

ACTIVE-CAD directly interfaces with your existing CAE tools and accepts such output files as netlists, JEDEC fuse maps, hex files, ABEL and VHDL designs, etc.. It imports these files into its own internal design data tables. ACTIVE-CAD also outputs netlist for all popular p.c.b. layout software and accepts their back annotation files. Special effort has been made to assure that all these CAE tools and files work directly with ACTIVE-CAD without any extra effort on your part.

Device Level Design Tools

ACTIVE-CAD is very popular with PLD, CPLD and FPGA designers, because it offers a complete design entry and verification at a very low cost. The device level products are called ACTIVE-(IC VENDOR), e.g. ACTIVE-XLX (for XILINX), ACTIVE-LAT (for LATTICE), etc. These tools include in addition a complete set of simulation libraries for specific device design and a proprietary Design Manager that automatically locates all software tools for the specific IC device and displays them as design flow buttons. Activating these buttons automatically transfers design data between applications and simplifies the interfaces to different design tools.

ACTIVE-CAD is for Board Level Designs

ACTIVE-CAD is particularly useful when troubleshooting designs with multiple PLDs, PALs, FPGAs and other forms of ASIC technology. Such designs can be analyzed both at the chip and system level. System level designs, such as communication and industrial controllers with multiple processors, memories, etc., can also be analyzed with ACTIVE-CAD.

Extensive System Level Libraries

ACTIVE-CAD is supported by extensive IC libraries for board-level design simulation. The TTL library comes with the system level ACTIVE-CAD package. Additional libraries such as CMOS, ECL, PLDs, memories and microprocessors can be purchased separately, as needed.

Supports All Popular Logic Synthesis Tools

ACTIVE-CAD has direct interfaces to Mentor Graphics (Exemplar), IST, and other logic synthesis tools. They are tightly integrated with ACTIVE-CAD for ease of use and error free operation. ALDEC provides turn-key FPGA design tools which include some of these logic synthesis tools.

Chapter 2

ACTIVE-CAD Schematic Editor

Introduction

The ACTIVE-CAD schematic editor is a PC-based design entry tool. It has many features not found in other similar products.

- ☐ As a true Windows application, ACTIVE-CAD has user interface that is very intuitive to anyone who has used any Windows application. The learning curve is short because of easy-to-use menus and toolbars, context sensitive help, and drag-and-drop operations.
- ☐ ACTIVE-CAD provides hierarchical design support for mixed Boolean, VHDL, and schematic designs.
- ☐ Both analog and digital schematic entry are allowed.
- ☐ ACTIVE-CAD allows PLD, CPLD, FPGA, MPU, DSP and PCB designs.
- ☐ Automatically connects device pins, which speeds schematic entry.
- ☐ An on-line simulator allows design testing on the schematic screen.
- ☐ Generates schematics directly from netlist files.
- ☐ Combines design blocks from different CAE tools on the same sheet. .
- ☐ EDIF support opens access to many UNIX EDA tools such as Mentor, Cadence and ViewDraw.

Schematic Fundamentals

Circuit definition

A schematic editor is typically used to create an electrical schematic drawing, which depicts the used components and their connections. The schematic drawing can be used in the hard copy form as documentation on how to assemble the circuit.

The schematic design can also be converted into electronic forms for use with other CAE tools such as PCB or FPGA layout programs or logic simulators. The file that contains information about parts and connections in the design is called the netlist. This file does not contain any graphical information about the design, but it is sufficient to create a physical board or an FPGA design. There are several netlist formats used by different tools. The ACTIVE-CAD schematic editor supports over 15 of the most popular ones.

Symbols and Libraries

The components used in the circuit design need to have graphical symbol representation. The ACTIVE-CAD schematic editor comes with a set of graphical symbol libraries with over 6,000 elements. In addition ACTIVE-CAD has a Symbol Editor program for creating new symbols and modifying the existing ones.

A schematic symbol consists of the symbol body and pins. The symbol body is a graphical drawing that may be used to describe the logical function of the component. The pin is a graphical line that represents the device connection point, used to connect to other devices. The pin can be used for a single connection or for an array of wires (bus pin).

Physical and logical connections

There are two (2) ways you connect devices on a schematic:

- ☐ A **physical connection** is a graphical line (called wire) drawn on the schematic between two or more pins.
- ☐ A **logical connection** is a connection made between two or more pins without drawing the connecting wire line. This type of connection is typically made by assigning identical names to two or more wires connected to device pins. Another example of logical connections is a bus line.

Support for large designs

The designs that do not fit on one schematic page can be easily managed by using multiple schematic pages or as a hierarchical design structure. The multi-page schematic is no different from a single page, except you cannot make physical connections between pins on two different sheets. Instead, you must use the logical connections such as net names or bus names.

NOTE: To connect circuits placed on multi-sheet designs, you must use the logical connections. You cannot use direct wiring or I/O terminals.

Hierarchical designs are based on dividing a design into smaller sections (macros). These sections are single-page schematics converted by the schematic editor into schematic symbols and saved in the project library. These macro symbols can then be used in other schematics and connected to other symbols.

The same schematic macro symbol can be used several times. The higher level schematic takes less space and is usually more clear to the user, because it uses higher level blocks to build complex designs. The connections between the hierarchical blocks can only be made through the I/O pins; i.e. in order to connect two symbols on different hierarchical levels, you have to create I/O pins in the hierarchical symbol and then connect these pins on the higher level schematic.

NOTE: Hierarchical macros (schematic sheets) can be connected at the higher hierarchical level only through their I/O terminals. The logical connections through net names are not permitted in hierarchical designs.

The schematic editor can generate designs of virtually any size, limited only by the available memory and disk space in your system.

ACTIVE-CAD handles sheet sizes A, B, C, D and E (metric sizes A4-A0). Custom and unlimited page sizes are also allowed.

Design verification

One of the major problems with most schematic editors is the number of steps to verify even the simplest design. The verification process consists of running a netlist generation program, checking that all connections made on the schematic are electrically correct, and then testing the circuit behavior with the help of the logic simulator. If any mistakes are found, the schematic must be corrected and the entire process must be repeated.

The ACTIVE-CAD schematic editor resolves this problem by integrating it with an on-line logic simulator. Since all schematic entries are dynamically passed to the simulator, they are automatically verified as they are placed on the schematic.

The the simulation data is also available in the schematic editor. You can select test points on the schematic, run simulation and watch how the device pins are changing their logical states. Any change to the schematic design instantly produces new logical signals in the simulator and on schematic sheet.

Reports

The schematic editor can create several types of design reports:

- ❑ **Bill of Materials** reports which parts are used in the design. It is very useful for prototyping and manufacturing because it helps in ordering parts and calculating costs. The Bill of Material helps to estimate the FPGA design size and if it will fit into a specific device.
- ❑ **Report** files can be converted into files compatible with spreadsheet and database applications. For example, you can use the **Report** in conjunction with Excel spreadsheet or Access database. This allows the use of company standard software tools to produce customized reports, with additional manufacturing information and comments.

Interfaces

Schematic Editor provides flawless interfaces to many design tools:

- ❑ **FPGA interfaces** include: design entry with a specific macro library, layout control by means of layout parameters assigned to symbols and nets, import and export of FPGA netlists, control of custom macros developed in VHDL or state machine languages, schematic generation for HDL macros, automatic update of schematics when their HDL sources change, and back annotation of layout data to the schematic (e.g. post-layout pin assignment).
- ❑ **PCB interfaces** include: symbol libraries for variety of active and passive components, PCB footprint control on the schematic, control of power connections, layout parameters like placement groups and net widths, PCB netlist generation, back annotation of changes made in PCB programs through ECO (Engineering Change Order) files, and forward annotation of changes made on the schematic after the PCB layout has been created.
- ❑ **Analog designs** support includes: a SPICE library of analog symbols and models, analog design rules checker that verifies if the design can be simulated, SPICE netlist output, selection of test points and means for design modification and viewing its operation.




Starting the Schematic Editor

The Schematic Editor can be invoked by clicking on the **Schematic Editor** button in the project flow section of the Project Manager or by selecting **Schematic Editor** option from the **Applications** menu. If the Schematic Editor is started directly from the File Manager or Windows 95 Explorer, the last project will be automatically loaded. If you start the

Project Manager first, then you will be able to select either an existing project or start a new one.

How To Quickly Start A New Project

The fastest way to start a new schematic is by following these rules:






1. Start the **Project Manager** by double clicking on the **ACTIVE-CAD** icon  in the ACTIVE-CAD group of icons.
2. When the **Project Manager** window appears, click on the **New Project** icon  to invoke the **New Project** window.
3. Enter the project name, its directory and type into the **New Project** window. Some design types will additionally allow to select device family, part number and device speed.
4. Select the libraries by clicking on the **Project Libraries** icon .
5. When the **ACTIVE-CAD Libraries** window appears, select the libraries to be used in the project. Select from the **Attached Libraries** field the necessary libraries and click the **Add>>** button to copy them to the right-hand side field. For description of the system libraries see the Appendix A - *ACTIVE-CAD Libraries*. To save on memory space and to improve the device search time, select only the necessary libraries. You can add more libraries to the current project at any time.
6. Close the **ACTIVE-CAD Libraries** window.
6. Start the schematic editor by clicking on the **Schematic Editor** button in the **Project Manager** window.
7. The schematic editor window will open a new schematic sheet (.sch). Its name will be the same as the current project's name. However, the number '1' will be appended to the project's name. For example, project *TEST* will open a new schematic sheet called *TEST1.sch*.

Before starting schematic entry, read the following section:

Basic Design Entry Operations

After setting up a new project, perform the following tasks:

- ☐ Set up the schematic page size, orientation and border type with the **Page Setup** option in the schematic **File** menu.
- ☐ Set the drawing table by clicking on the **File | Table Setup** option.

- ☐ Select the **Symbol Entry** mode by clicking on the  icon in the schematic vertical toolbar or pressing the **F2** keyboard key. Start entering devices by dragging them from the **SC Symbols** toolbox.
- ☐ Select the **Draw wires** mode by clicking on the  icon and draw connections
- ☐ To make any changes to schematic design, click on the **Select and Drag**  icon.
- ☐ Since ACTIVE-CAD operation is based on an incremental compiler, you can simulate devices as you place them on the schematic sheet. To activate the on-line simulator, click on the **Simulator**  icon.
- ☐ Activate the **Select Test Points**  icon and click on the desired test points on the schematic sheet. The selected test points are instantly displayed in the simulator and schematic windows.
- ☐ Use the simulator test vector editors to generate test vectors.
- ☐ Apply the test vectors to the desired schematic test points and click on the **Single Step** or **Long Step** simulation buttons in the Simulator toolbox.

What happens when you start the schematic editor

When you start the schematic editor, the following operations are performed automatically:

- ☐ The list of symbols is loaded into the memory. The list contains all the symbols from all libraries that have been selected for the project. The list contains only symbol names and not the actual symbols.
- ☐ The project contents file (*.PRJ) is opened to load the list of schematic sheets that belong to the current project.
- ☐ The previously used schematics will be open. They will look exactly as they were displayed on the screen, including window positions, zoom, and other settings that were saved during the last session.

NOTE: If you do not want to load the last project, select another one in the **Project Manager** before starting the schematic editor.

Screen Layout

Figure 2-1 shows the main Schematic window. Its arrangement has been designed to simplify design entry. Note that multiple schematic win-

dows can be opened within the main editor window to allow for simultaneous editing of multiple schematic sheets.

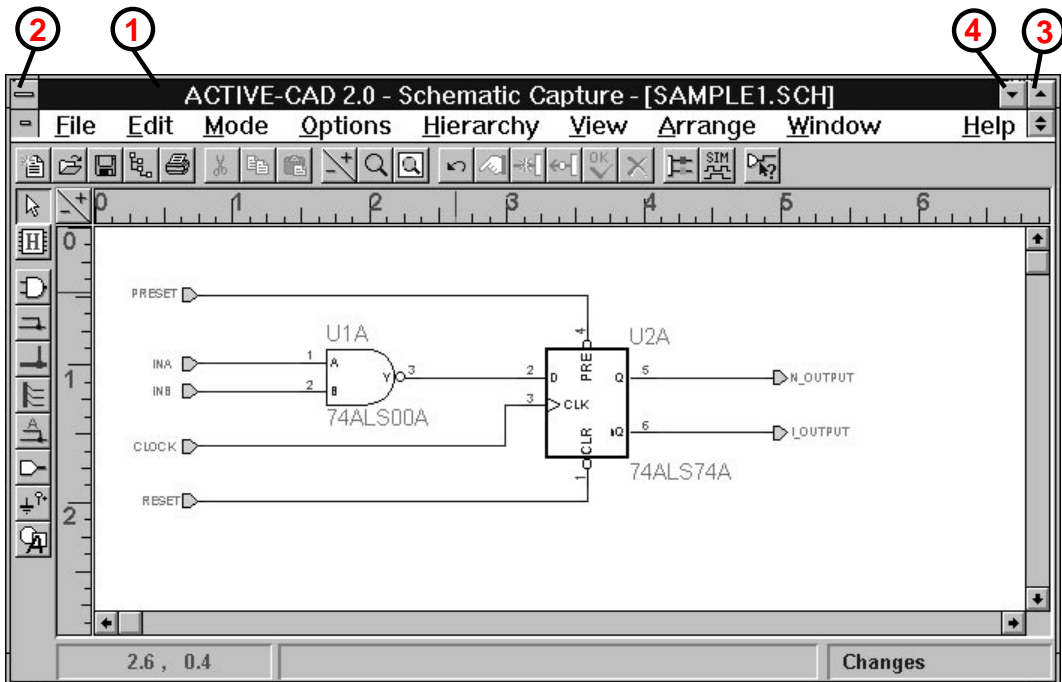


Figure 2-1. Schematic Editor Main Window.

Standard Windows Top Bar

The standard Windows top bar includes the **Maximize** button (3) for expanding the schematic window, the **Minimize** button (4) for reducing the schematic to an icon, **Title** bar (1) and **System** button (2). Their functions and behavior are the same as in all other Windows applications.

The equivalent Windows 95 buttons are described in the Appendix *Windows 95 User Interface Summary*.

ACTIVE-CAD Proprietary Toolbars

The ACTIVE-CAD operations are controlled by activating icons or buttons in vertical and horizontal toolbars. The horizontal toolbar is comprised of five (5) groups of icons. Since each of these groups handles different set of operations, they will be described separately.

Schematic Sheet Management Icons

The icons in Figure 2-2 are used for project management.

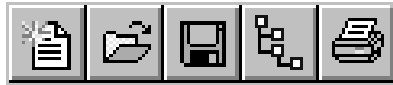


Figure 2-2. Project Management Icons.



- **New Schematic** icon allows to start a new schematic sheet.



- **Open a Schematic** icon allows to open one of the existing schematic sheets.



- **Save** icon allows saving current schematic designs. The icon is active only when some changes have been made to the schematic sheet since it was last saved. Otherwise, it will remain gray or inactive.



- **Projects** icon displays the Project Manager screen. Refer to *Design Process Management* chapter for details.




- **Print a Schematic** icon allows typical Windows print operation.

Cut And Paste Operations




Figure 2-3. Cut And Paste Icons.

The cut, copy and paste operations require setting ACTIVE-CAD into the

Select mode by clicking on the **Select and Drag**  icon. The icons in Figure 2-3 allow for typical cut and paste operations.



Cut icon allows to cut a selected schematic section into the Clip-

board. First, activate the **Select** mode by clicking on the **Select**  icon. Next, drag a rectangle around the selected items, or select them by holding down the **Shift** key and clicking on each item. Clicking on the **Cut** icon will remove the selected items from the schematic sheet and place

them in the Clipboard. All wires remaining entirely within the cut area, will also be copied into the Clipboard.



Copy icon allows to copy sections of the schematic to the Clipboard. Choose the items to be copied, and then click on the **Copy** icon.



Paste icon places the contents of the Clipboard at the selected screen location. First activation of the **Paste** icon displays a dotted outline of the Clipboard contents. The second click on the **Paste** icon places the contents of the Clipboard at the selected location.

Zoom Operations



Figure 2-4. Zoom Operations.

The group of icons in Figure 2-4 allows quick zoom operations.

Zoom buttons



There are two **Zoom** buttons. The button marked with plus (+) represents the **Zoom In** option, the one marked with minus (-) represents the **Zoom Out** option. The **Zoom In** and **Zoom Out** options allow quick selection of the next larger or smaller predefined viewing scale.



Zoom Area icon allows selection of the schematic area to be displayed over the entire screen. The first click on the **Zoom Area** icon activates the zooming cursor. Place that cursor at the first corner of the area to be enlarged. Press the mouse button and drag it to the opposite corner of the selected area. Upon release of the mouse button, the selected area will expand to cover the full computer screen.




Full Page Zoom icon displays the entire schematic sheet.


Editing Operations


The editing operations are controlled by the icons in Figure 2-5.





Figure 2-5. Editing Options Icons.

 **Undo** icon eliminates the last schematic change and restores the schematic to its state prior to that change.

 **Properties** icon allows to assign parameters to wires and symbols. Activating the **Select** schematic mode and clicking on a wire or symbol, followed by clicking on the **Properties** icon, will display the **Net Attributes** or **Edit Symbol** window, and will allow editing the wire or symbol parameters, respectively. For more details read sections *Wire Parameters* and *Symbol Parameters*.

 **Connect Symbol** icon connects the symbol pins with the overlapping end-wire symbols (crossed circles or crossed squares). This operation is used to replace parts and reconnect wiring.

 **Disconnect Symbol** icon disconnects all wires from the selected symbol and places wire-end symbols (crossed circles or crossed squares).

 **End** icon is used to end wires and buses at the current screen location. The wires will be marked with wire-end symbols.

 **Cancel** icon is used to cancel the wire and bus being drawn.

Design Verification

The design verification is comprised of connectivity checking and simulation. The icons in Figure 2-6 allow for simulation control and schematic connectivity query.



Figure 2-6. Schematic Design Verification Icons.



Simulation toolbox icon activates a toolbox with icons for placing test points on schematic, replacement of functionally equivalent parts, and execution of simulation steps. The selected test points can be used to apply stimulus signals (test vectors) or to monitor the design behavior.



Simulator icon generates schematic netlist and activates simulator. If the simulator was already active, it will pass control to the simulator.



Query Window icon lists connectivity for the selected items.

Schematic Entry Operations

The design entry operations are controlled by the icons in Figure 2-7. They are located vertically on the left-hand side of the screen. However, they can be dragged to other screen locations.



Figure 2-7. Schematic Entry Options Icons.



Symbols toolbox icon activates a toolbox with device symbols. The toolbox also includes icons for selecting desired types of symbols and allows searching for the selected parts or types of parts.



Draw wires icon activates the wire drawing mode. Clicking on a pin starts a wire. Clicking on another pin automatically draws a wire. Double clicking on an empty space starts a wire with wire-end symbol.



Draw buses icon allows drawing buses by clicking at any screen location. To end a bus, click the right mouse button or activate the **OK** icon.



Draw Bus Taps icon activates drawing connections between buses and pins - bus taps. Clicking on a bus and then clicking on symbol pins, will automatically connect these pins with buses and sequentially number the bus taps.



Add Net or Bus name icon is used for naming nets and buses. Clicking on a net or bus in the **Select** mode will turn the selected item red. Activating the **Add Net or Bus Name** icon will display a window for naming the selected item.



I/O Terminal icon allows naming I/O terminals and specifying their direction. This icon is active only when the **Draw Wires** icon is also activated.



Power Symbol icon allows for direct placement of power symbols on the schematic sheet. This icon is active only when the **Draw Wires** icon is also activated.



Graphics toolbox icon allows drawing graphical elements. All items drawn in this mode will have no electrical properties. This mode should only be used for non-electrical drawings or for adding additional information that does not need to be saved in the electrical database.

General Operations



Figure 2-8. General Operations Icons.



The **Select and Drag** icon is the most often used operation. It allows selecting objects in the schematic for further processing. The **Select** option is set by default by several schematic editor operations.



Hierarchy Push/Pop icon activates the hierarchical **H** cursor. Clicking with this cursor over a macro will display its internal schematic. Clicking on a primitive cell will display an advisory message that no schematic for the selected part is available. Clicking in an empty space, will display the higher hierarchical level.

The **Select** and **Hierarchy Push/Pop** icons do not produce any design changes. They only facilitate other operations on the schematics.

Menu

The menu bar includes the main schematic editor options. Each of these options is described in detail in the *Schematic Menu Reference* chapter.

Coordinates

The status line in the bottom left-hand corner displays the cursor coordinates as it moves over the schematic. The coordinate values are displayed in inches, millimeters or zone references. To select one of these ruler types, use the **Page Setup** option in the **File** menu.

Ruler

The ruler display is controlled by the **View | Ruler** option. The ruler is displayed on the left (vertical ruler) and top (horizontal ruler) of each schematic window.

The ruler can be marked in inches (English), millimeters (Metric) or Drawing Zones. The ruler type is selected from the **Page Setup** option in the **File** menu.

Scroll bar

The scroll bars are located on the right and at the bottom of each schematic window. The operation of the scroll bars is identical to all Windows applications, and allows for smooth scrolling of the schematic in vertical and horizontal directions.

Schematic Page Setup

A schematic can be comprised of multiple sheets or pages, which can be created with the **New Sheet** command in the **File** menu. You can draw a schematic on a new sheet immediately after it has been created. The new schematic sheet will have the default size and frame, and the same name as the current project.

You can change the default page settings by using the **Page Setup** option (Figure 2-9) in the **File** menu. Using the **Frame**, **Sheet Format** and **Ruler** fields in Figure 2-9, you can specify:

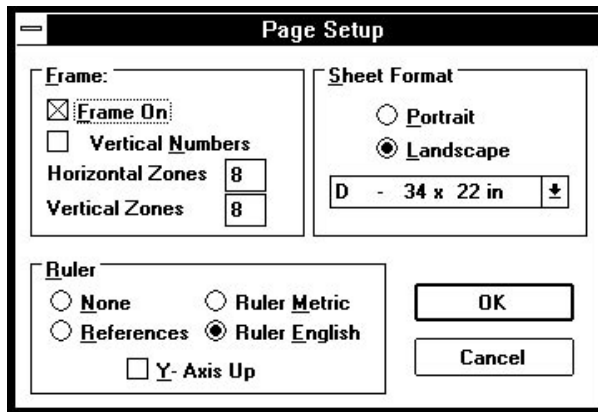


Figure 2-9. Page Setup Options.

- ☐ Frame on/off and the number of page zones
- ☐ Sheet size (A-E, A0-A4, any desired size)
- ☐ Sheet orientation (Portrait or Landscape)
- ☐ Ruler setting (None, Metric, English, References-Zones)
- ☐ Y-axis origin

ACTIVE-CAD allows to customize the design table that is displayed and printed in the bottom-right corner of the schematic sheet. To customize this table, use the **Table Setup** option in the **File** menu.

The **Table Setup** window (Figure 2-10) allows selecting the desired logo graphics file (metafile) to be displayed in the design table.

NOTE: The logo file has to be in the Windows Metafile format **without the header**. A Windows Metafile can be created with programs like CorelDraw.

Table Setup

Address:	
Line1:	ALDEC
Line2:	3525 Old Conejo Rd. #111
Line3:	Newbury Park, CA 91320
Logo MetaFile:	
	<input type="button" value="Browse ..."/>
Field name:	Date
Field description:	05/27/93
Table Contents:	
Project :	DYNET
Sheet :	2/2
Date :	05/27/93

Figure 2-10. Table Setup window.

Saving The Page Layout

The settings selected with the **Table Setup** option are saved in the project description file (*.PDF). These settings are in effect for all project sheets. However the sheet or page numbering is automatically controlled. All, settings in the **Page Setup** window (Figure 2-9) are saved in each schematic file (*.SCH), and can be different for each schematic sheet.

Building a Schematic

Each schematic consists of the following items:

- ☐ Page Frame and Table
- ☐ Symbols/Components
- ☐ Connections (wires or buses)
- ☐ Terminals and net names
- ☐ Graphics and Text

The page **Frame** settings are controlled using the **Page Setup** option (Figure 2-9), in the **File** menu.

Each symbol item consists of a graphical symbol, component name, component type and attributes. For example, the NAND gate item in Figure 2-1 consists of the graphical NAND symbol, the name *UIA*, the type (part number) 7400, and specific graphical attributes like rotation and mirror, which will be explained in detail later on.

Wire connections can be drawn:

- ☐ from pin to pin
- ☐ from pin to I/O terminal
- ☐ from pin to an existing wire connection
- ☐ from an existing wire connection to a pin

You can end a wire at any schematic location by double clicking. This will produce a dummy terminal, also called wire-end, which looks like a circle or little square with a red cross inside.


All connections are validated while they are drawn, and the editor does not allow starting or finishing a wire that does not represent a valid electrical connection. This restriction is due to the on-line simulation capability of the ACTIVE-CAD system. If the ACTIVE-CAD background simulator is active while you draw a schematic, then each schematic change is instantly (incrementally) passed to the simulator, and the design can be analyzed for electrical behavior as it is drawn.

Inactive Schematics


If the opened schematic does not belong to the current project, you will only be able to view it. However, you will not be able to edit it until you add it to the project with the help of the **Add Sheets To Project** option in the **Hierarchy** menu. The prompt message continuously warns you that the schematic does not belong to the current project and cannot be edited.

Accessing Schematic Areas (Zooming)

To view a specific schematic area, use the following options:

- ☐ zoom in on a selected area
- ☐ zoom on entire schematic sheet by clicking the **Full Page Zoom** ( icon)
- ☐ swap current page with a full page display (**PgUp**, **PgDn** keys)
- ☐ center the selected area on the schematic display
- ☐ use auto panning to access the desired schematic areas
- ☐ scan the window

Zooming in on a selected area

Select the **Zoom Area** () icon in the top bar and drag it from one corner to the opposite corner over the selected area. Upon release of the mouse button the marked area will be expanded to the full screen.

To zoom in on a selected area, click at one of the corners of the area to be expanded and press both the **Ctrl** key and the left mouse button. Holding the mouse button down, drag the cursor towards the opposite corner of the desired area. A dotted rectangle is displayed during this operation to mark the selected area. When the desired area is within the rectangle, release the mouse button. The selected area will expand over the entire display.

Centering the selected area on the schematic display

Activate the **F9** keyboard key or select the **Center** option from the **Display** menu. In response, ACTIVE-CAD displays a bulls eye cursor. As you move this cursor to the display borders, auto panning (shifting) of the display area takes place. If you click the left mouse button, the area under the cursor will be displayed in the center of the screen.

Using autopanning to access the desired schematic areas

You can move to the desired schematic areas by dragging an element, wire or bulls eye produced by selecting the **Center** option. Auto panning is one of the most natural and most often used options to access the hidden schematic sheet areas.

Scanning window

You can concurrently view the same schematic area in two windows with totally different magnifications. To open the scanner window for global schematic viewing, click on the **Scanner** option in the **View** menu. When the scanner window appears, slide its top bar to set the drawing magnification ration.

Next, click on the **Move Scanner** option in the **View** menu. A dotted rectangle starts following the cursor. Place it over the selected area and click the mouse button. The selected schematic area will appear in the scanner window.

Using the top bar button, adjust the scanner display magnification; Moving the top button to the left reduces the schematic magnification. Moving it to the right increases the scanner magnification.

All new wires, terminals and components are instantly displayed in the scanner window. This window looks identical to the main editing window but it does not allow for any editing or option selections. The scanner is primarily used for viewing the sheet layout while the design is in progress.

Placing Symbols

To place a new component on a schematic sheet, switch to the **Symbol** mode by either pressing the **F2** key or clicking on the **Symbols Toolbox**



icon in Figure 2-1. A symbol library window similar to Figure 2-11 is automatically displayed on the right side of the screen.

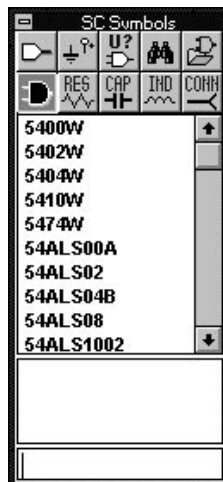


Figure 2-11. SC Symbols Toolbox.

Selecting symbols

To find the desired component, click on any symbol listed in Figure 2-11 to invoke the highlighted cursor. Next, scroll the symbol list with the cursor keys or the scroll bar. If you type the symbol name at the bottom of Figure 2-11, it will be automatically searched for and highlighted in the list.

The currently selected symbol is highlighted in color. If you move the mouse cursor to the schematic window (do not drag!), the graphical symbol of the selected component will follow the cursor.

TIP: You can view the symbols without placing them on the schematic by moving the cursor with the currently selected symbol over the sche-

matic area and then scrolling through the list of symbols with the arrow keyboard keys. The symbol for a current name on the list will appear at the cursor position.


Quick Symbol Finder

If you know the symbol name you want to place on the schematic, just type its name in the edit line which is located right at the bottom of the **SC Symbols** window (Figure 2-11). As you type the symbol name, the list is scrolled and Highlight Cursor points to the first matching symbol.

Placing a Selected Component


To place a symbol, move the symbol attached to the mouse cursor to the desired location and click the left mouse button.

As you move the symbol toward the outside of the window, the schematic will automatically scroll under control of the auto-panning software.

 **TIP:** You can rotate any component attached to the cursor by pressing **Ctrl+L** (left rotation), **Ctrl+R** (right rotation) or **Ctrl+M** (mirror image).


To place another symbol on the schematic, select it in the **SC Symbols** window and repeat the component placement process.

Duplicating symbols

With **Symbol Toolbox**  icon active, clicking on any symbol will produce its copy (duplicate), which will automatically follow the mouse cursor. You can place the symbol duplicate at any screen location by clicking the mouse button. This feature is available only if the schematic is in the **Symbol** (placement) mode and if no other symbol is currently attached to the cursor. If a symbol is attached to the cursor, click the right mouse button to release the existing symbol, and then click on the desired symbol that you want to duplicate. Each placed symbol is automatically given a new reference number.

Deleting symbols

To delete the symbols you have just placed, press the **Del** keyboard key. Each time you press this key, a new symbol is being deleted. The order of deletions is the reverse of the order in which the symbols were placed in the schematic. This delete operation is disabled once you switch out of the **Symbol Toolbox** mode. Returning back to the **Symbol Toolbox** mode will not reinstate the symbol delete option.

You can delete any symbol, terminal or wire in the **Select** () mode. This is the most popular way of editing a schematic. To delete an element, select it with the click of the mouse button and then press the **Del** keyboard key or select the **Delete** option from the **Edit** menu.

If you have mistakenly deleted a symbol, you can put it back by clicking on the **Undo** option in the **Edit** menu.

Symbol Overlap


All symbol outlines are rectangular. They are displayed when the symbols are selected in the **Select** mode. At least one grid step of free space is required between symbols. The schematic editor allows disabling of the symbol overlap so that the symbols may be placed closer than specified by their rectangular outlines.


To turn the symbol overlap control on/off, select the **General** option from the **View/Preferences** menu. Next, select the **Allow symbol overlap** option in the **General Settings** window.

NOTE: The ACTIVE-CAD schematic editor does not allow connections by placing two symbols next to each other so that the pins appear connected. Wires need to be drawn between pins to make a connection.

Symbol Filter

ACTIVE-CAD libraries include a large number of symbols. Each symbol has attributes that determine its class and logical function. These attributes can be used to simplify the search for the desired symbols. In order

to filter the symbols in the symbol list, press the **Filter**  icon in the **SC Symbols** toolbox. In response, the **Library Filter** window is displayed (Figure 2-12).

 **TIP:** If you want to find a symbol, but you don't know its entire name, enter the part of the name as a Filter mask, with an asterisk (*). For example, to find all TTL 245 devices, enter a mask *245* in the **Symbol Name** field of the **Library Filter** window.

The Library Filter window (Figure 2-12) shows available attributes and symbol classes. Select the desired symbol classes (technologies and IC families) and logical functions and press the **OK** button. A listing of components with the selected characteristics will appear in the **SC Symbols** window.

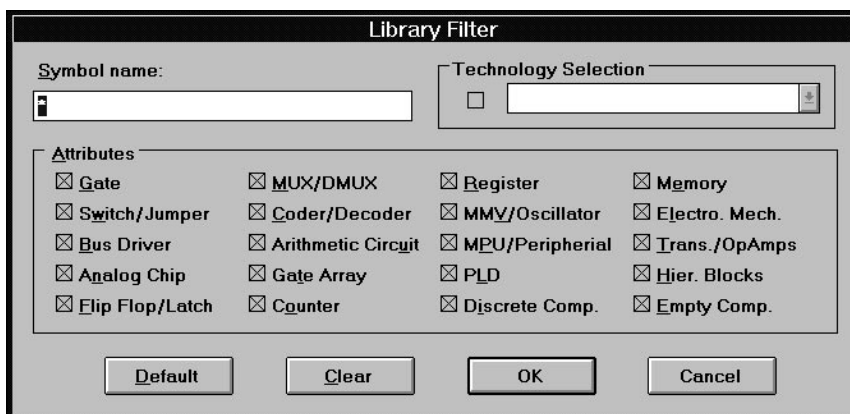


Figure 2-12. Library Filter Window.

Since all information about each device resides in its simulation model, you need to load the model library to effectively use this option.

Selecting resistors, capacitors and inductors



Figure 2-13. Selecting Resistors.

To speed drawing of resistors, capacitors and inductors, use one of the three icons serving these components in the **SC Symbols** toolbox (Figure 2-11). For example, by activating the **RES** icon, a special window for quick selection of the desired resistor appears (Figure 2-13). First, select the resistor type from the available menu (fixed, potentiometer, variable, etc.). Next, select its Value, Tolerance and Wattage from the appropriate fields. The selected resistor will follow the cursor. Clicking at the desired screen location will place the symbol.

The capacitors and inductors are selected in a similar manner, using the **CAP** and **IND** icons, respectively. The selected component value, tolerance and wattage can be viewed in the **Query** mode.

The resistor, capacitor and inductor symbols are built-in to the schematic editor and do not require selecting any library. Additional passive symbols, like resistor packs, switches, etc. can be found in the PASSIVE and DEVICE1 libraries. You need to attach these libraries to the current project to use their components.

To draw Spice analog circuits, select to the project the Spice analog library in the **Project Libraries** (**File** menu).

Connector

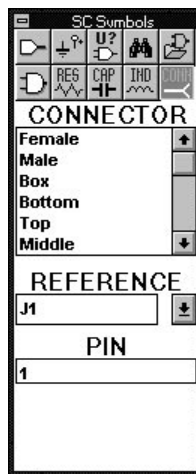


Figure 2-14. Selecting Connector Pins.

To put a single connector pin on the schematic, click the **CONN** icon in the **SC Symbols** toolbox. Select the connector type, reference name and pin number from Figure 2-14. The connector pin is placed as any other symbol on the schematic, with the exception that all pins of the connector with the same reference name are exported as the same connector symbol in the netlist.

To place a connector as one symbol instead of multiple separate pins, use the symbols from the DEVICE1 library.

Symbol Numbering

All symbols are automatically named when placed on the schematic. The symbol name is composed of a text string and a numerical suffix, which is incremented every time you place a symbol. Examples are U1, U2, U3, etc. You can choose several symbol name options by selecting the **Annotate** option in the **Options** menu. In response, the **Annotation** window (Figure 2-15) appears and allows renaming parts.

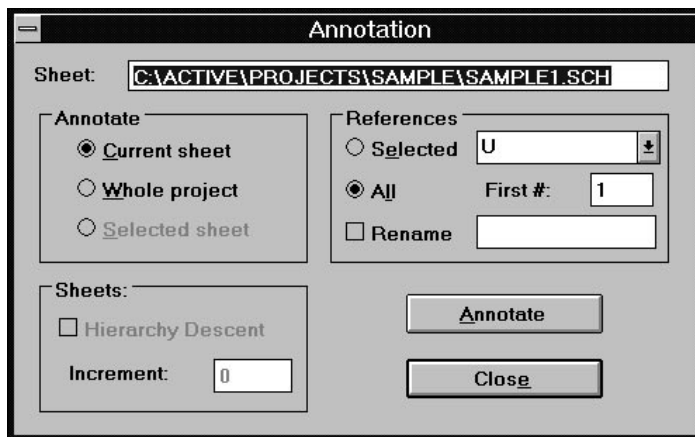



Figure 2-15. Symbol Naming Options.

Following is a description of the **Annotation** window:

- ☐ **Annotate field:** you can annotate parts on the current sheet, in the entire project, or on a selected schematic sheet.
- ☐ **Sheets field:** checking the **Hierarchy Descent** box, will force components in the hierarchical schematics to have unique names. For example, if you have two schematic devices in your design, both of which contain symbols called U1 and U2, ACTIVE-CAD will rename the U1 and U2 symbols in the hierarchical schematics to unique names. The **Increment** field can be used to specify the increment between hierarchical schematics of the renamed symbol names. For example, if you select an **Increment** of 100, then the U1 and U2 symbols will be renamed U100 & U101 in one schematic and U200 & U201 in the other one.
- ☐ **References field:** You can specify the reference prefix, (e.g. U for U1, U2) in the edit field. The **First #** setting defines the lowest reference number, (e.g. **First #** = 5 will start numbering components from U5, U6, etc.) Selecting the **Selected** option and the **Renamee** box option allows renaming existing components. Enter the prefix of the existing components into the prefix field (e.g. U) or select **All** to rename all components. Next, enter the new prefix in the **Rename** field and press the **Annotate** button.
- ☐ Clicking on the **Annotate** button will annotate schematic sheets according to the selected settings.

ACTIVE-CAD automatically tracks the device numbering. For example, if you add a new sheet to the current project, all devices on this sheet will be automatically renumbered in the sequential project order.

Displaying Component Attributes

After placing a component, you can specify its attributes. Switch to the **Select** mode, click on the symbol to which you need to assign parameters, and then click on the **Properties**  icon. When the **Edit Symbol** window appears, click on the **Attributes** button. In response, the **Symbol Attributes** dialog box (Figure 2-16) will appear..

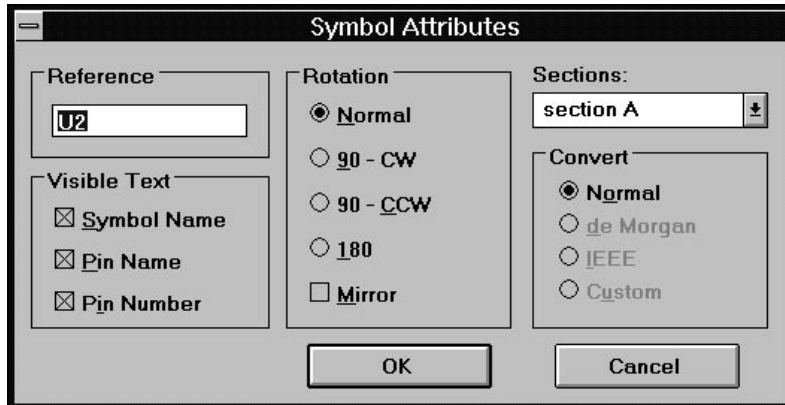


Figure 2-16. Selecting Symbol Attributes.

The component attributes include:

- ☐ Rotation and mirror
- ☐ Enable or disable the pin names, pin numbers, and symbol name appearance on the screen
- ☐ Auto Reference and Auto Section numbering control
- ☐ Manual section selection
- ☐ De Morgan and other symbol representations

Rotating a symbol

To rotate a single symbol counter-clock-wise (CCW), press the **{Ctrl+L}** keyboard keys while the symbol is attached to the cursor. Each **{Ctrl+L}** activation will rotate the symbol additional 90 degrees. To rotate symbol in the clock-wise direction (cw), click on the **(Ctrl+R)** keyboard keys. Pressing **(Ctrl+M)** keyboard keys will create a mirror image of the symbol.


If the symbol has already been placed on the schematic, highlight it in the **Select** mode and then click on the **Properties** icon. This will invoke the **Edit Symbol** window. From the **Edit Symbol** window select the **Attribute** option, and when the **Symbol Attributes** menu appears (Figure 2-

16), select the desired component rotation. To complete this operation click on the **OK**, **Update** and **Close** buttons.

Drawing connections

To enter the drawing connection mode, click on the **Draw wires** icon



() or press the **F3** key. The wire connectivity cursor, which displays a pencil and a wiring node, appears automatically. Clicking with the wiring cursor at pins or wire connections will start a new connection. Clicking on another pin or wire, will end the wire connection. The **Draw wires** icon activates drawing of single wires. In addition, there are several other modes of symbol connectivity:



Draw buses icon activates the bus drawing mode¹.



Add Net or Bus Name icon allows placing a label on wires and buses.



Draw bus taps icon allows drawing bus taps and add bus tap names to existing wires.



