

# MicroSim Schematics

---

Schematic Capture Software

## User's Guide

  
**MicroSim**  
MicroSim Corporation  
20 Fairbanks  
Irvine, California 92718  
(714) 770-3022

Version 6.3, April, 1996.

Copyright 1996, MicroSim Corporation. All rights reserved.  
Printed in the United States of America.

## TradeMarks

Referenced herein are the registered trademarks used by MicroSim Corporation to identify its products. MicroSim Corporation is the exclusive owner of "MicroSim," "PSpice," "PLogic," "PLSyn," "PLogic," "PLSyn" and "Polaris."

Additional marks of MicroSim include: "StmEd," "Stimulus Editor," "Probe," "Parts," "Monte Carlo," "Analog Behavioral Modeling," "Device Equations," "Digital Simulation," "Digital Files," "Filter Designer," "Schematics," "MicroSim PCBoards," "PSpice Optimizer," and variations thereon (collectively the "Trademarks") are used in connection with computer programs. MicroSim owns various trademark registrations for these marks in the United States and other countries.

SPECCTRA is a registered trademark of Cooper & Chyan Technology, Inc.

Microsoft, MS-DOS, Windows, Windows NT and the Windows logo are either registered trademarks or trademarks of Microsoft Corporation.

Adobe, the Adobe logo, Acrobat, the Acrobat logo, Exchange and PostScript are trademarks of Adobe Systems Incorporated or its subsidiaries and may be registered in certain jurisdictions.

EENET is a trademark of Eckert Enterprises.

*All other company/product names are trademarks/registered trademarks of their respective holders.*

## Copyright Notice

Except as permitted under the United States Copyright Act of 1976, no part of this publication may be reproduced or distributed in any form or by any means, or stored in a data base or retrieval system, without the prior written permission of MicroSim Corporation.

As described in the license agreement, you are permitted to run one copy of the MicroSim software on one computer at a time. Unauthorized duplication of the software or documentation is prohibited by law.

## Technical Support

Internet            Tech.Support@MicroSim.com

Phone            (714) 837-0790

FAX            (714) 455-0554

AUTOFAX        (714) 454-3296

BBS            (714) 830-1550

## Sales Department

Internet            Sales@MicroSim.com

Phone            800-245-3022

---

# Contents

## Before You Begin

Welcome to MicroSim . . . . .	xv
MicroSim Schematics Overview . . . . .	xvi
How to Use this Guide . . . . .	xvii
Typographical Conventions . . . . .	xvii
Mouse Conventions . . . . .	xviii
Related Documentation . . . . .	xix
On-Line Help . . . . .	xx
New Features in this Release . . . . .	xxi

## Chapter 1 Getting Started

Overview . . . . .	1-1
Using Schematics . . . . .	1-2
Example—Drawing a Schematic . . . . .	1-4
Starting a New Design . . . . .	1-6
Checking Symbol Libraries Configuration . . . . .	1-6
Selecting and Placing Parts . . . . .	1-7
Drawing and Labeling Wires . . . . .	1-10
Drawing and Labeling Buses . . . . .	1-11
Changing Reference Designators and Part Values . . . . .	1-13
Moving Parts, Wires and Text . . . . .	1-14
Placing Ports . . . . .	1-15
Placing Power and Ground Symbols . . . . .	1-16
Saving Your Work . . . . .	1-17

## Chapter 2 Using the Schematic Editor

Overview . . . . .	2-1
Components of a Design . . . . .	2-3
Parts . . . . .	2-3
Symbols . . . . .	2-4
Ports . . . . .	2-4

Attributes . . . . .	2-4
Annotations . . . . .	2-4
Connections . . . . .	2-5
Main Window . . . . .	2-6
Menus . . . . .	2-6
Toolbar . . . . .	2-7
Status Bar . . . . .	2-9
Keyboard . . . . .	2-9
Refreshing the Screen . . . . .	2-10
Configuring Schematics . . . . .	2-11
Configuring Symbol Libraries . . . . .	2-12
Changing Fonts . . . . .	2-17
Changing Page Size . . . . .	2-18
Changing Page Settings . . . . .	2-19
Changing Grid and Gravity . . . . .	2-21
Setting the Autosave Interval . . . . .	2-24
Controlling the Display in Schematics . . . . .	2-25
Configuring Colors . . . . .	2-26
Changing the Select Part List Size . . . . .	2-28
Changing Application Settings . . . . .	2-29
Zooming and Panning in Schematics . . . . .	2-32
Zooming . . . . .	2-32
Setting Zoom Parameters . . . . .	2-33
Fitting to a Page . . . . .	2-34
Panning . . . . .	2-34
Using the Message Viewer . . . . .	2-37
On-line Help . . . . .	2-38
Locating the Source of a Message . . . . .	2-38
Indicated Severity . . . . .	2-38
Additional Information . . . . .	2-39
Closing the Message Viewer . . . . .	2-39

## Chapter 3 Creating and Editing Designs

Overview . . . . .	3-1
Starting the Schematic Editor . . . . .	3-3
Opening a File . . . . .	3-3
Finding Parts . . . . .	3-4
Getting Parts by Name . . . . .	3-5
Searching for Parts in the Libraries . . . . .	3-6
Placing and Editing Parts . . . . .	3-9
Rotating and Flipping Parts . . . . .	3-10

---

Editing Part Attributes . . . . .	<b>3-12</b>
Global Editing of Attributes . . . . .	<b>3-18</b>
Editing the Default Attributes of a Symbol . . . . .	<b>3-19</b>
Repeating Part Placements . . . . .	<b>3-20</b>
Automatically Assigning Reference Designators . . . . .	<b>3-22</b>
Example—Using Auto-Repeat and Auto-Naming . . . . .	<b>3-23</b>
Replacing Parts . . . . .	<b>3-25</b>
Placing Power and Ground Symbols . . . . .	<b>3-27</b>
Placing Power and Ground Symbols . . . . .	<b>3-27</b>
Creating Custom Power and Ground Symbols . . . . .	<b>3-28</b>
Using Wires and Buses . . . . .	<b>3-29</b>
Drawing and Labeling Wires . . . . .	<b>3-29</b>
Drawing and Labeling Buses . . . . .	<b>3-31</b>
Automatically Labeling Wires and Buses . . . . .	<b>3-33</b>
Specifying Drawing Options . . . . .	<b>3-34</b>
Using Ports . . . . .	<b>3-38</b>
Selecting and Moving Parts, Wires and Attributes . . . . .	<b>3-40</b>
Selecting . . . . .	<b>3-40</b>
Moving . . . . .	<b>3-41</b>
Searching for and Selecting Parts . . . . .	<b>3-41</b>
Cutting, Copying and Pasting . . . . .	<b>3-43</b>
Creating and Editing Title Blocks . . . . .	<b>3-46</b>
Editing Page Title . . . . .	<b>3-46</b>
Entering Information into the Title Block . . . . .	<b>3-47</b>
Creating a Custom Title Block . . . . .	<b>3-48</b>
Creating and Editing Annotation Items . . . . .	<b>3-49</b>
Defining and Adding Annotation Symbols . . . . .	<b>3-51</b>
Creating and Editing Multi-sheet Designs . . . . .	<b>3-53</b>
Adding a Page to a Design . . . . .	<b>3-53</b>
Creating Connections between Pages . . . . .	<b>3-54</b>
Viewing Multiple Pages . . . . .	<b>3-55</b>
Cutting, Copying and Pasting between Pages . . . . .	<b>3-55</b>
Deleting a Page . . . . .	<b>3-56</b>
Printing Your Design . . . . .	<b>3-57</b>
Scaling . . . . .	<b>3-58</b>
Closing the Schematic Editor . . . . .	<b>3-63</b>