I/O Terminal icon is used to place I/O terminals on schematic sheets. They can be directly connected to wires and buses.



Power Symbol icon is used for connecting wires to power and ground.



Undo icon is used to undo the last wiring operation.



Cancel icon is used to cancel currently drawn wires and buses.




End icon is used to complete the drawing of a bus or wire at its current cursor location. Wires are terminated with crossed circles or squares.

¹ Drawing buses and bus taps is described in the **Buses** chapter.

Placing Wires

The schematic editor allows drawing a wire only if it is electrically correct and represents a physical connection between component pins or I/O terminals. To temporarily end a wire at any place on the screen, click

on the right mouse button, followed by clicking on the **End** () icon. This places wire-end symbol, which is a crossed red circle and allows continuation of the wire from the selected screen location.


To start drawing a wire, click on the component pin or in its proximity. The cursor will snap to the pin, and a wire will be drawn as you move the mouse. To approve a corner or a wire segment, click the left mouse button. The approved wire section will change its color. To temporarily exit this mode, click the right mouse button. In response, the currently drawn wire will freeze in its last position. If you click on the **End** icon, the wire will terminate with the wire-end symbol, which is a blue filled circle. If you do not click on the **End** icon, then clicking anywhere on the schematic screen will automatically connect the present cursor position to the last wire position.

Selecting the wiring mode puts the editor in the auto-panning mode. Every time you move the mouse cursor closer to the window border, it will automatically scroll to allow instant access to other parts of the schematic without using the scroll bars.

Adding I/O Terminals

I/O terminals represent hierarchical macro I/O pins. These are logical connectivity symbols only, and they do not represent any physical connections. Only I/O terminals on the top level schematic are automatically converted into device I/O pins and will be added to the layout netlist as connector pins.

NOTE: If you intend to place a physical connector, use the connector symbols instead of the I/O terminal symbols.

To activate the **I/O Terminal** icon, select either the **Draw wires** () icon or draw a bus line. To place an I/O terminal, click on the **I/O Terminal** icon. In response, **I/O Terminal** window appears (Figure 2-17) and allows entering the terminal name and its type (input, output, bi-directional or unspecified).

Enter the terminal name into the **Terminal Name** field and then select the terminal type from the **Terminal Type** list. Click on the **OK** button. A

box follows the cursor until you click to place the terminal on the schematic .

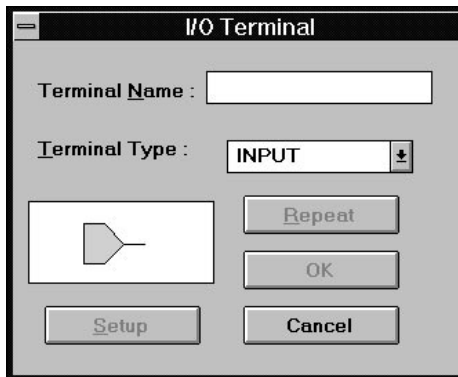


Figure 2-17. Entering I/O Terminals.

You can place a terminal on the schematic before you draw a connection or while you draw a connection. To end a wire with an I/O terminal, start drawing the wire and when you get to the location where the terminal should be placed, click the right mouse button to disconnect the wire from the mouse cursor, and click on the **I/O terminal** icon. In response, **I/O Terminal** window will appear. After selecting the terminal name and I/O type, press the **OK** button. The terminal will be placed at the location where the wire was previously terminated.

You can place a sequence of similar I/O terminals using the **Repeat** button displayed within the **I/O Terminal** window. Activating the **Repeat** button sets ACTIVE-CAD into the repeat mode and each mouse click will produce a new terminal on the schematic. The terminals are numbered sequentially, e.g. *IN1*, *IN2*, *IN3*,.... To exit this mode, click the right mouse button. For more details, please refer to the *Repeat Option* section.

The **Repeat** has two modes of operation:

- ☐ *Terminal symbol*; duplicates terminals with names incremented by '1'.
- ☐ *Terminal connection*: draws identical wire connections which start at symbol pins and end with I/O Terminals. The terminal names are incremented by one (1).

NOTE: The terminal name must always be the same as the net name. You can use multiple I/O terminals in the same node (e.g. input and output) but they must have the same name.

Naming Nets

The net names are used to name the wires (nodes) on the schematic. These names can also be used to connect multiple schematic sheets. All nets using the same name are connected together. Initially, wires are assigned hidden numbers. These numbers are used internally by ACTIVE-CAD to track wiring nets. However, for clarification and documentation, you can assign a name to any wire on the schematic. This name can, for example, describe the function of the signal in the design (e.g. Clock, Enable, Reset, etc.).

Adding A Net Name

To add a net name to an existing wire use the following procedure:


1. Click on the **Add Net or Bus Name** () icon.
2. When the **Net Name** window in Figure 2-18 appears, enter the desired name and click on the **OK** button.
3. A red rectangle appears attached to the cursor. Move it to the location where the net name should be placed and click the mouse button. The net name should be placed close to the wire itself. Otherwise, it will be displayed in a different (green) color than the net name (blue) and will not be attached to the wire. Changing the wire color in the **Preferences** menu, will change the above listed colors.



Figure 2-18. Entering Net Names.

NOTE: Each node can have only one name. If you attempt to join two nets that have different names, the schematic will display a warning. You have to change or delete one of the net names before you can join the two nets.

If you try to assign a new net name to a wire that has a name already assigned, a warning will be displayed and the operation will be aborted. You can assign a new net name only after deleting the original net name.

Editing a Net Name

To change a net name, double click on the net name and when the **Net Name** window appears, edit the name. If you double click on the wire instead of the net name itself, the **Net Attributes** window will appear. To change the net name from that window, click on the **Net Name** button which will display the **Net Name** window (Figure 2-18).

Deleting Net Name

To delete a net name, click on the name and press the **Delete** button.

Moving a Net Name

To move a net name to another location, switch to the **Select** mode, click on the net name and drag it to the desired location next to the wire.

Repeat Option

Because the **Repeat** option automatically updates the net names index, you can quickly draw nets and terminals connected to symbol pins. This is particularly useful when drawing buses with a large number of similar nets. The **Repeat** option helps you to place multiple net names, I/O terminals, and PWR symbols.

The net name and I/O terminal name consist of a name followed by an index (e.g. DATA1 or ADDR[14]). If you do not enter an index as part of the net name, ACTIVE-CAD will append to the name an index whose starting value defaults to one. Each time you place the name on the schematic, the index value is incremented by one.

The **Repeat** option for PWR symbols does not add any index to the PWR symbol. Instead, each click of the mouse button places the same power symbol in all schematic locations.

NOTE: To start numbering of nets and terminals from numeral '0', you must enter the full name, including the suffix 0, e.g. DATA0, ADDRESS0.

To stop the **Repeat** mode, click on the **Cancel** or **END** buttons, or click the right mouse button.


Drawing Buses

The **Draw buses** icon activates editing of buses. Buses can start and end at any screen location. They may also terminate at bus terminals and bus pins. A detailed bus description of the bus editor is provided in **Buses**, the 3rd chapter of this Guide.


Drawing Taps

The **Draw bus taps** icon activates a mode for automatic wiring of pins to buses (bus taps). Clicking on the selected bus starts automatic numbering of bus taps. Clicking on device pins automatically wires them to the selected bus and names in sequence. For more details refer to the 3rd chapter of this *Guide*.

END Icon Operations

The **END** () icon puts dummy terminals, also called wire-end terminals, on unfinished wires. To place a wire-end terminal, disconnect the wire from the mouse cursor by clicking the right mouse button. Once the cursor is free, move it over the **END** icon and click. The wire-end terminal instantly appears at the last mouse cursor location. You can connect the dumb terminals at a later date by clicking and dragging them to pins or terminals.

Assigning PWR Points

To assign a power signal to a selected pin or wire on the schematic, activate the **PWR** () icon. In response, ACTIVE-CAD displays a **PWR** window for selecting the power signal and its graphical representation. Make your choice and click on the **OK** button. A little square will follow the cursor. Place it at the desired pin or wire and click the mouse button. It will instantly produce a power connection. If you click on the **Repeat** button instead of the **OK** button, then each click at a pin or wire will connect them to a power symbol.

You can reverse the above process by drawing a wire first. Clicking the right mouse button will disconnect the wire from the cursor. Activating the **PWR** icon will display the **PWR** window which will allow the desired power signal selection. Clicking on the **OK** button will place the selected power symbol at the end of the wire.

Making Automatic Connections

You can automatically connect device pins without manually drawing any wires. To activate the automatic wiring, select the **View | Preferences | Wires&Buses** option. When the **Wire & Bus Settings** window appears (Figure 2-19), select the **Autorouting** box. Each time you click at the end point, ACTIVE-CAD will automatically complete the wiring.

In the same menu, you can also limit the time that it takes for the automatic routing algorithm to search for the best wiring connectivity.

Note: The autorouting is set active by default (on power on), unless you deselect it and save on exit from the schematic.

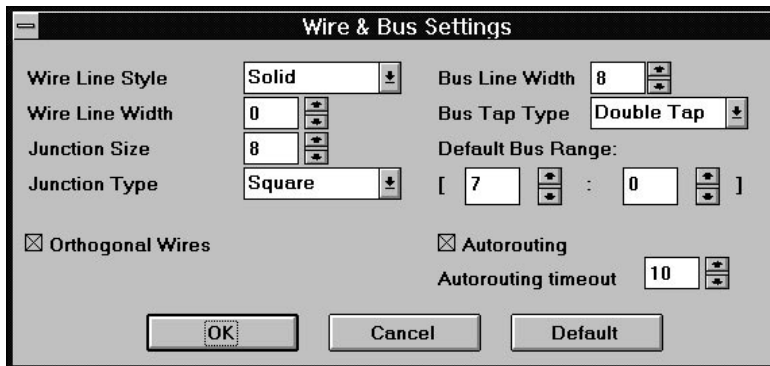


Figure 2-19. Wire And Bus Wiring Options.

The Wire & Bus Settings window applies to the entire project and offers several wiring options:

- ☐ **Wire Line Style;** allows to choose between solid, thick, dotted, etc.
- ☐ **Wire Line Width;** allows to select the width of the wires. The new width applies to all wires in the entire project (all schematic sheets).
- ☐ **Junction Size;** allows to select the junction size for the entire sheet.
- ☐ **Junction Type;** allows to choose the junction's shape (Circle or Square)
- ☐ **Orthogonal Wires;** allows only horizontal and vertical lines/wires. Disabling this feature allows drawing lines at any angle. This option should generally be set on.
- ☐ **Bus Line Width;** is used for setting bus thickness
- ☐ **Bus Tap Type;** offers several shapes of taps
- ☐ **Default Bus Range;** specifies if buses are drawn from the lowest order lines to the highest order lines, or vice versa.
- ☐ **Autorouting;** enables automatic routing of wires between symbols.
- ☐ **Autorouting timeout;** specifies the maximum time for wiring calculations (tryouts).

Disabling the auto-connecting feature

To disable the auto routing (wiring) feature select the **View | Preferences | Wires & Buses** option. When the Wire & Bus Settings window appears deselect the Autorouting option.

NOTE: If the autorouting feature is disabled, then the wiring needs to be manually anchored at each corner.

Manual wire connections

You can draw wires manually even in the auto routing mode by clicking the mouse button at each wire corner. This anchors the wire at the selected point. You can manually draw some wire segments and then complete the connection in the fully automatic mode by clicking at the end point, such as pin or net.

Automatic Table-Driven Connections

ACTIVE-CAD allows automatic connections between symbol pins selected from a table. The table for autorouting is activated by clicking on the **Autorouting Table** option in the **Edit** menu. In response, ACTIVE-CAD displays the **Connections** window that allows for selecting the start and end points for wiring.

To select a starting point, click on the symbol in the **Connect From | Symbols** field. The pins for the selected device will appear in the **For** field. Select from that field the pin which will start the wire connectivity.

Next, choose the end pin for the wire by selecting the symbol and its pin from the **Connect To** field. Clicking on the **Connect** button will start the autorouting process. Pins that are already connected are shown in gray and you cannot automatically connect wires to these pins.

The autorouting is controlled by the setups in the **Options** and **Auto Connections** fields. Detailed description of these items is provided below:

- ☐ **Ortho connection** allows drawing only vertical and horizontal lines. Disabling this feature allows drawing lines at any angle.
- ☐ **Check pin type** feature automatically verifies the compatibility of connected pins; e.g. connecting two totem pole outputs will produce an on-line warning.
- ☐ **Auto correction** helps to select any pin by pointing with the cursor in its neighborhood.
- ☐ **Cross chip** allows drawing wires through component symbols.
- ☐ **Auto connection** automatically draws wires between selected end points. The wiring editor will draw a wire that does not overlap with other wires and avoids going through components.
- ☐ **Optimization** gives the drawing algorithm more time for calculation and provides a higher percentage of automated connections.

- ☐ **Precise** option will use the smallest grid available for drawing connections in a highly populated area.

Auto connection option allows for fast and efficient wire connections. You can modify the automatically drawn wires by selecting the **Select** mode and manually dragging the wires.

Drawing graphics

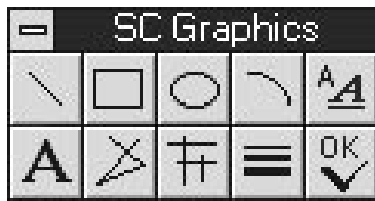




Figure 2-20. Graphics Toolbox Options.

The graphics editor is selected with the **Graphics toolbox** icon or the **F6** keyboard key. The elements drawn with this editor have no electrical parameters, and therefore they are not checked for electrical rules.


The Graphics Toolbox (Figure 2-20) has the following options:


Line  - draws a single line.


Polyline  - draws multiple-segment lines.


Ortho Line  - draws vertical and horizontal lines exclusively.


Rectangle  - draws a rectangle by selecting diagonal corners.


Circle  - draws a circle by selecting the center and then stretching its outline.

Arc  - draws a part of a circle by selecting the two endpoints of the arc, and then stretching its outline.

Text  - allows creating and placing text strings on schematics.

Text Style  - used for setting the text size, appearance (font) and color (Figure 2-21).

Line Style  - allows setting line shape, color and thickness (Figure 2-22).

Each graphics editing operation is started and ended by clicking the left mouse button. You can discontinue the current operational mode by clicking the right mouse button, followed by selecting the **End**  icon in the **SC Graphics** toolbox (Figure 2-20).

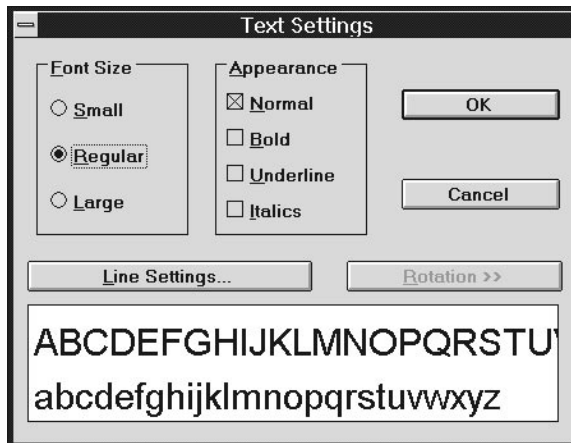


Figure 2-21. Text Setting.

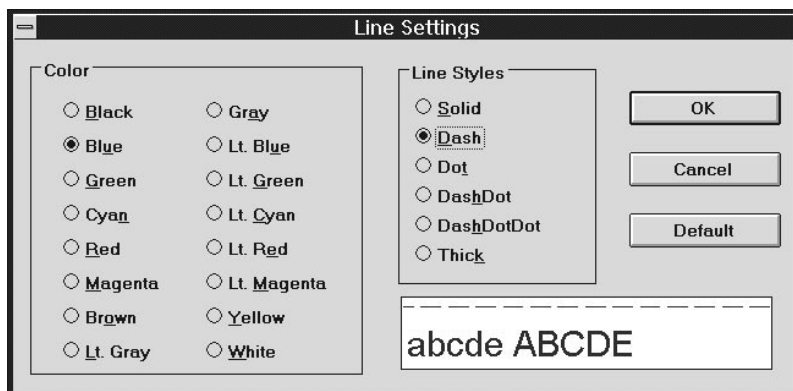


Figure 2-22. Line Setting.

Note: The text and graphics drawn in graphics mode represent supplemental schematic descriptions. They do not represent any electrical data and cannot be searched for or displayed selectively. For this reason, the graphical entries should be used seldom.

Examples of some graphical elements are shown in Figure 2-23

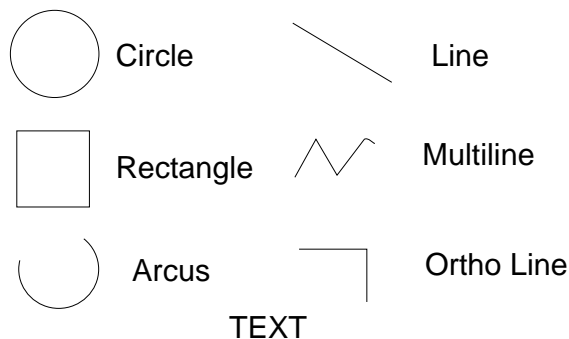




Figure 2-23. Examples Of Graphics Elements.

Editing A Schematic

The **Select and Drag** mode, activated by the **Select**  icon, allows changes to the existing schematic designs. Operations like deleting, moving, rotating components, dragging wires or buses can all be performed in this mode. To enter the schematic editor mode, press the **F4** key or click on the **Select and Drag**  icon.

Selecting Objects

An object is selected for editing by placing the cursor over its body and clicking with the left mouse button. The selected items are highlighted with an outline box. Selected wires and buses are highlighted by a change in color. Clicking outside the selected item deselects it. To select multiple items, press the **Shift** key and then continue clicking on additional components.

You can select the following items for modification:

- ☐ Component
- ☐ Wire

- ☐ Bus
- ☐ I/O Terminal
- ☐ Graphics and Text

Dragging Symbols And Wires

To drag or move a component or a group of components, position the cursor over one of the selected items and press the left mouse button. Next, move the cursor, keeping the left mouse button pressed down. The group of selected components will follow the cursor to the desired location. Releasing the mouse button will place the selected items at the new location. The auto-panning software allows moving items outside the visible schematic area, without using the scroll bars.

Both the components and their wires move with the cursor. The **Drag** mode causes the dragged symbols to bounce back if the **Allow symbol overlap** option is not selected (in **View | Preferences | General Settings**) and the symbols overlap at the new location with other symbols.

NOTE: You can force overlap of symbols and wires if you drag them with the **Alt** key depressed.

NOTE: You can disable automatic rewiring of the moved components by performing the move operation with the **Alt** key depressed.

To edit a wire, click on the section of the wire that you want to move, and drag it with the cursor, keeping the left mouse button pressed down. Releasing the mouse button will place the wire at its new location.

Overlapping symbol outlines

ACTIVE-CAD continuously checks if the symbols hidden (rectangular) outline does not interfere with another symbol's outline. If it does, ACTIVE-CAD will prevent you from placing the component at the new location. For this reason, you may observe that after dragging a component to the new location, it jumps back to its previous location. To allow for symbol outlines to overlap, drag the symbols with the **Alt** key depressed or select the **Allow symbol overlap** option (in **View | Preferences | General Settings**) .

Moving and shaping buses

To shape a bus line, click on the item and holding the mouse button depressed, move the mouse device. Notice that the bus may move in segments, and moving the entire bus may require several drag operations.

This extra flexibility allows bending buses into desired shapes, and to position them more precisely on schematic sheets.

NOTE: The bus shaping mode does not move the bus taps.


Dragging buses with bus taps

To drag a bus with its wire taps, you need to select the wire tap dragging mode by clicking next to the bus and dragging a small dotted square around any bus segment and one or more taps. Releasing the mouse button will turn the bus red and will select the wire tap dragging mode.


Clicking on the bus will move its entire structure, including the bus taps. To effectively move buses with taps, you need to shape the bus before attaching taps, because after the taps have been drawn, the bus structure will remain rigid.

Rotating Components

There are three (3) modes of component rotation:

1. *New component rotation* - You can rotate the component attached to the symbol placement cursor by clicking on the **Ctrl+L** keyboard keys. Each click will rotate the component 90 degrees in the counter clock-wise (CCW) direction. Clicking on the **CTRL+R** keyboard keys will rotate the component in the clock-wise (CW) direction. The **Ctrl+M** keyboard keys will create a mirror image of the symbol.
2. *Direct schematic component rotation* - To rotate a schematic component, enter the Select mode and click on the desired component. Once the red rectangle appears around the symbol, click on the **Ctrl+L** keyboard keys to rotate the component in the counter clock wise (CCW) direction. Each keyboard key activation will rotate the component by 90 degrees in the CCW direction. To rotate the component in the clock wise (CW) direction, click on the **Ctrl+R** keyboard keys. To generate a mirror symbol image, activate the **Ctrl+M** keyboard keys.
3. *Rotation by symbol editing* - Any symbol can be rotated through the editing process. First, select the symbol in the Select mode and then click on the **Properties**  icon. When the **Edit Symbol** window appears, click on the **Attributes** button which will display the **Symbol Attributes** window. Select the desired symbol rotation from that window.

Deleting objects without moving wires

You can delete an object or a group of objects on the schematic without deleting wires that have been connected to these objects. First, select the objects and then click on the **Disconnect symbol**  icon, as described in the *Replacing Symbols* section, below.

Copy, Cut and Paste

The Select mode allows copying objects from the schematic to the clipboard and then inserting the copied schematic section on the same or another schematic using the **Paste** operation.

To copy a section of the schematic, select the object you want to copy and then select the **Copy** option in the **Edit** menu. The selected objects will remain unchanged on the schematic sheet, and their copy will be placed in the schematic clipboard.

NOTE: All wires that are connecting selected symbols to other symbols, which have not been selected for the **Copy** operation, will not be copied to the clipboard.

Cut and Paste

The **Cut** option (**Edit** menu) copies the selected objects to the clipboard and deletes them from the schematic. It also deletes the wires between the deleted section and the rest of the schematic design. To avoid deleting of these wires, use the disconnect function described in the *Replacing Symbols* section.

To paste the contents of the clipboard on the schematic sheet, select the **Paste** option in the **Edit** menu. A rectangle representing the schematic section will be displayed. Move it over to the desired schematic sheet location and click the mouse button to place the clipboard contents on the schematic. Pressing the right mouse button cancels the operation.

Copying a schematic to other Windows programs

The **Copy** and **Cut** operations use the schematic clipboard that cannot be accessed by other Windows programs. You cannot directly paste the ACTIVE-CAD's clipboard contents in other programs. To copy a schematic section to another Windows program follow this procedure:

1. Arrange the schematic window so that you see the entire section of the schematic you want to paste into another program.
2. Press the **Alt+PrintScreen** keyboard keys. This copies the selected window to the Windows clipboard as a bitmap.
3. Open the Windows Paintbrush program and select the **Paste** option from its **Edit** menu. This will paste the schematic window bitmap to the Paintbrush window.
4. In the Paintbrush window, select the rectangle area that you want to copy to your program and select the **Copy** option from the Paintbrush **Edit** menu.
5. Go to the program in which you want to place the schematic bitmap (e.g. Microsoft Word) and paste the current Windows clipboard that contains the desired area of the schematic window.

Replacing symbols

To replace a symbol on the schematic, you have to first delete the existing one and place the new one. To do so, you need to be able to delete a symbol without removing the wires.

Removing a symbol without deleting wires


Select on the symbol to be removed and click on the **Disconnect symbol**



icon. This operation disconnects the wires from the symbol, so that they become hanging wires ended with crossed circles or squares, which are called wire-ends. After the wires have been disconnected from the symbol, the symbol can be deleted from the schematic.

Connecting a symbol to hanging wires

To connect a symbol to hanging wires, place it so that its pins are placed directly over the wire-ends (red crossed circles or squares), which mark the end of the hanging wires. Next, with the new symbol in place and se-

lected, click on the **Connect symbol**  icon. All wires that overlap the symbol's pins will be connected and the wire-ends will disappear.

All wires that did not overlap the symbol pins will remain disconnected. These wires can be connected to symbol pins by dragging their wire-ends and dropping them at unconnected pins. A message confirming the wire connection is displayed before a connection is completed by releasing the mouse button.

Editing Symbol Properties

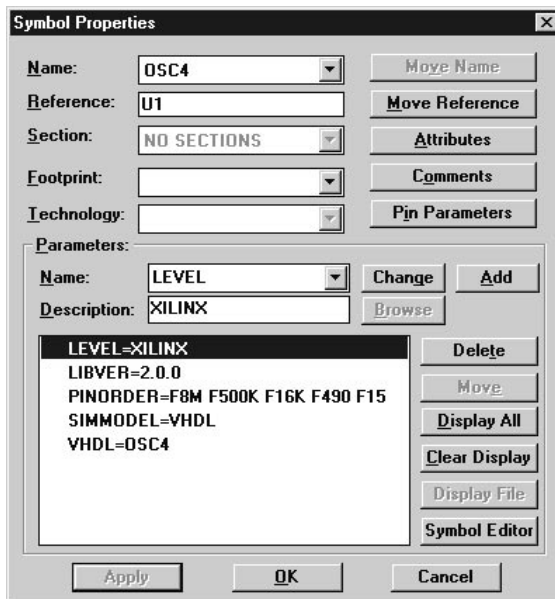


Figure 2-24. Symbol Properties Window.

You can change symbol attributes (reference name, part name, used section, etc.), by first selecting an object and then clicking on the **Properties**

icon. In response, the **Symbol Properties** window will be displayed (Figure 2-24). The same Symbol Properties window can be activated by double clicking on the desired symbol.

PCB Footprint

To add the device packaging information to a schematic symbol switch to the **Select** mode. Next, double-click on the selected component. When the **Symbol Properties** window (Figure 2-24) is displayed, enter any footprint name information into the **Footprint** field. This information will be stored in the design database, displayed in the inquiry mode and added to the netlist. When you assign a footprint name to a symbol for the first time, ACTIVE-CAD saves the information in the library. Every time this symbol is placed on schematic, its footprint name will be used by default.

Symbol attributes

You can add special attributes to each schematic component, such as power dissipation, voltage, current load, etc. To add these attributes to a schematic device, switch to the **Select** mode and double-click on the selected component. when the **Symbol Properties** window (Figure 2-24) is displayed, enter the attribute information into the **Parameters** field. This information will be stored in the design database and will be available for the **Query** window, **Bill of Material** report and any exported netlist.

Pin Parameters

You can define pin parameters for cell, macro and device pins. These parameters can contain information for the PCB layout program, FPGA routing system or simply for documentation. To define pin parameters, click on the **Pin Parameters** button in the **Edit Symbol** window (Figure 2-24). The **Pin Parameters** window (Figure 2-25) lists all device pins. Click on the desired pin and enter the name of the desired parameter in the **Name** field (e.g. Width) , and its value in the **Description** field (e.g. 0.25"), and click the **Add** button. The new parameter will be added to the parameter list. Note that each pin has its own list of parameters. The pin parameters are included in the **Bill of Material** report and in the exported netlist files. They can also be viewed in the **Query** window.

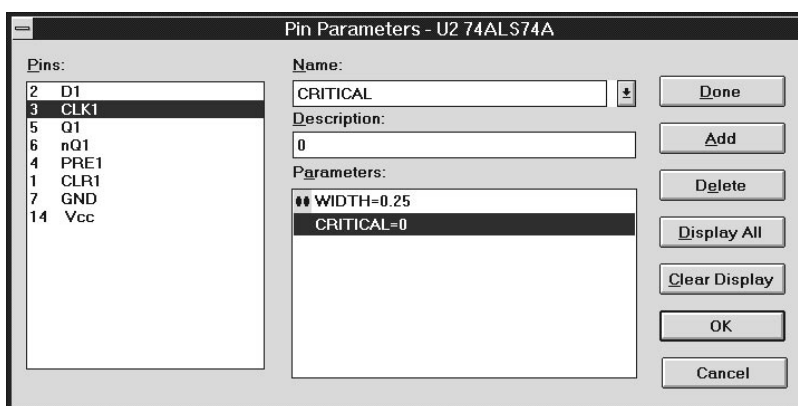




Figure 2-25. Pin Parameters Window.

Query

The **Query** mode is selected by pressing the **Query window**  icon. This mode allows you to inquire about any component or connection on the schematic. To switch to the Query mode, press the **F7** key or click on the **Query window**  icon. A window similar to Figure 2-26 will display all pertinent information about the selected component or connection.

To select a component or connection for the **SC Query/Find** window, position the cursor over its body and click the left mouse button. To select more than one item for the **SC Query/Find** window, press the **Shift** key while selecting additional components or wires.

You can inquire about connections by clicking directly on the schematic wire. To inquire about the components, you need to click on the schematic symbols. If the listing in Figure 2-25 displays a list of pins in the selected connection, you can instantly find the symbol associated with a selected pin by double clicking on the listed pin. If the **SC Query** window displays component information, clicking on any of its pins displays all wire connections for this pin.

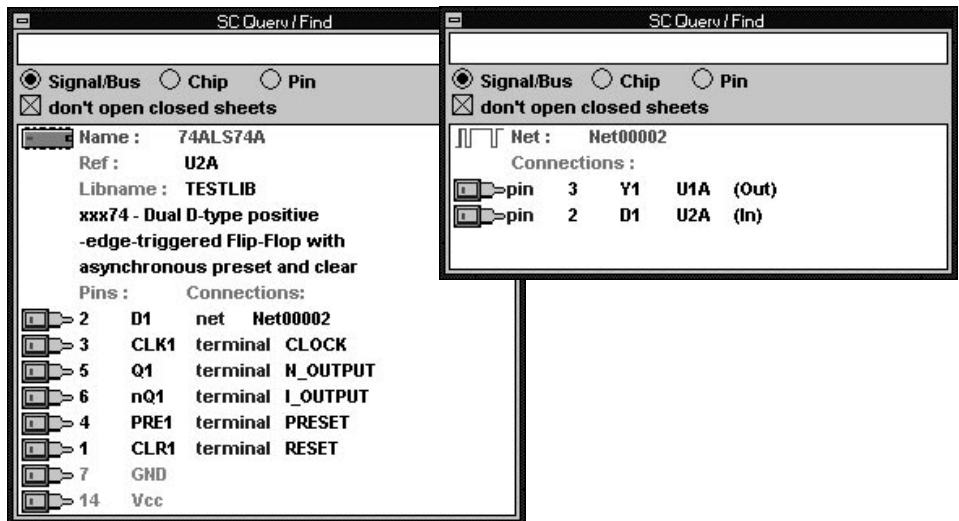


Figure 2-26. Query Windows For Symbols & Wires.

Back Annotation

All back- and forward-annotation in ACTIVE-CAD is based on the ECO (Engineering Change Order) standard. By using the ECO commands, you can automatically update the design database used for a project with the changes that you have made in any of the applications.

For example, if you have a schematic that has been used to lay out a PCB board, and the board designer makes some changes in the layout, you will need to update your schematic with these changes.

Manual schematic updates are labor-intensive and error-prone. Instead, you can modify the schematic by a PCB program that generates a list of ECO commands. This process is called back annotation. The ECO commands can be passed in a text file format or by using IPC (Inter Program Communication that uses Windows Dynamic Data Exchange) protocol. The ECO commands can be used to perform operations such as swap-

ping gates and renaming components. They also allow you some more advanced operations such as changing connections, adding and removing components, etc.

To import a back annotation file, click on the **PCB | Back Annotation** option in the **Options** menu. When the **Back Annotation** window appears, select the ECO file from the **File Type** field and select the file name from the **File Name** field and click on the **OK** button. ACTIVE-CAD reads the ECO file and updates the schematic. A detailed report about the performed changes is displayed in the **Messages** window, at the bottom of the computer screen. At the end of processing ACTIVE-CAD will ask if a report is needed. The report will be saved as a file with the same name as the back annotation file but with the *.BER extension.

The level of ECO commands depends on the specific application used in the design process. Forward-annotation takes place when a schematic is modified after creating a PCB layout, and these changes are used to update the PCB layout.

ACTIVE-CAD implements all ECO commands for both back- and forward-annotation. ACTIVE-CAD has a built-in IPC interface, so that other applications using the IPC protocol can exchange the design information with ACTIVE-CAD. These protocols can also perform other useful operations like cross-probing.

Working with the Simulator

Creating a netlist

A binary netlist is used to link the ACTIVE-CAD Schematic Editor with its background Simulator when the simulator is activated for the first time. The binary netlist, also called a connectivity database, can be generated manually using the **Create Netlist** option in the **Options** menu. The netlist is automatically stored with the project files by Project Manager. However, when you are not using the simulator, the netlist is not automatically updated until you select the **Create netlist** option again or start the simulator.

Each time you start the ACTIVE-CAD Simulator, the Schematic Editor automatically generates a new binary netlist to make sure that you are simulating the latest schematic changes. This new netlist is then loaded into the simulator. Once the simulator is started, every schematic change is instantly fed into the simulator and you do not have to generate another netlist to account for the schematic changes.

To create a netlist in the ASCII text format, select the **Export Netlist** option in the **Options** menu. This option is described in detail in the *Menu Reference* chapter.

Hidden pins in the netlist

The power pins are typically not displayed with the schematic symbols. They are defined as hidden pins and have the appropriate pin numbers assigned. You can inquire about the device power pins by selecting them in the **Query** mode.

Manual wiring of power pins

If you want to display the power pins on the schematic, you have to edit the symbol with the **Symbol Editor**. You can change the attributes of the power pins from PWR to PAS and place them in the symbol body. This allows you to make power pin connections like any other pins on the schematic.

Automatic wiring of power pins

If the power pins are hidden, ACTIVE-CAD can automatically connect all pins of the same type. When you use the **Create Netlist** option, ACTIVE-CAD prompts you in the **Create Netlist** window (Figure 2-27) about inclusion of the power signals in the netlist. ACTIVE-CAD checks all symbols for pins that are not displayed on the schematic, and allows you to connect pins with the same names. For example, you can connect all GND pins together. If the list has some pins that should not be connected, you need to manually exclude them from the netlist.

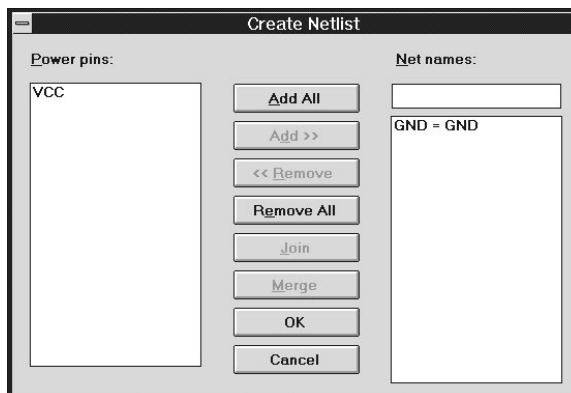



Figure 2-27. Power Pin Selection.

The same rules apply to other hidden pins. For example, Xilinx macro library symbols have hidden reset pins (GR or GSR) which can be connected together in the netlist to allow resetting of the design in the simulator.


Linking with the simulator

The schematic editor can operate on-line with the simulator. This happens when both applications are active at the same time. In the on-line mode, any changes to the schematic are automatically passed to the simulator. You can experiment in real-time with schematic changes. Each and every design change can be instantly simulated without any compilation or new netlist generation.

If you activate the schematic first, the simulator can be started either from the Program Manager or by clicking on the **Simulator**  icon. This will start the simulator and load the previously generated netlist. From this point on, all schematic changes will automatically be fed into the simulator so that the schematic and the simulator's design database always match.

If the simulator is started before the schematic editor, all test points (probes) selected within the simulator will be placed on the schematic once it is loaded. Also if you make any design changes in the simulator (e.g. **Change Technology**), the schematic will be automatically updated.

Similarly, you can select test points in schematic even if the simulator is inactive. The selected test points will be displayed in the simulator once it is opened. If the simulator is on-line and operating in background mode, you can simulate single steps from the schematic by pressing the **Step** or the **Next event** button in the **SC Probes** toolbox (Figure 2-29),

which is activated by pressing the **Simulation toolbox**  icon. The background simulator shows logical states at schematic test points after every clock cycle, and allows for on-line design corrections.

Simulating a Schematic

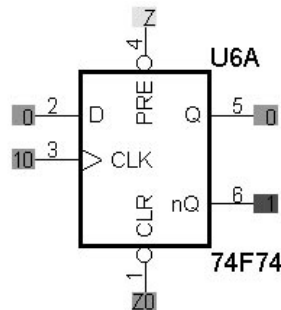







Figure 2-28. Pins With Probes.

Since the schematic editor has a background simulator, it behaves like a real hardware breadboard. You can freely select schematic test points and feed them with test vectors. The simulation results are displayed right at the selected IC pins (Figure 2-28) both as alpha numerical characters (0,1, X, and z) and with different colors (red=1, green=0, blue=X, yellow=z).

You can monitor logic states of selected test points by clicking on either the **Step**  or **Next event**  icon in the **SC Probes** toolbox which is activated by pressing the **Simulation toolbox**  icon.

The **Step** icon is similar in operation to the **Short** step button in the simulator. However, its simulation step time is set with the **General | View-Preferences** menu in the schematic editor. This setting can be different from the simulation steps set in the simulator.

The **SC Probes** toolbox (Figure 2-29) contains two icons, **Next event**  for simulation and **Previous event**  icon for returning to previous simulation events. A simulation event is a change of logical state on any of the monitored test points (probes). **Next event** will go to the next event and update the probes on the schematic. **Previous event** will go back to the previous event on the selected test points so you can analyze the past design behavior. Both icons use the background simulation results. If the **Next event** step goes beyond the previously simulated waveforms, the simulation will be activated and the new results will be displayed.

Selecting Simulation Test Points

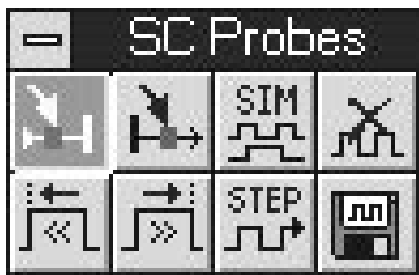


Figure 2-29. Simulation Toolbox.

To add a new test point to a schematic, select the **Simulation toolbox** icon or press the **F8** key. The **Simulation toolbox** in Figure 2-29 will be displayed. To place test probes on the schematic, activate the **Set probes**



icon, located in the top left corner of the **Simulation toolbox**. Next, place the cursor at the tip of the device pin and click the mouse button. A gray square appears at the cursor location. This square will be filled with color corresponding to the pin's logical state that is produced by the simulation process. To delete a selected test point, click on it with the mouse again.

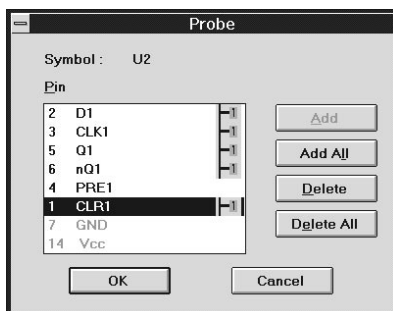


Figure 2-30. Probe Window.

To select multiple pins of the same device, click on the device body instead of its pin. If you select an IC, all its pins will be displayed in the **Probe** window (Figure 2-30). To select an IC pin for test point display, double click on it, or highlight it and then press the **Add** button. The selected pins are displayed on the schematic after you press the **OK** button.

You can also click directly on the desired test point on the schematic to place a probe.

Device Pin Event Tracking

Since ACTIVE is a closely integrated schematic editor and simulator, whatever you do to the design from the simulator affects the schematic display and vice versa. For example, the two schematic-based



Event icons in the **SC Probes** toolbox allow searching for events to the left and right of the current simulator (waveform) cursor location.

Clicking on a selected signal name (it will become blue) in the simulator window, and then clicking on any one of the **SC Probes** toolbox double-arrow icons, will move the simulator cursor to a new simulation event on that selected signal line and at the same time will update the schematic LVIs (Logic Value Indicators) display, which perform the function of LED lamps on hardware breadboards. Similarly, you can select multiple signal lines in simulator and perform concurrent event searches on these lines in the schematic editor.

Displaying logical states

The logical states are displayed on all probes in the form of one or two characters on a colored background square (Figure 2-28). The characters are not displayed when the font is too small, but the background color will always be visible, allowing you to easily determine the pin's logical state. The colors and symbols displayed on the probes are shown in Table 2-1.

Table 2-1. Probe symbols on the schematic

Symbol	Color	Description
1	red	high
0	green	low
X	dark blue	unknown or don't care
Z	yellow	high impedance or not connected
R	cyan	reference voltage (ECL)
V	cyan	high voltage (e.g. 12V, -5V)
?	green	unknown activity low
?	red	unknown activity high
	gray	state not defined

Component libraries

The component libraries of the ACTIVE-CAD system are stored internally by the **Library Manager** software program and the associated database. Schematic symbols and corresponding IC (simulation) models, packages, etc., are all linked together within the **Library Manager** database.

To review the directories and contents of these libraries, select the **Library Manager** program from the **Project Manager**. You can also invoke the **Library Manager** by clicking on the **Lib Manager** button in the **Libraries** window, which is activated by clicking on the **File | Project Libraries** option.

For details, please refer to the *Library Manager* chapter of this Guide.

Chapter 3

Buses

A bus is used to represent multiple wire connections by a single graphical line. Instead of drawing separate wires, you can create a single bus line that contains all of the wires. Traditionally buses represent the signal lines of the same type, e.g. all address lines or all data lines. However, the bus line can be used to connect any types of signals, not necessarily related to one another. The bus lines are designed to make schematics more readable, and their utilization depends on specific applications.

Because bus connections depend on how you name the bus members and not how they appear on the schematic, it is easier to make a mistake using the bus connections instead of single wire connections. Be sure to check all bus connections with the **Query** option.

Bus Definition

Bus name

A generic bus name should consist of the array name (e.g. DATA) and the definition of upper and lower signal indexes (e.g. [0:15]). A typical bus name is a combination of name and index:

NAME[X:Y]

where X, Y are integer numbers greater or equal 0.

The number of signals N in a bus is calculated from the this formula:

$N = \text{absolute of } (X - Y) + 1.$

Note that it is possible to have $X=Y$, i.e. a bus that has only one signal, e.g. DATA[3:3].

Each bus signal line *NAME[X:Y]* has multiple single wires with their own names defined as *NAMEA*, where *A* is the number within the range of X, Y (including both limits). For example, bus *DATA[0:3]* contains signals *DATA0*, *DATA1*, *DATA2*, *DATA3*.

☞ **The bus name** should not end with a digit. It can lead to unexpected connections. For example, buses *DATA1[0:7]* and *DATA[0:10]* will have a connection on signal *DATA10* that is a member of both buses.

Bus signal order

Buses *NAME[X:Y]* and *NAME[Y:X]* contain the same sets of signals, but they define a different order of signals in the bus. The **signal definition order** is important when a bus is connected with a bus pin, as it is explained later in this chapter. The significance of the bus members is implied by software for which Schematic Editor produces data. In XILINX designs buses should be defined with the most significant bit (the one with higher index) as the first and the least significant bit as the last. For example, four-member *DATA[3:0]* bus has the most significant *DATA3* bit defined as the first and the least significant bit *DATA0* defined as the last.

NOTE: Using the same signal definition order (i.e. either from MSB to LSB or from LSB to MSB) in the whole design is strongly recommended. Mixing the definition orders can be a source of errors.

Bus pin order

When connecting a bus to a bus pin, the connections between signals contained in these two objects will be established in the signal definition order, i.e. from the **bit defined first** to the **bit defined last**. If the number of signals in the bus signal line and the bus pin are different, the signals with indexes exceeding the signals count on the smaller bus will not be connected at all.

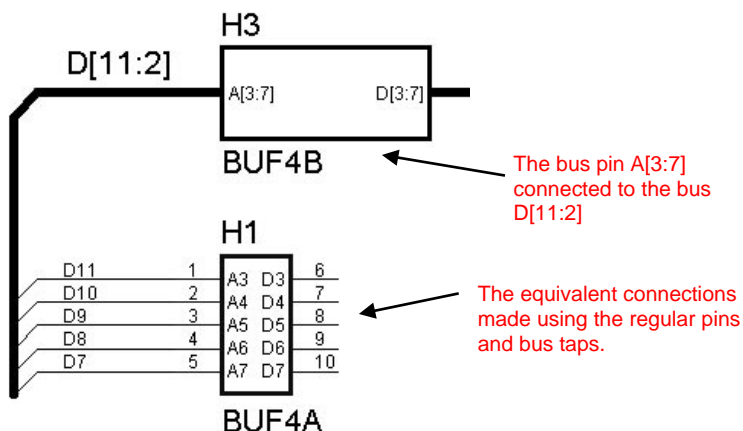


Figure 3-1. Example of bus connection with bus pin.

For example, connecting a symbol bus pin A[3:7] to the bus line D[11:2] (Figure 3-1), will create the following connections:

pin A3= signal D11, pin A4=signal D10, pin A5= signal D9, pin A6= signal D8, pin A7= signal D7. Signals D6, D5, D4, D3 and D2 will not be connected to the bus pin A[3:7].

NOTE: The names of buses and bus pins are irrelevant for making physical connections. For example, connecting the bus pin A[0:3] to bus A[3:15] will produce the following connections: pin A0=signal A3, pin A1 = signal A4, pin A2 = signal A5, pin A3 = signal A6.

Bus terminal

Bus terminals that represent bus pins in hierarchical designs, are subjected to the same rules of connectivity as buses and bus pins. The bus pins are used to connect a bus line directly to symbol pins. Bus terminals save space on the schematic and make it more clear .

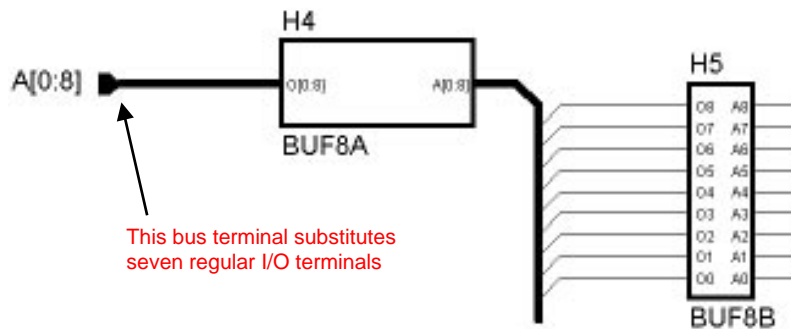


Figure 3-3. Bus terminal on the schematic.

NOTE: Unless you assign to a bus line a specific name, the bus terminal name will automatically be assigned to the connected bus signal line.

Unnamed Bus

Bus pins can be connected by unnamed bus lines. In this case, the bus signals are connected via unnamed bus members, and the connections order is governed by the bus pin definition order. For example, connecting the bus pin A[0:8] to bus pin D[3:0] via an unnamed bus line will produce the following connections: *pin A0 = pin D3, pin A1 = pin D2, pin A2 = pin D1, pin A3 = pin D0*. Pins A4 through A8 will not be connected.

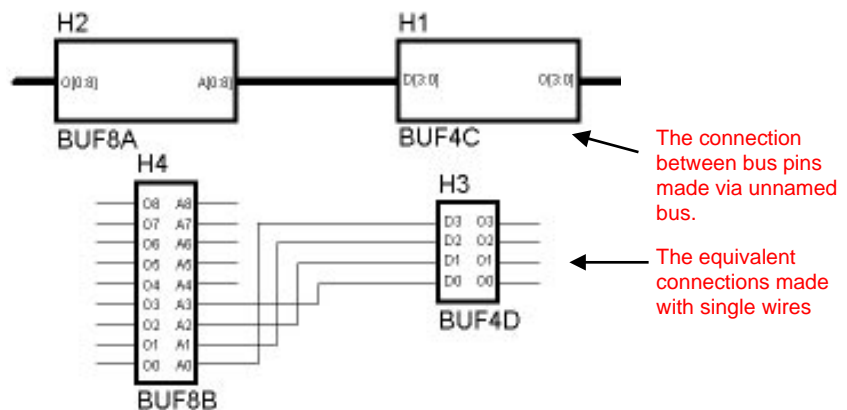


Figure 3-2. Connecting bus pins with an unnamed bus.

Generic buses

To connect a bus to a bus pin or bus terminal you must use a generic bus that consists of the bus name and indexes (NAME[X:Y]). You cannot directly connect a bus with disparate signal names (complex bus) to a bus pin or bus terminal. The *Editing Complex Buses* section explains how to connect such buses to pins and terminals.

Complex buses

A bus line may be comprised of several generic buses and individual signal lines. For example you can combine buses ADDR[0:15] and bus ADDR[16:31] into one bus. The complex bus is a set of generic buses which contains all signals from these buses. The complex bus can also contain non-indexed signals, e.g. CLOCK, RESET, etc. Such buses are often called the control signal buses.

NOTE: You cannot directly connect non-indexed bus signals to bus pins or bus terminals. Instead, single I/O terminals and single symbol pins must be used to connect these non-indexed signals.

When connecting a generic bus to a complex bus, the name of the generic bus has to be explicitly listed on the appropriate generic bus sections. For example, Figure 3-4 shows a complex bus comprised of generic buses CTRL[0:3] and STS[0:4]. The bus stubs connected to the input terminals (CTRL[0:3] and STS[0:4]) do not need to be named. However, the bus stubs connected to the symbol bus pins need to be named explicitly, as shown on input to symbols H6 and H7, below.

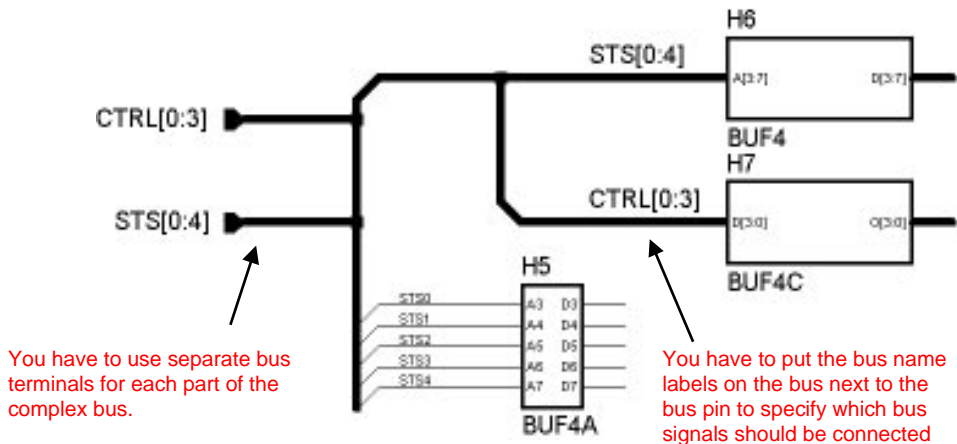


Figure 3-4. Example of a complex bus.

Bus Connection Rules

Busess can be connected to single pins and bus pins. The single pins are connected to buses via bus taps. The bus taps have to be named with concrete signal names, so that it is clear which bus member is connected to each single pin. If you draw a bus tap wire and do not assign any name, the schematic editor will not know which bus member has been connected..

Connecting wires (taps) to a bus

In order to connect a single wire to a bus, you need to draw a wire that starts or ends at the bus line. This connection is called the bus tap. The shape of the tap symbol is defined in the **Edit Bus** window (Figure 3-5), which is invoked by double clicking on a bus in the **Select** mode.

Connecting a bus tap to the bus does not guarantee any connection. For example, connecting a signal to a bus and naming it A1 will produce a valid connection only if there is another signal line with the A1 name.

The bus tap name is the same as the associated net name. The bus tap symbol is only a graphical confirmation that the net name is part of the bus. However, it does not have any other significance.

Bus pins

Symbols that have bus pins can be directly connected to bus lines. If the bus ADR[0:3] is connected to the bus pin D[2:4], ACTIVE-CAD will automatically generate the following connections:

ADR0=D2, ADR1=D3, ADR3=D4. The pin ADR4 will not be connected.

Connecting buses in a multi-sheet design

If you draw buses on schematics in a multi-sheet design, then the bus signals with the same name will automatically be connected.

☞ Do not use bus terminals to connect the same bus signal on different sheets of the multi-sheet schematic design. Bus names will automatically connect signals between schematic sheets.



NOTE: Terminals should be used in the multi-sheet design only if the signal I/O type (input, output, bi-directional) needs to be defined.



Connecting buses in a hierarchical design

All hierarchical macros (schematic sheets) interface with other macros through their I/O terminals. If a hierarchical macro has a bus, it can be connected to other sheets only if it is assigned an I/O terminal. Such an I/O terminal will be represented at a higher hierarchical level as a single symbol bus pin, which can be connected to other symbols at the higher hierarchical level.

Drawing a bus


To draw a bus, follow this procedure:

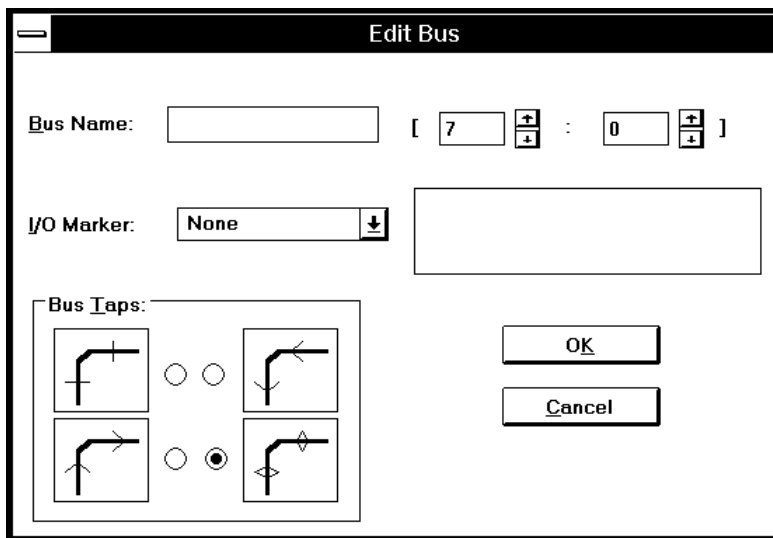
1. Click on the **Draw buses**  icon.
2. Click on the schematic where you want to start a new bus line.
3. Move the cursor and click the mouse button on every corner of the bus. A thick, red bus line confirms the last section of the bus line.
4. To assign a terminal to the bus, press the right mouse button to disconnect the bus from the cursor. Next, click on the **I/O Terminal**  icon. When the **Edit Bus** window appears, enter the bus name, select the lower and upper range and choose the I/O type.

5. To have an unnamed bus, press the right mouse button and then click on the **OK**  icon.
6. To entirely delete the newly drawn bus, click the right mouse button, and then click on the **Cancel**  icon or press the **Esc** keyboard key.

Adding a bus name

To assign a name to an unnamed bus line:

1. Activate the Draw Buses mode.
2. Click on the **Add Net or Bus Name**  icon.
3. When the **Add Terminal/Label** window appears (Figure 3-5), enter the bus name, low and high index ranges, and click the **OK** button.
4. Move the cursor with the attached rectangle to the desired bus location and click the mouse button.
5. The new name should be displayed on the selected bus.



Edit Bus

Bus Name: [7 : 0]

I/O Marker: None

Bus Taps:

OK

Cancel

Figure 3-5. Defining a bus name.

Moving a bus name

If the bus name has been assigned while the bus was drawn, it will be permanently attached at one of its ends, and you will not be able to move it to a new location. However, if the bus name has been added after the bus has been completed, it will be placed somewhere along the bus line.

Such a bus names can be moved by entering the **Select** mode, clicking on the bus name and dragging it to a new location on the same bus.

Using bus terminals

Bus terminals are generally used for two purposes:

- ☐ To define bus pins in the hierarchical macro schematic
- ☐ To define the bus signals direction, e.g. input, output, bi-directional

The bus terminal defines the direction, name and index of the bus pin of a hierarchical macro. A bus terminal on a single schematic sheet or hierarchical schematic, defines the direction of the signal. However, an I/O terminal on the FPGA top level schematics sheet defines in addition an external port of the FPGA device.

NOTE: You cannot assign parameters to the bus terminals. You can assign them only to the individual bus wires.

☞ If you need to assign parameters to a bus, use the single wire terminals, or assign the parameters to the individual bus taps.

Drawing bus terminals

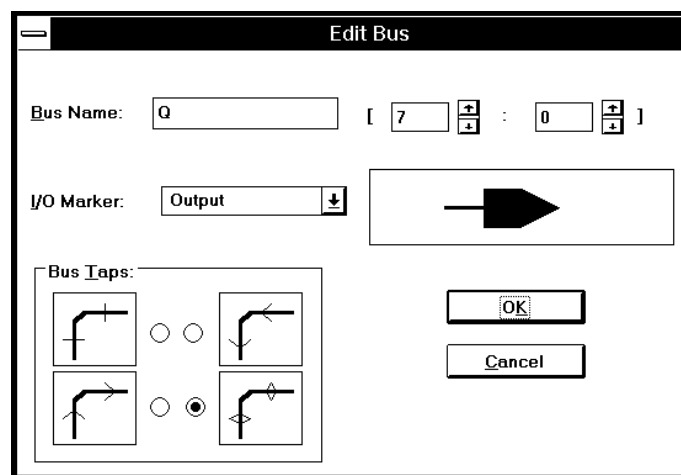




Figure 3-6. Defining a bus terminal.

To draw a bus terminal at the end of the bus line:

1. Drag the bus wire to the location, where you want to place the bus terminal and click the right mouse button to free the cursor.

3. Click on the **I/O Terminal**  icon, which will invoke the **Edit Bus** window.
4. In the **Edit Bus** window, enter the bus terminal name, indexes and direction. Click on the **OK** button to complete the operation.

Manual bus taps



Single wires can be connected to a bus with the bus taps. To draw a bus tap, select the **Draw bus taps**  icon. Start the bus tap wire by clicking at a pin, I/O terminal or a net. A black line will follow the cursor. Clicking at the bus will create a new bus tap.

The schematic editor draws a bus tap on the same side of the bus where the wire came from.

Automatic bus taps



If the symbol is placed in such a way that its pins can be connected with straight lines to the bus, you can use the automated bus tap option.

Drawing bus taps with names:

- ☐ Click on the **Draw bus tap**  icon and then on the bus signal line. The name of the bus tap is displayed in the status line at the bottom of the schematic screen, e.g. A0.
- ☐ Use the **Up** and **Down** arrow keyboard keys to select the desired bus signal index. The signal name in the status line changes according to the bus definition, e.g. for the bus A[3:7], the signal name can change anywhere from A3 to A7.
- ☐ Clicking on a symbol pin will automatically draw a wire from the pin to the bus, and will assign to it the selected bus tap name. The bus tap name listed in the status line is automatically advanced by one (1) or decremented by one, depending whether the **Up** or **Down** key was used for setting the current bus tap index.
- ☐ Clicking on the remaining symbol pins will connect them to the bus and assign to them sequential bus tap names.
- ☐ To end the bus tap drawing, press the **Esc** keyboard key or click on the **OK**  icon.

NOTE: The automatic bus tap feature works only if the bus is found in a straight line from the symbol pin, independent of the horizontal or vertical orientation. Also, the symbol pin cannot be connected to any wire.



Drawing bus taps without names:

- ☐ Select the **Draw bus tap**  icon.
- ☐ Click on the symbol pin.
- ☐ Wire is automatically drawn from the pin to the nearby bus.
- ☐ Keep clicking on other symbol pins that you want to connect to the bus.
- ☐ To end this procedure press the **Esc** keyboard key or the **OK**  icon.

NOTE: The bus taps without names will not be connected in the netlist. You have to name them before generating a netlist.

Adding bus tap names

If the wire connected to a bus is unnamed, you need to assign it a bus tap name.

1. Click on the **Draw bus tap**  icon.
2. Click on the bus line. The first bus member name will appear in the status line and a small rectangle will follow each cursor movement.
3. Use the arrow keyboard keys to select the desired bus member name (index) in the status line.
4. Click on the desired bus tap line. The selected name will be assigned to that wire. If the wire was already named, the new name will override the old one, and the tap name will be advanced by '1' in preparation for a new tap naming operation.
5. Press the **Esc** keyboard key or click on the **OK**  icon to end the tap naming operation.

Renaming bus taps

To rename bus taps, follow the procedure described in the *Adding bus taps* section, right above.

Changing bus tap type

You can select the type of bus tap symbol that will be used on your schematics. To select the bus tap type, go to the **Wires & Buses** option in the

View/Preferences menu and select the **Bus tap type** option. Choose one of the four allowed values:

Normal, **Left**, **Right** and **Double**.

You can also change the bus tap type by entering the **Select** mode and double clicking on a bus. When the Edit Bus window appears, select the appropriate bus tap. Select one of the four bus tap settings shown in Figure 3-7

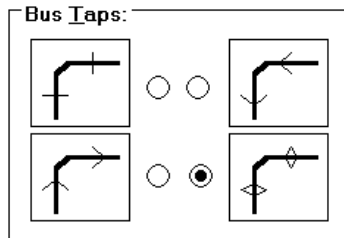


Figure 3-7. Bus tap types.

NOTE: All bus tap settings apply only to the bus taps drawn after the new setting has been made. To change existing bus taps you have to re-draw them with the new settings.

Editing a bus

To edit a bus, activate the **Select** mode. The Select mode supports the following bus editing operations:

Stretching

You can extend or shrink existing bus by clicking and dragging one of its ends.

Dragging a bus independent of taps

You can move and drag bus segments. Click on the bus segment you wish to move and drag it. The wires attached to the selected bus section are NOT moved along with the bus. You can manually move the bus taps to the new bus location.

Dragging a bus with its taps

To automatically drag the bus taps, select the *bus tap dragging* mode. This mode is selected by clicking in proximity of the bus and dragging a small dotted-line rectangle that includes a section of bus and at least one tap. Upon release of the mouse button, the bus will turn red. Dragging bus will pull all bus taps with the bus.

To exit the bus tap dragging mode, click outside the bus.

☞ In the *bus tap dragging* mode, the entire bus structure is moved as one piece. Use the *bus dragging* mode for routing of the bus segments, and select the *bus tap dragging* mode for automatic dragging of taps.

Deleting a bus


To delete a bus, click on the bus to be deleted and then press the **Del** keyboard key. All bus taps and bus terminals will also be deleted.

Renaming a bus

To rename a bus, double-click on the bus name. When the **Edit Bus** window (Figure 3-6) is displayed, edit the bus name and its lower and upper range indexes.

Querying a bus

Bus connections with bus pins are not automatically verified by the ACTIVE-CAD software, the way it is done with single wires. The single wires are physically connected to pins and their connections are easily verifiable. Buses on the other hand are connected through signal names and there may be some unconnected pins that look connected. You should always verify the bus connections in the **SC Query/List** window

(Figure 3-8), which is activated by clicking on the **Query window**  icon. This window lists all the bus members and bus pins connected together.

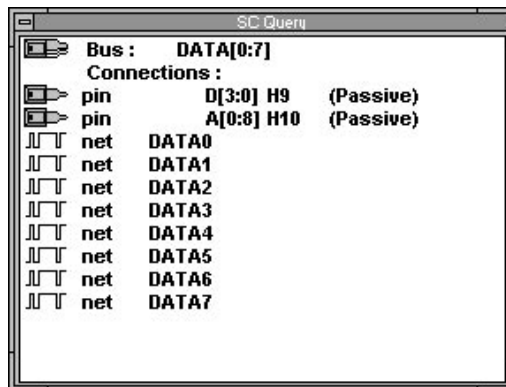




Figure 3-8. Example Of Bus Display In The Query Window

NOTE: Because bus connections highly depend its signal names, it is recommended to always verify the bus connections in the **Query** window.

Bus probes

The logical state of bus signals can be displayed directly on the schematic. Clicking on the **Simulation toolbox**  icon displays the **SC**

Probes toolbox. Selecting the **Probe**  icon from that toolbox and clicking on the bus pin or bus names will place a gray rectangle next to the selected pin or bus name, and will display the current logical state of the bus signals, if the simulator was active.

The bus probe displays the logical states provided by the on-line logic simulator. The simulator can be activated either from the simulator window, or from the **SC Probes** toolbox in the schematic.

Schematic design inputs are stimulated by signals called test vectors. If the bus probe is placed on an input bus terminal (Figure 3-9), the test vectors have to be defined in the simulator window before running the simulation. If the bus probe has been placed on an output bus terminal, then it will display the logical states generated by the design itself.



Figure 3-9. Bus Probes Display On The Schematic.

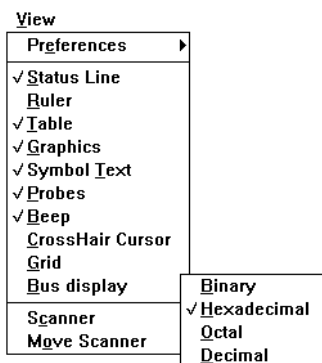


Figure 3-10. Bus Probes Display Type Selection.

The bus probes can display the values in binary, hexadecimal, decimal and octal format. To select the display format, choose the desired setting from the **View/Bus Display** menu (Figure 3-10).

The displayed values include only high and low logical states. If the bus signals have other logical states, such as Z, X, Resistive, etc., the * character will be displayed. For example, if the DATA[3:0] bus has signals D0=z, D1=1, D2=0 and D3=0, the hexadecimal format will display * (asterisk), but the binary format will display 001*. Note, that the order of the signals in the bus determines how the bit values are displayed. If the same bus was defined as DATA[0:3], the binary display would be *100.

Buses in the netlist

The ACTIVE-CAD binary netlist format provides a full support for bus constructs. This means that the bus name and its members are defined as such in the netlist created by the schematic editor. The netlist with buses is used by the associated simulator to simplify signal selection, i.e. you do not have to redefine buses in the simulator. When you export the netlist to other formats, the bus definition is not supported and the buses are converted into single signal lines.

Using buses with the on-line simulator

Schematic editor buses which are fed into the simulator via the schematic netlist are represented in the simulator by single bus lines. If a new bus is added to the schematic and transferred into simulator through the incremental design process, it will be listed in the simulator as a set of discrete wires. To have the newly created bus shown as a single bus line in the

simulator, you need to reload the simulator with a netlist by clicking on the **Options | Update Simulation** option.



Figure 3-11. Bus Display In The Simulator.

The buses can be viewed and selected from the **Signal Selection** field in the simulator's **Component Selection for Waveform Viewer** window, shown in Figure 3-11.

The bus signals drawn in the incremental mode will not be displayed in the simulator's Waveform Viewer as a single bus due to the lack of support for the buses in the IPC cross-probing protocol. These buses can be manually recreated in the simulator (see *Simulator User's Guide*).

If you are editing the schematic while the simulator is active you may be informed by the simulator that a design netlist must be regenerated to account for the latest bus changes. Upon your acknowledgment, the simulation will reload the new netlist and automatically restore the test vectors. If you do not update the simulation, the bus changes on the schematic will not be reflected in the current simulation database.

Editing Complex Buses

ACTIVE-CAD has powerful editing capabilities for creating designs with complex bus connections. Many FPGA cell or macro symbols utilize bus pins to enhance the drawing clarity, and they typically group from 8 to 16 I/O pins into one bus pin. These symbol bus pins can be connected to other bus pins by bus lines.

However, in some cases the bus pins can be inconvenient to use, particularly if you want to mix signals from different buses or reorder the signal lines within the buses. This section shows how to use the advanced bus editing features to achieve any desired connections.

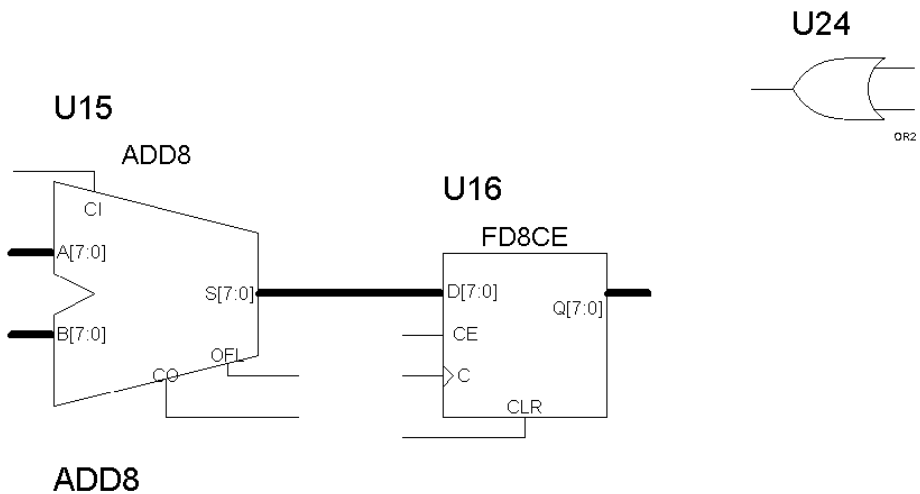


Figure 3-12. A Schematic For Complex Bus Editing.


Let us consider the schematic in Figure 3-12. The symbols U15, U16 and U24 need to be connected in the following way:

Table 3-1.

Source pin	Destination pin
U24.O	U15.B7
U16.O6	U15.B6
U16.O5	U15.B5
U16.O4	U15.B4
U16.O3	U15.B3
U16.O2	U15.B2
U16.O0	U15.B1
U16.O1	U15.B0

Notice that the output of the OR gate (U24.O) connects to the most significant bit of U15 (pin B7) and that the lowest two bits U15.B0 and U15.B1 are swapped and connect to U16.O1 and U16.O0, respectively.

To connect the U15 pins with discrete wires would be cumbersome and time consuming. The ACTIVE-CAD schematic editor allows you to make these connection through the bus editing process:

1. Draw a bus line from U16.O[7:0] to U15.B[7:0]. The pin connection order is displayed next to the bus pin to clearly list what is actually connected.
2. Add the bus name AD[7:0]
3. Click on the **Draw bus taps**  icon and connect the U24.O pin to the bus and name it M7; the bus will now have 9 members: AD0 through AD7 and M7
4. You should end up with the schematic similar to Figure 3-13

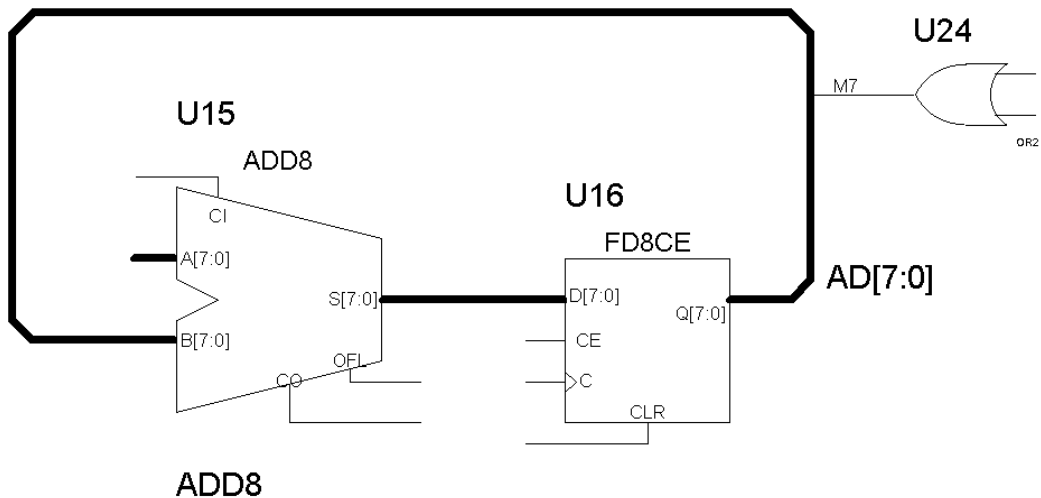


Figure 3-13. Simple Bus With Connects Bus Pins.

5. Activate the Select mode and double-click at the end of the bus line where it connects to the U15.B[7:0] pin. The window in Figure 3-14 lists all bus members that are connected to U15 through that bus pin.

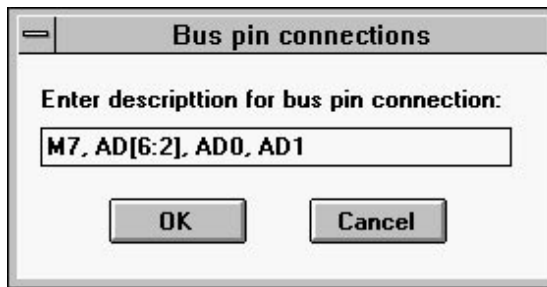


Figure 3-14. Editing Bus Pin Connections.

6. To rearrange the order of signals connected to the U15 bus pin edit the signals listed in Figure 3-14. Since the M7 signal should be connected to B7, enter it first into the window as shown in Figure 3-14. Next, put the comma (,) and list the six signals that are connected in the proper order (AD[6:2]). Because the last two signals of that bus are swapped, you must list them explicitly (AD0 and AD1).
7. Click on the **OK** button to exit Figure 3-14. In response ACTIVE-CAD displays Figure 3-17 which lists the new bus members directly at the **U15-B** input pin.

Note that if the B bus pin had a different signal order, for example B[0:7]) instead of B[7:0], then you would have to list the signals in the reverse order to provide the connections listed in Table 3-1. In such a case the proper definition for the bus pin would be: AD1, AD0, AD[2:6], M7.

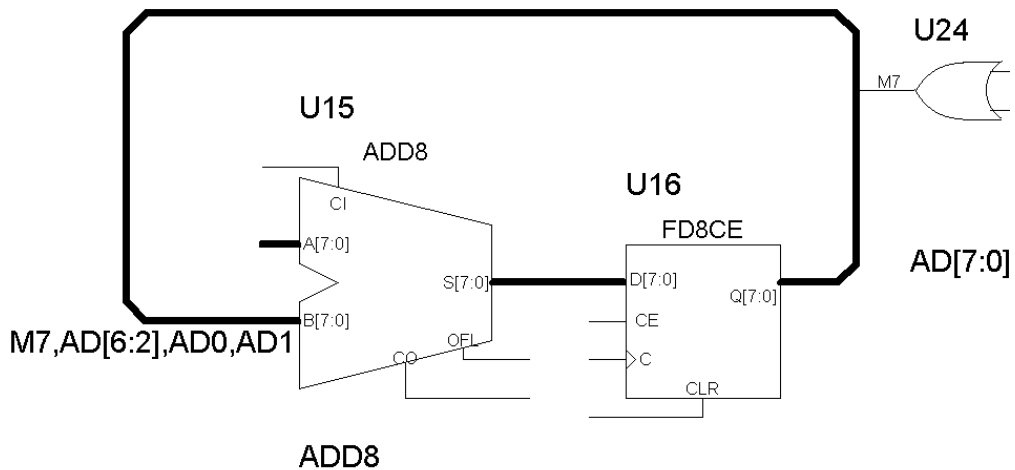


Figure 3-15. Completed Schematic With Complex Bus.

You should always double check the complex bus connections. The best way to check bus connections is to start the simulator and use its **View Connections** option to verify the critical bus connections.

6. Start the simulator and select the **Add Signals** option from the simulator **Signal** menu. The **Chip selection** window in Figure 3-16 will list all devices, cells and macros. Double click on U15. In response, ACTIVE-CAD displays explicitly all pins of U15 in the **Pins For** window.
7. Select the pin U15.B7 and press the right mouse button to invoke the local menu shown in Figure 3-16.

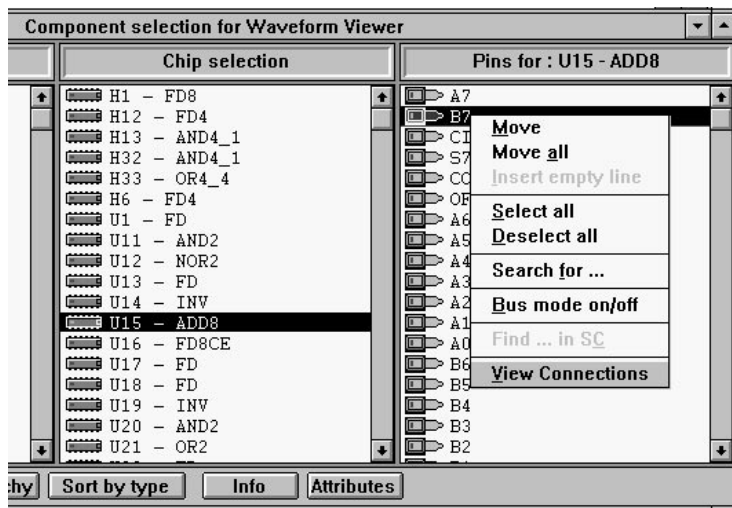


Figure 3-16. Selecting View Connections Option.

8. Select the **View Connections** option in Figure 3-16 and notice that the new window shows a direct connection between U15(ADD8).B7 and U24(OR2).O (Figure 3-17).

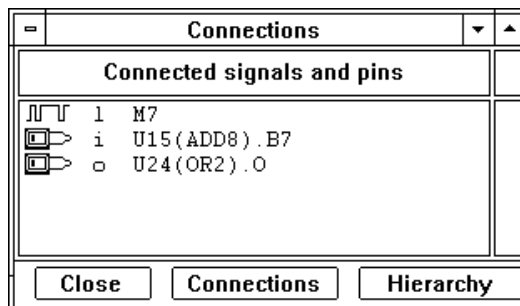


Figure 3-17. List Of Connections To The M7 Signal.

9. Similarly, you can verify that the two lower bits (U15.B1 and U15.B0) are properly connected. Figure 3-18 shows the connectivity for U15.B1.

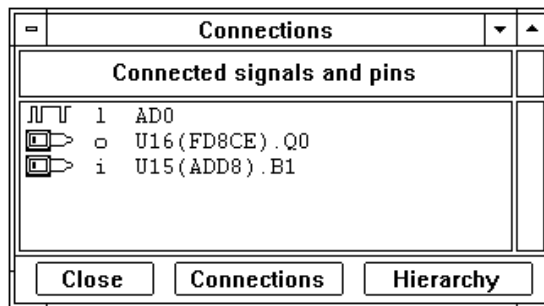


Figure 3-18. Connections to the U15.B1 pin

Chapter 4

Managing Schematic Designs

Design Structure

You can create in ACTIVE-CAD the following design structures:

- ☐ single sheet design
- ☐ multi-sheet design
- ☐ hierarchical design

Choosing a specific design structure depends on the particular design size (number of symbols and connections), purpose (board or chip design), and company standards. The following sections describe each of these design types and their utilization.

Single sheet schematic

Single page designs are typically used for small designs. The largest page size is 44"x34" (size E). The major advantage of a single sheet schematic is that you can use physical connections for the entire design. This makes tracking of the connections easier when a board is made.

The disadvantages of using large pages are:

- ☐ Slow redraws. A schematic with large number of symbols takes longer time to scroll than a schematic with fewer parts.
- ☐ Difficulties to print on laser printer. Large schematic sheets need to be printed on plotters. Laser printers can usually handle pages of size B (11"x17"). The four-page tile printing option allows printing a

single schematic page on four sheets of paper. This eliminates the need to use plotters for medium-size drawings.

Multi-page flat schematic

When a design becomes too large to print on a single page, you can use a multi-page design structure. When you create a new sheet, it automatically becomes a part of the current project. Whatever you draw on that page belongs to the same board or chip design as the first page. To make connections between schematic pages, you have to make logical connections by using the same net names, I/O terminals or bus taps on all schematic sheets. For example, entering the *CLOCK* I/O terminal on sheet 1 and *CLOCK* net name on sheet 2, will automatically connect both items. These connections can be confirmed by using the **Query** option.

The following are advantages of using the multi-sheet design structure:

- ☐ You can use small sheet sizes which are easier to print on laser printers and easier to handle in manufacturing, test and other departments.
- ☐ You can handle unlimited design sizes without condensing the schematics.

The following are the disadvantages of using multi-sheet schematics:

- ☐ Connectivity errors between sheets are detected only when the netlist is created. The **Query** window helps to trace all connections.
- ☐ All net names are global. This means that you can accidentally connect two nets by assigning the same name.
- ☐ Symbols with multiple sections, e.g. NAND gates of the 7400 TTL device, can fully be utilized only within a single page. This can later be corrected by manually selecting section (gate) reference numbers.

NOTE: All reference designators for symbols in the multi-sheet schematics have to be unique and ACTIVE-CAD automatically assigns these unique numbers. If you manually assign the same reference to two different devices, an error will be reported when you create a netlist.

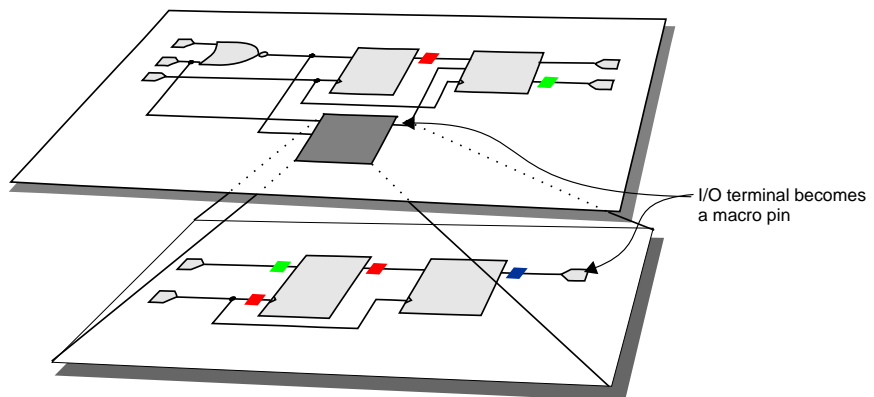
Hierarchical schematic

Because of the large number of symbols used in ASIC designs (typically thousands), it is very difficult to handle such designs using the simple multi-sheet design structures. These designs typically require thousands of simple primitives like gates and flip-flops to be used for PLD, FPGA or ASIC. To simplify the schematics, designers prefer to use high level components that have clear functionality, e.g. TTL-equivalent devices.