## Chapter 4 Using the Symbol Editor

Overview . . . . .	4-1
Components . . . . .	4-3
Symbols . . . . .	4-3
Packaging Information . . . . .	4-3
Footprints . . . . .	4-4
Simulation Models . . . . .	4-4
Starting the Symbol Editor . . . . .	4-5
Loading a Library for Editing . . . . .	4-5
Saving your Changes . . . . .	4-6
Returning to the Schematic Editor . . . . .	4-7
Starting Automatically . . . . .	4-7
Symbol Editor Window . . . . .	4-8
Refreshing the Screen . . . . .	4-8
Menus . . . . .	4-8
Toolbar . . . . .	4-9
Title Bar . . . . .	4-10
Keyboard . . . . .	4-11
Changing Text Characteristics . . . . .	4-12
Attribute Text . . . . .	4-12
Pin Name and Number . . . . .	4-14
Free-Standing Text . . . . .	4-15
Changing Grid and Gravity . . . . .	4-16
Grid On . . . . .	4-16
Snap-to-Grid . . . . .	4-16
Stay-on-Grid . . . . .	4-17
Grid Spacing . . . . .	4-18
Gravity . . . . .	4-18
Text Stay-on-Grid . . . . .	4-19
Zooming and Panning . . . . .	4-20
Printing Symbols . . . . .	4-21

## Chapter 5 Creating and Editing Symbols

Overview . . . . .	5-1
Creating New Symbols . . . . .	5-3
Using the Symbol Creation Wizard . . . . .	5-3
Creating a Symbol by Copying Another Symbol . . . . .	5-4
Making a Copy of a Symbol . . . . .	5-5
Importing a symbol definition . . . . .	5-6
Using AKO Symbols . . . . .	5-7
Editing Existing Symbols . . . . .	5-10

Drawing Symbol Graphics . . . . .	<b>5-11</b>
Elements of a Symbol . . . . .	<b>5-11</b>
Rotating and Flipping . . . . .	<b>5-15</b>
Selecting . . . . .	<b>5-16</b>
Moving . . . . .	<b>5-17</b>
Cutting, Copying and Pasting . . . . .	<b>5-17</b>
Defining and Editing Pins . . . . .	<b>5-19</b>
Specifying Pin Types . . . . .	<b>5-19</b>
Defining and Editing Hidden Power and Ground Pins . . . . .	<b>5-22</b>
Specifying Symbol Origin and Bounding Box . . . . .	<b>5-22</b>
Editing Symbol Attributes . . . . .	<b>5-24</b>
Using Symbol Aliases . . . . .	<b>5-26</b>
Specifying Part Packaging Information . . . . .	<b>5-27</b>
Pin Assignment Lists . . . . .	<b>5-27</b>
Packaging Definitions . . . . .	<b>5-28</b>
Creating a New Package Definition . . . . .	<b>5-28</b>
Copying a Package Definition . . . . .	<b>5-30</b>
Editing a Package Definition . . . . .	<b>5-31</b>
Deleting a Package Definition . . . . .	<b>5-39</b>
Configuring Package Types . . . . .	<b>5-39</b>
Configuring Custom Libraries . . . . .	<b>5-41</b>

## **Chapter 6 Creating and Editing Hierarchical Designs**

Overview . . . . .	<b>6-1</b>
Hierarchical Design Methods . . . . .	<b>6-3</b>
Creating and Editing Hierarchical Blocks . . . . .	<b>6-4</b>
Associating an Existing Schematic . . . . .	<b>6-8</b>
Creating and Editing Hierarchical Symbols . . . . .	<b>6-9</b>
Creating a Hierarchical Symbol . . . . .	<b>6-9</b>
Converting Hierarchical Blocks to Symbols . . . . .	<b>6-11</b>
Using Interface Ports . . . . .	<b>6-12</b>
Setting Up Multiple Views . . . . .	<b>6-13</b>
Translators . . . . .	<b>6-13</b>
Navigating through Hierarchical Designs . . . . .	<b>6-15</b>
Assigning Instance-Specific Part Values . . . . .	<b>6-17</b>
Passing Information between Levels of Hierarchy . . . . .	<b>6-18</b>
Example—Creating a Hierarchical Design . . . . .	<b>6-20</b>
Drawing the Top-Level Schematic . . . . .	<b>6-20</b>
Drawing the Lower-Level Schematic . . . . .	<b>6-24</b>

## Chapter 7 Preparing Your Design for Simulation

Overview . . . . .	7-1
Creating Designs for Simulation and Board Layout . . . . .	7-3
Specifying Part Attributes . . . . .	7-3
Handling Unmodeled Pins . . . . .	7-4
Specifying Simulation Model Libraries . . . . .	7-5
Creating Symbols for Existing Simulation Models . . . . .	7-6
Editing Simulation Models from Schematics . . . . .	7-8
Adding and Defining Stimulus . . . . .	7-9
Placing Stimulus Sources . . . . .	7-9
Using the Stimulus Editor . . . . .	7-10
Setting Up Analyses . . . . .	7-10
Starting the Simulator . . . . .	7-10
Viewing Results . . . . .	7-11
Viewing Results as You Simulate . . . . .	7-11
Using Markers . . . . .	7-11
Configuring Probe Display of Simulation Results . . . . .	7-12

## Chapter 8 Targeting Your Design for Programmable Logic

Overview . . . . .	8-1
Targeting Parts for Programmable Logic . . . . .	8-3
Creating and Editing DSL Blocks . . . . .	8-4
Simulating a Programmable Logic Design from Schematics . . . . .	8-5
Using PLSyn . . . . .	8-6
Updating the Schematic with the PLD(s) . . . . .	8-7

## Chapter 9 Preparing Your Design for Board Layout

Overview . . . . .	9-1
Connectors . . . . .	9-3
Placing Connectors . . . . .	9-3
Creating Single-pin Connector Symbols . . . . .	9-5
Packaging the Parts in Your Design . . . . .	9-6
Assigning Reference Designators Manually . . . . .	9-7
Assigning Reference Designators Automatically . . . . .	9-9
Setting Package Class Priorities . . . . .	9-10
Generating a Bill of Materials Report . . . . .	9-12
Printing and Saving the Report . . . . .	9-13
Customizing the Format of the Report . . . . .	9-14
User Defined Component Information . . . . .	9-15
Exporting to a Spreadsheet or Database Program . . . . .	9-16



Swapping Pins . . . . .	9-17
Interfacing to MicroSim PCBoards . . . . .	9-18
Specifying Trace Properties . . . . .	9-18
Specifying Component Locations . . . . .	9-19
Cross-Probing . . . . .	9-20
Applying Backward ECOs . . . . .	9-21
Applying Forward ECOs . . . . .	9-22
Interfacing to Other Board Layout Products . . . . .	9-23
Layout Mapping Files . . . . .	9-24
Back Annotation . . . . .	9-28