The solution for this problem is the concept of the hierarchical macro. Such a macro is a device in the library, which appears as a standard component, but is internally implemented as a netlist or schematic. For example, you can create an equivalent of the 74161 TTL counter by drawing a macro schematic with only gates and flip-flops. This macro can then be saved in the library and used several times in your designs. Several FPGA libraries already come with a number of hierarchical macros so that you don't have to create them yourself.

Hierarchical designs are very effective with IC designs (PLD, FPGA, ASIC, etc.). In hierarchical macro schematics all net names and reference names are local to these schematics. This means that you can use the same signal names several times in different macros and they will not be automatically connected.

The connections between hierarchical schematics and the rest of the design are made via I/O terminals. These terminals are converted into hierarchical symbol pins, and after the symbol is placed on the schematic sheet, wire connections can be made to these pins. This allows showing explicitly all connections in hierarchical schematic. Since only signals shown as symbol pins need to be connected, fewer wires need to be placed, which simplifies designs.



Hierarchical schematic is converted into a library symbol and can be placed on any schematic.

Figure 4-1. Hierarchical Design Structure.

The following are the advantages of using hierarchical designs:

- ❑ The symbols in libraries can represent large functional blocks implemented in detail on lower hierarchical levels. By viewing the high level schematic, you can see the general design structure without being overwhelmed by the implementation details.
- ❑ Top-down or bottom-up methodology available in hierarchical design tools allows team effort. Each design section can be given to different designer. All conflicts between design sections are eliminated because all interfaces are through explicitly defined symbol pins.
- ❑ You can use multiple instance of the same macro. If you were to use the same schematic sheet in a flat design, you would have to duplicate it for each instance. Any correction to the schematic macro, would have editing of all its instances. The hierarchical macro needs to be modified only once because all its instances will be automatically updated by the hierarchical design editor.
- ❑ Macros can be used in multiple projects. You can develop a set of reusable modules that can be stored as hierarchical macros and used in several designs.

The disadvantages of using hierarchical designs are the following:

- ❑ Net names can become very long. Many IC design tools do not accept hierarchical netlists. This requires producing a flat netlist out of a hierarchical design. The method used to create unique reference identifiers adds the hierarchy reference name to each symbol reference. For example, a symbol U58 in a macro called H8 will be called H8/U58. In the multilevel hierarchical designs these names can become very long, depending on the number of hierarchy levels.
- ❑ Design back-annotation is difficult to implement, particularly for PCB designs. Also, there are no PCB design programs that accept hierarchical netlists. For that reason, all the net names and reference names have to be converted into unique identifiers. The default method of doing this is to add the hierarchy prefix for each hierarchical name (e.g. H50/H2/U30). The **Annotate** option allows you to produce the unique references without the hierarchy designator, however, there is no provision for renaming net names. The back-annotation changes in the macros are also not supported.
- ❑ It is impossible to split IC sections of the same device between macros. The sections can be manually packed into one device in the PCB layout program, but they cannot be back annotated to the hierarchical schematic.

- ❑ Updating macros often requires changing their symbols. For example, if you want to add or remove some signals from a hierarchical macro, the symbol for that macro will be changed. This may require corrections of wiring on all schematic sheets which use that macro.

Multi-sheet Design Management

Most of the operations in multi-page designs are performed from the Hierarchy Browser, which is located within the Project Manager (Figure 4-2). Additional schematic related operations can be invoked by clicking on the icons in Figure 4-2.

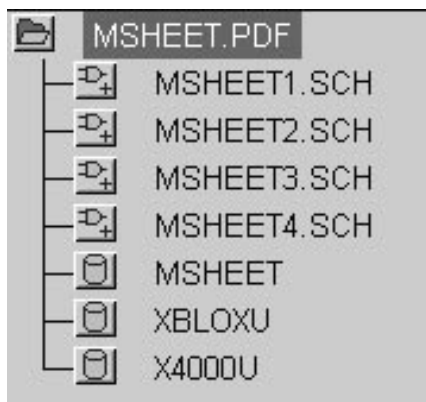



Figure 4-2. Multi-sheet Project Contents.

Adding new sheets

To create a new empty sheet select the **New Sheet** option in the **File**

menu, or click on the **New Schematic**  icon in the schematic editor. The new sheet receives the name of the project, with the sequential sheet number assigned to it automatically. This sheet can be saved under a different name using the **Save As** option from the **File** menu. Each new sheet is automatically added to the current project contents. To open a sheet that does not belong to the project use the **Scratchpad** option in the **File** menu. The scratchpad schematic can be used to draw some temporary schematics that are then copied to another schematic or deleted without changing the project schematics. To add a scratchpad to the project use the **File | Save As** option and enter the schematic name.

Adding existing sheets to the project

To add to the current project a schematic from another project, click on the **Add Sheets To Project** option in the **Hierarchy** menu.

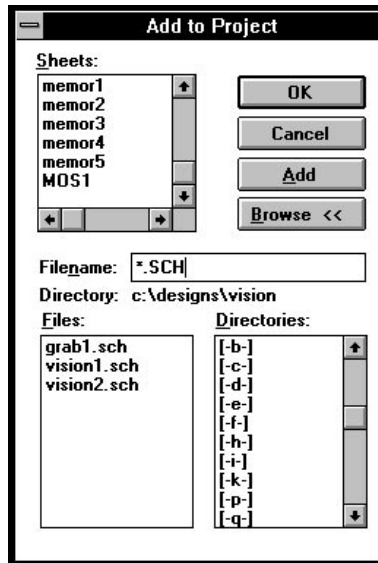



Figure 4-3. Adding sheets to project.

Select in the **Add to Project** window the schematic file(s) you want to add and click on the **Add** button. The schematic editor loads each added sheet and verifies if the symbols used in these schematics are available and if there are no duplicate reference numbers. To select files from other drives and directories, click on the **Browse** button. Additional window, displayed in the lower part of Figure 4-3, will allow selecting schematic sheets from other locations. To end adding sheets to the project, click on the **OK** button. The list of project sheets will be instantly updated and displayed in the **Hierarchy Browser**.

NOTE: The schematic editor automatically adds to the current project the libraries used by newly added schematic sheets.

Opening non-project sheets

When you click on the **Open a Schematic**  icon or on the **File | Open** option, only the sheets that belong to the current project are shown. If you want to open a sheet that does not belong to the current project, use the **Browse** button, which allows searching disks and directories for additional schematic files.

The newly opened schematics can be edited. However, they do not belong to the project till the **Add Current Sheet to Project** option in the **Hierarchy** menu is activated.

NOTE: As a schematic sheet is from another project is opened, it is copied to the current project directory, and any schematic changes will not affect the projects from which it was copied.

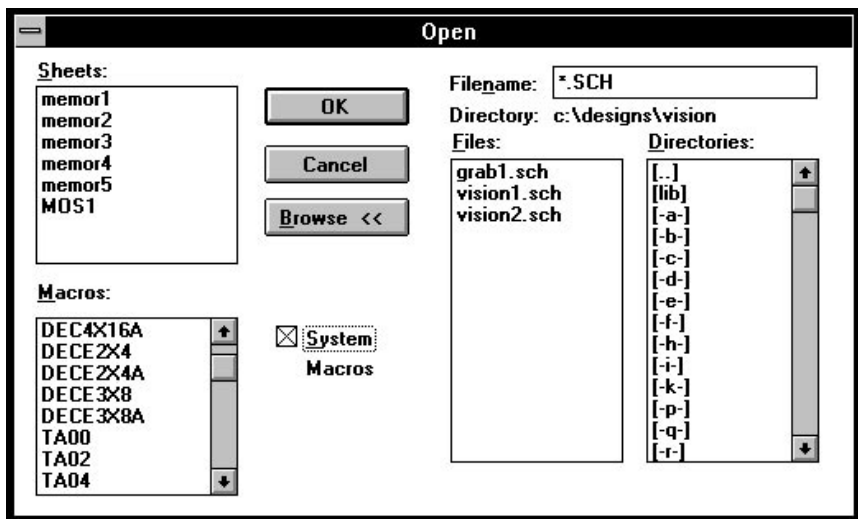


Figure 4-4. Open window.

The reference names of components on the added schematic sheets are verified for possible conflicts with reference names on other project sheets. If duplicate reference numbers are detected on the new sheet, they will be changed.

Removing sheets from the project


To remove a sheet from the current project, select the schematic sheet in the Hierarchy Browser (Project Manager), and click the **Del** keyboard key. Another way to delete a sheet from the project is to activate it and then select the **Delete Sheet from Project** option from the **Hierarchy** menu. The deleted schematic will no longer be a part of the project and will disappear from the Hierarchy Browser listing..

NOTE: The delete option deletes the sheet from the project contents, but it does not delete the schematic file from the disk. To delete these files permanently, use the Windows **File Manager**.

☞ The deleted files are readily available for a review and adding back to the project by clicking on the **Hierarchy | Add Sheets to Project** option.

Opening project sheets

To open schematic sheets that belong to the current project, click on the

Open a Schematic  icon, or select the **File | Open** option. In response, the **Open** window (Figure 4-4) appears and lists all project sheets. The project sheets can also be invoked by double clicking on their names in the Hierarchy Browser.

Renumbering Symbol References

The symbol reference numbers are assigned sequentially in the order in which they are placed on schematic sheets. As a result, the symbol reference numbers in a multi-sheet schematic design can be random. To order symbol numbers by page, use the **Options | Annotate** option. It will allow you to force the desired reference designations on each sheet.

For example, symbols on the first sheet may start with U100. On the second sheet they may start with U200, etc. This naming convention simplifies tracking of symbols on schematic sheets.

To renumber project sheets and the associated symbol reference numbers, perform the following procedure:

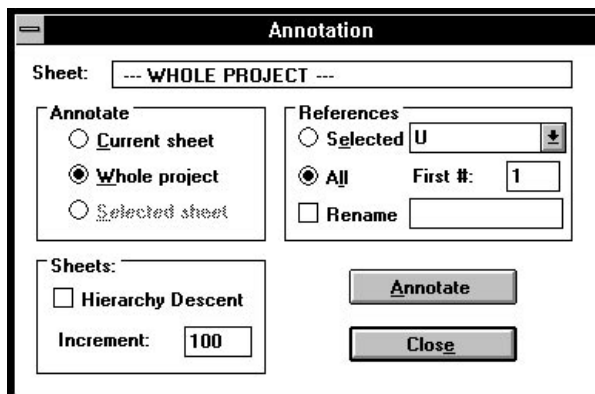


Figure 4-5. Annotation window.

1. Select the **Annotate** option in the **Options** menu.
2. In the new **Annotate** window select the **Whole project** option.
3. In the **Sheet** field, enter the sheet increment, e.g. **100** (**Increment** box).
4. Press the **Annotate** button.
5. Press the **Close** button.

To renumber sheets in the hierarchical descent order, select the **Hierarchy Descent** box and enter the sheet increment number into the **Increment** field. All subsequently drawn lower hierarchical levels will have their reference numbers increased by the selected number.

Creating a netlist for a multi-sheet design

A netlist generated from a multi-sheet design always carries the project's name and has the .ALB file name extension. This file is a binary netlist that can be used by the simulator and can be exported to other netlist formats. Since the design is contained in one file, the exported netlist is also contained in one file.

If you need to divide a project into two or more separate netlists, you can use the following procedure:

1. Using the Hierarchy Browser , select the sheets to be included in the first netlist. All other sheets should be deleted from the project.
2. Select the **Options | Export Netlist** option to export the current netlist.
3. Using Windows **File Manager**, rename the generated output netlist file.
4. To add to the project the previously deleted schematic sheets, select the **Hierarchy | Add Sheets to Project** option. Select the desired schematics from the new **Add to Project** window.
5. Export the second netlist. Its name will be the same as the project's name, but it will contain only the currently selected sheets.

NOTE: If you do not rename netlists generated for the preceding design sections, they will be overwritten by the last created netlist.

Troubleshooting project contents

- ☐ The project contents is stored in the *.PRJ binary file. The file name is the same as the current project's name, and it is located in the project directory. If the project contents file is corrupted, you can delete it and recreate it by adding all sheets to the project again.
- ☐ If a netlist creation error is reported, try removing sheets from the project until the netlist can be successfully created. Then analyze the last-removed sheet for any possible errors.

Multi-sheet schematic connectivity rules

All nets with identical names are automatically connected by the editor. This rule applies both to single-sheet schematic designs and to multi-sheet designs. Because of that, you do not need to connect circuits on different sheets with connector or terminal pins.

All wire nets with identical names are connected together. Buses with identical names are connected only through its member wire names.

Hierarchical Design Management

The design has a hierarchical structure if any of the symbols contains a netlist or a schematic. The hierarchical symbol is called a macro because it is built of other symbols internally. The hierarchical macros can be created by the user or may already exist in the library. Some libraries contain symbols that are hierarchical macros. If you use one of these symbols, your design becomes hierarchical.

NOTE: Only FPGA system libraries contain hierarchical macros. Libraries such as TTL, ECL, CMOS, PLD, DSP, etc. do not contain macros.

Recognizing hierarchical macros

The symbols that are hierarchical macros can be recognized by their color. By default, all schematic-based macros are dark blue. The netlist-based macros are purple. These default colors can be changed in the **View/Preferences** menu.

Netlist-based macros

Importing a netlist file into a schematic automatically creates a macro which is stored with its own symbol in the project library. The saved macro can then be placed on the schematic as any other symbol. The netlist-based macro cannot be edited in schematic editor¹.

The applications for the netlist based macros are the following:

1 **NOTE:** When importing a netlist, you can automatically convert it into a schematic. Once it becomes a schematic macro, you can then edit this macro schematic.

- ❑ To reuse netlist files created in other schematic editors. This eliminates a need for reentering the schematics again.
- ❑ To download netlist files generated by logic synthesis tools. A netlist from VHDL or state machine editor can be converted into a macro and placed on the schematic together with other parts of the design.
- ❑ FPGA post-layout netlists can be simulated with board-level components. For example, XILINX routed netlist can be imported and placed as a macro at the board-level schematic. This macro can then be interconnected and simulated together with other devices such as memories, PLDs, MPUs, DSPs and other components.

Creating Netlist-based Macros

A hierarchical macro can be created from any netlist file, having one of the supported netlist formats. Additionally, a schematic can be generated from such a netlist if the symbols for all the used components are readily available in the project library.

To create a macro symbol from a netlist, follow this procedure:

1. Select the **Import Netlist** option from the **Hierarchy** menu.
2. Select from the **Import Netlist** window, the netlist format (from the **From Format** list) and the netlist file (**Input Netlist**) to be imported.
3. Click the **OK** button. This starts the conversion of the selected netlist format to the ACTIVE-CAD binary format.
4. A new symbol is automatically added to the project library and to the list of symbols in the **SC Symbols** toolbox.
5. You can place the newly created symbol on the schematic.
6. If you are not satisfied with the automatically generated symbol shape, use the Symbol Editor program to make changes.
7. To edit pin names or the internal macro design, import the netlist as a schematic (see *Generating schematics from the netlist* section, below).

ACTIVE-CAD automatically creates a symbol that is based on input and output terminals listed in the netlist.

- ☞ Some netlist formats do not contain information about input and output terminals. In such a case, the symbol will have no pins and is not usable. Some netlist formats do not define the direction of the I/O terminals and thus all pins are defined as bi-directional. This results in generating a symbol that has all pins on the right-hand side.

Importing a hierarchical netlist

Some netlist formats can contain hierarchy information. In such a case the imported netlist is converted into multiple macros in the project library. Each macro corresponds to the original hierarchical netlist block in the source file and has its own symbol in the project library after the netlist import.

Generating schematics from the netlist

ACTIVE-CAD can generate schematics from netlist files being imported as macros. To create a schematic sheet from a netlist, select the **Import Netlist** option in the **Hierarchy** menu. In the **Import Netlist** window, select the **Schematic** option and then click on the **OK** button. The schematic generation process requires that all symbols must be found in the currently selected libraries. The generated schematic is built on a single sheet and contains only logical connections (net names).

Schematics generated from imported netlists are saved in the project library, and they can be viewed and edited the same way as any other hierarchical schematic macros.

Updating netlist macros

A library macro that has been generated from imported netlist will include in the comment field a reference to the original netlist location on disk and its netlist file format. When you create a netlist of the design that contains netlist-based macros, the version of the macros is compared with the file on the disk. If the netlist on the disk is newer than the macro, a message is displayed asking if you want to update the macro. Answering Yes, will automatically update macro with the new netlist.

You can update macros at any time by importing their netlists and overwriting the old macros.

Assigning a Netlist to a Symbol

To add an FPGA netlist to a board-level design for the purpose of simulation, use the **Assign Netlist** option in the **Hierarchy** menu. This option allows assigning a netlist to an existing symbol. The **Assign Netlist** option creates a new object in the project library that contains the imported netlist and the symbol from the schematic. The new object receives the name that is a combination of the original symbol name and the imported netlist name. For example, when you assign a netlist TEST.NET to a symbol PGA175, The new symbol will be called PGA175@TEST.NET.

To assign a netlist to a symbol on the schematic, select the **Assign Netlist** option from the **Hierarchy** menu, and click on the symbol that should be loaded with the imported netlist. In response, the **Netlist Conversion** window is displayed. Select the desired netlist format (from the **From Format** list), select the file name from **Input Netlist** field and click the **OK** button. The netlist will be imported and the schematic symbol will be saved along with its netlist, under the new symbol name.

NOTE: You cannot assign a netlist to a symbol that has a simulation model. You can assign it only to a symbol without its own model.

Creating Schematic Macros

To create a hierarchical macro that can be placed as a symbol on a higher level schematic, follow this procedure:

1. Open a new sheet and enter a schematic design.
2. Use I/O terminals for the schematic inputs and outputs. These signals will be used as symbol pins for the macro symbol.
3. Choose the **Create Macro Symbol From Current Sheet** option in the **Hierarchy** menu.
4. In the **Create Symbol** window (fig. 4-6) do the following:
 - ☐ Enter **Symbol Name** (up to 8 characters).
 - ☐ Enter a **Comment** (optional).
 - ☐ Verify the **Input** and **Output** pins.
 - ☐ Make sure that the **Netlist** option is selected.
 - ☐ Select the symbol shape (**Dual in line** or **Square**).
5. Click the **OK** button. The schematic is instantly converted into a macro with the name selected in step 4 and saved in the project library.
6. A confirmation message will be displayed upon successful macro generation and will ask if you want to continue editing of this macro. Answering *No* ends conversion of the schematic sheet into a macro.
7. To use the new macro, select it from in the **SC Symbols** window and place on the schematic as any other symbol.
8. To modify the automatically created symbol (change its shape or move pins around), use the **Symbol Editor** option in the **Options** menu.

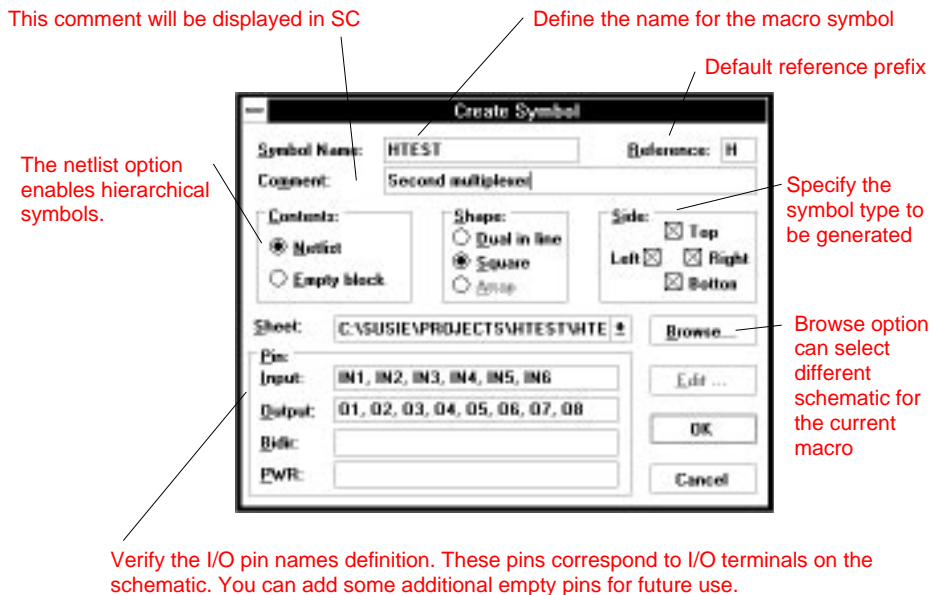


Figure 4-6. Create Hierarchical Macro option.

Creating a hierarchical schematic

Before drawing a schematic, make sure that all necessary libraries have been assigned to the project (schematic). First, select the **Project Libraries** option from the schematic **File** menu, or click on the **Project Libraries**



icon in the Project Manager. When the **Libraries** window appears, select the appropriate libraries and then press the **Add** button. This transfers the selected libraries from the **Attached libraries** field to the **Project libraries** field and makes them available to the schematics editor.

Next, enter the schematic design, just like any other flat design but remember the following constraints:

- ☐ Each schematic macro is a self-enclosed entity; any connection to the other sheets can only be made through the I/O pins.
- ☐ Specify the I/O pins explicitly as either **Input**, **Output** or **Bi-directional**. This is very important because ACTIVE-CAD automatically generates a rectangular DIP symbol and the location of the pins on the symbol outline depends upon the pin's I/O type (only Input pins

are placed on the left-hand side of the symbol outline). If needed, the symbol can be edited in the Symbol Editor.

Bus pins in a macro symbol

Use the bus I/O terminals to add bus pins to a hierarchical symbol. Each bus I/O terminal is automatically converted into one bus pin of the hierarchical symbol. This is making the symbol smaller because buses substitute for multiple discrete wires.

To convert individual terminals into a bus pin, edit them directly in the **Create Symbol** window:

If I/O terminals have the same name but different indexes (e.g. A1, A2, A3 ... A15), you can delete them from the pin listing in Figure 4-7 and replace with a single bus pin name, preceded and followed by the double equal signs. For example, `==A[1:15]==` will create a bus pin with the A name and [0:15] index.

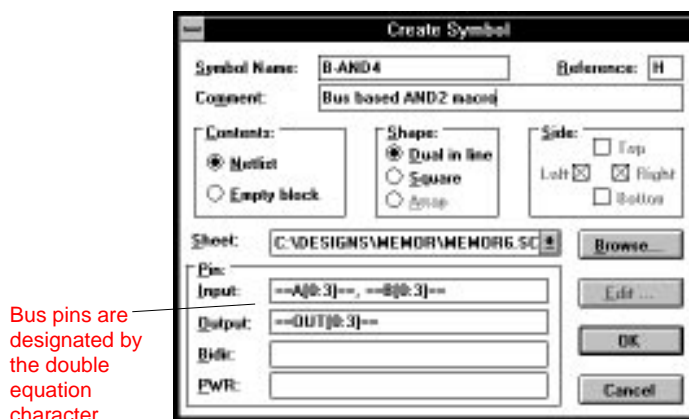


Figure 4-7. Creating a macro with bus pins.

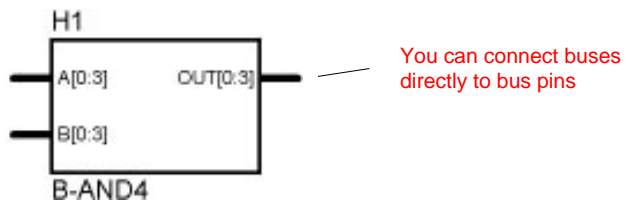


Figure 4-8. New B-AND4 macro symbol with bus pins.

Using macros in the schematic

Hierarchical macros are used in schematics the same way as other symbols. The symbol color (dark-blue) indicates that a schematic macro is associated with the symbol. It can be opened with the **Hierarchy Push/Pop** option. Use the **Query** mode to find out the location of the macro in project libraries. You can place multiple copies of the same macro on a schematic sheet.

Remember that only signals defined as macro I/O pins can be used for direct connection with other devices and macros on the schematic sheet.

Opening a hierarchical macro schematic

You can open a hierarchical macro schematic window in two ways:

- ☐ Double-click on the macro's name in the Hierarchy Browser section of the Project Manager. This opens the macro's schematic. To save Windows resources, close the schematic sheet as soon as it is not needed.
- ☐ Invoke the **H** cursor (**Ctrl+H**) and click it on the macro symbol. This opens the macro's schematic temporarily. The schematic is automatically closed upon the **H** cursor exit from the hierarchical level.

Navigating a hierarchy

Double-clicking the **H** cursor on a hierarchical symbol displays the internal schematic of that symbol. The **H** cursor is invoked either by pressing the **Ctrl+H** key or by clicking on the **H** icon in the vertical toolbar.

While the **H** cursor is active, you can open as many macros as desired. To go back to a higher hierarchical level, double-click on an empty space in the macro schematic. If you click on a symbol that does not have any schematic, ACTIVE-CAD displays a message *Symbol is a Primitive Cell*. If the **H** cursor is at the top hierarchical level, clicking the mouse button outside schematic symbols, displays the message: *Already at the Top Level*.

To exit the **H** cursor mode, click the right mouse button or press the **Esc** keyboard key. You can also view and navigate the hierarchical structure of the design from the Hierarchy Browser in Project Manager (Figure 1-22). Please refer to the *Hierarchy Tree Operations* section in Chapter 1 of this manual.

NOTE: Double-clicking on the schematic name in Hierarchy Tree loads that schematic to the screen for detailed viewing and editing.

Opening too many schematics in the Hierarchical Browser may strain the Windows resources. Delete the unused schematic sheets from the **Window** menu.

Modifying Macros

If you want to make some changes to an existing macro schematic, use the following procedure:

1. Bring up the schematic of the macro using any of the three (3) methods described above. ACTIVE-CAD displays header: *Hierarchy Level - Hn*
2. Select the **Edit Hierarchical Symbol** option from the **Hierarchy** menu.
3. In response, ACTIVE-CAD displays header: *Hierarchical Symbol -*. This indicates that the selected schematic is enabled for editing.
4. Make necessary changes to the schematic.
5. Select the **Save** option in the **File** menu. This updates the macro.

When you change and save a hierarchical symbol, you change all the instances of this macro in the entire design. If the modified macro schematic has different I/O pins, its symbol will change and the pins may not match their previous locations on the schematics. If this happens, ACTIVE-CAD will automatically disconnect the wrong wires and mark them with crossed circles. These wires will have to be manually reconnected by switching to the **Select and drag** mode, dragging the crossed circles over the target pins and releasing the left mouse button.

If you edit a macro from the system library that comes with the product, you will not be allowed to save it in the system library. You can only save it in the project library. For clarity, use a different name (use the **Save As...** option) for the modified macro so that you can always be sure which symbols are currently used on the schematics.

NOTE: All macros created by modifying the ACTIVE-CAD (system) macros can only be used with the keylock with which they were saved. For this reason, network keylocks are strongly recommended because they allow designers to share their custom-made macros.

The difference between a macro and a schematic

The following example explains what happens with the hierarchical macro schematic when you create a macro. Assume that the project TEST contains schematic sheets TEST1 and TEST2. Lets create a macro out of the schematic sheet TEST2.

1. Using the **Create Macro Symbol** option, convert the TEST2 schematic into a macro called MACRO1 in the TEST project library.
2. The old schematic sheet TEST2 is instantly removed from the project contents, but the file TEST2.SCH still exists in the project directory (e.g. in C:\ACTIVE\PROJECTS\TEST). You can open this schematic file only with the **Browse** option (available from the **File | Open** menu), but there is no longer any relationship between the TEST2.SCH schematic file and the TEST project or MACRO1.
3. You can directly modify macros schematics. They are automatically updated when you activate the **File | Save** option.
4. The schematics from which the macros were generated are stored in the projects library, e.g. C:\ACTIVE\PROJECTS\TEST. Using the Windows **File Manager**, you can delete the file TEST2.SCH from this directory because it is now stored as the MACRO1 schematic.

What happens when the hierarchy symbol changes?

When a hierarchical macro is updated, its symbol is not modified unless there is a different number of I/O pins. In such a case a new symbol is generated based on the new I/O pins. This may result in incorrect connections on the schematics that have previously used that symbol.

Clicking on the **File | Save** option will automatically update all opened schematics with the new symbol. Other schematics will be updated when you open them in the schematic editor.

- ☐ When the symbols do not match the previous connections, the wires will be disconnected from that symbol, and a crossed circle or large dots will be displayed at the end of these wires. To correct these connections, you can drag the circles in the **Changes** mode and drop them at the appropriate pins.
- ☐ You can edit the newly created symbol in the **Symbol Editor** program so that it matches the old pin locations.

Editing system (ACTIVE-CAD) macros

An edited system macro (from the system library), cannot be saved back into the system library because of built-in protections against such modifications. They will be automatically stored in the current project library with the **Save** and **Save As...** operations from the **File** menu.

Exporting a hierarchical netlist

Use the **Export Netlist** option in the **Options** menu to export a netlist. A hierarchical design netlist, can be exported as either hierarchical or flat.

If you choose the **Flat** option in the **Export Netlist** window, the netlist will be exported as one netlist file. To avoid duplication of reference numbers and net names, the schematic editor adds the hierarchy designator name in front of each hierarchical symbol reference and net name. For example, if the macro H2 that contains the U45 symbol, the exported netlist will list it as H2/U45.

NOTE: If you do not select the **Flat** option, all hierarchical macros will be exported as a set of separate netlist files located in the project directory.

Hierarchical Annotation

The **Options | Annotation** option allows exporting hierarchical designs as single (flat) netlists with unique reference names for every symbol instance. This eliminates the hierarchical prefixes in symbol names, and makes for shorter netlists.

To renumber the entire design select the **Annotate** option in the **Options** menu. In the new **Annotation** window(Figure 4-10) select the **Whole Project**, **Hierarchy Descent** and **All** (References) options.

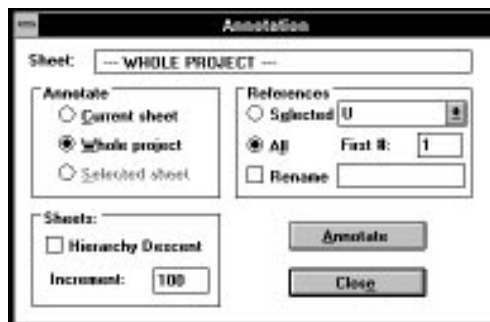


Figure 4-9. Hierarchical design annotation.

Creating a library of user macros

Each newly created macro is saved by default in the current project directory (e.g. C:\ACTIVE\PROJECTS\TEST\LIB). However, you can create a special library that contains all your macros for future use on different projects:

- ☐ Select the **Create** option from the **Library** menu of the **Library Manager**
- ☐ Enter the desired USER MACROS *library name* and its *directory*.
- ☐ Draw the macros schematic, and select **Create Macro Symbol From Current Sheet** in the **Hierarchy** menu; use the **Browse** option to select the USER MACROS (destination) library directory.
- ☐ Click on the **OK** button.
- ☐ Select **Project Libraries** in the **File** menu and attach the USER MACROS library to the project. Otherwise, the newly created macro will not be seen by the current project.

You can protect the proprietary macros library against accidental damage, by changing its access mode from the R/W (read and write) to R/A or R/O in the **Library Manager**. You can also copy macros from different project libraries to a desired library using the **Library Manager** (LM) (see *Library Manager* Chapter). Such library can contain all user macros.

Deleting macros

To delete a macro from the library, select the **Library Manager** (see *Library Manager* Chapter) window and double-click on the project library that contains the macro. Click on the desired macro in the listing of objects, and without closing that window, select the **Delete** option from the **Object** (LM) main menu.

Renaming macros

There are two (2) ways to rename a macro:

- ☐ Open the selected macro in schematic editor, and then using the **File | Save As** option save it under a different name
- ☐ Change the macro name in the **Library Manager**.

If you have decided to change the macro name in the **Library Manager**, invoke this application by selecting **File | Project Libraries** option, and then click on the **Lib Manager** button. This opens the **Library Manager** window. Next, double click on the project library that contains the macro, highlight the macro in the **Logic Name** window, and without closing that window, select the **Rename** option in the **Object** main menu. Enter the new name in the **Rename** window and confirm the change.

Top down methodology

To implement a top-down design, first create empty symbols for hierarchical macros. These empty symbols will then be implemented as schematic macros.

To create an empty symbol, select the **Hierarchy | New Symbol Wizard** option and then:

- ☐ click on **Next>** button in the **Design Wizard** welcome screen.

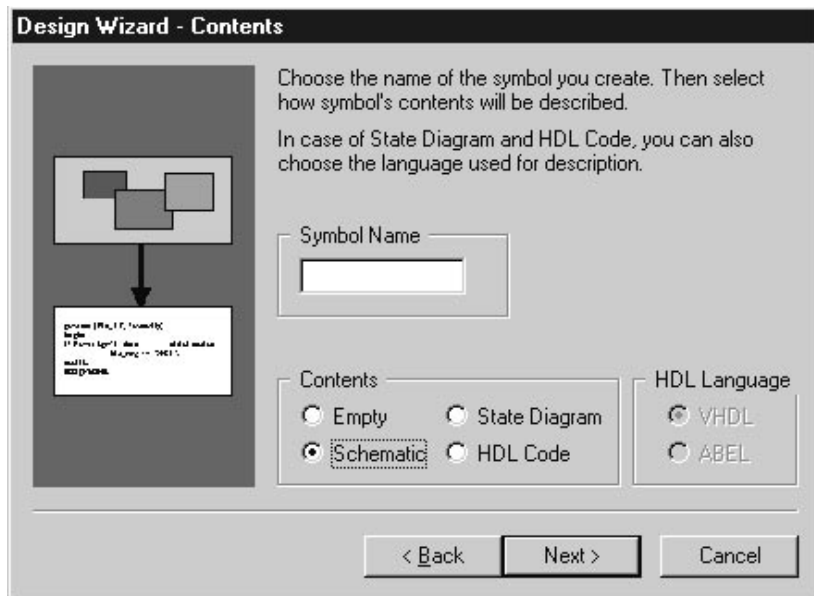


Figure 4-10. Design Wizard - Contents.

- ☐ In the **Contents** window shown in figure 4-10 enter symbol name and select *Schematic* as the contents of the macro. (*State Diagram* or *HDL Code* can also be chosen; selecting *Empty* means that macro contents type will be picked later).
- ☐ Click on the **Next>** button
- ☐ In the **Ports** window (figure 4-11) add ports to the symbol by clicking on the **New** button, entering port name in the **Name** field and selecting port **Direction**. Selecting clock input type and combinatorial/registered output type can be performed after clicking on the **Advanced** button.

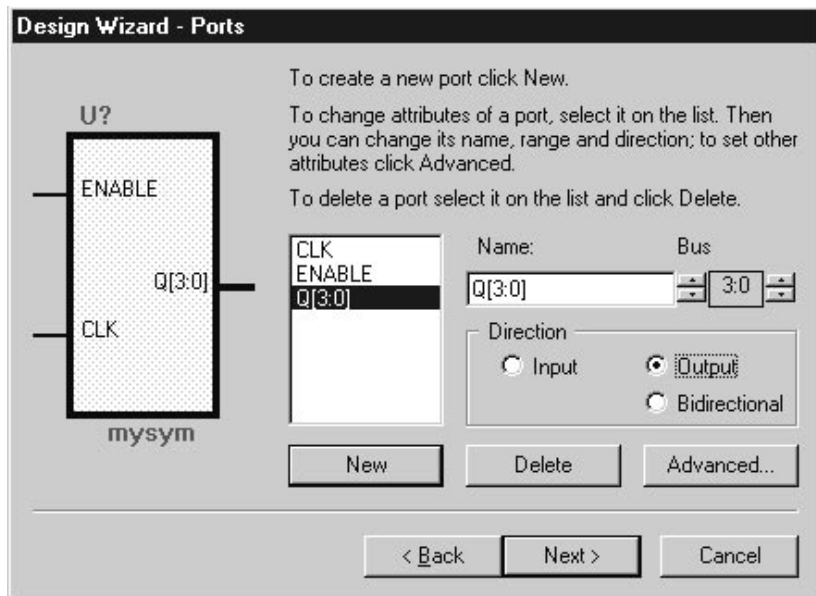


Figure 4-11. Design Wizard - Ports.

- ☐ Click on the Next> button.

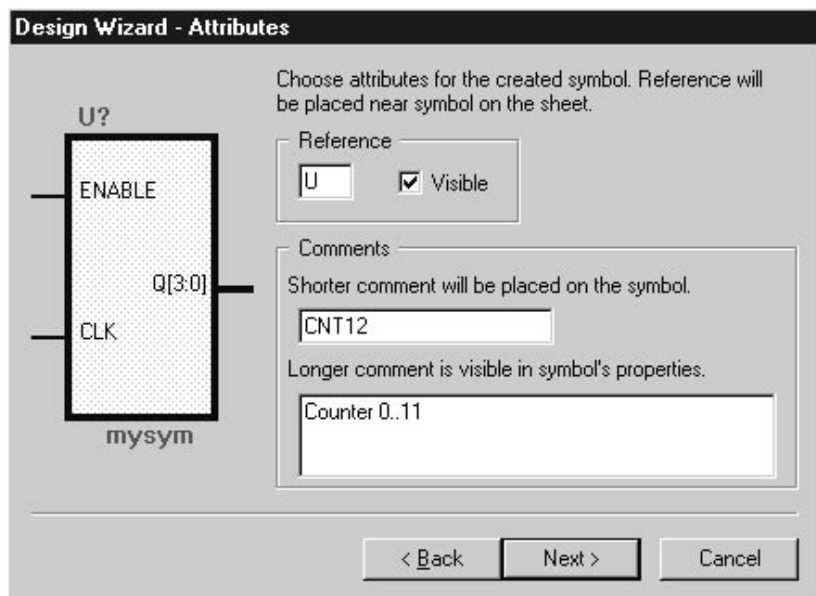


Figure 4-12. Design Wizard - Attributes.

- ☐ In the **Attributes** window shown on figure 4-12 enter required symbol reference letter and symbol comments.
- ☐ Click on the **Next>** button.
- ☐ Click on the **Finish** button in the final Design Wizard window.
- ☐ A new window opens, and displays the previously entered symbol pins at the left and right edge of the schematic sheet. Enter the schematic design and connect to the I/O pins.
- ☐ Click on the **Save** option in the **File** menu. This updates the macro for future use on schematic sheets. However, the existing symbols will not be updated by the save operation.

Instead of drawing a schematic inside an empty symbol, you can assign to an empty symbol a ready-made design netlist by selecting the **Assign Netlist** option from the **Hierarchy** menu and clicking on the empty symbol. The **Netlist Conversion** window (Figure 4-13) allows selecting the desired netlist. The imported netlist will be automatically converted into a schematic if the **Schematic** check-box in the **Netlist Conversion** window has been selected.

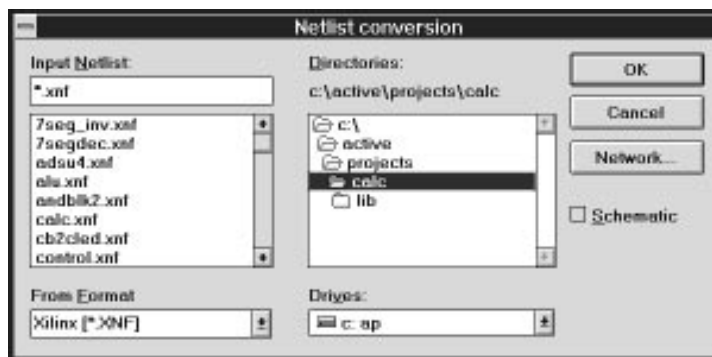


Figure 4-13. Netlist Conversion Options.

Hierarchy Errors

The hierarchical designs are checked for hierarchy errors such as circular macros. The circular hierarchy structure means that one of the macros in the design uses some hierarchical symbols that include itself. When the hierarchy error is reported, you have to analyze your design in the Hierarchy Browser and make the necessary corrections.

Hierarchy Menu Options Summary

<u>H</u> ierarchy	<u>V</u> iew	<u>D</u> isplay	<u>W</u> indow
Add Sheets to Project			
Add Current Sheet to <u>P</u> roject			
Create Macro Symbol From <u>C</u> urrent Sheet			
New Symbol <u>W</u> izard ...			
<u>H</u> ierarchy Push			Ctrl+H
<u>H</u> ierarchy Pop			Ctrl+I
<u>I</u> mport Netlist ...			
Assign <u>N</u> etlist			


Figure 4-14. Hierarchy menu.

- ☐ **Add Sheets to Project** - Opens a window for adding schematic files to the current project
- ☐ **Add Current Sheet to Project** - Adds a sheet to the current project
- ☐ **Create Macro Symbol From Current Sheet** - Converts schematic sheet into macro symbol.
- ☐ **New Symbol Wizard** - creates a schematic or an empty macro symbol
- ☐ **Hierarchy Push** - selects the Push mode (activates the **H** cursor) for navigating hierarchical schematics.
- ☐ **Hierarchy Pop** - allows exit from the current hierarchical level.
- ☐ **Import Netlist** - Creates a macro from any netlist file
- ☐ **Assign Netlist** - Downloads a netlist into an existing symbol and creates a macro from the existing schematic symbol

Example of Hierarchical Design

The following is a step-by-step process which explains how to create and use the hierarchical designs in ACTIVE-CAD. To better explain hierarchical designs, a simple design will be created and simulated interactively. For simplicity, only TTL components will be used for the schematic.

Step 1. Create a new project called MACROS.

Click on the New Project  icon in the **Project Manager**. When the New Project window appears, type **MACROS** in the Name field, select **ALDEC** project Type, and click the **OK** button or press **Enter**.

Step 2. Select the TTL library for the project.

Select the **Project Libraries** option in the schematic **File** menu. In the **Libraries** window select the TTL library from the **Attached libraries** list and click on the **Add** button. The TTL library will be added to the **Project Libraries** list. Activate the **Close** button to exit the **Libraries** window.

Step 3. Create the MACROS1 schematic.

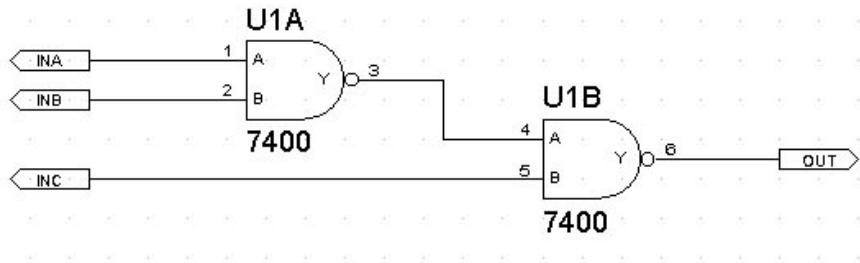



Figure 4-15. MACROS1 schematic.

Click the **Symbols toolbox**  icon. When the **SC Symbols** toolbox appears, select the 7400 symbol. Next, draw the schematic in Figure 4-15, including input terminals *INA*, *INB*, *INC* and output *OUT*. Save the schematic using the **Save** option in the **File** menu. The schematic will automatically be named by ACTIVE-CAD as *MACROS1*.

Step 4. Create the MACROS1 Symbol.

Select the **Create Macro Symbol From Current Sheet** option within the **Hierarchy** menu. The **Create Symbol** window in Figure 4-7 should display the symbol name *MACROS*. Click on that name and change it to *ONE*. This will be the name of the hierarchical macro symbol. In the **Input pins** section there should be a list of pins: *INA*, *INB* and *INC*. The output pin name should be *OUT*. Note that only I/O terminals are recognized as pins for the symbol. Click on the **OK** button. The netlist and the *MACROS1* schematic will be saved in the project library and a graphical symbol will be created automatically.

Note that the newly created symbol can be named differently from the schematic itself. For example, the schematic in Figure 4-15 is called *MACROS1*, but the new symbol created from that schematic is called *ONE*. Upon successful macro generation, a message window will be displayed and you will be asked if you want to continue editing the macro schematic. If you answer *No*, the schematic will be closed.

Step 5. Create the second schematic (MACROS2).

Click on the **New Schematic** icon or select the **New** sheet option from the **File** menu. A new schematic *MACROS2* will automatically be opened. Draw the schematic shown in Figure 4-16. Remember to use as input terminals D, CLK and RST and call the output terminal OUT. Save the schematic using the **Save** option.

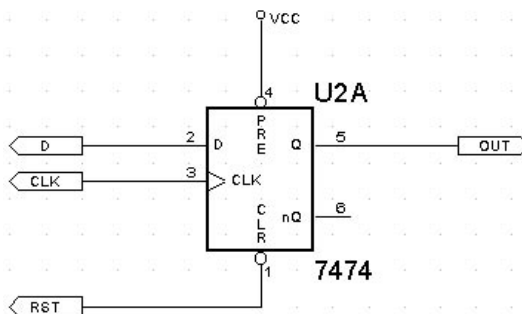


Figure 4-16. *MACROS2* schematic.

Step 6. Create symbol MACROS2.

Select the **Create Macro Symbol From Current Sheet** option within the **Hierarchy** menu and in the **Create Symbol** window change the **Symbol Name** from *MACROS* to *TWO*. Then click on the **OK** button. The new symbol *TWO* will be created and placed along with the schematic in the *MACROS* library.

Step 7. Create new sheet for top level.

Click on the **New Schematic** icon or select the **New** sheet option from the **File** menu. A new schematic *MACROS3* will be opened. Draw the schematic shown in Figure 4-17. Typing *ONE* in the bottom line of the **SC Symbols** toolbox, will highlight the item for selection. Place two copies of that symbol, which are automatically called H1 and H2, on the schematic. Similarly, place two copies of the *TWO* symbol on the schematic. They will automatically be called H3 and H4 (Figure 4-14). Interconnect the symbols as shown in Figure 4-17.

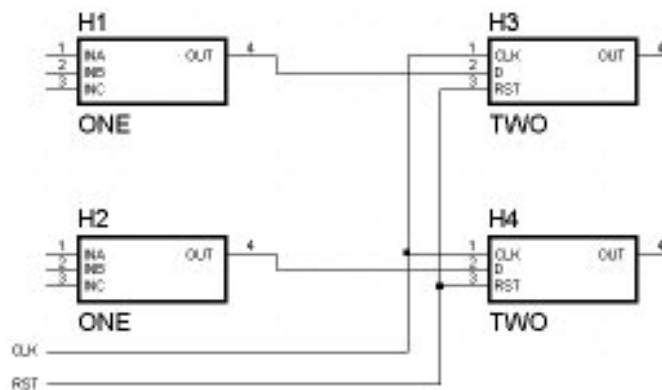


Figure 4-17. Top level schematic.

Step 8. Use Push/Pop option to view schematics.

Select **Hierarchy Push** option within the **Hierarchy** menu or press **Ctrl+H**. A cursor with the letter H will appear. Point the cursor at the symbol H1 and double-click the mouse button. The schematic ONE will open, showing internal design of the symbol H1.

Step 9. View the Project Contents.

Switch to the Hierarchy Browser (in Project Manager). Notice that the list of sheets is now showing the top level schematic and four sub-levels (H1..H4). Double-clicking on any of these schematic names will open the desired sheet.

Step 10. Edit the hierarchical levels

The schematics displayed in the Step 8. and 9. can be directly edited. Use the File | Save option to save the changed macro and update all schematics.

Chapter 5

Schematic Menu Reference

This chapter describes the menu options and commands in detail.

File Menu

New sheet

File	Edit	Mode	Option
New Sheet			Ctrl+N
Open			Ctrl+O
Scratchpad			
Close			
Close All			
Save			Ctrl+S
Save As ...			
Save All			
Project Manager			
Project Libraries			
Update Libraries			
Go to Simulator			
Import			▶
Export			▶
Print ...			Ctrl+P
Printer Setup ...			Ctrl+T
Page Setup ...			
Table Setup ...			Ctrl+B
Exit			Alt+X

The **New sheet** command creates a new sheet for editing. The new sheet name consists of the project name and sequential page number, which is automatically added to the project name. To create a schematic sheet that does not belong to the project, use the **Scratchpad** option in the **File** menu in Figure 5-1.

Figure 5-1. File Menu.

Open Sheet

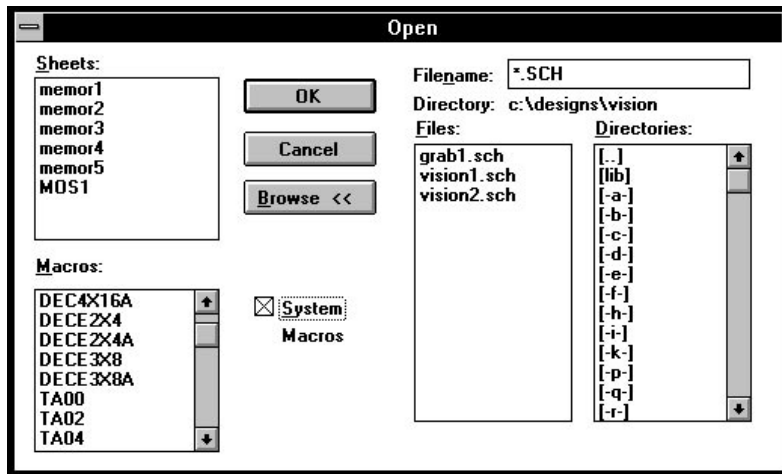


Figure 5-2. Open Sheet Allows Loading A Schematic.

The **Open** option in the **File** menu allows loading any schematic sheet in the current project for viewing and editing. Clicking on the **Browse** button in the **Open** window displays Figure 5-2, which allows loading schematics from another projects. The loaded schematic is in the viewing mode and cannot be edited till it is added to the current project. To do so, click on the **Add Current Sheet to Project** option in the **Hierarchy** menu.

NOTE: Opening a schematic file or browsing a schematic does not append it to the current project. To append it to the project, you must click on the **Add Current Sheet to Project** option in the **Hierarchy** menu.

Scratchpad

The **Scratchpad** option allows creating a schematic that are outside the current project. A scratchpad schematic cannot be simulated until it is added to the project resources.

To add a scratchpad schematic to the project, select the **Add Current Sheet to Project** option from the **Hierarchy** menu. When a prompt window appears, enter the sheet name and exit.

Close

The **Close** option closes the currently active schematic window. If the schematic sheet in this window has been modified you will be prompted

to save the changes before the schematic window is closed. The **Close** option does not affect the project contents, and if the schematic was a part of the current project it will still be included in the netlist when it is generated. To delete a sheet from the project, use the **Delete Sheet From Project** option in the **Hierarchy** menu.

Close All

The **Close All** option closes all active schematic windows. If the schematic sheets in any of these windows have been modified, you will be prompted to save the changes before the schematic windows are closed.

Save

The **Save** option saves only the active sheet. If the **Save** button is gray, the active sheet has either been saved (no new changes) or is not a part of the current project. If a name has not been specified for this sheet, you will be prompted to select the sheet name.

Save As

The **Save As** option allows making an extra copy of the active sheet and saving it under a new name. This option allows for having different versions of similar sheets.

Save All

The **Save All** option saves all opened schematic sheets and the Schematic Editor settings. The settings are also automatically saved upon exit.

Project Manager

This option displays the **Project Manager** window and allows opening new projects, etc. (see *Project Manager* Section in Chapter 1).

Project Libraries

This option is used (ACTIVE-CAD requires that all schematic wires be electrically correct at all times). for selecting libraries for the current project (see *Library Manager* Chapter).

Update Libraries

If a symbol is modified while a schematic sheet is open, the changes will be stored in the schematic database when the **Update Libraries** option is selected, or the schematic is closed and then opened again.

Go to Simulator

This option passes control to the Simulator window. If the Simulator is not running, it is launched and loaded with the current design.

Import

This menu allows importing symbols and schematics from other EDA tools, after which the imported schematic designs will look like the original designs. You will be able to mix the original vendor symbols with the ACTIVE-CAD symbols in the same schematic. In addition, they will operate with the on-line background simulator in real-time mode.

Import OrCAD symbols

This option allows importing symbols saved in the OrCAD ASCII format. To import an OrCAD library from OrCAD, you need to decompile it with the **DECOMP** program from OrCAD. The resulting text file can then be imported into ACTIVE-CAD using the **Import/OrCAD Symbols** option in the **File** menu (Figure 5-3).

Click on the **Select OrCAD (ASCII) library** button to browse disks and select library files. ACTIVE-CAD will display a list of symbols found in the library file. If the library is very large, you can use the **Filter** button to specify a mask for the listed symbols (e.g. *574). To view the symbols before importing them into ACTIVE-CAD, use the **View Symbol** and **View Next** buttons.

To import all symbols, press the **Put All** button, or select the desired symbols from the list in Figure 5-3 and press the **Put Selected** button. The selected symbols will be placed in the current project library. You can then use the **Library Manager** to copy them to other libraries and the **Symbol Editor** to edit these symbols.

The appropriate ACTIVE-CAD libraries must be selected to the current project before importing symbols. This will attach the appropriate models to imported symbols. Otherwise, the imported schematics will have empty symbols and will not simulate.

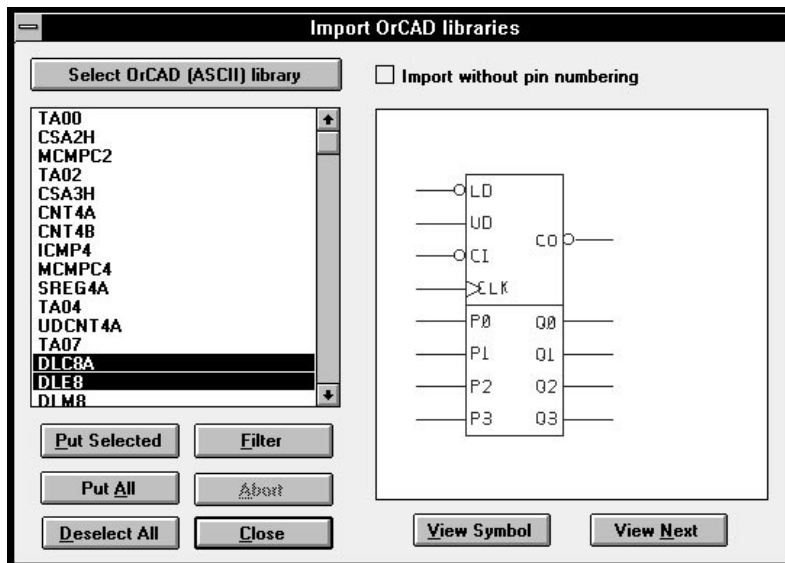


Figure 5-3. Import OrCAD Symbols.

Import PADS Symbols

The **Import/PADS Symbols** option in the **File** menu imports graphical symbols from the PADS Logic schematic editor. Before you can import PADS symbols you need to convert them to ASCII format using the ASCII OUT option in PADS Logic. The files with the *.PT and *.PD extensions need to be converted to *.PTA and *.PDA, respectively.

Import EDIF Symbols

This option imports schematic symbol libraries from other schematic capture programs that have EDIF 2.0.0 export capabilities. Using the **Import/EDIF Symbols** option in the **File** menu, you can import EDIF symbol files. The imported symbols will be added to the current project directory.

Import ViewLogic Symbols

This option imports symbol libraries from the ViewLogic schematic editors. The imported symbols will be added to the current project directory.

Import OrCAD[®] Schematics

After the OrCAD symbols have been imported, you can convert OrCAD schematics into ACTIVE-CAD database. This option converts OrCAD passive schematics into ACTIVE-CAD active schematics for direct interactive design analysis in conjunction with the on-line logic simulator.

NOTE: Import OrCAD Schematic is an add-on option. A keylock is needed to enable this option in the Schematic Editor.

The **Import OrCAD Schematic** option allows importing OrCAD schematics in their native format (binary schematic files).

☞ Before you import an OrCAD schematic, you need to import all its symbols using the **Import/OrCAD Symbols** option.

To import an OrCAD schematic, do the following:

1. Determine which libraries are used in your OrCAD schematic. These libraries have to be imported into ACTIVE-CAD, so that the graphical symbols will look the same as in OrCAD. Convert all the libraries, which have been used in the OrCAD schematics, with the DECOMP program that comes with OrCAD schematic tools. For example, to convert the TTL library use the following command:

DECOMP TTL.LIB TTL.F

This creates an ASCII TTL.F file, which contains all symbols from the OrCAD TTL library. Use a similar command for other OrCAD component libraries.

2. Create a new project in ACTIVE-CAD. To allow the imported symbols to be assigned to ACTIVE-CAD models, select all libraries in ACTIVE-CAD that contain models used in your OrCAD schematics. These libraries will be used to assign simulation models to the symbols used in your OrCAD schematics (during the import process). For example, if you use the TTL symbols in OrCAD, add the ACTIVE-CAD TTL library to the current project, using the **Project Libraries** option within the **File** menu.

3. Invoke the **Import/OrCAD Symbols** option in the ACTIVE-CAD schematic **File** menu and import all OrCAD decompiled library files that will be used in the current project. This option allows importing all symbols from the OrCAD library or just selected symbols. Make sure that you import all of the symbols that are used in your OrCAD schematics. During this conversion, ACTIVE-CAD will restore the graphical symbols from the decompiled OrCAD library, and assign to them the simulation models from the ACTIVE-CAD library.

4. Import the schematic using the **Import OrCAD schematic** option. ACTIVE-CAD restores all electrical connections based on the graphical wires in the OrCAD schematic. If some of the wires are not electrically connected to device pins or labels, they will be converted into non-electrical graphics and shown as light blue lines. You can then manually cor-

rect these wires into ACTIVE-CAD electrical wires and complete the schematic.

NOTE: If you are importing a 32-bit OrCAD schematic into ACTIVE-CAD, you first need to convert these files into the 16-bit format, using the conversion software provided by OrCAD.

Import EDIF schematic

You can import a schematic drawings from other schematic capture programs like Mentor Graphics, Cadence, Viewdraw, which can output a schematic drawing file in the EDIF 2.0.0 format. The EDIF file must contain not only the netlist information but also the symbols and wires graphics. The **Import/EDIF Schematic** option loads and converts the EDIF file into ACTIVE-CAD schematic. After the conversion, you can save the schematic using the **Save As** option in the **File** menu.

Import ViewLogic schematic

You can import schematic drawings from ViewLogic schematic capture programs like PowerView and Pro Series. The **Import/ViewLogic Schematic** option loads and converts the ViewLogic ASCII files directly into ACTIVE-CAD schematic. After the conversion, you can save the schematic using the **Save As** option in the **File** menu.

Export

This option allows exporting ACTIVE-CAD schematic designs to other popular standards, such as EDIF and DXF (Autodesk). This option is used for storing designs in the company standard format and for further processing by other EDA tools.

NOTE: You must generate the design netlist file before using the **Export** options.

Export Schematic to EDIF

To export schematics into other schematic capture programs, that have EDIF import capability, use the **Export/EDIF** schematic option in the **File** menu. Clicking on this option, opens a special window for selecting the design source file. The resultant EDIF file is stored by default in the PRO-JECTS directory.

Export Schematic to DXF

Clicking on the **Export/DXF** option, opens special window for selecting the design netlist file to be exported. The resultant DXF file is stored by

default in the PROJECTS directory. The DXF format has no provision for storing complete electrical information. Once you export the design in the DXF format, it cannot be used to recover the electrical design properties. You must keep a duplicate design file in the ACTIVE-CAD or EDIF format for the backup purpose. The DXF format is used by some companies for archival purposes.

Print

The **Print** option (Figure 5-4) allows making hard copies of the schematic sheets on the selected Windows output device (printer or plotter). Note that the output device has to be installed in the Windows system with an appropriate device driver. For information on installing printers or plotters in the Windows, refer to the **Windows Users Guide**.

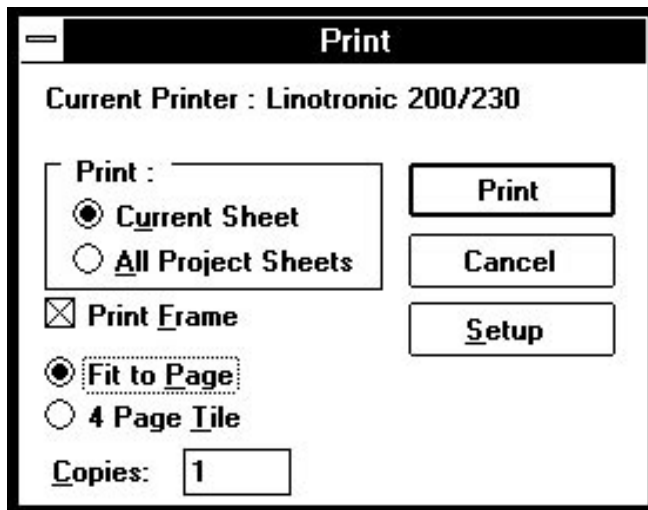


Figure 5-4. Print Dialog Box.

The Print window activates printing of the Current Sheet or All Project Sheets. It allows printing multiple copies of the schematic by changing the number in the **Copies:** field. The **4 Page Tile** option prints a schematic sheet on four pages for greater detail. The **Fit to Page** option will scale the schematic sheet to a single page. If you have multiple printers or plotters installed, specify the output device in the **Printer Setup** option, which can be activated from either the **File** menu or the **Setup** button in the **Print** window in Figure 5-4.

Printer Setup

The **Printer Setup** option (Figure 5-5) allows selecting a printer or plotter as the current output device, and setting the device specific options, such as Landscape or Portrait.

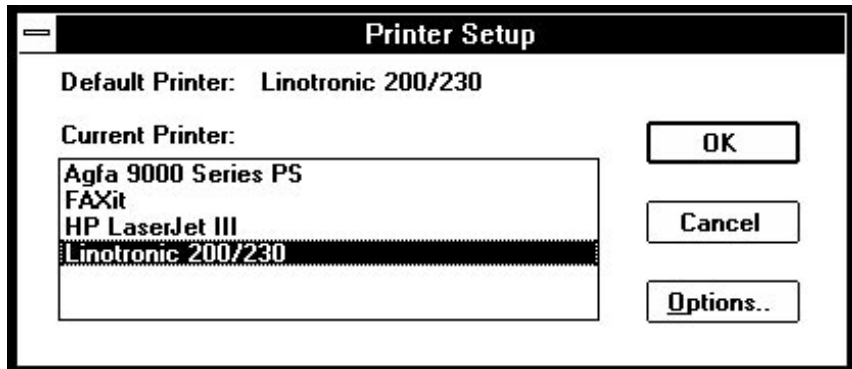


Figure 5-5. Printer Setup Window.

Page Setup

ACTIVE-CAD has a default page setup, including Frame, Ruler and Sheet Format. You can change this setup using the **Page Setup** option (Figure 5-7), which allows to specify:

- ☐ **Frame on** - If checked, a border is placed around the schematic when it is printed.
- ☐ **Vertical Numbers** - If checked, the vertical zones are 1,2,3 and the horizontal zones are A,B,C. The number of zones can be set using the **Horizontal** and **Vertical Zones** fields. These options are active only if **References** is selected as the **Ruler** type.
- ☐ **Sheet Format** - selects the sheet size(A-E) and orientation (Portrait or Landscape)

Ruler - allows specifying the coordinate system of the cursor coordinates that are displayed in the upper left corner of the main Schematic Editor window. **Ruler Metric** will display millimeters, **Ruler English** will display inches, **References** will display the zone numbers as specified in the **Horizontal** and **Vertical Zone** fields. The **Y Axis Up** option controls the direction of vertical zone numbers.

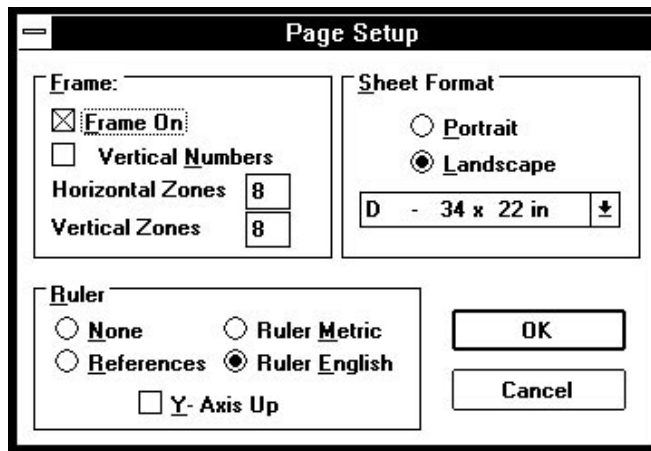


Figure 5-7. Page Setup Window.

Table Setup

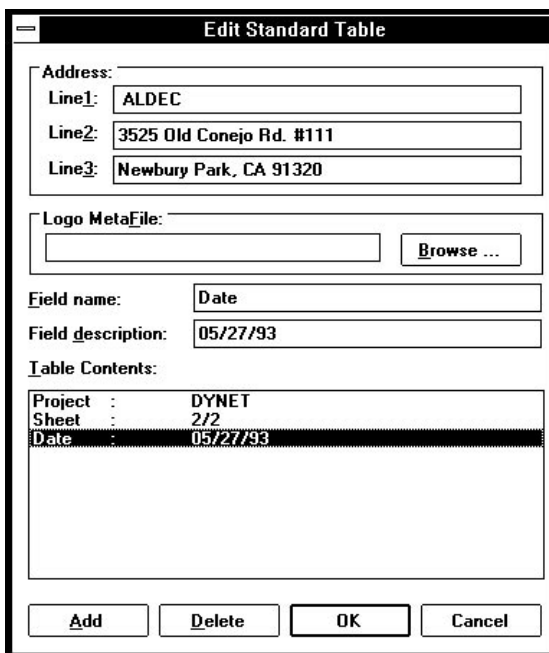


Figure 5-6. Table Setup Window.

The **Table Setup** option is used for configuration of the the design table (also called title block by some) that is displayed and printed in the bottom-right hand corner of each schematic sheet. You can select the desired logo graphics file to be displayed in the design table.

NOTE: The logo file has to be in the Windows Metafile format **without the header**. A Windows Metafile can be created with programs like CorelDraw.

Exit

The **Exit** option exits the Schematic Editor. You will be prompted to save all modified and un-saved schematic sheets. If the simulator is active, exiting the schematic editor will automatically disconnect the simulator-schematic link. After closing the Schematic Editor, the simulator will continue to operate as a stand-alone application.

Edit Menu

Edit	Mode	Options	Hierarchy	View	Di
Undo	Ctrl+Z, Alt+Bksp				
Redo	Ctrl+A, Alt+Shift+Bksp				
Cut	Ctrl+X				
Copy	Ctrl+C				
Paste	Ctrl+V				
Delete	Del				
Deselect All	Ctrl+U				
Show Clipboard					
Export Clipboard ...					
Import to Clipboard ...					
Redraw Wires	Ctrl+W				
Autorouting Table					

Figure 5-8. Edit Menu.

Undo and Redo

All editing operations are recorded so that you can undo the last changes, in case you made a mistake. The **Undo** option will undo all operations since the last change of editor mode. The **Redo** option restores the **Undo** changes (it undoes the **Undo** option).

Cut

The **Changes** mode allows selecting a group of items for editing. The **Cut** option deletes the selected group from the schematic and places it in the clipboard. If the selected items are connected to other components on the schematic, the connecting wires will also be deleted. However, the wires

within the group will stay intact. The items in the clipboard can be placed in another part of the same schematic or on another sheet using the **Paste** command. You can view the items in the clipboard using the **Show clipboard** option.

Copy

The **Copy** option is similar to the **Cut** option, but it does not delete anything. **Copy** places the selected group of items in the clipboard but does not delete them from the schematic.

Paste

The **Paste** command places the items from the clipboard into the current schematic.

Delete

The **Delete** option removes the selected group of items from the schematic. This function can also be performed by pressing the **Del** key. The deleted items are not placed in the clipboard and can be restored only by using the **Undo** operation.

Deselect All

The **Deselect All** option deselects all items previously selected for editing. You can also deselect all items by clicking with the left mouse button on any blank schematic area.

Show Clipboard

The **Show Clipboard** option is used to display the clipboard, which will contain the last cut or copied item. You can edit the clipboard schematic the same way as a regular schematic after you add it to the current project (**Hierarchy | Add Sheets To Project** option) or after you paste it on the current schematic sheet.

To add the clipboard schematic to a schematic sheet, select the **Edit/Paste** option.

Export Clipboard

The **Export Clipboard** option saves the contents of the clipboard as a schematic file. This file can later be loaded for editing as a regular schematic file.

Import to Clipboard

The **Import to Clipboard** option loads the selected schematic file to the clipboard. This schematic can then be added to any other schematic using the **Paste** command.

Clicking on the **Import To Clipboard** option displays the **Import To Clipboard** window. Double-clicking on the **Projects** directory in the **Directory** field displays all schematic sheets in the **File Name** field. Double-clicking on any of these schematic sheets will automatically transfer it to the clipboard.

Redraw Wires

The **Redraw Wires** option redraws every wire connected to the selected symbol. Using this option (or **Ctrl+W** keyboard shortcut) is suggested when wires look untidy after moving the symbol.

Autorouting Table

This option is described in the *Automatic Table Driven Connections* section of the *ACTIVE-CAD Schematic Editor* chapter.

Mode Menu

<u>M</u> ode	<u>O</u> ptions	<u>H</u> ier
✓ Select and <u>D</u> rag		F2
<u>S</u> ymbols		F3
Draw <u>W</u> ires		F4
Draw <u>B</u> uses		F5
Draw Bus <u>T</u> aps		F6
<u>Q</u> uery		F7
Test Points		F8
<u>G</u> raphics		

Figure 5-9. Mode Menu.

The **Mode** menu directly corresponds to the schematic tool bar. You can use either one to switch to the appropriate mode.

Options Menu

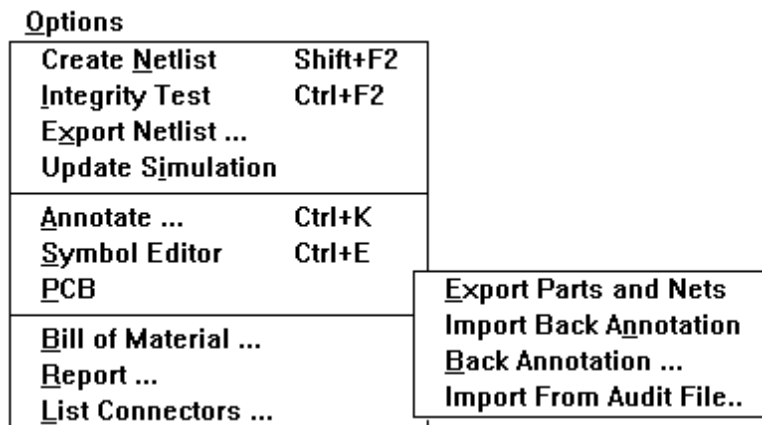


Figure 5-10. Options Menu.

Create Netlist

The **Create Netlist** option generates a binary netlist file that can be converted to one of the ASCII formats such as Future Net, PADS, Schema, etc. The binary netlist is used to link the Schematic Editor with its background Simulator when the Simulator is opened for the first time.

The netlist is stored with the project file and is updated every time you invoke the Simulator while editing a schematic. If you start the Simulator without the Schematic Editor active, you need to make sure that the netlist is up to date.

The **Create Netlist** option connects all sheets specified in the project contents. If any errors are detected, you will be notified about the location and type of error. Examples of netlist errors are: duplicated parts and terminal names, and unnamed components. Most of the electrical errors are checked while you draw the individual schematic sheets. The errors detected during the netlist generation are related only to the multiple sheet connections, which are not detected when individual sheets are drawn.

Integrity Test

The **Integrity Test** option performs a comprehensive analysis of the project netlist. It does not create a netlist but shows all design errors such as electrical violations and design inconsistencies.

Export Netlist

The **Export Netlist** option allows you to create an ASCII netlist in one of the following formats:

- ☐ SUSIE ASCII (SUSIE 6)
- ☐ CALAY
- ☐ CUPL
- ☐ EDIF 2.0.0
- ☐ FUTURENET
- ☐ INTUSOFT SPICE
- ☐ LATTICE
- ☐ MASSTECK ASCII
- ☐ MENTOR
- ☐ PADS
- ☐ P-CAD[* .NDE]
- ☐ P-CAD[* .ALT]
- ☐ QUICKLOGIC
- ☐ RACAL-REDAC
- ☐ SCICARDS
- ☐ SPICE
- ☐ TANGO
- ☐ XILINX

Clicking on the **Options | Export Netlist** option opens the **Export Netlists** window. Select the netlist file to be exported from **Input Netlist** field, and the format of the exported netlist from the **To Format** field. All project netlist files have the .ALB extension. The exported netlist will be written to a file with the same base name as the project netlist and with an extension name that corresponds to the selected export format. It will be placed in the same directory as the project netlist. The **Flat** option allows you to flatten the hierarchical designs. Otherwise, a separate netlist is created for each hierarchical macro, and the project hierarchical netlist will be comprised of top level netlist having the projects name and individual netlists for each hierarchical macro.

Update Simulation

After the Schematic Editor and the background Simulator have been running for a while, and numerous changes have been made, the system can start to slow down and memory allocation can become less efficient, since changes are saved incrementally. The **Update Simulation** option optimizes the interactive design changes by performing the following operations:

1. Saving Test Vectors and simulation results.
2. Saving the current schematic.
3. Creating a new netlist and loading it to the simulator.
4. Loading test vectors and previous simulation results.
5. Restoring and updating schematic probes.

After this procedure is performed, ACTIVE-CAD will operate faster and more memory will be available in the system.

Annotate

Clicking on this option displays the **Annotation** window allows sequential renumbering of all project components. All schematic sheets are affected, including the closed ones but included in the project contents. For more details refer to the **Annotation** in the *Placing Symbols* section of Chapter 2.

Symbol Editor

This option loads the Symbol Editor program which is described in more detail in the **Symbol Editor** Chapter 6.

PCB / Export Parts and Nets

The **Export Parts and Nets** option is used to load schematic designs into PCB layout programs, using the IPC protocol (DDE). The PCB program receives on-line all components and net information and therefore no file transfer is necessary. If the PCB layout program is not active, the information is exported to an audit file which can then be loaded into it off-line.

PCB / Import Back Annotation

The **Import Back Annotation** option is used to receive on-line ECO changes from the PCB layout programs, via the IPC protocol (DDE).

PCB / Back Annotation

The **Back Annotation** option reads the back annotation files from the PCB layout programs and updates the schematic. You can use the back annotation files from Massteck, Racal-Redac, Schema, PADS, Tango and PCAD. When you select a back annotation file, ACTIVE-CAD will update changes such as pin swaps, section swaps, package, name and type changes.

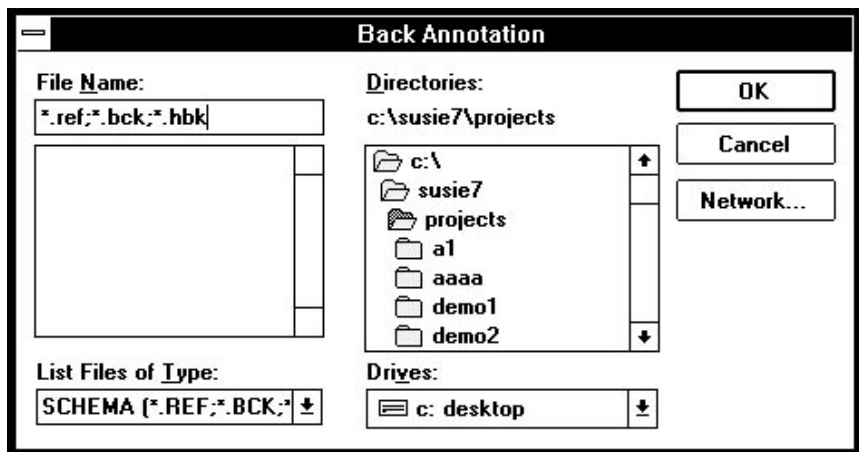


Figure 5-11. Back Annotation Window.

PCB / Import from Audit File

The **Import From Audit File** option is used to load the changes recorded by the PCB program to a file. The Audit files are used when the on-line IPC interface is not efficient because of memory limitations or if the PCB program runs on a different computer from the one with the schematic.

Bill of Material

The **Bill of Material** option generates reports about the components used in your design. This report can be used for ordering parts for production, cost calculations, etc. You can generate either an **Engineering** or **Manufacturing Summary**. The **Engineering Summary** produces a detailed description of the devices used, their location on the schematic, etc. The **Manufacturing Summary** displays device stock numbers and other information that is needed for procuring the parts.

Bill of Material

Choose Bill of Material options :

☒ Engineering Summary
☐ Manufacturing Summary

☒ Location
☒ Description
☐ Comment

☒ Totalled
☐ Separate lines

☒ No sections
☐ List sections

Create

Edit Cancel

Figure 5-13. Bill of Material Window.

To create the report press the **Create** button in Figure 5-13. After the report is created it can be printed or edited by pressing the **Edit** button. This loads the report to the Windows Notepad, shown in Figure 5-12.

PART	USED	NAME	REFER	DESCRIPTION	COMMENT
1	1	22V10-5	U6	22V10 - Programmable Array Logic Device	
2	1	27S281	U17	27S281 : 1k*8bit TTL PROM.	
3	1	54ALS162	U24	xxx162 - Synchronous BCD Counters With Clear	
4	1	54ALS374	U25	xxx374 - OCTAL D-TYPE EDGE-TRIGGERED FLIP-FLOPS	
5	1	7474	U2A	xxx74 - Dual D-type positive-edge-triggered Flip-Flop with asynchronous preset and clear	
6	1	7485	U16	74xx85 - 4-Bit Magnitude Comparators	
7	1	74ALS08	U5A	74/5408 - Quadruple 2-input positive-AND standard TTL gates.	
8	1	74ALS161	U11	xxx161 - Synchronous Binary Counters With Direct Clear	
9	1	74ALS175	U10	xxx171, xxx175 - Quadruple D-type flip-flops with asynchronous clear	

Figure 5-12. Editing Bill of Material Report.

Report

The **Report** option allows exporting design information in the ASCII format. This file can then be used with any database or spreadsheet program to create a customized Bill of Material report. For example, you can create a bill of material using the Microsoft Access software. You can select the attributes that you want to export and their order, as shown in detail in Figure 5-15 .

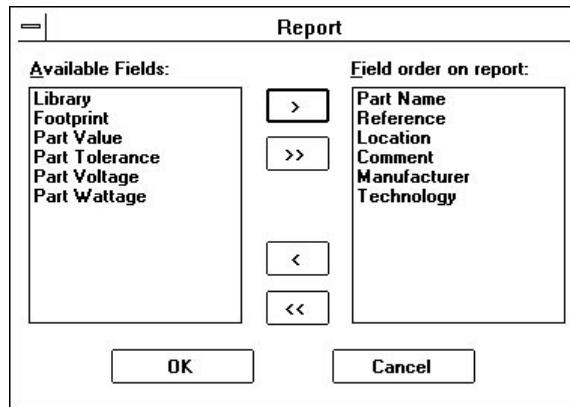


Figure 5-15. Creating Report File.

List Connectors

The connectors from the **SC Symbols** toolbox are placed separately on the schematic. To generate a summary of all connector pins, use the **List Connectors** option (Figure 5-15).

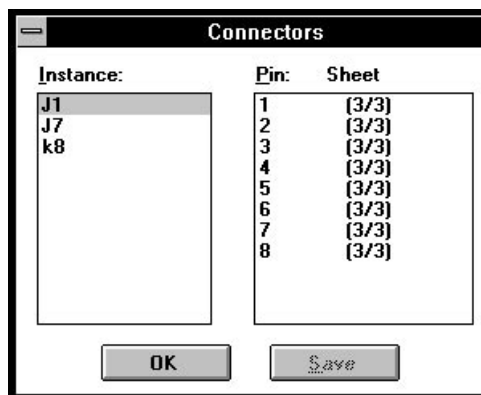


Figure 5-14. List Connectors Window.

Hierarchy

Hierarchy	View	Display	Window
Add Sheets to Project			
Add Current Sheet to Project			
Create Macro Symbol From Current Sheet			
New Symbol Wizard ...			
Hierarchy Push			Ctrl+H
Hierarchy Pop			Ctrl+I
Import Netlist ...			
Assign Netlist			

Figure 5-17. Hierarchy Menu.

The **Hierarchy** menu is described in detail in the *Managing Schematic Designs* chapter.