## Appendix AImporting OrCAD SDT Schematics

Importing OrCAD Files . . . . .	A-1
Import Options . . . . .	A-3
Package Types Dialog Box . . . . .	A-4
PSpice Simulation Device Types Dialog Box . . . . .	A-5
Translating Multi-Page Schematics . . . . .	A-6
Translating Hierarchical Schematics . . . . .	A-6
Translating Large Designs . . . . .	A-7
Text Size . . . . .	A-7
Connecting Signal via Labels . . . . .	A-8
Differences between OrCAD SDT and Schematics . . . . .	A-9

## Appendix BLibrary Expansion and Compression Utility

Running the Utility . . . . .	B-3
-------------------------------	-----

## Appendix CAdvanced Netlisting Configuration Items

Specifying PSpice Node Name Netlisting Preferences . . . . .	C-1
Specifying Board Layout Node Name Netlisting Preferences . . . . .	C-3
Customizing EDIF Netlists . . . . .	C-4

## Appendix DAttribute List

## Appendix ESymbol Libraries

## Glossary

## Index

---

# Figures

Figure 1-1	Interaction of MicroSim Software Programs and Files . . . . .	<b>1-3</b>
Figure 1-2	Opto-isolated, Addressable Serial-to-parallel Converter Circuit . . . . .	<b>1-5</b>
Figure 3-1	Rotating a Part . . . . .	<b>3-10</b>
Figure 3-2	Flipping a Part . . . . .	<b>3-11</b>
Figure 3-3	Placing Resistors with Various Vertical and Horizontal Offsets . . . . .	<b>3-21</b>
Figure 3-4	Auto-Naming for Bus Labels . . . . .	<b>3-23</b>
Figure 3-5	Orthogonal Wire Drawing . . . . .	<b>3-34</b>
Figure 3-6	Rubberbanding of Wires . . . . .	<b>3-36</b>
Figure 3-7	Off-page Port . . . . .	<b>3-38</b>
Figure 3-8	Global Ports . . . . .	<b>3-38</b>
Figure 3-9	Region of Interest Box . . . . .	<b>3-40</b>
Figure 3-10	Printing with Auto-Fit Enabled . . . . .	<b>3-59</b>
Figure 3-11	Zoom Factor Set to 100% with Printer Configured in Portrait Mode . . . . .	<b>3-59</b>
Figure 3-12	Zoom Factor Set to 200% with Printer Configured in Portrait Mode . . . . .	<b>3-60</b>
Figure 3-13	User-definable Zoom Enabled in Portrait Mode . . . . .	<b>3-61</b>
Figure 3-14	User-definable Zoom Enabled in Landscape Mode . . . . .	<b>3-62</b>
Figure 5-1	Rotating a Drawing Element . . . . .	<b>5-15</b>
Figure 5-2	Flipping a Drawing Element . . . . .	<b>5-15</b>
Figure 5-3	Pin Types . . . . .	<b>5-19</b>
Figure 6-1	Top-level Schematic Drawing for CMOS Inverter . . . . .	<b>6-20</b>
Figure 6-2	Schematic of CMOS Inverter . . . . .	<b>6-24</b>
Figure 9-1	Entire Connector Symbol . . . . .	<b>9-4</b>
Figure 9-2	Single Pin Symbol . . . . .	<b>9-4</b>
Figure 9-3	Bill of Materials Report . . . . .	<b>9-12</b>

---

# Tables

Table 1-1	Remaining Parts to be Placed . . . . .	<b>1-9</b>
Table 2-1	Schematic Editor Toolbar Icons . . . . .	<b>2-7</b>
Table 2-2	Schematic Editor Function Keys . . . . .	<b>2-9</b>
Table 2-3	Schematics Colors Items . . . . .	<b>2-27</b>
Table 3-1	Text Characteristics . . . . .	<b>3-16</b>
Table 4-1	Symbol Editor Toolbar Icons . . . . .	<b>4-9</b>
Table 4-2	Symbol Editor Function Keys . . . . .	<b>4-11</b>
Table 4-3	Display Characteristics . . . . .	<b>4-13</b>
Table 4-4	Content Options . . . . .	<b>4-22</b>
Table 9-1	Distinctions between Connectors and Ports . . . . .	<b>9-3</b>
Table 9-2	Trace Properties Attributes . . . . .	<b>9-18</b>
Table 9-3	Component Location Attributes . . . . .	<b>9-19</b>
Table 9-4	Supported Layout Packages and File Formats . . . . .	<b>9-23</b>
Table B-1	List File Format . . . . .	<b>B-2</b>
Table E-1	Symbol Libraries . . . . .	<b>E-1</b>

---

# Before You Begin

---

## Welcome to MicroSim

Welcome to the MicroSim family of products. Whichever programs you have purchased, we are confident that you will find they meet your circuit design needs. They provide an easy-to-use, integrated environment for creating, simulating and analyzing your circuit designs from start to finish.

# MicroSim Schematics Overview

MicroSim Schematics is a schematic capture front-end program with a direct interface to other MicroSim programs and options.

All in one environment you can do the following using Schematics:

- design and draw circuits
- simulate circuits using MicroSim PSpice or MicroSim PLogic
- analyze simulation results using MicroSim Probe
- graphically characterize simulation stimuli using the fully integrated Stimulus Editor so stimulus definitions are automatically associated with the appropriate symbols
- graphically characterize simulation models using the fully integrated MicroSim Parts utility so model definitions are automatically associated with the appropriate symbols
- interface to MicroSim PSpice Optimizer for analog circuit performance optimization
- interface to MicroSim PLSyn for programmable logic synthesis
- interface to MicroSim PCBoards for printed circuit board layout

The MicroSim family of products is fully integrated, giving you the flexibility to work through your circuit design in a consistent environment. The MicroSim family of products work together, with Schematics as the central point of control.

# How to Use this Guide

This guide is designed so you can quickly find the information you need to use Schematics.

This guide assumes that you are familiar with MicroSoft Windows (3.1, NT or 95), including how to use icons, menus and dialog boxes. It also assumes you have a basic understanding about how Windows manages applications and files to perform routine tasks, such as starting applications and opening and saving your work. If you are new to Windows, please review your *MicroSoft Windows User's Guide*.

For UNIX users:

All screen captures in this manual are of Windows dialog boxes and windows. Most options in these dialog boxes and windows are available in your operating environment. When certain options are not available to you, or you must do something differently than what is primarily outlined, information specific to your platform is provided.

## Typographical Conventions

Before using Schematics, it is important to understand the terms and typographical conventions used in this documentation.

This guide generally follows the conventions used in the *MicroSoft Windows User's Guide*. Procedures for performing an operation are generally numbered with the following typographical conventions.

Notation	Examples	Description
<code>[Ctrl] + r</code>	Press <code>[Ctrl]+r</code>	A specific key or key stroke on the keyboard.
monospace font	<code>analog.slb</code>  Type VAC...	Library files and file names.  Commands/text entered from the keyboard.

## Mouse Conventions

- If you have a multiple-button mouse, the left mouse button is the primary mouse button, unless you have configured it differently.
- “Point” means to position the mouse pointer until the tip of the pointer rests on whatever you want to point to on the screen.
- “Click” means to press and then immediately release the mouse button without moving the mouse.
- “Right-click” means to press the right mouse button and then immediately release the mouse button without moving the mouse.
- “Drag” means to point and then hold down the mouse button as you move the mouse.

# Related Documentation

The documentation for all MicroSim products is available in both hard-copy and on-line. The documentation you receive depends on the software configuration you have purchased.

Manual Name *	Description
MicroSim Schematics User's Guide	Provides information about how to use Schematics, which is a schematic capture front-end program with a direct interface to other MicroSim programs and options.
MicroSim PCBoards User's Guide	Provides information about MicroSim PCBoards, which is a PCB layout editor that allows you to specify printed circuit board structure, as well as the components, metal and graphics required for fabrication.
MicroSim PCBoards Autorouter User's Guide	Provides information on the integrated interface to Cooper & Chyan Technology's (CCT) SPECCTRA autorouter in MicroSim PCBoards.
MicroSim PCBoards Tutorials	Provides a collection of tutorials to help you quickly get started using MicroSim PCBoards.
MicroSim PSpice A/D & Basics+ User's Guide	Describes the capabilities of PSpice A/D, Probe, Stimulus Editor, and Parts. It provides examples for demonstrating the process of specifying simulation parameters, analyzing simulation data results, editing device stimuli, and creating models.
MicroSim PSpice A/D User's Guide <i>for the Sun &amp; HP</i>	Describes the capabilities of PSpice A/D, Probe, Stimulus Editor, and Parts. It provides examples for demonstrating the process of specifying simulation parameters, analyzing simulation data results, editing device stimuli, and creating models.
MicroSim PSpice & Basics User's Guide	Provides information on PSpice & and PSpice Basics which are circuit analysis programs that allow you to create, simulate and test circuit designs containing analog components.
MicroSim PSpice A/D Reference Manual	Provides reference material for PSpice A/D. Also included: detailed descriptions of the simulation controls and analysis specifications, start-up option definitions, and a list of device types in the analog and digital model libraries. User interface commands are provided to instruct you on each of the screen commands.
MicroSim Application Notes	Provides a variety of articles that show you how a particular task can be accomplished using MicroSim's products, and examples that demonstrate a new or different approach to solving an engineering problem.



Manual Name*	Description
MicroSim PSpice Optimizer User's Guide	Provides information for using the PSpice Optimizer for analog performance optimization.
MicroSim PLSyn Programmable Logic Synthesis	Provides information for using programmable logic synthesis.
MicroSim/AMD PLD Design System User's Guide	Provides information about the implementation of a PLD design targeted for using one or more of AMD devices.
MicroSim Filter Designer User's Guide	Provides information about designing electronic frequency selective filters.
Library Reference Manual**	Provides a complete list of the analog and digital parts in the model and symbol libraries.

\* On-Line documentation is available only to those users who install MicroSim products by CD-ROM.

\*\* This manual is provided in on-line format *only*.

## On-Line Help

Pressing **F1** or selecting Contents from the Help menu brings up an extensive on-line help system.

The on-line help includes:

- Step-by-step instructions on how to use Schematics features.
- Reference information about menu and dialog box options.
- Technical Support information.

If you are not familiar with the Windows (3.1, NT or 95) Help System, select How to Use Help from the Help menu.

# New Features in this Release

## Graphical Part Browser with Search Capability

You can now preview a part while browsing the alphabetical list of available parts. With this list you can search for specific part names using wildcards and also search based on the part description. You can still browse by library if desired.

The Part Browser dialog box can now be left open while you place parts.

## Select Part List Box on the Toolbar

A drop-down list box has been added to the toolbar. It keeps a list of the names of the last ten parts placed. To place another instance of a part in the list, click on the drop-down arrow and select from the list.

You can also type in the name of a part in the Select Part text box to place a part without using the Part Browser dialog box.

## Improved Error Message Handling (Windows Only)

Schematics now provides a Message Viewer that logs and displays error messages occurring during netlisting, packaging and other operations. The Message Viewer can be displayed simultaneously with Schematics so you can see messages as they are logged and as you resolve errors.

Double-clicking on an error will take you to the source of the error on the schematic. Pressing **[F1]** on a selected error will bring up on-line help, further describing the message and offering possible solutions. Error messages are saved between sessions.

## **Symbol Creation Wizard**

A symbol creation wizard has been added to the symbol editor, which leads you through creating a new symbol.

## **Text Editor Included (Windows Only)**

Schematics now ships with its own text editor that can handle large files.

## **Cross-Probing with MicroSim PCBoards (Windows Only)**

You can now select nets and components in MicroSim PCBoards and have them highlight in Schematics. You can also select a part or wire in Schematics and have the corresponding part or net highlight in MicroSim PCBoards.

---

# Getting Started

---

# 1

## Overview

This chapter describes Schematics: what it is, what it can do and how you can use it.

This chapter has the following sections:

*Using Schematics on page 1-2* provides a broad overview and describes various functions.

*Example—Drawing a Schematic on page 1-4* provides a step-by-step example of creating a schematic.

# Using Schematics

Schematics is a schematic capture front-end program that provides a convenient system for:

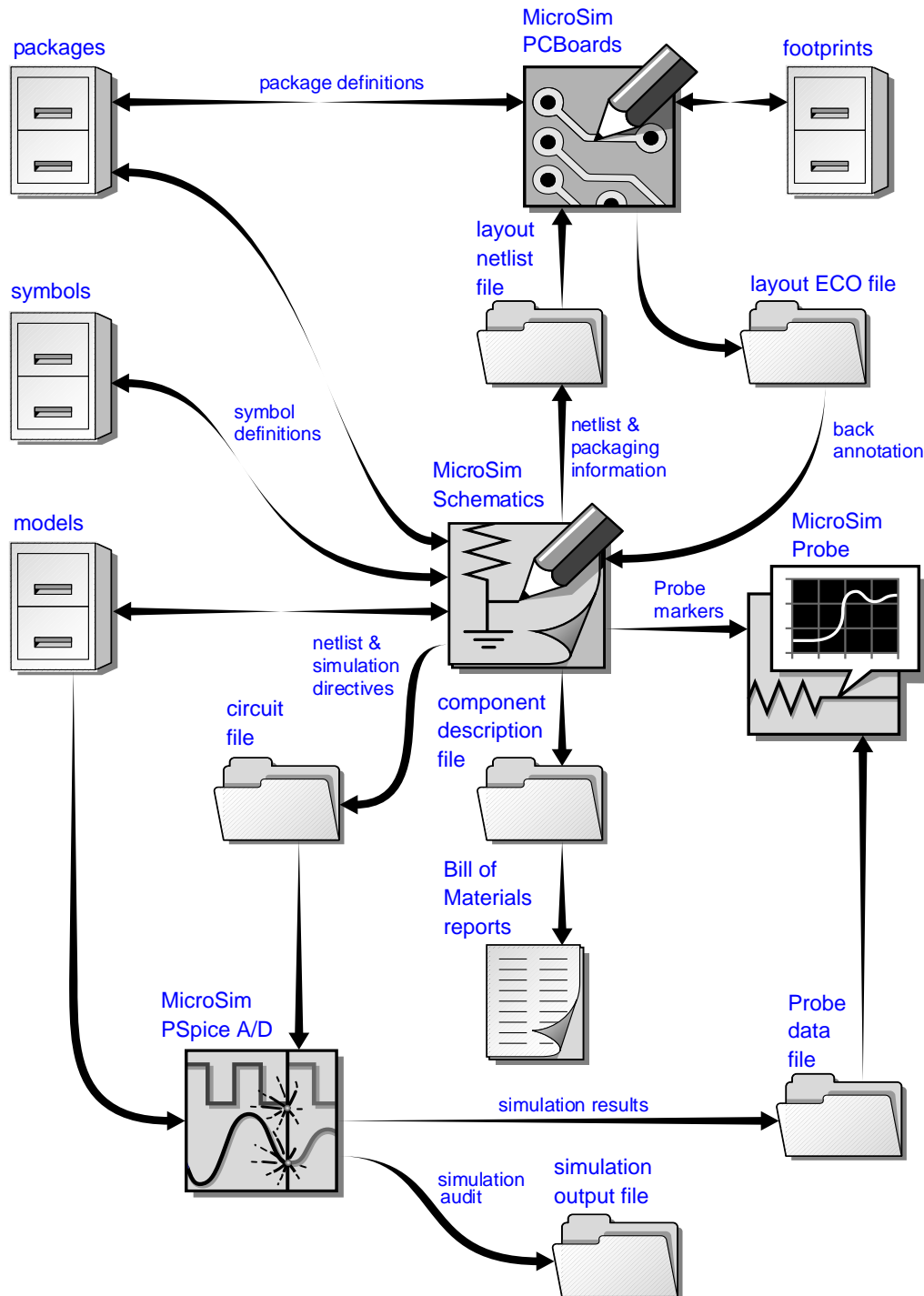
- creating and managing circuit drawings
- setting up and running simulations
- evaluating simulation results using MicroSim Probe
- creating netlists (for MicroSim PCBoards and other external PCB layout packages)