View Menu

View	Display	Window
Preferences		
✓ Status Line		Colors
Ruler		Text
✓ Table		Display
✓ Graphics		Grid
✓ Symbol Text		Wires and Buses
✓ Probes		General
✓ Beep		✓ Save Settings on Exit
CrossHair Cursor		Save Settings Now
Grid		
✓ Snap to Grid Ctrl+G		
Bus display		
Scanner		
Move Scanner		

Figure 5-16. View Menu.

The **View** menu enables and disables the following items:

- ☐ **Status Line** with the current cursor position and messages
- ☐ **Ruler** display along the top and left side of the window

- ☐ **Table** with the design information table, at the schematic bottom
- ☐ **Graphics** (circles, lines, comments, etc.)
- ☐ **Symbol Text** (pin names and numbers)
- ☐ **Probes** on/off control (logical states display)
- ☐ **Beep** for illegal editing operations
- ☐ **Cross-Hair Cursor** (very large cross-hair cursor)
- ☐ **Grid** (grid display on the schematic background)
- ☐ **Bus Display** for selecting the bus display format (hex, binary, etc.)

Preferences

The **Preferences** menu options allows you to customize the look of the schematic editor windows. You can select any window element and set the font, line style, color, or enable / disable options. Some users refer to these options as the Edit Library.

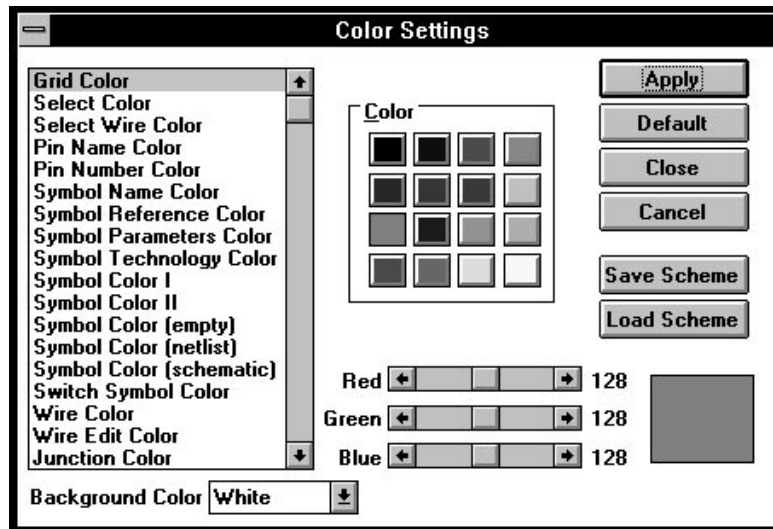


Figure 5-18. Color Preferences window.

Colors

The **Color Settings** window (Figure 5-18) allows selection of colors for different schematic objects. Each schematic element is listed in that window. Clicking on any of these items displays its current color setup in the bottom right-hand corner. To change this color, click on the appropriate button in the **Color** field. The fields **Red**, **Green** and **Blue** allow for precise tweaking of the colors by regulating the amount of these basic colors in the selected (by the **Color** button activation) color.

Upon completing the color setup of the schematic element, click on the **Apply** button. You can save the current settings to a file using the **Save Scheme** button. To restore the proprietary color setups, click on the **Load Scheme** button. Exit the color setups by clicking on the **Close** button. If you click on the **Cancel** button, ACTIVE-CAD will ignore your current setups and will return back to the previous color setups.

Screen background color

The **Background Color** field in Figure 5-19 allows you to select some basic background colors used in the industry. Always start your color selections with the background color because it affects the readability of all other colors on the schematic sheet.

Symbol colors

ACTIVE-CAD has some default colors for various symbols:

- ☐ a *black* symbol indicates that it has a built-in (C) model
- ☐ *blue* denotes a symbol that has a simulation (VHDL) model
- ☐ a *brown* symbol outline indicates an empty symbol

To restore the default colors set at the factory, click on the **Default** button.

Text

You can assign different font sizes and attributes to selected text items with the help of the **Text Settings** window. Figure 5-19 shows the available settings.

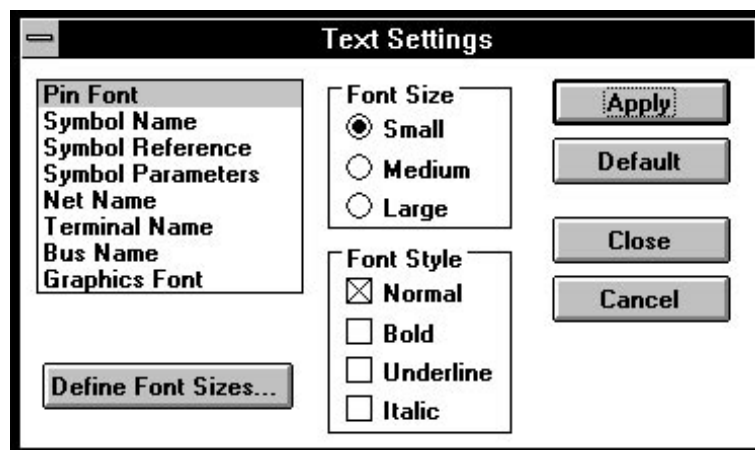


Figure 5-19. Text Preferences window.

Customizing the font size

ACTIVE-CAD has three (3) font sizes: small, medium and large. You can assign any of these fonts to pins, symbol names and reference numbers. The height of the small, medium and large fonts (expressed in pixels) for the entire schematic is set in the **Define Font Sizes** window (Figure 5-23) invoked with the **Define Font Sizes** button in the **Text Preferences** option.

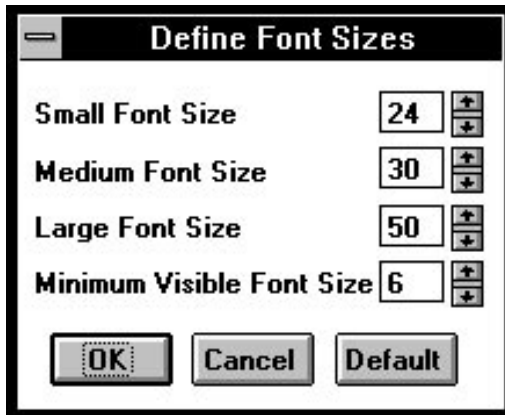


Figure 5-20. Setting the schematic font sizes.

Minimal visible font size

As you rescale the schematic drawing, the fonts change in height and at some point they may become unreadable and obstruct the drawing. To avoid this, you can set the minimal visible font size by selecting the **Minimal Visible Font Size** option in the **Define Font Sizes** window.

NOTE: Set the **Minimal Visible Font Size** to a lower value when your display does not show the text in small zoom scales. The typical setting is between 5 and 11, and depends upon your display card resolution.

Display

The **Display Configuration** window (Figure 5-21) allows controlling the appearance of the Status Window and Ruler marker.

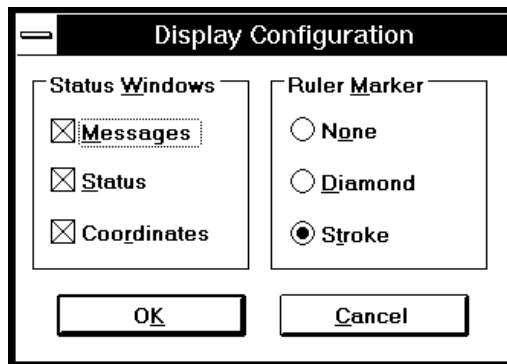


Figure 5-21. Display Configuration Window.

Grid

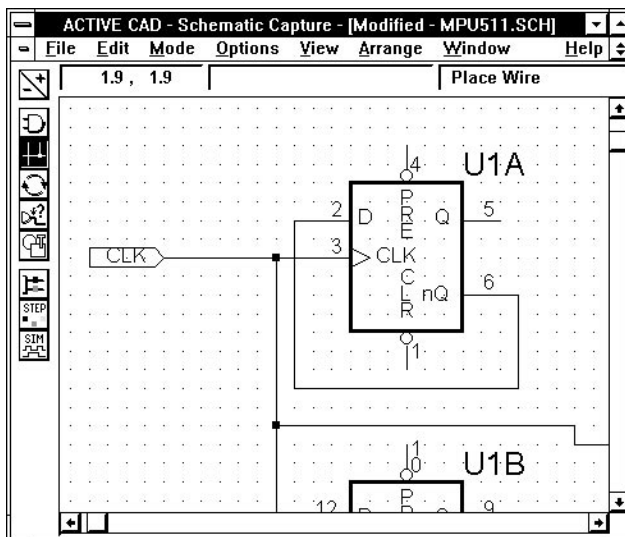


Figure 5-22. Schematic Grid.

The schematic grid can be helpful when you draw wires and place symbols. It allows you to align the schematic elements with the horizontal and vertical lines displayed in the background.

The **Grid Configuration** (Figure 5-23) window allows you to turn the grid on and off, and set the spacing and style. To activate or de-activate the grid, use the **Grid Active** check box. The snapping of the drawn elements to the grid will take place independent of whether the grid has been selected for display.

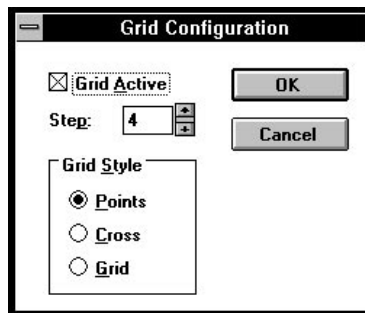


Figure 5-23. Grid Configuration Window.

To choose the grid resolution, select the appropriate grid **Step** by editing the **Step** field or using the up and down arrows. The resolution in inches is determined by multiplying the number of grid steps by 0.025". So a Step of 3 will result in a resolution of 0.075". Since the smallest Step you can enter is 1, the smallest grid resolution is 0.025 inches.

The **Grid Style** allows you to select either **Points** (dots), **Cross** or **Grid**. Choose the grid that is most suitable for the work you do. The **Grid Configuration** can be used to enable and set the minimum distance between the wires. By default, the minimum distance between wires is 1/8" and can be increased by the user.

Wires & Buses

The **Wires & Bus Settings** window allows you to set the preferred options for drawing wires and buses on the schematic.

Wire Line Style

This option allows you to select either **Thick**, **Solid**, **Dash**, **Dot**, **Dash Dot** or **DashDotDot** line style for single wire connections. Since the most popular setup is **Solid** for individual wires, this is the default ACTIVE-CAD setup. However, you can use any other wire style.

Wire Line Width

You can set the width of single wires on the schematic to suit the company policy and/or your preferences.

Junction Size

It is used for determining how big will be the wire junctions.

Junction Type

It allows to select either square or circle wire junctions.

Bus setups

Use the **Bus Line** (line thickness) and **Bus Tap Type** (geometry) options to set the specification for all buses. You can dynamically change the tap geometry on the schematic but the bus line thickness can be changed only from this window.

Orthogonal Wires

This option forces all lines to be drawn at right angles, which is a standard with electrical schematics. The **Orthogonal Wires** option should be turned on (checked) for all schematics, unless special conditions prevail.

Autorouting

ACTIVE-CAD is an advanced schematic editor which has an autorouting feature. This feature may save you in excess of 50% of time when drawing complex drawings with extensive wiring. Since the autowiring algorithm may take a long time in a cramped schematic area, you may wish to terminate it and complete some wire segments manually.

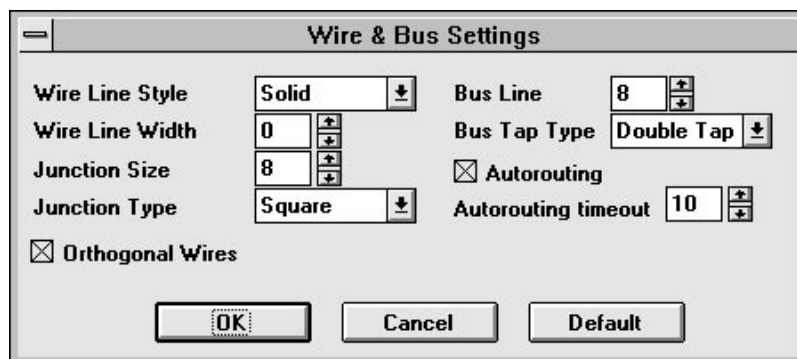


Figure 5-24. Wires and Buses Preferences.

The **Autorouting Timeout** option allows you to set the time for the autorouting algorithm to terminate. Typical user setups are between 6 seconds to 10 seconds.

General

The **General Settings** window in Figure 5-26 is used for controlling some of the basic operational modes of the schematic editor and simulator.

Allow symbol Overlap

ACTIVE-CAD by default allows overlapping of symbol outlines. By unchecking the **Allow symbol overlap** option, you will not be able to place new symbol if its outline overlaps with the other outline.

Interactive simulation mode

ACTIVE-CAD operates by default in the interactive schematic-simulator mode, which means that you do not need to generate a netlist each time you modify the schematic because the simulator database is updated incrementally with each design entry. If you disable the interactive simulation mode, you need to activate the **Update Simulation** option in the **Options** menu before again invoking the simulation process.

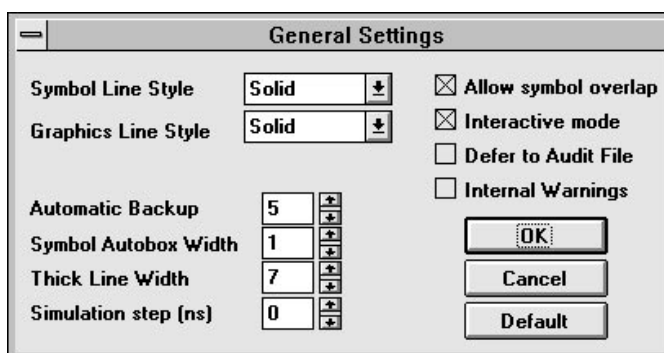


Figure 5-25. General Preferences

Defer to Audit File

This option saves all schematic changes to an audit file with the AUD extension in the project directory. These changes are in the ECO format and they include operations like add symbol, delete symbol, add connection, add net name, etc. These changes can be used for automatic forward annotation of PCB designs and for documentation purposes.

NOTE: Select the **Defer To Audit File** option if you need to keep track of all new design changes that need to be added to the current p.c.b. layout.

Automatic Backup

By default, all edited schematics are saved to a backup file every 5 minutes. The backup files have the .BSC extensions. You can change the setting for automatic backup from 0 seconds (no backups) to saving designs every 60 minutes.

Simulation step

The ACTIVE-CAD schematic editor has a patented on-line background simulator. The operation of this simulator can be controlled from the **Probes** toolbox, which has a **Step** button for controlling the duration of the simulation process. The **Simulation step** option allows to set the duration of the simulation step from 1 nanosecond to 999 nanoseconds.

Controlling schematic graphics

The **General Settings** option has five (5) parameters that control the schematic graphics, including symbol outline thickness and type:

- ☐ **Symbol Line Style** controls the thickness of the symbol outline and its pins. It can be set and reset at any time during schematic editing and it has instant effect on all schematic symbols.
- ☐ **Graphics line Style** controls the line style in the graphics mode, activated by the button 14 in Figure 2-1.
- ☐ **Symbol Autobox Width** sets the thickness of the hierarchical blocks outline. It does affect the symbols from the system library or made in the symbol editor.
- ☐ **Thick line width** is used for controlling the width of the lines drawn in the graphics mode, activated by pressing the button 14 in Figure 2-1.

Save Settings Now

The **Save Settings Now** option saves all schematic settings and window sizes to the SUSIE.INI file.

Save Settings on Exit

The **Save Settings on Exit** updates the schematic parameters and settings when exiting the program. When this option is on, the current window sizes and preferences are saved in the SUSIE.INI file.

Status line

The **Status** line displays the current position of the cursor and schematic editor messages. This line can be disabled to allow more space for the schematic sheet.

The **Status Line** aids you in understanding what the schematic editor is doing or will do. You need to have this line enabled when adding bus taps, performing autorouting and similar time consuming operations.

Ruler

The **Ruler** can be disabled to allow more space for the schematic sheet. The ruler settings are available in the **Page Setup** window, which is activated by clicking on the **Page Setup** option in the **File** menu. You can set the ruler to **English**, **Metric**, **References** and **None**. If the **Ruler** option in the **View** menu does not display any ruler or displays the wrong ruler, check the **Page Setup** window for the actual ruler setups.

The ruler marker is selectable from the **View/Preferences/Display** option. When selected, this option displays the **Display Configuration** window which allows you to set the ruler marker to either **Diamond**, **Stroke** or **None**.

Table

The design table is enabled by default because it is required by most corporations. The table, which is located in the bottom-right hand corner of the schematic sheet, typically contains the company information and logo, the project or design name and sheet number. If the table is not required, you can disable its display. The design table contents is specified in the **Table Setup** window of the **File** menu.

Graphics

You can disable the graphics and text display to speed redraw of a schematic. All existing graphics or text items will show up on the screen only if you enable this option again.

The **Graphics** option in the **View** menu enables the buttons of the **Graphics** toolbox, activated by pressing the button 14 in Figure 2-1. If the **Graphics** option in the **View** menu is inactive, you will not be able to draw any graphics elements on the schematic, except for the text.

Symbol Text

Symbol Text, such as part names and types, and pin names and numbers, are displayed as long as the zoom is large enough to clearly display the fonts. By using the **Symbol Text** option, you can disable the text display, and thus save time when redrawing a design.

The size of the text is set in the **View | Preferences | Text** option, which activates the **Text Setting** window. You can assign to each text type a spe-

cific font category (small, medium and large) and exactly control its size with font adjusting buttons.

You can control the minimum visible font size by adjusting the font size in the **Minimum Visible Font Size** field, located in the **Define Font Sizes** window. This window is activated by clicking on the **Define Font Sizes** button in the **Text Setting** window (activated by the **View | Preferences | Text** option).

NOTE: Do not set the minimum visible font too large because symbols may lose their text. The recommended minimum font size is 5.

Probes

When the simulator is active, all selected simulator probes and stimulators are displayed next to the pins and I/O terminals. By selecting the **Probes** option, you can toggle on/off the display of these probes.

Scanner

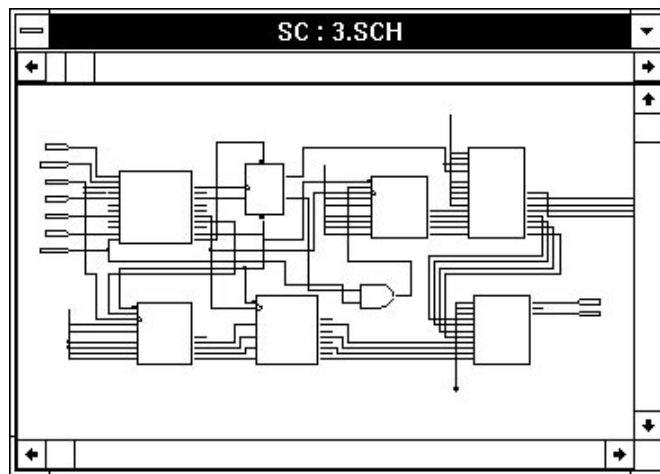


Figure 5-26. The Scanner Window.

The **Scanner** option allows you to simultaneously view the entire schematic in a smaller window superimposed over the original schematic. The **Scanner** window, shown in Figure 5-26, has an extra scroll bar at the top that can be used to set the scaling factor for this secondary display window. This bar allows you to minimize and enlarge the schematic view. This is very useful while editing large schematic sheets because

you can view the entire sheet in the **Scanner** window and edit details in the primary schematic window.

The scanner can also be used to magnify the schematic areas. The scanner is only for viewing the schematic designs and you cannot edit the scanner schematics.

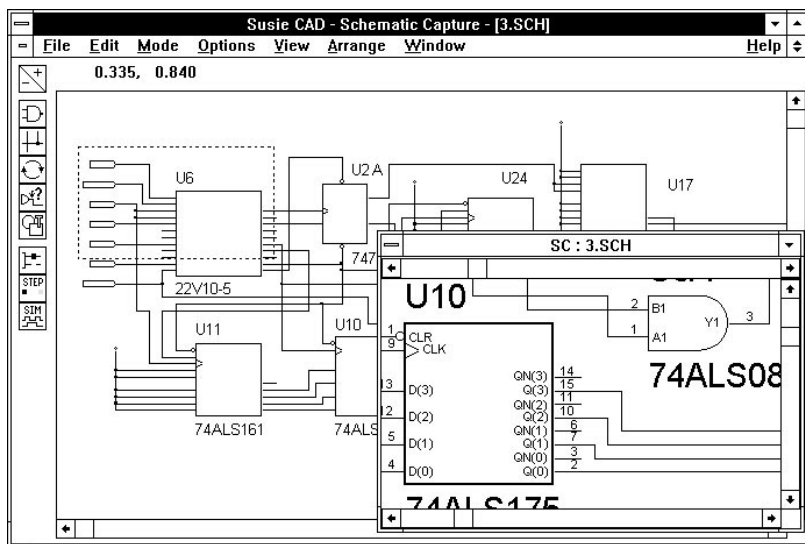


Figure 5-27. Example of Magnifying Scanner Window.

Move Scanner

The **Move Scanner** option allows select the scanner display area . When you select **Move Scanner**, a rectangle representing the scanned area is following the cursor. Click the mouse to select the section to be moved into the **Scanner** window. You can set the scanner window to enlarge or reduce the schematic scale.

If needed, the **Scanner** window can be moved outside of the main schematic editor window.

Display Menu

<u>D</u> isplay	<u>W</u> indow
<u>Z</u> oom <u>I</u> n	Ctrl +
<u>Z</u> oom <u>O</u> ut	Ctrl -
F <u>u</u> ll <u>P</u> age	PgDn
P <u>r</u> ev <u>i</u> ous <u>Z</u> oom	PgUp
<u>Z</u> oom <u>S</u> elect	▶
<u>C</u> enter	F9
<u>R</u> edraw	F10

Figure 5-28. Display Menu.

The **Display** menu (Figure 5-28) allows you to display a schematic at a desired scale and screen position. Note that in addition to the menu commands, you can use the dynamic zoom to select the block displayed in the window (described in the section *Viewing a Schematic* in Chapter 2). You can also use the scroll bars to scroll the schematic horizontally and vertically.

Zoom In and Zoom Out

The Schematic Editor allows continuous scaling of the edited schematic sheet, so that you can view it at any desired scale. However, for faster operation you can use the **Zoom In** and **Zoom Out** options, which are equivalent to the **Zoom** buttons in the schematic toolbar.

Full Page

Displays the entire schematic in the active window.

Previous Zoom

Returns to the previous zoom level.

Zoom Select

Quickly switches to one of the predefined zoom settings from 1:5 to 5:1.

Center

The **Center** option or the **F9** key is used to select the central point of the schematic window. When you select this option, a special cursor will appear. Pointing this cursor at a schematic location will move that location into the center of the window. Note that the selected location will remain in the center of the window, even if you change the scale using the **Zoom In** or **Zoom Out** commands.

Redraw

The editor does not redraw the screen after every operation because it would slow down such operations as editing, moving and dragging. Parts of the screen can sometimes become unclear, so you need to periodically redraw the screen to keep it free of spurious elements. To redraw the screen use the **Redraw** option or press the **F10** key.

Window Menu

Window	
Cascade	Shift+F5
Tile	Shift+F4
Arrange Icons	
Close All	
1 Netlist	
✓ 2 CALC.SCH	

Figure 5-29. Window Menu.

The **Window** menu allows you to arrange the schematic windows within the main Editor window.

Cascade

The **Cascade** option arranges the windows so that they overlap, but you can see all of the Window titles. The active window is on top. This allows you to quickly select the desired schematic sheets.

Tile

The **Tile** option arranges the windows so that they do not overlap, and they are approximately the same size. This arrangement of windows is useful when several sheets need to be viewed at the same time.

Arrange Icons

The **Arrange icons** option allows arranging the minimized windows icons in the main Schematic Editor window. This option arranges only the minimized schematic windows and not the open or maximized ones.

Close All

The **Close All** option closes all schematic windows. You will be notified about all modified but not saved sheets. Proceed with caution when using this option.

Chapter 6

Symbol Editor

The Symbol Editor allows you to create new symbols and modify the existing ones. To invoke the Symbol Editor, double-click on the **Symbol**

Editor  icon in the Project Manager toolbar or select the **Symbol Editor** option from the schematic **Options** menu.

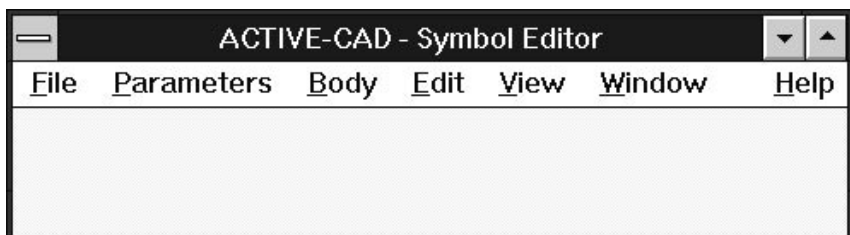


Figure 6-1. Symbol Editor First Screen.

Creating a new symbol

Select the **Symbol Editor** option from its icon or the schematic **Options** menu. In response, the window in Figure 6-1 will appear. Selecting the **New** option in the **File** menu, will display the **New Symbol** window (Figure 6-2). This window lists all user libraries which can store the user generated symbols.

NOTE: The user generated symbols cannot be stored in the system libraries provided on installation diskettes or CD ROM.

The user designed symbols can be stored within projects and user libraries, which can be created with the help of the *Library Manager* (Chapter 7).

To create a new symbol, enter the new symbol name in the **Symbol** field. Clicking on a library name in the **Libraries** field will select the new symbol library name.

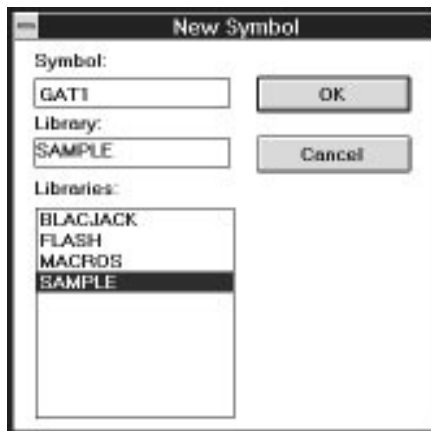


Figure 6-2. *New Symbol Window.*

Next, click the **OK** button. In response, ACTIVE-CAD displays the **Symbol Editor** window. The **Symbol Editor** window operations are described in detail in the *Editing an existing symbol* Section, below. The operations for editing an existing symbol and creating a new one are similar.

Editing an existing symbol

Editing library symbols

Select the **Symbol Editor** option from the **Options** menu. When ACTIVE-CAD displays the screen in Figure 6-1, select the **Open** option in the **File** menu and the **Edit Symbol** window in Figure 6-3 will be displayed. This window allows you to select for editing the desired part from any library listed in that window.

Double-clicking on the library name in the **Libraries** field will list all library symbols in the **Symbols** field of Figure 6-3.

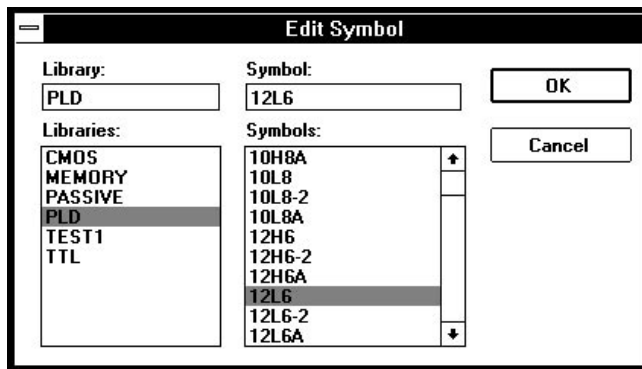


Figure 6-3. Open Symbol Window.

Double-clicking on the desired symbol in the **Symbols** field will activate the **Symbol Editor** window in Figure 6-4 and will display the selected symbol. You can select a symbol for display in the **Symbol Editor** window by clicking on the desired symbol in the **Symbols** field and then clicking on the **OK** button.

Editing of schematic symbols

You can select any schematic symbol for editing in the **Symbol Editor** window. First, select the **Select** mode and click on the symbol to be edited. The selected symbol will be highlighted by a red rectangular frame. Next, select the **Symbol Editor** option from the **Options** menu. In response, ACTIVE-CAD instantly displays the **Symbol Editor** screen in Figure 6-4.

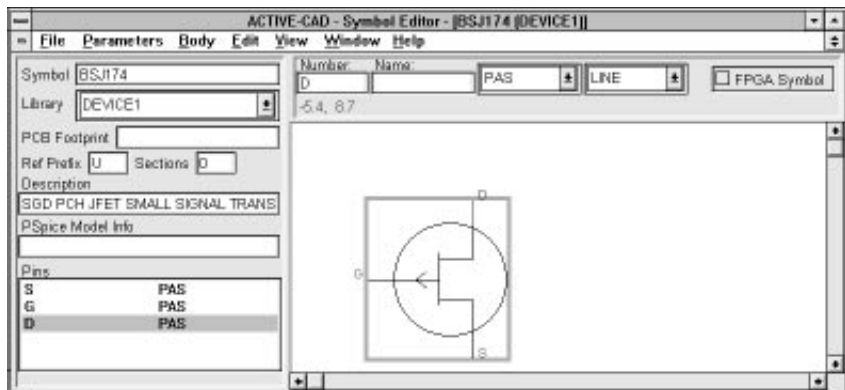


Figure 6-4. Symbol Editing Screen.

NOTE: You can edit any schematic or library symbol but you can store it after editing only in the user library.

Description of Symbol Editor Window

The following is a description of the **Symbol Editor** fields in Figure 6-4:

- ☐ **Symbol** - symbol name under which the graphical symbol is stored; it can start with any character and can be up to 8 characters long.
- ☐ **Library** - name of the library in which the original symbol resides.
- ☐ **PCB Footprint** - name of the p.c.b. layout footprint that is used as the default parameter when exporting a netlist to a p.c.b. program (e.g. PADS, TANGO, P-CAD, Massteck).
- ☐ **Ref. Prefix** - default reference prefix, used for numbering parts placed on a schematic, e.g. U, R, C, L, D, Q, etc.
- ☐ **Sections** - indicates how many sections the symbol contains. For example, 7400 has four 2-input NAND gates, shown as separate sections.
- ☐ **Description** - describes the part function, e.g. 4Kx8 RAM, 2-input AND gate, counter, J-K flip-flop, etc.
- ☐ **Spice Model Info** - analog simulation parameters.
- ☐ **Pins** - list of symbol pin numbers and the associated names.
- ☐ **Graphical symbol display** - the graphical display of the symbol in a blue rectangular outline.
- ☐ **Pin parameters** - there are four fields above the graphical symbol display (**Number**, **Name**, signal type and graphical representation) which display the key parameters of the selected pin.

Symbol Text Editing

Changing pin location

To change the pin location within the symbol, click on the desired pin in the graphical symbol display, and holding down the left mouse button, drag the desired pin to its new location. The edited pin is displayed in red.

Changing pin numbering

To change the pin number, click on the desired pin in the graphical symbol display area, and when the pin turns red, its number will appear in the **Number** window. Place the cursor in the **Number** window and enter the new pin number. The new pin number will instantly be shown on the graphical symbol outline. Double-clicking anywhere in the graphical screen area will make this change permanent.

Changing a pin name

To change the pin name, click on the desired pin in the graphical symbol display area, and when the pin turns red, its name will appear in the **Name** window. Place the cursor in the **Name** window and enter the new

pin name. The new pin name will instantly be shown on the graphical symbol outline. Double-clicking anywhere in the graphical screen area will make this change permanent.

Changing the pin's I/O designation

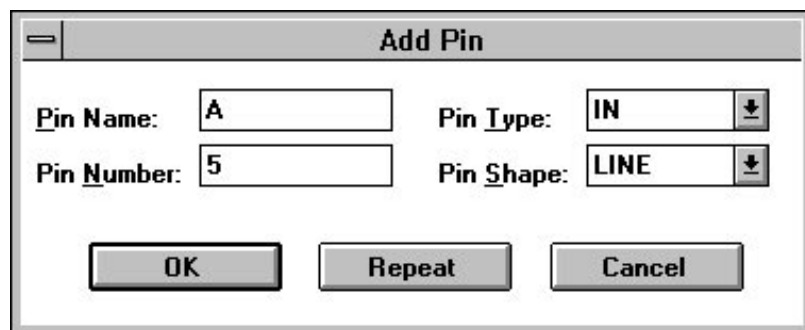
To change the pin's I/O designation, click on the desired pin and when the pin turns red, its current I/O designation will appear in the **I/O** field, located to the right of the **Name** field. Click on this window to display the I/O options like IN, OUT, OC, OE, etc. Select the desired I/O type and double-click the mouse button anywhere in the graphical symbol display. The pin's I/O type will be instantly updated in the pin list.

Changing pin graphics

To change the graphical pin display, click on the desired pin and when the pin turns red, its current graphical name will appear in the window to the right of the **I/O** field. Click on this field to display the graphical options like **LINE** (a straight line), **CLK** (a triangular clock marker), **DOT** (a line with a circle), etc. Select the appropriate graphical pin representation. Double-clicking anywhere in the graphical screen area will make this change permanent.

Adding symbol pins

To add a new pin, either select the **Add Pin** option from the **Body** menu or click on the **Pin** button in the **Symbol Editor** toolbox. This displays the **Add Pin** window (Figure 6-5). Enter the pin name, pin number, pin type and pin shape in the corresponding fields in Figure 6-5. Pressing the **Enter** key or clicking on the **OK** button, completes the pin description entry.



Add Pin	
Pin Name:	A
Pin Type:	IN
Pin Number:	5
Pin Shape:	LINE
<div>OK Repeat Cancel</div>	

Figure 6-5. Add Pin Window.

Notice that following the pin data entry, the cursor is automatically placed in the symbol outline and a red line is attached to the cursor.

Place the cursor at the desired pin location and click the mouse button. The new pin is instantly anchored at the selected location.

If the symbol outline needs to be modified, place the cursor over the blue symbol outline and when the double arrow cursor appears, press the mouse button and drag the symbol outline in the desired direction.

NOTE: You can save the edited symbol only in the USER libraries. You cannot save the edited symbol in the ACTIVE-CAD system (R/S) or Read Only (R/O) libraries. If you attempt to save an edited symbol in these libraries, ACTIVE-CAD will automatically issue the **Save as** prompt.

Pin editing options

ACTIVE-CAD allows typical Windows operations that make editing pins an easier task. These options are available in the **Edit** and **Body** menus of the **Symbol Editor**.

- Undo** - (**Edit** menu) cancels the last operation.
- Cut** - (**Edit** menu) deletes the pin and saves it to the clipboard.
- Copy** - (**Edit** menu) copies the pin to the clipboard without deleting it.
- Paste** - (**Edit** menu) places a pin from the clipboard.
- Delete** - (**Edit** menu) deletes the pin without saving to the clipboard.
- Add** - (**Body** menu) allows you to add a pin (Figure 6-5).

Editing Symbol Graphics

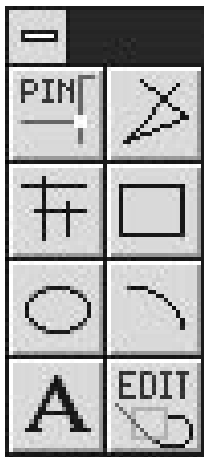


Figure 6-6. Symbol Editor Toolbox.

The Symbol Editor toolbox (Figure 6-6) allows you to select between various graphical elements such as lines, arcs, circles and squares. Instead of the toolbox, you can also use the **Body** menu to select the graphics element such as lines, arcs, circles and rectangles.

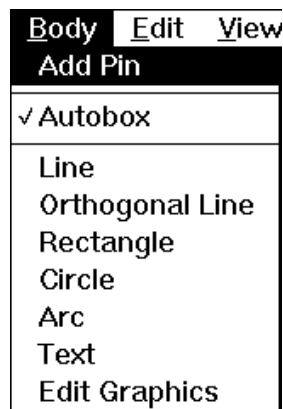



Figure 6-7. The Body Editing Options.


Before you start editing a symbol, you must set its size. Otherwise, if you run out of space, you will be forced into extensive changes of the symbol.

The buttons of the Symbol Editor toolbox are described below. The corresponding **Body** options are listed in Figure 6-7.

Entering pins


The  **PIN** button activates pin editing window. You can enter the pin name, pin number, pin type and pin shape. If you select the **Repeat** option, you will be able to place pins that have the same name but different indexes. Also, the pin number will increment automatically.

Orthogonal lines

When you activate the  **Orthogonal Lines** button, all lines will be drawn at the right angle. By clicking the mouse button at each corner, you will be able to draw more complex items.


You can edit only the area marked by the blue rectangle box. You must adjust this area to suite your needs.

Circles


The  **Circles** button activates drawing of circles. To draw a circle click the mouse button inside the blue rectangle and drag the mouse cursor. A red circle appears and grows as you move the mouse cursor away

from its initial position. Release the mouse button when the circle reaches the desired size.


Arcs

The  **Arcs** button activates drawing of arcs. Since arcs are based on two reference points, click at the first spot and then at the second spot and drag the mouse cursor. An arc based on the two spots will follow the mouse. Adjust the arc size and shape by dragging the mouse. When the desired shape has been created, release the mouse button.


Unrestricted lines

The  **Polyline** button allows drawing lines at any angle. Click the mouse button at each corner to anchor the line. Click the right mouse button to end the line editing.

Rectangles

The  **Rectangles** button allows drawing rectangles. Click at one of the corners of the rectangle and drag the mouse button to the opposite corner of the rectangle. When the rectangle reaches the desired shape and size, release the mouse button.

Graphical text

If there is a need for a non-electrical or graphical text, click on the  **Graphical Text** button. The text entered in this mode will have no bearing on the electrical symbol features and will not be included in any symbol data base, except for the graphics alone.

Do not use this text mode for naming pins, part numbers, etc., because this information must be entered as electrical data. Text that you enter with the other Symbol Editor options is automatically fed into the electrical data base and can be used for electrical circuit analysis and documentation.

Deleting graphics

To delete graphics items, select the  **Edit** button, click on the selected elements and when they turn red, press the **Delete** keyboard key.

Note: You can choose one of the three text sizes but you cannot change the thickness of the lines, circles and arcs. These are controlled by the ACTIVE-CAD schematic **Preferences** option.

Symbol parameters

With today's complexity of electronic designs, you must be able to assign multiple electrical parameters to pins, devices, wires, etc. To assign parameters to symbols, select the **Parameters** option in the **Parameters** menu.

In response, the **Symbol Editor** displays the Figure 6-8, which lists the previously defined symbol parameters in the left-hand window. To edit any of these parameters, select it with the mouse cursor and click the **Edit** button. ACTIVE-CAD will display an editing window (Figure 6-8) which will allow you to edit the current parameter name and its value. In addition, you can change the parameter display conditions:

- ☐ **Display All** - displays both the parameter name and description
- ☐ **Display Name** - displays only the name of the selected parameter
- ☐ **Display Description** - displays the parameter description alone.

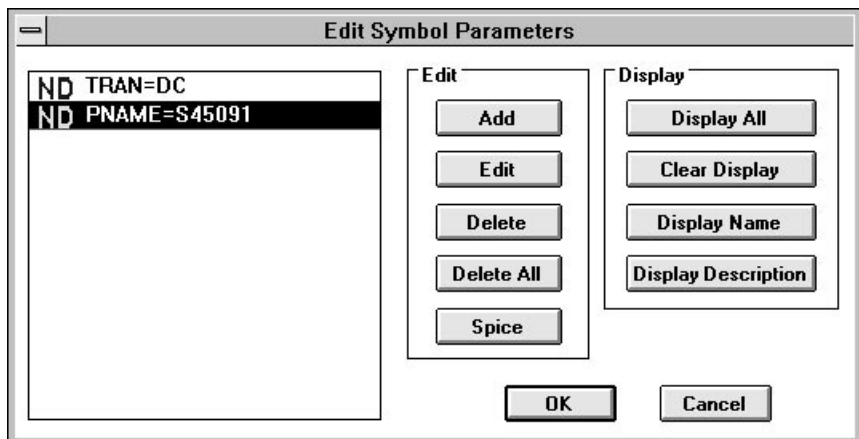


Figure 6-8. Editing Symbol Parameters

To add a new symbol parameter, click on the **Add** button. When the window in Figure 6-9 appears, enter the parameter name, its value and define the display conditions.