An important prerequisite to building a schematic is availability of the necessary devices (in the form of symbols) for assembly. Schematics has extensive symbol libraries and a fully integrated symbol editor for creating your own symbols or modifying existing symbols.

The main functions of Schematics are:

- creating and editing designs
- creating and editing symbols
- creating and editing hierarchical designs
- preparing a design for simulation
- preparing a design for board layout

These primary functions are described in the following chapters.



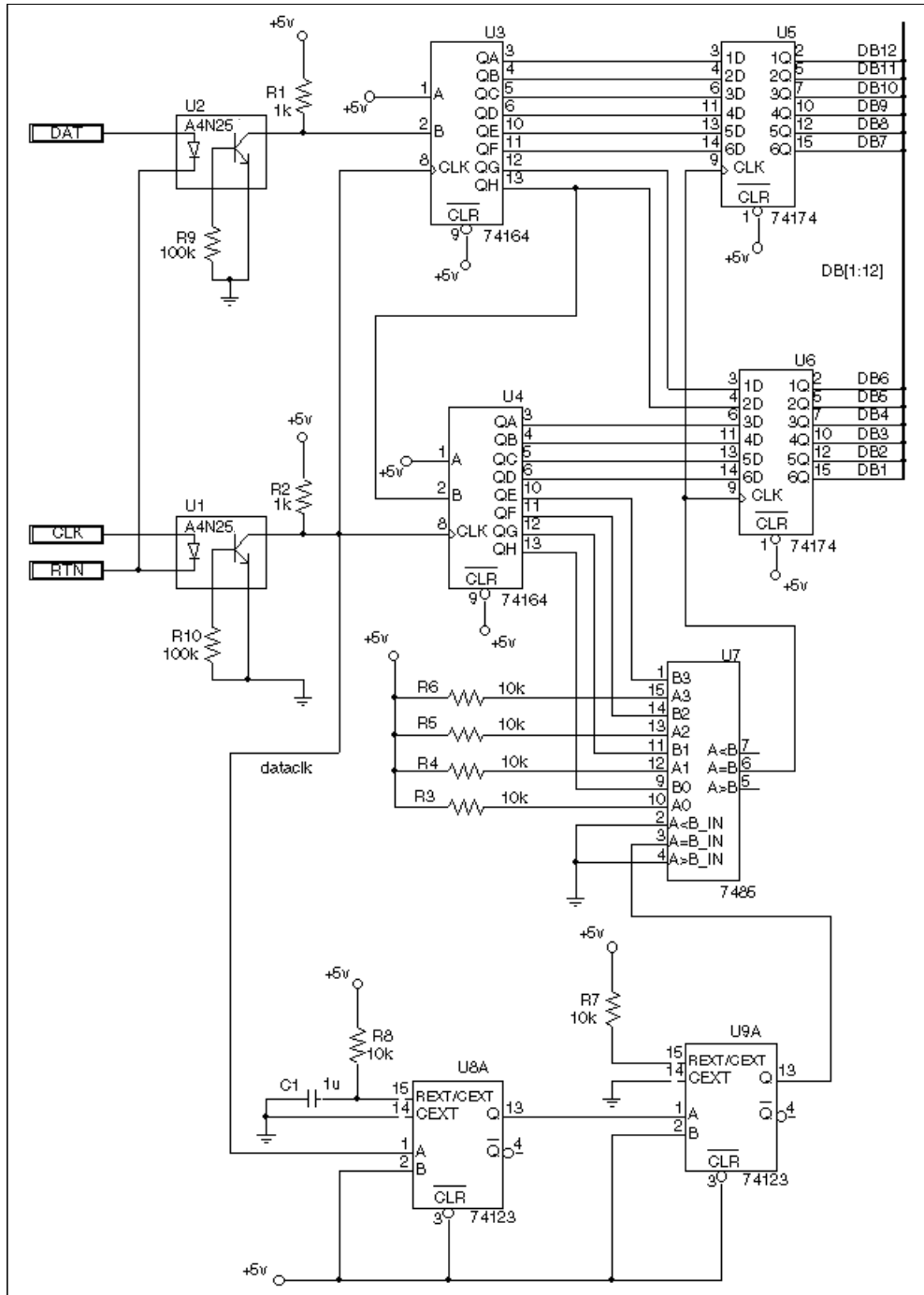
**Figure 1-1** *Interaction of MicroSim Software Programs and Files*

## Example—Drawing a Schematic

The following example demonstrates the basic drawing features of how to draw a schematic. It shows you:

- how to start the schematic editor and begin a new design
- how to check to see which libraries are configured for Schematics
- how to place parts on a schematic
- how to connect the part using wires and buses
- how to label wires and buses
- how to change reference designators and part values
- how to move parts, wires and text
- how to use ports on a schematic
- how to place power and ground symbols on a schematic
- how to save your design

Follow this example to create the circuit shown in Figure 1-2.



**Figure 1-2** Opto-isolated, Addressable Serial-to-parallel Converter Circuit





## Starting a New Design

Start the schematic editor by double-clicking on the Schematics icon in the MicroSim program group. An empty schematic page displays.

If you already have Schematics running with another schematic displayed, click the New File icon to start a new schematic.

## Checking Symbol Libraries Configuration

When you installed Schematics, you selected a set of libraries to be installed. These are global libraries, which means the symbols contained in them are available to be used in any new or existing schematic.