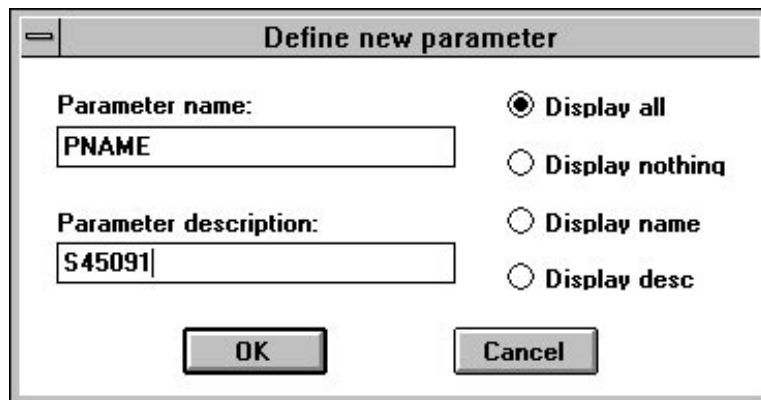


Figure 6-9. Adding New Symbol Parameter.

The symbol pin parameters are entered directly in the schematic editor. See *Pin Parameters* in the *Editing Schematic* Section (Chapter 2)

Testing symbols

The symbols can be as simple as a gate diode or as complex as 302 pin grid ASIC. Since there can be many errors created in the process of symbol editing, ACTIVE-CAD provides for quick verification of the symbol integrity.

To check symbol integrity, click on the **Test Symbol** option in the **File** menu or activate the **Ctrl-T** keyboard keys. The **Symbol Editor** will automatically check for duplicate pin name and numbers, and similar items, and will provide a list of errors. Always use this option before attempting to save a symbol.

Effective symbol editing

- ☐ Since editing graphics is the most difficult part of the symbol editing, start with the graphics and draw the arcs and circles. No pins should be added till the symbol graphics is perfect.
- ☐ Make the blue symbol outline as big as the screen. It will give you a great freedom in moving pins around. You can always reduce the symbol outline after the pins have been entered.
- ☐ Next, add the pins and their locations on the symbol body.
- ☐ Enter the symbol parameters. Decide their display modes carefully so that they will not cramp the schematic display, but at the same time they will provide all vital design information.

- ☐ Plan the user library structure well in advance so that you will remember where to find the symbols.
- ☐ Always delete the wrong symbols or copy them to the junk library for reference. Do not keep the current and outdated symbols together.

To save time on unnecessary editing, always start with a larger symbol (blue rectangle) than expected because increasing the symbol size incrementally is tedious and requires extra editing operations.

Symbol access priority

Sometimes you may have a hard time pulling a symbol to the schematic screen. It may be listed in the library, yet it may not show on the screen. The reason for that is that ACTIVE-CAD searches its symbol libraries in the order in which they are listed in the **Library** window. If you want a specific symbol to take priority over other symbols with the same name, place its library in front of other libraries in the **Libraries** window. The top-most libraries are scanned first. The library listed at the bottom is scanned the last. Use the following procedure to reorder the libraries:

1. Select **Project Libraries** from the schematic **File** menu.
2. When the **ACTIVE-CAD Libraries** window appears (Figure 1-7), select in the **Project libraries** list the library that you want to reorder.
3. Press the **Reorder** button.
4. Place the cursor at the desired location in the **Project libraries** list and click the mouse button. The selected library is moved to the new location, and thus it will have a new search priority.

View

This set of options allows you to zoom in and out, redraw, and enable and disable viewing the display grid, the PWR pin, the pin name and the pin number.

Window

This set of options allows you to select a window, and cascade or tile all active windows. It is similar to the standard **Window** menu in other Windows applications.

Chapter 7

Library Manager

The **Library Manager** helps you to review, sort, print and display system libraries. It also allows you to find synonyms, create libraries of your own components, list projects, edit the existing IC models, create new models from existing models, etc. As you get involved with the ACTIVE-CAD product, you will discover that the **Library Manager** is the most useful tool for housekeeping of all symbol and model libraries.

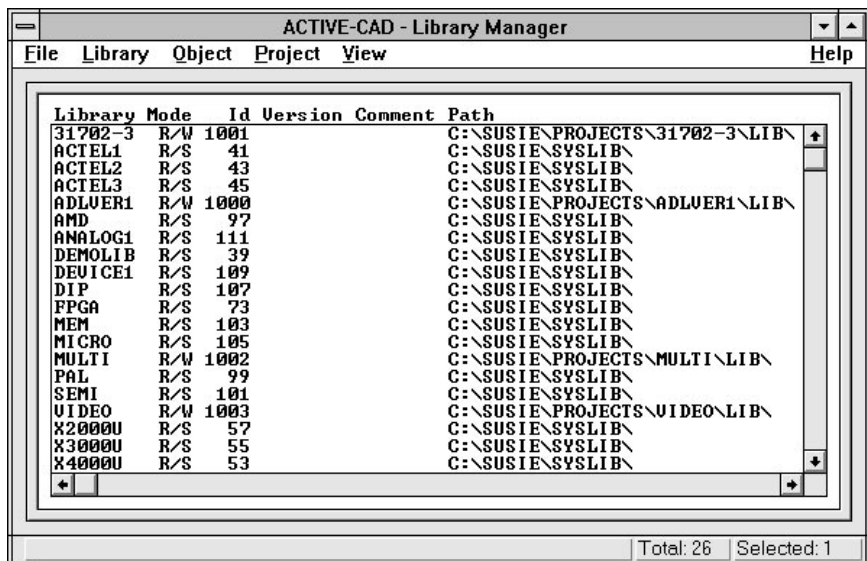


Figure 7-1. Library Manager Window.

To provide for quick access, the **Library Manager** option can be selected by clicking on the **Library Manager** icon in the Project Manager. It can also be invoked by selecting the **File|Project Libraries** option within the schematic or the simulator and then clicking on the Lib Manager button in the **Project Libraries** window. The main **Library Manager** window is shown in Figure 7-1.

The **Library Manager** displays six menus: **File**, **Library**, **Object**, **Project**, **View and Help**. Each of these menus is described in a separate section. The main display area in Figure 7-1 is used for listing the libraries.

File Menu

The **File** options include:

- ☐ **Print Setup** - invokes the standard Windows printer setup window
- ☐ **Rebuild** - allows to rebuild the ACTIVE-CADlibrary catalog

Rebuild

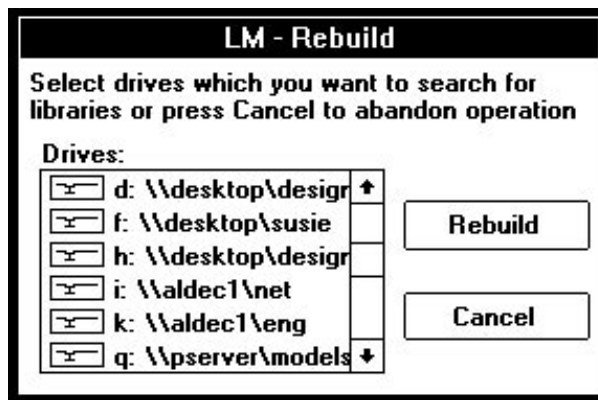


Figure 7-2. Rebuild Window.

ACTIVE-CAD displays and allows to use only these libraries which are 'attached' to the program. The information about attached libraries is stored in the binary file LIB_DIR. If this file gets corrupted or removed, ACTIVE-CAD cannot operate normally and the **Rebuild** option has to be activated to restore the LIB_DIR. file.

To rebuild ACTIVE-CADlibrary, select the drives where the libraries are installed and click the **Rebuild** button. ACTIVE-CAD will automatically attach and list all libraries installed on these drives.

Library Menu

The **Library** menu allows for operations on libraries such as copy, move, delete, attach, detach, create new libraries, list library components, etc.

List Objects

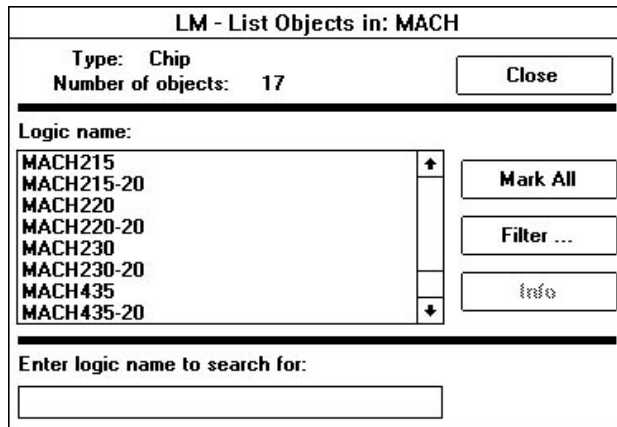


Figure 7-3. List Objects Window.

Clicking on a library name in the Library Manager window (Figure 7-1) and selecting the **List Objects** option displays the contents of the selected library (Figure 7-3). You can also double click on the selected library in Figure 7-3 to list its contents. You can scroll up and down the listing to view the components.

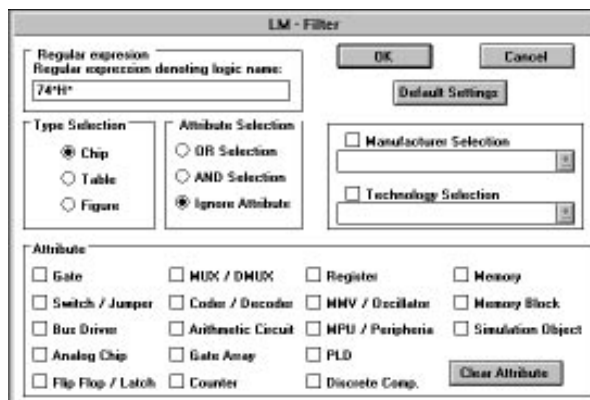


Figure 7-4. Filter Window For Selecting Objects.

To selectively display the library objects, click on the **Filter** button. The new **Filter** window (Figure 7-4) allows selecting the desired model attributes. Clicking on the **OK** button starts the search for the objects with selected attributes. For example, to list all H technology devices within the CMOS library, enter **74*H*** into the **Regular Expression** field, click on the **Technology Selection** box and enter **CMOS** in the field below. The asterisk (*) is used as a wild character.

Attach

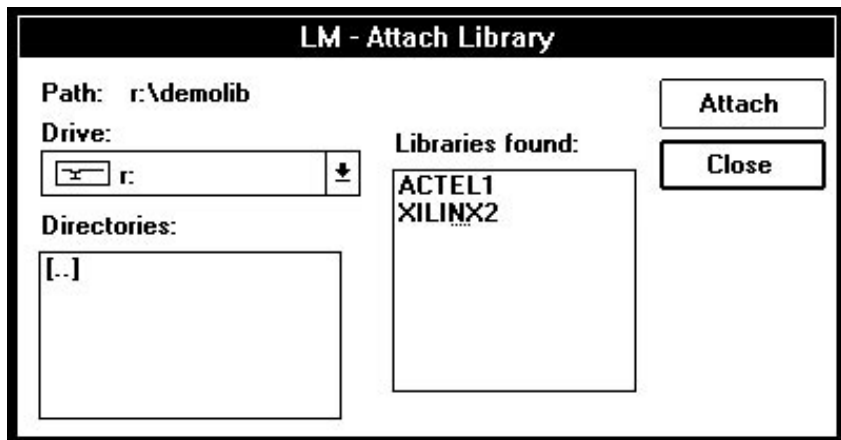


Figure 7-5. Attach Library Window.

Only the libraries attached to ACTIVE-CAD can be used in designs. If you click on a library in the **Library Manager** window and select the **Attach** option, ACTIVE-CAD will display the **Attach** window (Figure 7-5) which allows you to attach the desired library from the selected disk and directory location. If you select a directory, ACTIVE-CAD displays in the **Libraries Found** window all libraries that were found in that directory. The libraries that are already attached to ACTIVE-CAD will not be displayed in the **Libraries found** window.

Create

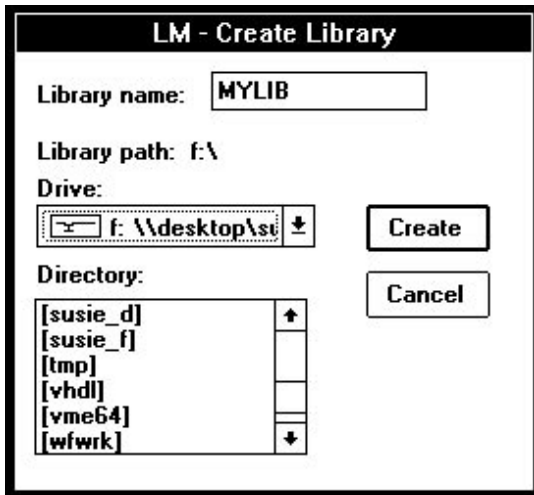


Figure 7-7. Create Library Window.

If you want to create a new empty library, click on the **Create** option and select the directory in the **Create Library** window (Figure 7-6). The new library is instantly added to the listing of the libraries in the main window. You can then perform desired operations on that library, e.g. copy some components from other libraries, etc.

Copy

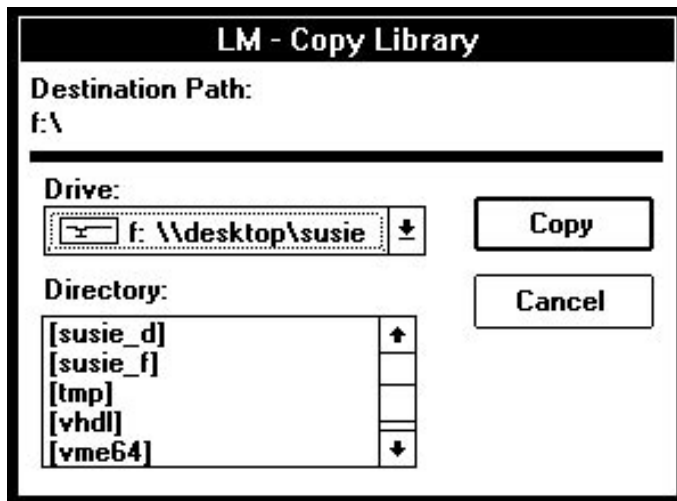


Figure 7-6. Copy Library Window.

The **Copy** option copies the selected library to a new directory and a duplicate of the existing library is created in the process. This option can be used for backup purposes or to copy the library to another computer or diskette. The location of the library in the library listing does not change

and ACTIVE-CAD will continue using the library in the current location. You need to use the **Copy** option if you want to give a library to someone else or use on another computer.

Delete

To delete a library from both the list of libraries and from a hard drive, use the **Delete** option in the **Library** menu. Be careful when using this option because it may delete valuable information from the hard disk.

Detach

To remove a library from the ACTIVE-CAD list of libraries, without deleting it from a hard drive, use the **Detach** option. The current project library cannot be detached but libraries used by the project itself can be detached. However, this may cause a *missing library* error.

Move

To copy a library to another location and remove it from the current directory, use the **Move** option. The library will remain on the ACTIVE-CAD library list but will be relocated to the new directory. This option is useful if you want to move libraries to or from the network drives or between local hard drives.

Access Mode

You can define the access mode of your libraries. The access mode defines if the library can be modified. There are four access modes:

- ☐ **R/W - Read and Write** - this mode allow you to make any modifications to the library. To prevent library modifications by several users, the R/W library can be attached only by one user. Other users on the network cannot access this library unless the current user of the library detaches it or changes the access mode, e.g. to **R/O**.
- ☐ **R/O - Read Only** - this mode allows the library access but prevents you from any modifications. R/O libraries can be shared by multiple users on the network.
- ☐ **R/A - Read and Append** - this mode allows adding new parts to the library, but you cannot modify the existing ones. This type of library can be shared on a network by multiple users.
- ☐ **R/S - Read System** - The ACTIVE-CAD system libraries are protected by keylock and they cannot be modified by the user. These libraries can be shared by multiple users on the network. You can

copy objects from the system libraries to R/W libraries to make the desired modifications of the library components or objects.

To change the access mode of a selected library, select the **Access mode** option in the **Library** menu and select the new mode in the **Access mode** window of Figure 7-8.

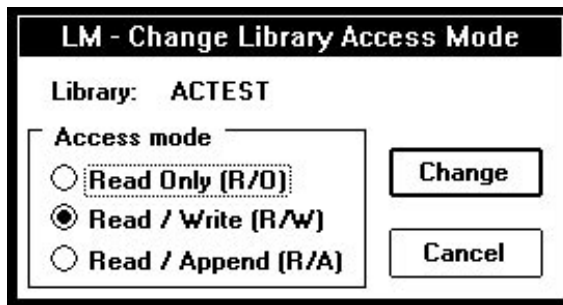


Figure 7-9. Setting Library Access Mode.

NOTE: You cannot set or change the R/S library access mode. All ACTIVE-CAD libraries come factory set to the R/S mode.

Comment

This option is active only with the user libraries. It is inactive for system libraries that come with ACTIVE-CAD. You can enter a library description, so that other users on the network can see what this library contains. This option is active only with the user libraries and is inactive for ACTIVE-CAD system libraries. To enter a comment, select the **Comment** option within the **Library** menu and enter the comment text in the **Library Comment** window (Figure 7-9).

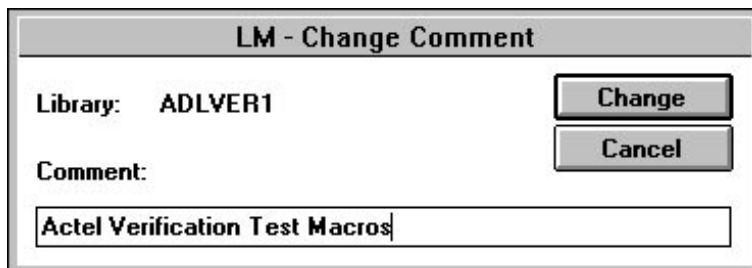


Figure 7-8. Setting library description.

Optimize

After performing several operations on user library objects, the user library can become fragmented, which increases its size and access time. To compact the library and optimize it for the best performance, select the desired library and use the **Optimize** option in the **Library** menu. Note that this operation can take several minutes if the library contains large number of objects. The system libraries cannot be subject of any modifications, editing, etc., and thus do not require optimization.

Rename

To change the name of a library, select it in the main window and select the **Rename** option in the **Library** menu. In the **Rename** window enter the new name and press the **Rename** button. Only the user libraries can be renamed. The system libraries can never be renamed.

Print

To print the list of all or selected libraries used in the system (currently attached), use the **Print** option in the **Library** menu. This option prints library names, comments, locations and access modes. It does not print the contents of the libraries. To set up the printer, use the **Printer Setup** option in the **File** menu.

Report

The **Report** option in the **Library** menu allows you to create a listing of the objects (IC models) within selected libraries. You can choose to include **Chip comments**, **Attributes**, **Manufacturers** and **Technology** (Figure 7-10). When you click on the **Create** button, ACTIVE-CAD opens the **Library Report** window, which allows you to select the drive (**Drives** field), directory (**Folders** field) and the name under which the file should be stored (**File Name** field). Clicking on the **OK** button generates a report which can be read by any text editor such as Notes. The report file can then be used by other programs to create custom reports, listings, etc.

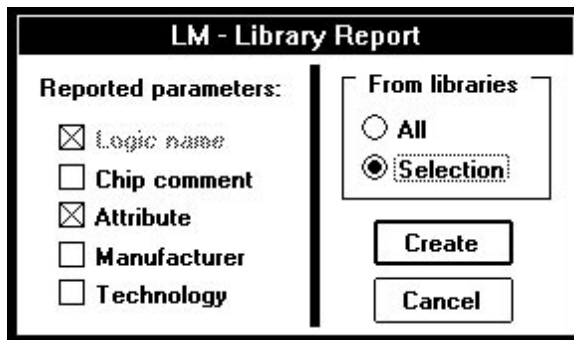


Figure 7-10. The Report Window.

Project Menu

To view all objects residing in project libraries, select the **List All Objects** option within the **Project** menu which opens the **List All Project Objects** window. This window allows you to review the list of ICs that are used for the current project, and shows a similar information to the **SC Symbols** toolbox in the Schematic Editor.

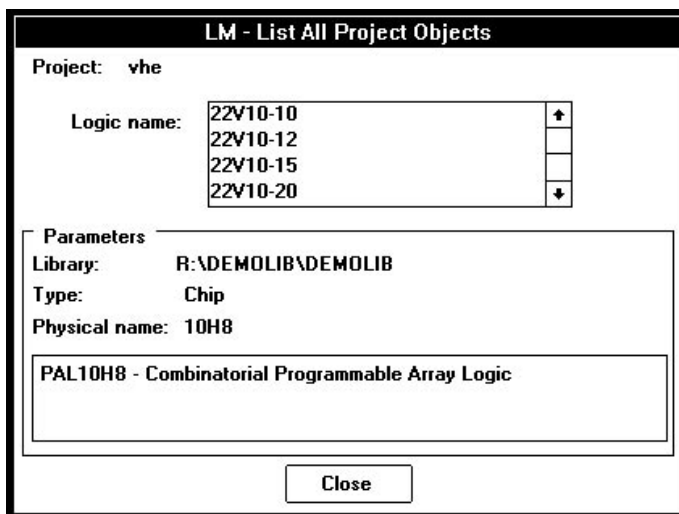


Figure 7-11. List All Project Objects Window.

The **List All Project Objects** window in Figure 7-11 allows you to review the **Description**, **Type**, **Physical Name** and **Library** of each object in the project libraries. The **Physical Name** is used to store the custom part name, the **Type** specifies if the object is a netlist, chip (simulation model), or just a graphical symbol. This option can also be useful to detect which

library has been used for a selected object, if there are several components with the same part name.

View Menu

This menu has four options for selective display of libraries information:

- ☐ *Status Bar* shows advisory notes at the bottom of the window.
- ☐ *Show Project Libraries* lists all libraries used by the current project.
- ☐ *Show Comment* lists comments assigned to libraries.
- ☐ *Show Path* displays the library path.

To add or remove libraries from the current project, use the **Add** and **Remove** buttons in the **Project Libraries** option that is available in the simulator and schematic **File** menus. To show all project libraries again, toggle the **Show Project Libraries** option.

Object Menu

General

Before you begin any operations on library objects, you need to select one or more components in the **List Objects** window (Figure 7-3). The **Object** menu is inactive until you select at least one object in this window. Only then you can select options from the **Object** menu.

Each library object may contain the following information:

- ☐ Logical Name
- ☐ I/O Pin names
- ☐ I/O Pin Numbers
- ☐ I/O Pin Loading (number of loads for CMOS parts)
- ☐ Description (comment about the component)
- ☐ Timing specification (Generics)
- ☐ Contents (default file e.g. JEDEC, hex)
- ☐ Technical Info (Manufacturer, Technology)
- ☐ Attributes (e.g. gate, PLD, memory)
- ☐ Graphical Symbol
- ☐ Simulation Model
- ☐ Netlist in the .ALB file format (hierarchy macro)
- ☐ Schematic for hierarchy macro
- ☐ Footprint for PCB layout
- ☐ Part Number (user defined part name)

- ☐ Spice Model for analog simulation
- ☐ ISE window for editing internal registers in the simulator
- ☐ Model Source code for VHDL and MBC debugger

NOTE: Do not close the **List Objects** window with selected objects until you perform the desired operations from the **Object** menu.

Not all of these parameters have to be defined for each part. **Library Manager** allows you to modify these parameters with the exception of:

- a. VDS (VHDL Development System) exclusively manages simulation models, ISE windows and model source code.
- b. The Symbol Editor exclusively controls graphical symbols.
- c. The Schematic Editor controls schematics for hierarchy macros.
- d. Only the Footprint and Part number can be edited in both the symbol editor and the schematic editor

NOTE: You can change **Footprint** and **Part number** of the system library objects and make them available to all network users. To change any other parameters you need to copy the selected objects to user library first and then perform the desired changes.

Copying Library Object

The **Copy** option within the **Object** menu allows you to copy selected objects to another library. It copies all information about the component, including simulation model, netlist, etc. You can copy objects from the system libraries to customize the standard components to your requirements. You can copy objects only to the libraries that have **R/W** and **R/A** access mode. To simplify the library search, **Library Manager** will automatically list all libraries that have these access codes.

Select in the **Copy Object** window (Figure 7-12) the object to be copied and the destination library where the component should be copied, and click on the **Copy** button.

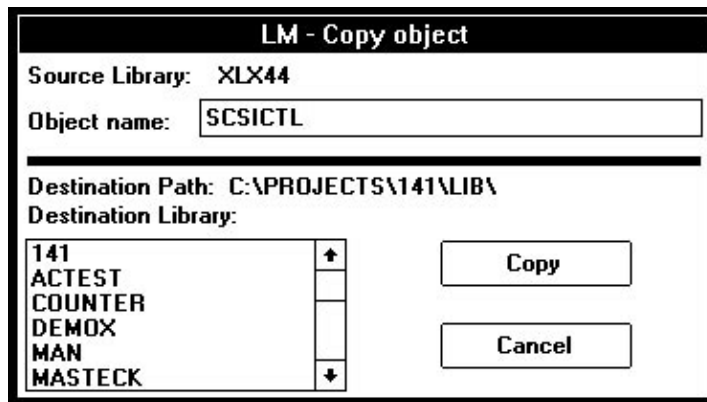


Figure 7-12. Copying Object to another library.

Deleting Library Objects

To delete an object from a library with the R/W access, use the **Delete** option within the **Object** menu. First, select in Figure 7-3 the object to be deleted. Next, select the **Delete** option from the **Object** menu, and when the **Delete Library** window appears, click on the **Delete** button.

NOTE: The deleted objects are permanently deleted from the library and cannot be recovered.

Renaming Library Objects

To change the name of a library object use the **Rename** option within the **Object** menu. Only the objects in the R/W user libraries can be changed. Other types of libraries do not allow for object name changes.

To change an object name, select in Figure 7-3 the object to be renamed. Next, select the **Rename** option from the **Object** menu, and when the **Rename Object** window appears (Figure 7-13), enter the new object's name and click on the **Rename** button.

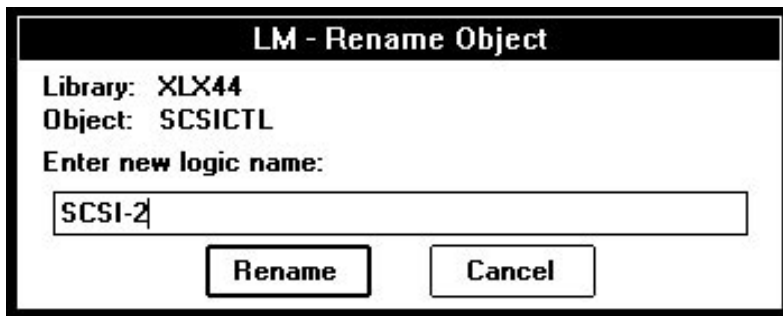


Figure 7-14. Renaming Object.

NOTE: This option must be used very carefully because the original objects will disappear from the library and will not be found under their old names when you load some old designs again.

Copying Functional Equivalents

To copy a group of functional equivalents to another library, select any one of its members and click on the **Copy Func. Equivalents** option (**Object** menu). When the **Copy Functional Equivalents** window (Figure 7-14) appears, select the destination library and click on the **Copy** button.

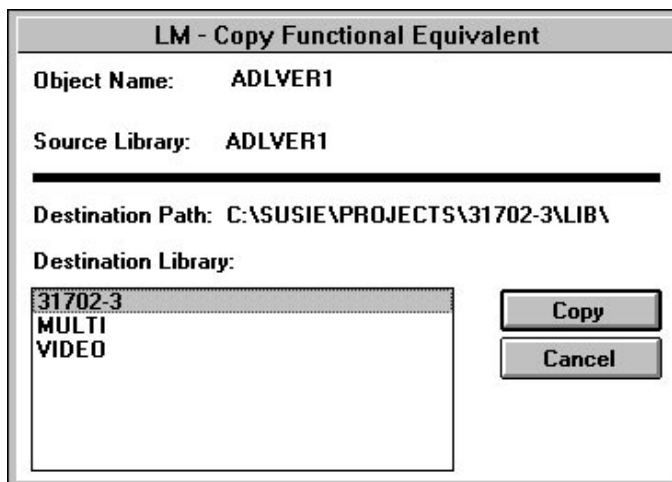


Figure 7-13. Copy Functional Equivalents Window.

Cloning New Timing Models

The ACTIVE-CAD allows loading different timing versions of the functionally equivalent devices without any compilations. A functional equivalent is a device which has the same logical function (simulation model) but has a different timing specification.

ACTIVE-CAD allows you to create a copy of an existing functional model, change its name and timing parameters and add to the library as a new timing model. This process of creating new timing models from existing functional equivalents is called “cloning.”

NOTE: Only R/W library objects can be cloned into new timing models.

To create a new timing model by cloning, select a part in Figure 7-3. Next, select the **Create Functional Equivalent** option from the **Object** menu. When the window in Figure 7-15 appears, enter the new part name in the **New Name For Equivalent** field, and click the **Create** button.

After creating a new component, you can use the **Object/Generics** menu to define new timing parameters for the new component. For example, the 22V10-15 PLD model can be instantly cloned into 22V10-5, which has 5 nanoseconds propagation delay between the clock inputs and the device outputs. The new part name (22V10-5) is created in the window in Figure 7-15, and the timing parameters are edited in the **Change Propagation Times** window (Figure 7-18), described in the **Generics** section.

Later on, whenever you use this part in your design you will be able to quickly switch between different versions of the component using the **Change Technology** option in the Simulator and Schematic Editor.

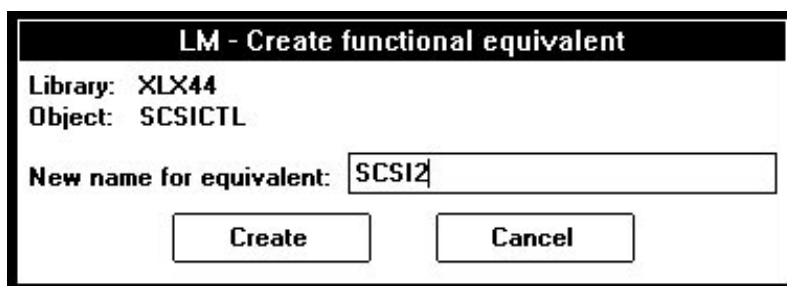


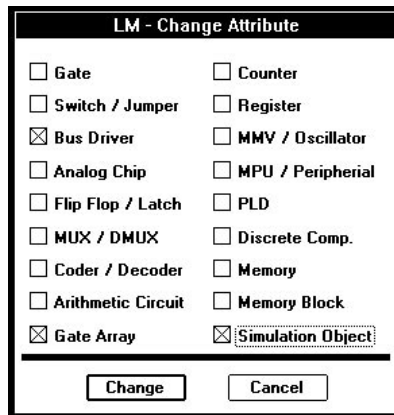
Figure 7-15. Create Functional Equivalents Window.

Note: The simulation model is not duplicated and is the same for all functional equivalents.

The new part will be added to the R/W library in which it was created. You can copy this part to any user library and put it on the network for corporate-wide use.

Attributes

Attributes allow for faster search and filtering of the library components.



To assign object attributes, select the **Attribute** option within the **Object** menu. In the **Change Attribute** window (Figure 7-16), click on the selected attributes and then on the **Change** button to save the setting.

Figure 7-16. Editing Object Attributes.

Each object has a set of attributes which describe its type. Attributes are used by the schematic editor to filter the symbols and by the simulator to determine which objects can have a JEDEC or hex file loaded. It is important that you always assign correct attributes to all components so that ACTIVE-CAD can correctly identify them in different filtering options.

Comment

Each user defined object can be assigned a description. This text is displayed when selecting an object in the schematic and in other ACTIVE-CAD programs. All ACTIVE-CAD library objects have factory provided description of their logic functions. However, if you want to add a comment to your custom objects, use the **Comment** option within the **Object** menu. Enter the comment in the **Chip Comment** line in Figure 7-17 and press the **Modify** button.



Figure 7-17. Setting Object Description.

Generics

Simulation models created with the VHDL compiler can have timing parameters assigned to the model. These parameters are usually specified by the manufacturer of the device in the databook. ACTIVE-CAD libraries keep three values for each timing parameter (Minimum, Average and Maximum). The simulator by default takes the average timing parameters of objects. However, using simulator options you can set these parameters to either the Min or Max values.

To change timing parameters of a library object, select the part in Figure 7-3 and click on the **Generics** option in the **Object** menu. When the **Change Propagation Times** window appears, select the desired parameter from the **Propagation Times** field (Figure 7-18) and then edit its values in the **Propagation Time Values** field. The new values are permanently assigned to the object when you click on the **Modify** button.

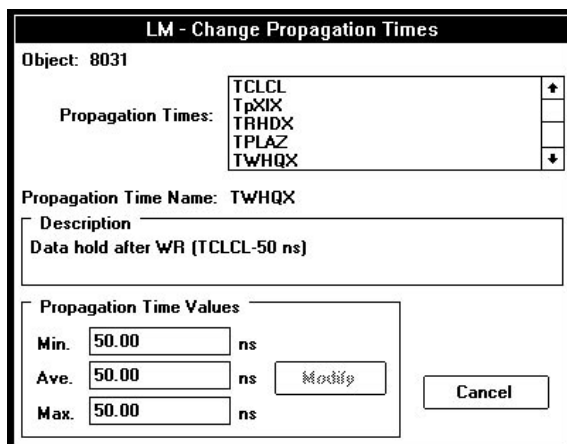


Figure 7-18. Editing timing parameters.

The simulator also allows you to directly edit any parameter value, but these changes are done locally in the simulator design tables. If you want to permanently change timing parameters of the library objects, you should use the **Generics** option within the **Object** menu.

NOTE: To protect the integrity of the ACTIVE-CAD system library objects, you can view their timing parameters but you cannot modify them. To modify their timing parameters, copy them first into your own (user) library and then change their timing values.

Pin Loading

Some of the IC components change their timing specification depending upon the loading or the number of input pins connected to their output pins. Output pin loading specifies how many input pins can be connected to the selected output. Most of the manufacturers define the maximum number of loads that can be connected to a single output pin. A load is usually the same as a single pin. However, some input pins take more current and are equivalent to more than one load. Also, some output pins have higher driving capability than normal. **Library Manager** allows you to modify the loading factor of such pins to account for the extra loading capacity. .

The pin loading is used in ACTIVE-CAD to detect any errors in the design caused by connecting too many input pins to a single output. Some models are also able to change their timing parameters according to the manufacturers loading table.

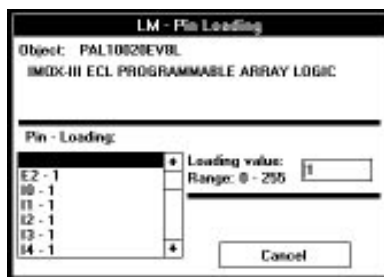


Figure 7-19. Editing Pin Loadings.

To edit pin loading parameters, use the **Pin Loading** option within the **Object** menu. In the **Pin Loading** window (Figure 7-19), select the desired pin and change its loading value in the Loading Value field. To save the new loading parameters, click on the **Save Loading** button.

NOTE: You can only save the loading values for the user libraries. To protect the system libraries, the **Save Loading** button is automatically disabled when a system library object is loaded into Figure 7-19.

Pin Numbering

Each component can have numbers assigned to its pins. Pin numbers can be assigned to any type of objects (simulation model, netlist, graphical symbol). Each object that has pins must also have pin names. Most of the objects that represent physical components have both pin names and pin numbers. Pin numbers are required for all designs that are exported to a PCB layout program. Some of the objects have only pin names (e.g. FPGA libraries). To change pin numbers, select an object in Figure 7-3 and activate the **Pin Numbering** option in the **Object** menu.

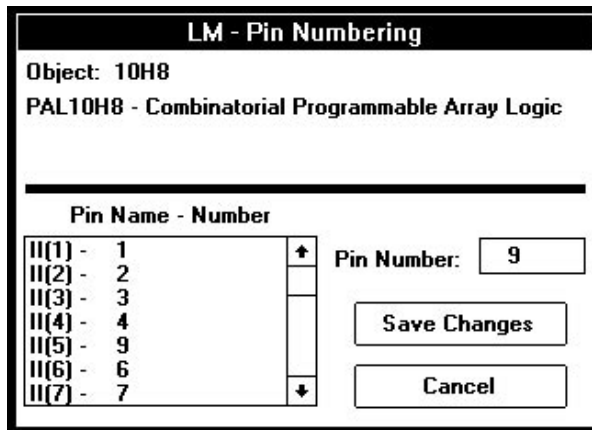


Figure 7-20. Editing Pin Numbers.

In the **Pin Numbering** window (Figure 7-20), select a pin from the **Pin Name** list and change its number in the **Pin Number** window. When all pin numbers have been changed, click on the **Save Changes** button to save the new pin numbers.

NOTE: You cannot change pin numbers for the object from system libraries. In such case, the **Save Changes** button is not displayed.

NOTE: You cannot change the pin *names* in Library Manager. The pin names can only be changed in the symbol editor and only if the objects do not have any simulation model or netlist assigned to it.

Synonyms

In order to minimize the number of different IC names in the library, ACTIVE-CAD uses a mechanism called synonyms. A synonym is an exact equivalent of an object that already exists in the library. For example,

the 7400 gate has several synonyms: SN7400, SN7400N, 00, etc. All of these parts have the same pin numbers, timing parameters, loading, etc. The synonyms are used for importing netlists from external schematic editors because they allow ACTIVE-CAD to recognize parts that have different names but have their equivalents in the ACTIVE-CAD library.

Adding Synonym Parts

To add a synonym to the library, select a specific object from Figure 7-3. Without closing that window, select the **Synonym** option from the **Object** menu. In response, the **List Object Synonyms** window is displayed, listing all synonyms of the selected object (Figure 7-21).

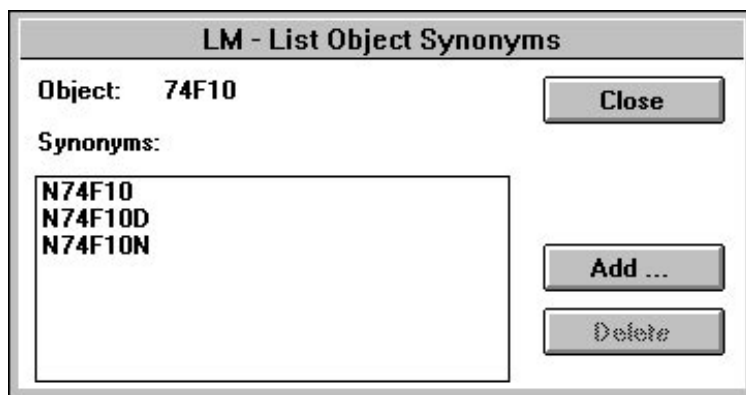


Figure 7-21. Synonyms Window.

To add a new synonym, click on the **Add** button in the **List Object Synonyms** window. The **Add Synonym** window will appear and will prompt you to enter the new synonym name. Click on the **Add** button to add it to the synonym list.

To delete a synonym, highlight it in the **List Object Synonyms** window and then click on the **Delete** button.

Technical Info

Each object can have assigned the **Manufacturer** and **Technology** parameters. These parameters can be useful for searching and filtering components in the library as well as for reports and **Bill of Material** options in the Schematic Editor. To edit these parameters, use the **Tech Info** option within the **Object** menu.

Select the desired parameters from the **Manufacturer**, **Technology** and **Package** fields in the **Technical Info** window (Figure 7-22), and click on the **Modify** button. You can only use one entry from each list.

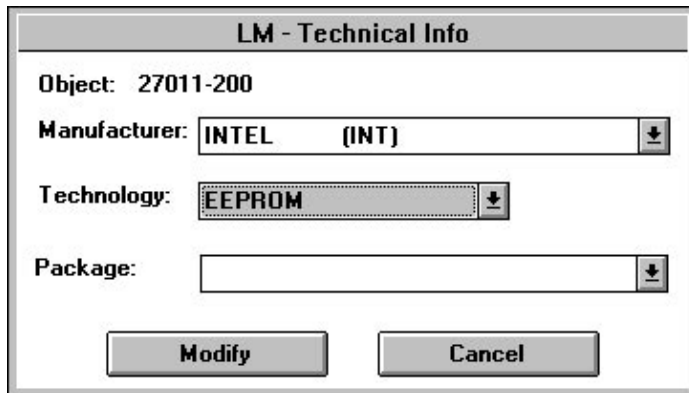
The image shows a dialog box titled "LM - Technical Info". It contains four fields: "Object:" with the value "27011-200", "Manufacturer:" with a dropdown menu showing "INTEL (INT)", "Technology:" with a dropdown menu showing "EEPROM", and "Package:" with an empty dropdown menu. Each dropdown menu has a small arrow icon on the right. At the bottom of the dialog box are two buttons: "Modify" and "Cancel".

Figure 7-22. Editing Technical Information.

You cannot enter your own text into the **Manufacturer**, **Technology**, or **Package** fields. You can only make a selection from the predefined lists.