Check to see that you have the correct symbol libraries configured for this example:

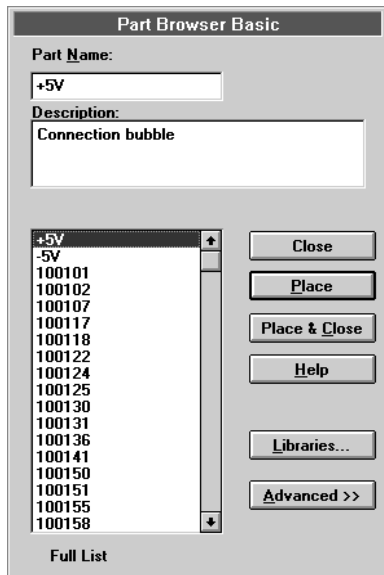
- 1 Select Editor Configuration from the Options menu.
- 2 Check that the following libraries are included in the Libraries list box in the upper left of the Editor Configuration dialog box:

7400 [.slb,.plb]  
analog [.slb,.plb]  
opto [.slb,.plb]  
port [.slb]

**Note** *If you are using the evaluation version of Schematics, you will be using "eval.slb".*

## Selecting and Placing Parts

- 1 Click the Select Part icon to display the Part Browser dialog box.

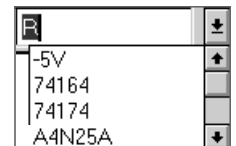


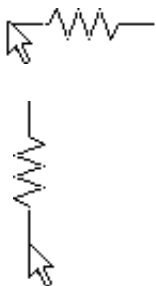
One of two Part Browser dialog boxes may be displayed: the Part Browser Advanced or the Part Browser Basic. If the Part Browser Advanced dialog, click <<Basic to display the Part Browser Basic.

The Full List in the Part Browser dialog box represents all the parts in the configured symbol libraries that are available for your use.

- 2 There are several ways to select a part in the Part Browser dialog box:
  - If you know the name of the part, type the name in the Part Name text box.
  - Select the part name from the Full List of part names.
  - Click Libraries to view the Library Browser dialog box, select a library, and select the part name from that library's list of parts.
- 3 Click Place to place the part (with the browser remaining open) or click Place & Close (to place the part and close the browser). If you choose to leave the browser open, you can move it out of your way by clicking on the title bar of the dialog box and dragging it to a new location.

Another method of selecting a part is to use the Select Part list box on the toolbar. You can scroll and select a previously placed part, or you can type in the name of the part you want to place.





As you place parts, the numerical portion of the reference designator is automatically assigned. For instance, if you place resistor R2, the next resistor you place will be designated R3.

## Placing resistors R1 and R2

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on 1-7).
- 2 Type R in the Part Name text box.
- 3 Click Place & Close.

With the mouse, move the outline of the resistor.

Note that as you move the pointer, the X and Y coordinates at the left of the Status Bar (bottom of the window) change. These coordinates show the location of the pointer on the drawing to the closest 0.01 inch (or closest mm if you are using a metric page size).

- 4 Press **[Ctrl]+R** to rotate the resistor before placing it.
- 5 Move the pointer to 2.40, 1.80 coordinates (within a few hundredths of the inch is close enough) and click to place the resistor on the schematic. If the Stay-on-Grid option is on, parts are automatically placed on the nearest grid point.
- 6 Move the pointer to 2.40, 3.90 and click again to place the second resistor on the schematic.
- 7 Right-click to stop placing the part.

## Placing resistors R3 through R6

You can quickly place resistors R3 through R6 using the Auto-Repeat function.

- 1 Select Auto-Repeat from the Options menu to display the Auto-Repeat dialog box.
  - a Set Horizontal Offset to 00.00 and Vertical Offset to -00.20.
  - b Select the Enable Auto-Repeat check box.
  - c Click OK.
- 2 Select R from the Select Part list box on the toolbar.
- 3 Place the pointer in the approximate position for the placement of R3 and click to place the part.
- 4 Press **[Space]** three times to place three more resistors above the first.

### Placing resistors R7 through R10

- 1

Select R from the Select Part list box on the toolbar.
- 2

Press Ctrl+R to rotate the resistor before placing it.
- 3

Place four resistors in the approximate locations of R7, R8, R9 and R10.
- 4

Right-click to stop placing resistors.

### Placing the remaining parts on the schematic

- 1

Click the Select Part icon.
- 2

In the Part Browser dialog box, select each part listed in Table 1-1 from the Part list box.
- 3

Place the part on the schematic in the approximate location shown in Figure 1-2.

**Table 1-1** *Remaining Parts to be Placed*

Reference Designator	Part Name
U1	A4N25
U2	A4N25
U3	74164
U4	74164
U5	74174
U6	74174
U7	7485
U8A	74123
U9A	74123

## Drawing and Labeling Wires

Draw the wire labeled `dataclk` to connect pin 8 (CLK) on U3 and pin 1 (A) on U8A.



### Drawing the dataclk wire

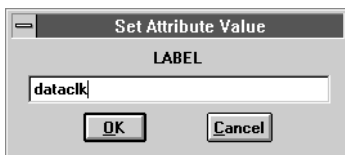
- 1 Click the Draw Wire icon.

The pencil pointer indicates that you are ready to draw a wire.
- 2 Click pin 8 of U3 to begin the wire.
- 3 Following the illustration in Figure 1-2, click where you want each vertex of the wire. Each click ends a wire segment and starts a new one.
- 4 Click pin 1 of U8A

Notice that the wire is now ended since you clicked on a pin. The pointer remains in the shape of a pencil and you are ready to start another wire.
- 5 Wire the rest of the schematic, except for the wires from the right sides of U5 and U6 to the bus.
- 6 Right-click to stop drawing wires.

### Labeling the dataclk wire

Label the wire connecting the CLK pin on U3 to the A pin of U8A.



- 1 Double-click any segment of the wire to display the Set Attribute Value dialog box.
- 2 Type `dataclk` in the LABEL text box.
- 3 Click OK.

## Drawing and Labeling Buses

Draw the bus labeled DB[1-12].

### Drawing the bus

- 1 Click the Draw Bus icon.

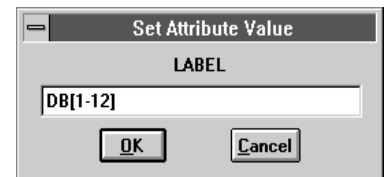
The pointer is now shaped like a pencil (as it was when you were drawing wires).

- 2 Click where you want to start the bus.
- 3 Move the mouse and click where you want to end the bus.
- 4 Right-click to stop drawing buses.



### Labeling the bus

- 1 Double-click any segment of the bus to display the Set Attribute Value dialog box.
- 2 Type DB[ 1-12 ] in the LABEL text box.
- 3 Click OK.



### Connecting wires to the bus

You can use the Auto-Repeat function to place the wires that connect the pins to the bus, since the wires will be the same length and fixed distances.

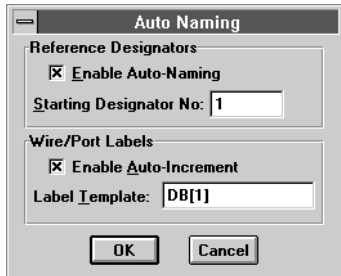
- 1 Select Auto-Repeat from the Options menu to display the Auto-Repeat dialog box.
  - a Set the Vertical Offset to 00.10.
  - b Check that Enable Auto-Repeat is enabled.
  - c Click OK.
- 2 Click the Draw Wire icon and draw a wire from pin 2 of U5 to the bus.
- 3 Press Space five times to place five more wires.
- 4 Click the Draw Wire icon and draw a wire from pin 2 of U6 to the bus.
- 5 Press Space five times to place five more wires.



Buses must be labeled.  
Examples of legal bus names are:

DB[0-12]  
DB[0:12]  
DB[0..12]  
DB0, DB1, CLK

**Note** Each wire connecting to a bus must be labeled with the name of one of the signals on the bus.



### Labeling the wires connected to the bus

You can use Auto-Naming to label a uniform collection of wires.

- 1** Select Auto-Naming from the Options menu to display the Auto Naming dialog box.
  - a** In the Wire/Port Labels area, select the Enable Auto-Increment check box.
  - b** The Starting Designator No. should be 1.
  - c** In the Label Template text box, type DB, the label for the first wire in the series.

Wires will be labeled incrementally as DB1, DB2, DB3, etc.
  - d** Click OK.
- 2** Select the first (lower-most) wire to be labeled.
- 3** Press **Ctrl+E** to label the wire.
- 4** Repeat steps 2 and 3 for each wire segment, in the order they are to be labeled (from bottom to top).

## Changing Reference Designators and Part Values

Part values and reference designators are changed by double-clicking on them and entering a new value in the displayed dialog box.

### Changing U8A to U9B

- 1 Double-click U8A to display the Edit Reference Designator dialog box.
- 2 Type U9 in the Package Reference Designator text box.
- 3 Type B in the Gate text box.
- 4 Click OK.

When you place a part on the schematic, the part is automatically assigned a reference designator, e.g., U8, and a 'gate' (if it is a multi-part component). For instance, when you placed the 74123 part, it was assigned something like U8A (i.e., reference designator U8 and gate A).

### Changing R9 from 1 Kohm to 100 Kohm

- 1 Double-click 1K (next to resistor R9) to display the Set Attribute Value dialog box.
- 2 Change 1K to 100K.
- 3 Click OK.

Now, change the value of R10 to 100K and the values of R3 through R8 to 10K.

If you placed any of the components in an order other than the sequential order shown in Figure 1-2, use this feature now to change the reference designators to match the schematic in Figure 1-2.



Rubberbanding is one of the Display Options under the Options menu.

## Moving Parts, Wires and Text

Parts, wires, buses and text are moved by clicking on them to select them, and dragging them to a new location. To maintain connectivity when moving parts, wires or buses, enable the rubberbanding option.

### Moving resistor R1 up one grid

- 1 Click the resistor to select it.  
A red resistor indicates that it is selected.
- 2 Place the pointer on the resistor and drag to move the resistor up one grid.
- 3 Release the mouse button to place the resistor at the new location.

Part values, reference designators and other text are moved in the same way.

### Moving the value of R10

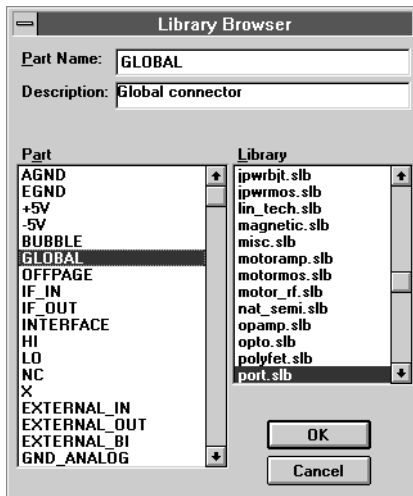
- 1 Click the 100K value of the resistor.  
The box outline around the value indicates that it is selected.  
The dotted box outline around the resistor shows that the resistor is the “owner” of the selected value.
- 2 Place the pointer on 100K and drag the value to a new location.  
The box representing the 100K value follows as you drag the mouse.
- 3 Release the mouse button to place the 100K value at the new location.

## Placing Ports

Ports in Schematics are used to identify signals that are inputs or outputs to a schematic. Ports are placed in the same way that you place other parts.

### Placing the port

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on 1-7).
- 2 Click Libraries to display the Library Browser dialog box.



- 3 In the Library list box, select port.slb.
- 4 In the Part list box, select GLOBAL (which is the name of a global port symbol).
- 5 Click OK.
- 6 In the Part Browser dialog box, click Place & Close.
- 7 Move the pointer to the location for the DAT port and click to place the part.
- 8 Right-click to stop placing ports.

### Labeling the port

- 1 Double-click the port symbol to display the Set Attribute Value dialog box.
- 2 Type DAT in the LABEL text box.
- 3 Click OK.

Now place and label the CLK and RTN ports as shown in Figure 1-2.

## Placing Power and Ground Symbols

Power and ground symbols are types of global port symbols in Schematics. The label on the port defines the name of the power supply.



### Placing +5-volt power supplies

- 1 Type +5V in the Select Part list box on the toolbar.
- 2 Press **Enter** to activate the part selection.
- 3 Move the pointer to the location of the +5V symbol and click to place the symbol.
- 4 Move the pointer and click to place the other nine +5V symbols.
- 5 Right-click to stop placing parts.

To place 12 volt supplies, first place +5V symbols. Then double-click the +5V symbol and change +5V to +12V.

All signals tied to power supplies of the same name are connected.

## Placing ground symbols

- 1 Type EGND in the Select Part list box on the toolbar.
- 2 Press  to activate the part selection.
- 3 Move the pointer to the location of the ground symbol and click to place the symbol.
- 4 Move the pointer and click to place the other four ground symbols.
- 5 Right-click to stop placing parts.



## Saving Your Work

Click the File Save icon, or select Save (or Save As) from the File menu to save the schematic.



If this is a new design, you are asked to enter a file name where the new schematic will be saved.

---

# Using the Schematic Editor

---

## 2

### Overview

This chapter provides background information about the schematic editor. For the specific step-by-step instructions for creating a design, see **Chapter 3, *Creating and Editing Designs***.

This chapter has the following sections:

*Components of a Design on page 2-3* introduces and explains the components of a design.

*Main Window on page 2-6* describes the user-interface to the schematic editor. This section describes the uses of menus, the toolbar and toolbar icons, the status line and the keyboard.

*Configuring Schematics on page 2-11* provides information on configuring the schematic editor to suit your requirements.

*Zooming and Panning in Schematics on page 2-32* tells how to zoom in and out of the drawing, refresh the screen display, pan to various sections of the drawing and fit the drawing to the page.

*Using the Message Viewer on page 2-37* describes the Message Viewer that displays system messages and explains the various displays and controls.

# Components of a Design

A schematic consists of:

- symbols
- attributes
- wires
- buses
- text items

Schematics can have either a flat or hierarchical structure, depending on the way you choose to implement your design.

## Parts

Parts are electrical devices that make up a circuit, such as:

- resistors
- operational amplifiers
- diodes
- voltage sources
- digital gates

The graphical representation of a part is a symbol. Symbols are stored in symbol libraries.

Schematics uses two basic types of parts: primitive and hierarchical.

Primitive parts are at the lowest level and explicitly contain all of the information required by the netlister. Most symbols in the symbol libraries are primitive parts.

Hierarchical parts have the same appearance as primitive parts. The difference is that hierarchical parts represent one or more levels of schematics and primitive parts do not.

A hierarchical part is modified by pushing into it from within the schematic editor or symbol editor and editing the associated schematic(s).