Attach Netlist

The library objects can have assigned either IC models or netlists. The netlists are used as simulation macros that use lower level components. You can attach a netlist to the selected object. This netlist will be used by the simulator as a model. The netlist has to be in the .ALB binary format and can be assigned to any object that has no simulation model. The netlist is stored with its object in the library, and if it ever changes, you must assign the new netlist to the object again.

All netlists created by ACTIVE-CAD schematic editor are in the .ALB format. If you want to assign a netlist imported in another format, it has to be converted first.

For example, loading a netlist to the simulator results in an automatic conversion to the .ALB format. The converted netlist is stored by ACTIVE-CAD in the current **Project** directory and can be downloaded directly into an object.

To assign a netlist to the selected library object, use the **Attach Netlist** option within the **Object** menu and select the desired .ALB file.

You can also create a netlist-based object directly in the schematic editor. When you use the **Import Netlist** option in the schematic **Options** menu, it converts the imported netlist to the .ALB format, creates a new object in the library and assigns to it the converted netlist. The new object will be listed in the **SC Symbols** window in Schematic Editor, can be placed on schematic and edited in Library Manager.

Placing an FPGA symbol on the schematic sheet and invoking the **Assign Netlist** option in the **Hierarchy** menu (Schematic Editor), will assign the selected netlist to the symbol. The selected netlist can be in any format .

To assign a netlist to the selected library object in the Library Manager, use the **Attach Netlist** option within the **Object** menu and select the desired .ALB file.

You can attach netlist only to empty symbols. Library Manager does not allow you to attach a netlist to the system library symbols.

Create with Contents

One of the ways to create custom parts for simulation is by downloading a JEDEC or Memory file into an empty object.. For example, loading a JEDEC file for a 4- bit Johnson counter into the 22V10 object, will create a custom model of that Johnson counter. Any time you load that 22V10-based object to the screen, it will simulate the Johnson counter.

The best way to create such models is by copying system library parts into user libraries and then assigning to them concrete JEDEC or hex files. You should rename such parts with the **Rename** option in the **Object** menu. This will allow you to clearly differentiate the new parts from the standard 22V10 devices.

To assign a contents to the selected object, use the **Create with Contents** option within the **Object** menu, and then select the desired file from any disk directory.

NOTE: The **Create with Contents** option can be used only if the selected object is either a PLD or a Memory device. JEDEC and memory HEX files are saved in the library.

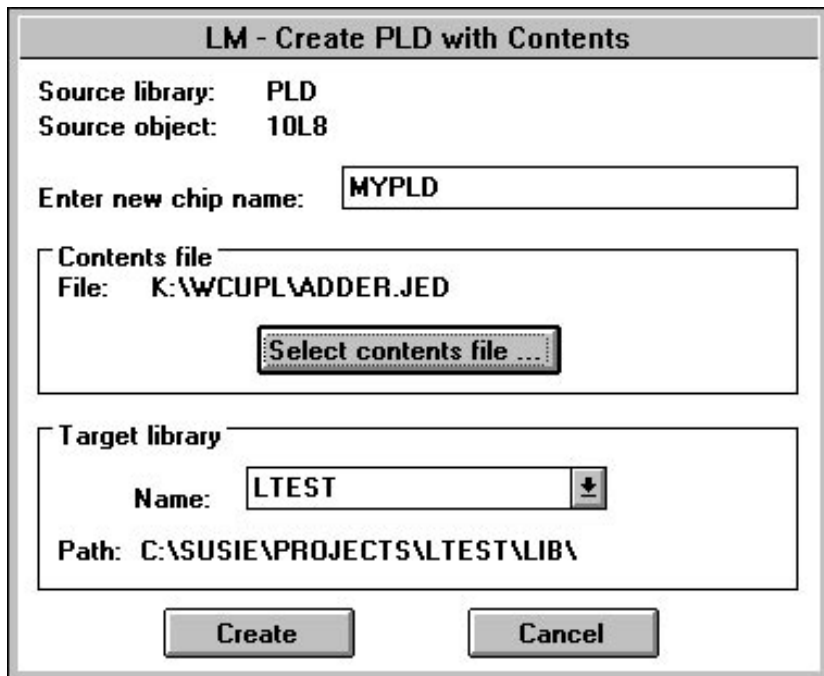


Figure 7-23. Create PLD with Contents window.

Change Contents

If the JEDEC or hex files that have been used to create custom simulation models ever change, you have to update the models by using the **Change Contents** option.

NOTE: The **Change Contents** option allows overriding the existing JEDEC and hex files with new ones and creating new simulation models.

Chapter 8

Interface to the Intusoft SPICE Simulator

Introduction

The ACTIVE-CAD SPICE option includes full support for creating analog SPICE designs and simulating them in the Intusoft IsSPICE 4 analog simulator. The ACTIVE-CAD SPICE option includes schematic libraries with analog symbols, a direct interface to the IsSPICE database for library model selection, and a SPICE netlist generation capability with SPICE simulation commands.

Installation

ACTIVE-CAD and IsSPICE programs both need to be installed on the same computer for the interface to operate properly. You can install these two programs according to the installation instructions for each product. The order in which you install the products is unimportant.

The interface to IsSPICE utilizes a direct access to the Intusoft library database. This database is located in the following files:

C:\SPICE4\ICAPS.EXE

C:\SPICE4\SN\DBASE.@@@

C:\SPICE4\SN\INDEX.@@@

You need to set the PATH in your AUTOEXEC.BAT file to include the C:\SPICE4 directory. If you installed IsSPICE in a different directory, please verify that your AUTOEXEC.BAT file shows the correct PATH setting.

If the IsSPICE database files cannot be found, you will receive an error message when you try to create a SPICE design.

Creating a SPICE project

Schematics created for analog SPICE simulation have to be entered in a SPICE project. To create a SPICE project you need to open the **Project Manager** window and select SPICE from the **Type** selection list (Figure 8-1).

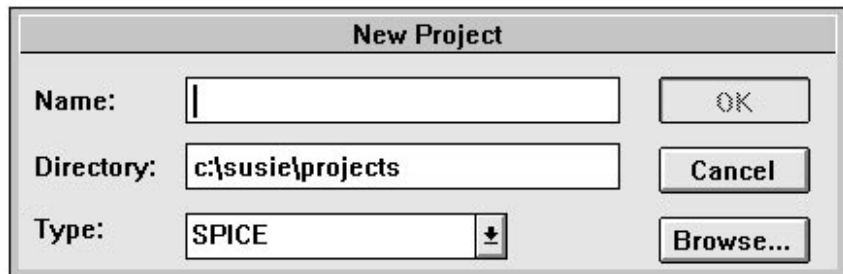


Figure 8-1. Creating a new SPICE project

After creating a SPICE design the **Design Manager** window shows a SPICE flow with the SPICE icon (Figure 8-2).

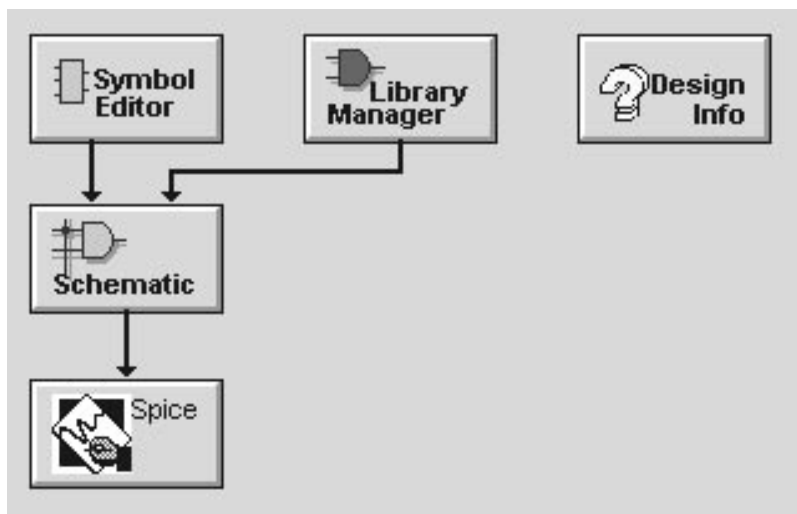


Figure 8-2. SPICE Design Flow

NOTE: You cannot use other project types to interface to IsSPICE simulator. Only SPICE projects are set up to generate a correct SPICE netlist for simulation.

Libraries

When you create a SPICE project, the SPICE library is automatically assigned for this project. In the Project Libraries list, you should see the two project libraries: SPICE and a working library with the same name as the project itself (Figure 8-3). If the library is not installed you will see an error message.

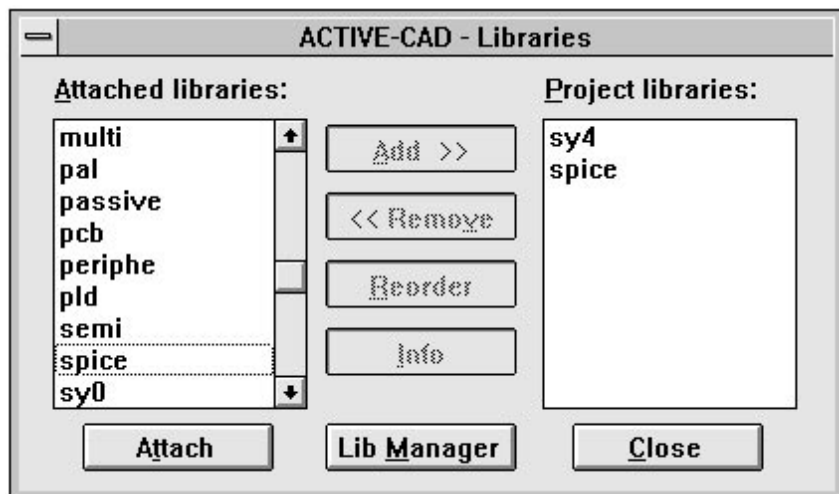


Figure 8-3. ACTIVE-CAD library window

NOTE: Do not select any additional libraries for the SPICE project. Only the symbols from the SPICE library can be accepted for simulation in the IsSPICE simulator.

Creating an analog schematic

Because of certain features unique to SPICE, placing symbols in SPICE projects is different from placing symbols in other projects. The symbol toolbox contains a hierarchical database of over 5,000 symbols grouped in several categories.

The **SC Symbols** toolbox has two SPICE buttons that allow you to browse through the IsSPICE symbol database and select the desired symbols or simulation objects (Figure 8-4).

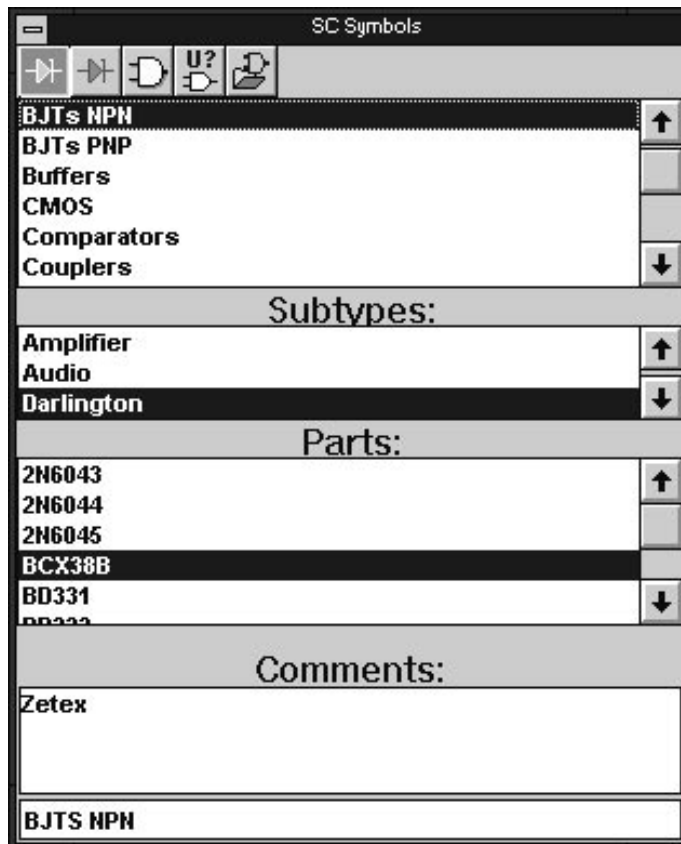


Figure 8-4. SC Symbols toolbox

The first SPICE button displays SPICE symbols grouped into Type, Subtypes and Parts (Figure 8-4). The second SPICE button displays simulation probes and voltage/current sources. (Figure 8-5).

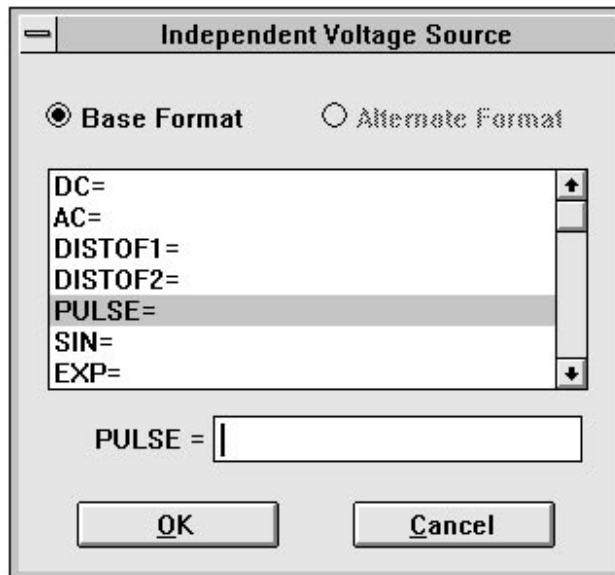


Figure 8-5. Simulation probes and voltage sources

Please refer to the IsSPICE manuals for a complete library guide and a description of library types.

NOTE: When you click on the DIGITAL button, the symbols list shows the generic symbols. These symbols do not have any SPICE properties assigned and will not load to the SPICE simulator. The properties for these symbols are assigned when you select them from the Type/Sybtype/Group selection.

Editing SPICE parameters

Most SPICE symbols contain specific attributes and parameters that you can assign for simulation. To edit symbol-specific parameters, you need to switch to the **Changes** mode and double-click on the desired symbol. Then from the **Edit Symbol** window, click on the **Spice** button to display symbol SPICE parameters.

Selecting a SPICE analysis

Before you can simulate the design, you need to select the type of SPICE analysis you want to perform. To do so use the **Spice Analysis** option in

the schematic **Options** menu. This information will be included in the generated SPICE netlist.

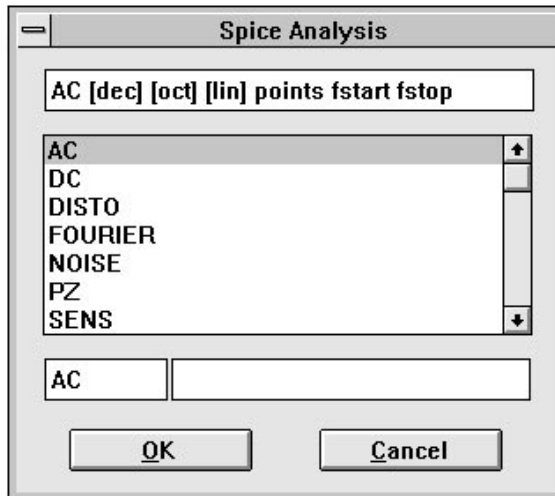


Figure 8-6. The SPICE analysis window

To simulate the SPICE design click on the **SIM** button in the schematic toolbar. The ICAPS program is then started instead of the regular logic simulator.

You can also start the SPICE simulation from the **Design Manager Flow** window by pressing the **SPICE** button.

Running a SPICE simulation

For instructions on running the SPICE simulation, please refer to the Is-SPICE4 User Guide.

Appendix A

Library Listing

The following is an abbreviated list of SUSIE libraries. The parts can be available in all technologies, packaging and from all manufacturers.

TTL	77	164	266	456	621	777	1604
	78	165	269	465	622	779	1645
	83	166	273	466	623	786	1760
00	85	167	276	467	638	804	1760A
01	86	168	279	468	639	805	1762
02	89	169	280	490	640	808	1779
03	90	173	282	518	641	810	1894
04	91	174	283	519	642	811	1895
05	92	175	286	520	643	821	1896
06	93	178	292	521	644	822	1897
07	94	180	294	522	645	823	2232
08	95	181	295	526	646	824	2233
09	96	182	297	527	647	825	3037
10	97	183	298	528	648	826	3038
11	107	190	299	533	649	827	3040
12	109	191	348	534	651	828	3893
14	111	192	350	537	652	832	5074
15	113	193	352	538	653	841	8003
20	116	194	353	539	654	842	8960
21	120	195	354	540	655	843	8961
22	121	196	355	541	656	844	30240
25	122	197	356	543	657	845	30244
26	123	198	365	545	673	846	30245
27	125	199	366	547	674	850	30640
30	126	221	367	548	676	851	50109
31	128	229A	368	560	677	861	50728
32	131	225	373	561	678	862	50729
33	132	230	374	563	679	863	
34	133	231	376	568	680	864	
36	134	232A	377	569	682	866	
37	135	233A	378	574	685	867	
42	136	234	379	575	686	869	
45	137	235	381	577	687	870	
46	138	236	382	579	688	873	
48	139	240	385	583	689	874	DIP314A
49	140	241	386	588	711	876	DIP314B
50	143	242	390	590	712	878	DIP316A
51	147	243	393	591	723	879	DIP316B
53	148	244	396	592	725	881	RES
54	150	245	398	593	732	882	RPAK10
55	151	246	399	594	733	899	RPAK14
56	153	248	410	595	746	990	RPAK16
57	154	250	412	596	747	991	RPAK8
64	155	251	422	597	755	996	SIP106A
65	156	253	423	598	756	1000	SIP106B
68	157	256	432	604	757	1004	SIP108A
69	158	257	436	605	758	1032	SIP108B
70	159	258	440	606	759	1241	SIP110A
72	160	259	441	607	760	1242	SIP110B
73	161	260	442	614	762	1243	
74	162	264	444	615	763	1244	
75	163	265	455	620	776	1245	
76							

List of available libraries:

- ☐ **8051**(Intel 8051, 8052, 8031, 8032)
- ☐ **ACTEL1, ACTEL2, ACTEL3** (Actel FPGA macro library)
- ☐ **ATMEL** (Atmel EPLDs)
- ☐ **ALTERA** (Altera MAX & FLEX)
- ☐ **CMOS** (TTL-CMOS & CD4000)
- ☐ **ECL** (10K, 100K, 10H, 10E)
- ☐ **EPLD** (Altera & Intel EPLD, e.g. EP300, EP1800)
- ☐ **ISA** (Intel ISA Simulation library)
- ☐ **LATTICE** (Lattice pLSI macro library)
- ☐ **M6809** (Motorola MPU)
- ☐ **MACH** (AMD CPLD, MACH1, 2, 3 & 4)
- ☐ **MEMORY** (RAM, ROM, SRAM, DRAM, FIFO, Dual Port, etc.)
- ☐ **PASSIVE** (Resistors, Capacitors, Inductors, Switches)
- ☐ **PERIPHE** (Intel and Motorola Peripheral Devices)
- ☐ **PLD** (Classical PLD and PAL devices e.g. 10H8, 22V10)
- ☐ **QUICKLOG** (Quicklogic PASIC Macro library)
- ☐ **TTL** (LS, S, F, ALS, etc.)
- ☐ **XILINX2, XILINX3, XILINX4** (Xilinx FPGA macro library)
- ☐ **X2000U, X3000U, X4000U, X7000U, XBLOXU** (Xilinx Unified Libraries)
- ☐ **Z80** (Zilog Z80 MPU)
- ☐ **AMD** (symbols only, AMD devices)
- ☐ **ANALOG1** (symbols only, Analog devices, amplifiers)
- ☐ **DEVICE1** (symbols only, discrete components, transistors, diodes)
- ☐ **DIP** (symbols only, footprint symbols)
- ☐ **MEM** (symbols only, Additional memory devices)
- ☐ **MICRO** (symbols only, Microprocessors)
- ☐ **PAL** (symbols only, additional PAL & PLD symbols)
- ☐ **SEMI** (symbols only, CD4XXX symbols)

NOTE: Libraries marked as “symbols only” do not contain simulation models.

INDEX

A

ACTIVE-CAD

- New project 1-5
- Annotation 2-22 - 2-23, 4-19, 5-16
- Arc 2-33, 6-8
- Arrange menu 5-32 - 5-33
 - Center 5-32
 - Full page 5-32
 - Previous zoom 5-32
 - Redraw 5-33
 - Zoom in 5-32
 - Zoom out 5-32
 - Zoom select 5-32
- ASCII 1-26 - 1-27, 5-4 - 5-7, 5-14 - 5-15, 5-19
- Assign Netlist 5-16
- Audit file 5-27
- Automatic Connections 2-30 - 2-32, 2-34
- Autorouting 5-26

B

- Back Annotation 2-42, 5-16
- Backup, schematic 5-27
- Bill of Material 5-17 - 5-18
- Bus taps
 - Adding names 3-10
 - Automatic 3-9
 - Dragging 2-37
 - Drawing 2-30
 - Manual 3-9
 - Names 3-10
 - Type 5-26
 - Type, changing 3-10
- Buses 2-29, 3-1 - 3-3, 3-5 - 3-15, 3-17, 3-19
 - Complex 3-4, 3-6, 3-15 - 3-20
 - Connection rules 3-5 - 3-6
 - Deleting 3-12
 - Dragging 2-37, 3-11 - 3-12
 - Drawing 3-6 - 3-7, 5-25 - 5-26
 - Editing 3-11 - 3-13, 3-15 - 3-16, 3-18, 3-20
 - Generic 3-4
 - Label 2-25, 3-1 - 3-4, 3-7, 3-12
 - Name 3-1 - 3-4, 3-7, 3-12
 - Netlist 3-14
 - Pin order 3-2
 - Pins in macro symbol 4-15

- Probes 3-13
- Querying 3-12, 3-14
- Renaming 3-12
- Signal order 3-2
- Simulator 3-14
- Stretching 3-11
- Taps, adding names 3-10
- Taps, automatic 3-9
- Taps, hanging type 3-10
- Taps, manual 3-9
- Terminals 3-3, 3-8
- Unnamed 3-3

C

- Capacitors 2-21
- Centering
 - Displayed circuits 5-32
- Check design integrity 5-14
- Check electrical rules 2-3, 2-16
- Check for duplicate references 2-23
- Check symbol integrity 6-10
- Circle 2-33, 6-7
- Circuit
 - Definition 2-1
 - Symbols 2-2
- Colors, screen 2-28, 2-46, 2-48
- Comments 4-12, 4-14, 5-21, 7-7
- Compile schematic 1-4
- Component
 - attributes 2-24, 2-26
 - Libraries 2-49
 - placing 2-18 - 2-21, 2-23 - 2-24, 2-26
 - rotation 2-24
 - selecting 2-18
- Connection
 - auto 2-31 - 2-32
 - Drawing 2-25, 2-27 - 2-29, 2-31
 - Logical 2-2
 - manual 2-32
 - ortho 2-32
 - Physical 2-2
- Connectors 2-22
- Create hierarchical netlist 5-14
- Cursor Coordinates 2-13
- Custom libraries 1-9

D

- Delete
 - Graphics items 6-8
 - Objects 2-38, 5-11 - 5-12

- Pins 6-6
- Project 1-12
- Project resources 1-25
- Resources 1-28
- Sheet 5-3
- Symbols 2-19
- Undo 2-20, 5-11 - 5-12
- Design Manager
 - Starting 1-6
- Design structure
 - flat 4-2, 4-4, 4-19, 5-15
 - Hierarchical 4-1 - 4-5, 4-10 - 4-19, 4-21, 4-24 - 4-25, 4-27, 5-15
- Design Table 5-10, 5-29
- Designs
 - Checking duplicate references 2-23
 - Checking electrical rules 2-3, 2-16
 - Hierarchical 2-23
 - Large 2-2
 - Management 4-5 - 4-11, 4-13, 4-15 - 4-21, 4-23
 - Reference numbers 2-19
 - Structure 4-1, 4-3
 - Top-down 4-21 - 4-22
 - Verification 1-29, 2-3
- Double-click, defined 1-4
- E**
- EDIF
 - Import 5-5, 5-7
- Edit
 - Attributes 2-24, 2-26
 - Labels 2-23, 6-4
 - Names 2-23, 6-4
 - Objects 2-35 - 2-41
 - Part orientation 2-24, 2-37
 - Part reference designators 2-23, 6-4
 - part values 6-9
 - Schematic 2-35, 2-37, 2-39, 5-11 - 5-13
 - Symbol 2-40, 6-1, 6-3 - 6-4, 6-6 - 6-10
 - Title block 5-10
- Edit Library 5-21
- Edit menu 5-11 - 5-13
 - Copy 5-12
 - Cut 5-11
 - Delete 5-12
 - Deselect all 5-12
 - Export clipboard 5-13
 - Import to clipboard 5-13
 - Paste 5-12
 - Redo 5-11
 - Show clipboard 5-12
 - Undo 5-11
- Event 2-46
 - tracking 2-47
- Export
 - Clipboard 5-13
 - Netlist 4-19, 5-15
 - Parts and Nets 5-16
- Export Schematic
 - to DXF 5-7
 - to EDIF 5-7
- F**
- File menu 5-1 - 5-7, 5-9, 5-11
 - Exit 5-11
 - Export 5-7
 - Go to simulator 5-4
 - Import 5-4 - 5-6
 - New sheet 5-1
 - Open sheet 5-2
 - Page setup 5-9
 - Print 5-8
 - Printer setup 5-9
 - Project libraries 5-3
 - Project manager 5-3
 - Save 5-3
 - Save all 5-3
 - Save as 5-3
 - Table setup 5-10
 - Update libraries 5-4
- Flat netlists
 - Creating from hierarchical designs 4-4, 4-19
 - Creating from hierarchical designs 5-15
- Formats
 - .ALB 7-20
 - ASCII 5-4, 5-14, 5-19
 - DXF 5-7 - 5-8
 - ECO 5-27
 - EDIF 5-7
 - FutureNet 5-14
 - OrCAD 5-4, 5-6 - 5-7
 - PADS 5-14
 - Schema 5-14
 - Windows Metafile 5-11
- FutureNet netlist format 5-15

G

Graphics

- Deleting 6-8
- Draw 2-33, 6-7
- editing 6-6, 6-8
- Enable/Disable 5-29

Grid

- Configuration 5-24
- Displaying 5-21, 5-24 - 5-25
- Resolution 5-25

H

- Hierarchical netlist formats 4-11 - 4-13, 5-15
- Hierarchical schematic 2-1 - 2-2, 4-1 - 4-7, 4-9 - 4-14, 4-16 - 4-20, 4-23 - 4-27
 - Annotation 4-19
 - Buses, connecting 3-6
 - Converting to flat 4-4, 4-19, 5-15
 - Creating 4-14
 - Errors 4-23
 - Example 4-24 - 4-27
 - Inter-sheet connections 2-3, 4-3, 4-14
 - Management 4-10 - 4-13, 4-15 - 4-21, 4-23 - 4-24
 - Navigating 4-16
 - Numbering reference designators 2-23, 4-2, 4-6, 4-8, 4-19
 - Push/Pop 5-16
 - SEE ALSO Schematic
- Hierarchy
 - Menu 4-24 - 4-26, 5-20

I

I/O Terminal 2-26

Import

- Back Annotation 5-16
- Clipboard 5-13
- EDIF Schematic 5-7
- EDIF Symbols 5-5
- from Audit File 5-17
- Hierarchical netlist 4-12
- Netlist 5-15
- OrCAD schematic 5-5 - 5-6
- OrCAD symbols 5-4
- Viewlogic schematic 5-7

Incremental design 2-16

Inductors 2-21

Interfaces 2-4

J

Junctions 5-25

L

Library

- Access mode 7-6
- Attach 1-8, 7-4
- Browsing through parts 2-18, 2-20, 7-3
- Changing order 6-11
- Comment 7-7
- Contents 7-3
- Copy 7-5
- Create 7-1, 7-3, 7-5
- Delete 7-6
- Deleting parts 7-12
- Detach 7-6
- Edit 5-21
- Editing 5-4
- Import EDIF 5-5
- Import OrCad 5-4
- Import PADS 5-5
- Interfaces, and 2-4
- Macros 4-19
- Move 7-6
- Objects 2-2, 7-10
- Optimize 7-8
- Print 7-8
- Rename 7-8
- Report 7-8
- Search Order 6-11
- Symbols 2-49
- System-level P-xii
- Library Filter 2-20
- Library Manager 7-1, 7-3 - 7-7, 7-10, 7-12, 7-15, 7-17 - 7-19, 7-21 - 7-22
 - Access mode 7-6
 - Attach 7-4
 - Comment 7-7
 - Copy 7-5
 - Create 7-5
 - Delete 7-6
 - Detach 7-6
 - Library menu 7-3
 - Library objects operations 7-10, 7-12, 7-15, 7-17 - 7-19, 7-21
 - List objects 7-3
 - Move 7-6
 - Optimize 7-8
 - Print 7-8

- Rebuild 7-2
- Report 7-8
- System options 7-2
- Library objects
 - Attributes 7-15
 - Comment 7-15
 - Copy 7-11
 - Copy functional equivalents 7-13
 - Create functional equivalents 7-14
 - Create with contents 7-21
 - Custom 1-9
 - Delete 7-12
 - Generics 7-16
 - Netlists 7-20
 - Pin loading 7-17
 - Pin numbering 7-18
 - Rename 7-12
 - Synonyms 7-18 - 7-19
 - Technical information 7-19
- Line 2-33, 6-7 - 6-8
- Line Set 2-34
- List Connectors 5-19 - 5-20
- M**
- Macros
 - Creating 4-13
 - Deleting 4-20
 - Hierarchical 4-10
 - Library 4-19
 - Modifying existing 4-17
 - Netlist-based 4-10 - 4-12
 - Opening hierarchical 4-16
 - Renaming 4-20
 - Schematic, difference 4-17
 - Symbol, changing 4-18
 - Top-down 4-21 - 4-22
 - Update libraries 5-4
 - Updating netlist 4-12
 - Using 4-16
- Managing schematics 4-1, 4-3
- Menu
 - Arrange 5-32 - 5-33
 - Edit 5-11 - 5-13
 - File 5-1 - 5-5, 5-7, 5-9 - 5-10
 - Hierarchy 4-24 - 4-26, 5-20
 - Library 7-3
 - Mode 5-13
 - Options 5-14 - 5-17, 5-19
 - View 5-20 - 5-21, 5-23, 5-25, 5-27

- Window 5-33 - 5-34
- Messages 5-21
- Mode menu 5-13
- Mouse 1-28, 2-26
- Multisheet schematic 4-5 - 4-9
 - Connectivity rules 4-10
 - Creating a netlist for 4-9
- N**
- Net name
 - add 2-28 - 2-29
 - edit 2-28 - 2-29
 - Move 2-29
 - Repeat option 2-29
- Netlist
 - Buses 3-14
 - Create 2-43 - 2-45, 4-9, 5-14
 - Defined 2-2
 - Design verification 2-3
 - Export 2-44, 4-19, 5-15
 - External 1-28
 - Flat, creating 4-2, 4-4
 - Flat, multiple sheets 4-2 - 4-3, 4-9
 - Flat, single sheet 4-1
 - For simulation 2-43, 2-45, 4-12
 - Format 1-28
 - Formats 2-2, 2-4, 5-15
 - Generating schematics from 4-12
 - Hidden pins 2-44
 - Import 4-12, 5-15
 - Library objects 7-20
 - Macros 4-10, 4-12
 - Post-layout 1-28
 - Symbol 4-12
 - Updating 5-14, 5-16
 - VCC, GND 2-44

O

- Object
 - Cut and paste 2-38
 - Deleting 2-38
 - Selecting 2-35
- Options menu 5-14 - 5-17, 5-19 - 5-20
 - Annotate 5-16
 - Bill of material 5-17 - 5-18
 - Create netlist 5-14
 - Export netlist 5-15
 - Integrity test 5-14
 - List connectors 5-19 - 5-20
 - PCB, back annotation 5-16

- PCB, export parts and nets 5-16
- PCB, import back annotation 5-16
- PCB, import from audit file 5-17
- Report 5-19
- Symbol editor 5-16
- Update simulation 5-15
- OrCAD
 - Import 5-4 - 5-6
- Ortho Line 2-33
- P**
- PADS
 - Import 5-5
- Page Setup
 - Schematic 2-13 - 2-15, 5-9
- Parts
 - Creating 4-13, 4-15
- PCB
 - Back Annotation 5-16
 - Creating netlists for 5-16
 - Export Parts and Nets 5-16
 - Footprint 2-40
 - Import Back Annotation 5-16
 - Import from Audit File 5-17
 - Layout tools 5-16, 5-27
- pin
 - Adding 6-5, 6-7
 - Bus 4-15
 - editing 6-6
 - graphics 6-5
 - I/O Designation 6-5
 - Input 6-5
 - location 6-4
 - macro 4-13 - 4-14
 - name 2-24, 6-4
 - Netlist 2-44
 - numbering 2-22, 6-4
 - Numbers 2-24, 6-4
 - Output 6-5
 - Parameters 6-4
 - power 2-30, 2-44
 - power pin visibility 2-44
 - Types 2-32, 6-5 - 6-6
- Pin Parameters 2-41, 6-4
- Placing
 - Test points 2-45
 - Test probes 2-47
- Print
 - Page setup 5-9
- Schematic 2-14, 5-8
- Table setup 5-10
- Title block 5-10
- Printer Setup
 - Schematic 5-9
- Probes
 - Enable/Disable 5-30
 - Select on schematic 2-47
- Project
 - Add to 5-3
 - Adding existing sheets 4-6
 - Create 1-5
 - Delete 1-14
 - Information 1-14
 - Libraries 1-28, 7-9
 - Netlist 1-28
 - Resources 1-25, 1-27, 1-29
 - Save 1-28
 - Troubleshooting 4-9
- Q**
- Query 2-41
 - Bus 3-12, 3-14
- R**
- Rectangle 2-33, 6-7 - 6-8
- Redo 5-11
- Reference designators 2-19, 2-22 - 2-24, 2-40, 4-14, 4-19
 - Duplicates 4-19
- Repeat Drawing 2-29
- Repeat option 2-27, 2-29 - 2-30
- Report 2-4, 5-19, 7-8
- Resistors 2-21
- Ruler 2-13
 - Enable/Disable 5-29, 5-31
- S**
- Scanner 5-30
 - Move 5-31
- Schematic
 - Adding existing sheets 4-6
 - Adding new sheets 4-5
 - Backup 5-27
 - building 2-15
 - Center 5-32
 - Circuit definition 2-1
 - Clipboard 5-12
 - Connections 2-2
 - Copying to other programs 2-38

- Creating macros 4-13
- Design verification 2-3
- Display Configuration 5-20 - 5-21, 5-23, 5-25, 5-27
- Generating from a netlist 4-12
- Graphics 5-28
- Hierarchical 4-14
- Interfaces 2-4
- Large designs 2-2
- Libraries 2-2
- Link with Simulator 2-45
- Logical States 2-48
- Macro, difference 4-17
- Macros 4-16
- Multi-sheet 4-2 - 4-9
- Redraw 5-33
- Reports 2-4
- Ruler 5-29, 5-31
- Save 5-3
- Selecting objects 2-35
- simulating 2-46, 5-28
- Single sheet 4-1
- Symbols 2-18 - 2-23, 2-36 - 2-38, 2-40
- viewing 2-16 - 2-17
- Zoom 2-16 - 2-18, 5-32
- Schematic editor 2-1 - 2-2, 2-4 - 2-7, 2-13, 2-15 - 2-17, 2-19 - 2-27, 2-29, 2-31 - 2-32, 2-35, 2-37, 2-39 - 2-40, 2-43 - 2-47, 2-49
 - Coordinates 2-13
 - Menu 2-13
 - Page setup 2-13 - 2-14
 - Ruler 2-13
 - Screen layout 2-6 - 2-7, 2-9, 2-11, 2-13
 - Scroll bar 2-13
 - Starting 2-4 - 2-6
 - Table setup 2-15, 5-10
 - Top bar 2-7
 - Zoom buttons 2-9
- Scratchpad 5-2
- Screen layout, schematic editor 2-6 - 2-7, 2-9, 2-11, 2-13
 - Coordinates 2-13
 - Menu 2-13
 - Ruler 2-13
 - Scroll bar 2-13
 - Top Bar 2-7
- Scroll bars 2-13
- Sheet
 - Adding existing 4-6
 - Adding new 4-5
 - Close 5-2
 - Layout 2-13 - 2-15
 - Multi 4-2, 4-10
 - New 5-1
 - Open 4-6, 4-8
 - Removing 4-7
 - Single 4-1
- Signal
 - Power 2-30, 2-44
- Simulation
 - Enable/Disable Interactive Mode 5-27 - 5-28
 - SEE ALSO Probes
 - step 5-22
 - update from schematic 5-15
- Simulator 2-43, 2-45 - 2-47, 5-4
 - Buses 3-14
 - Connectivity database 2-43
 - Creating a netlist 2-43
 - Linking with 2-45
 - Selecting on schematic 2-47
- SPICE 8-1 - 8-6
- SPICE netlist 2-4, 8-1, 8-3, 8-6
- States
 - Logical display 2-48
- Status line
 - Enable/Disable 5-28
- Stretching buses 3-11
- Switches 2-22
- Symbol
 - Access 6-11
 - Adding pins 6-5
 - Annotation 2-22 - 2-23
 - Attributes 2-24, 2-26, 2-40
 - Capacitors 2-21
 - Connectors 2-22
 - Create 5-16, 6-1
 - Cut and paste 2-38
 - Delete 2-19, 2-38
 - Drag 2-36
 - Duplicate 2-19
 - Edit 2-40, 6-2, 6-10
 - Filter 2-20
 - Finding 2-19
 - Graphics 6-6 - 6-8
 - Libraries 2-49
 - Macro 4-13, 4-15, 4-18
 - Move 2-36

- Netlist 4-12
 - Overlap 2-20, 2-36, 5-27
 - parameters 6-9
 - PCB footprint 2-40
 - Pin parameters 2-41, 6-4, 6-6 - 6-7
 - placing 2-18 - 2-21, 2-23
 - Query 2-41
 - References 4-8
 - Replacing 2-39
 - Resistors 2-21
 - Rotating 2-24, 2-37
 - selecting 2-18
 - Testing 6-10
 - Type 2-20 - 2-21, 2-40
 - View 6-11
 - Symbol Editor 6-1 - 6-9, 6-11
 - Window 6-4
 - Symbol text
 - Editing 6-4
 - Enable/Disable 5-29
- T**
- Table
 - Enable/Disable 5-29
 - Setup 2-15, 5-10
 - Terminal
 - I/O 2-26
 - Text
 - Graphical 6-8
 - place a string 2-33
 - set size, font, and color 2-33
 - Title block 5-10
 - Tool bar 2-13 - 2-14
 - Toolbox
 - Editing 2-35, 2-37, 2-39
 - Graphics 2-33
 - Wires 2-26, 2-28, 2-30
 - Top-down design 4-21 - 4-22
 - Troubleshooting
 - Project contents 4-9
- U**
- Undo 5-11
 - Update Simulation 5-15
- V**
- View menu 5-20 - 5-21, 5-23, 5-25, 5-27
 - Display 5-23
 - General settings 5-26
 - Graphics 5-29
 - Grid 5-24
 - Move scanner 5-31
 - Preferences 5-21, 5-23, 5-25
 - Probes 5-30
 - Ruler 5-29, 5-31
 - Save settings now 5-28
 - Save settings on exit 5-28
 - Scanner 5-30
 - Symbol Text 5-29
 - Table 5-29
 - Text 5-22 - 5-23
 - Wires and buses 5-25 - 5-26
 - Viewlogic
 - Import schematic 5-7
- W**
- Window menu 5-33 - 5-34
 - Arrange icons 5-33
 - Cascade 5-33
 - Close all 5-34
 - Tile 5-33
 - Wire
 - Automatic 2-30 - 2-32, 2-34
 - Buses 2-29, 5-26
 - Connecting signals without wires 2-2
 - Draw 2-25 - 2-27, 2-29, 2-31
 - End 2-30
 - I/O terminal 2-26
 - Junctions 5-26
 - Labels 2-28 - 2-29
 - Line style 5-25
 - Manual 2-32
 - Net names 2-29
 - Orthogonal 5-26
 - PWR points 2-30
 - Wires and bus settings 5-25
 - Wires, drawing 2-26
- Z**
- Zoom
 - Buttons 2-13
 - Full Page 5-32
 - In/Out 5-32
 - previous 5-32
 - select 5-32