A primitive part is modified by editing its graphics, pins and attributes.

## Symbols

Symbols are the graphical representation of parts, ports and other schematic elements. They are grouped by functionality into symbol libraries. Each symbol contains a specific set of attributes that define the symbol. You can edit these attributes as well as create new attributes. Symbols can share similar attributes and graphics. Hierarchical symbols represent schematics and are the mechanism that you use to create hierarchical designs.

## Ports

Ports are not physical connectors. If you want a specific pin (such as a DB25 pin) you must use a symbol for such a connector from the “connect.slb” symbol library.

Ports are symbols that form connecting points leading into or out of the schematic page. Ports provide connectivity between schematic pages and between levels of hierarchy. They play an important role in determining names of electrical nets.

## Attributes

Two attributes of a resistor are PKGTYPE (package type) and VALUE.

Parts, ports, wires (nets), buses and most other symbols have associated attributes. An attribute consists of a name and an associated value. Attributes are used for Bill of Materials reports, and simulation and layout netlists.

Attribute Name		Value
PKGTYPE	=	RC05
VALUE	=	1K

## Annotations

Annotation symbols are used to show non-electrical information on the schematic, such as comments and tables. Annotation symbols may consist of text and/or graphics. Title blocks and page borders are considered annotations.



## Connections

Parts and ports contain one or more pins to which connections are made. Electrical connections are formed by wire and bus segments joining pins and other wire and bus segments. Connections are also formed by attaching pins directly to pins. Schematics represents each such electrical connection by a junction. Junctions are made visible when three or more connected items converge at the junction. Junctions are created and removed automatically.

Some parts have hidden pins. Hidden pins are most often used for power and ground connections to digital parts. Hidden pins are not connected by wires and buses, but rather through an attribute that names the net to which they belong (the `IPIN(<pinname>)=<netname>` attribute).

## Main Window

When you start Schematics, a schematic editor window opens and displays a single schematic page. You have the option of opening additional schematic editor windows. Use these windows to:

- display different schematics
- display different portions of a single schematic page
- display different pages of the same schematic
- display different levels of hierarchy from the same schematic
- display a separate symbol editor window

## Menus

**Note** *Drop-down menu items sometimes appear grayed-out. This means they are not currently available. In some cases, you must select an object first or perform some other operation before you can choose grayed-out items.*













The display and operation of the menus and submenus in Schematics follows a standard Windows layout and operation.

## Toolbar







Toolbar icons provide shortcuts for initiating common actions. To turn the toolbar display on and off, select Display Options from the Options menu and click the Toolbar check box.

When you move the pointer over an icon, the status bar displays the operation that will take place if you click that icon.

**Table 2-1** *Schematic Editor Toolbar Icons*

Icon	Name	Function	Page
	New File	opens a new schematic file	3-3
	Open File	opens a schematic file	3-3
	Save File	saves a schematic file	3-63
	Print (immediate)	prints the current schematic	3-57
	Zoom In	views smaller area of schematic	2-32
	Zoom Out	views a larger area of schematic	2-32
	View Area	views a selected area of schematic	2-32
	View Fit	fits the view to show all items on the page	2-34
	Draw Wire	enables the drawing of wires on schematic	3-24
	Draw Bus	enables the drawing of buses on schematic	3-23
	Draw Block	enables the placing of blocks on schematic	6-4
	Draw Text	enables the placing of text on schematic	3-49

**Table 2-1** *Schematic Editor Toolbar Icons*

<b>Icon</b>	<b>Name</b>	<b>Function</b>	<b>Page</b>
	Select Part	displays Part Browser dialog box for selecting part for placement	<b>3-5</b>
	Edit Attributes	edits the attributes of selected objects	<b>3-18</b>
	Edit Symbol	opens the symbol editor for the selected symbol	<b>4-5</b>
	Analysis Setup	sets up the simulation analyses for the active schematic	<b>7-9</b>
	Simulate	simulates the active schematic	<b>7-10</b>
	Redraw	refreshes the active schematic page screen display	<b>2-10</b>

## Status Bar

The status bar is located at the bottom of the schematic editor window and provides the following information:

- X and Y coordinates of the pointer. Use the Display Options selection under the Options menu to turn the display of X and Y coordinates on and off.
- A message area that provides a brief description of the function that will be performed if you click the toolbar icon at the present pointer location.
- A brief description of the function to be performed by selecting the menu item at the present pointer location.
- Prompts and warning messages.
- Name of the function to be performed when you use the Repeat function.

3.69, 4.56

Draws a new wire

Imports an OrCAD schematic

Cmd: Show Selected

Use the Display Options selection from the Options menu to turn the status bar on or off.

## Keyboard

Table 2-2 lists the function keys in the schematic editor that allow you to choose, enable or disable certain menu items. For those items that toggle, pressing the function key enables the feature, and pressing ⇧ plus the function key disables the feature.

**Table 2-2**    *Schematic Editor Function Keys*

Key	Action	Menu	Selection
<span>F1</span>	on-line help	Help	Help
<span>F2</span>	move to lower level in the schematic hierarchy	Navigate	Push
<span>F3</span>	move up one level in the schematic hierarchy	Navigate	Pop

**Table 2-2** *Schematic Editor Function Keys (continued)*

Key	Action	Menu	Selection
F4	text stay-on-grid	Options	Display Options
F5	orthogonal	Options	Display Options
F6	stay-on-grid	Options	Display Options
F7	auto-increment	Options	Auto-Naming
F8	auto-repeat	Options	Auto-Repeat
F9	rubberband	Options	Display Options
F10	view errors	File	Current Errors
F11	start the simulator	Analysis	Simulate
F12	start MicroSim Probe	Analysis	Run Probe

## Refreshing the Screen



To refresh the screen, click the Redraw icon on the toolbar.

# Configuring Schematics

Schematics is installed with a set of default settings for configurable options. Customizing the configuration allows you to use Schematics in the way that best suits your needs and requirements.

This section describes the following configuration items:

- Changing the set of symbol libraries configured for use by the system. Describes procedures for modifying the search path, and adding or deleting a symbol library from the list of libraries used by Schematics.
- Changing the font used for displaying and printing text on your schematics.
- Setting the page size for your schematics.
- Specifying other page settings such as paper size, pin spacing and borders.
- Changing the grid and gravity settings of the schematic editor.
- Setting the time interval between automatic saves of your schematics.
- Specifying the use of a different text editor.
- Specifying the size of the list in the Select Part list box on the toolbar.
- Determining which elements of a schematic are to be displayed and printed.
- Changing the colors used for drawing and displaying objects.
- Changing where to find the .exe files for MicroSim programs with which Schematics interfaces. Also allows you to specify an initialization file other than the installed default initialization file.

# Configuring Symbol Libraries

When the libraries are listed (as in the Libraries list box in the Editor Configuration or in the Browse Libraries dialog box), the local libraries are listed first.

- The global libraries are preceded by an asterisk (\*).
- The file extension .slb indicates a symbol library; the file extension .plb indicates a package library.

When you open a schematic or create a new schematic, the symbols available for use in the schematic are those contained in *configured* libraries. Schematics searches through the libraries in the order in which they are configured. It uses the first symbol encountered that has the name being searched.

Global libraries are available to all schematics. Local libraries are available only to the schematic for which they are configured. Local libraries are searched before global libraries.

## Adding libraries

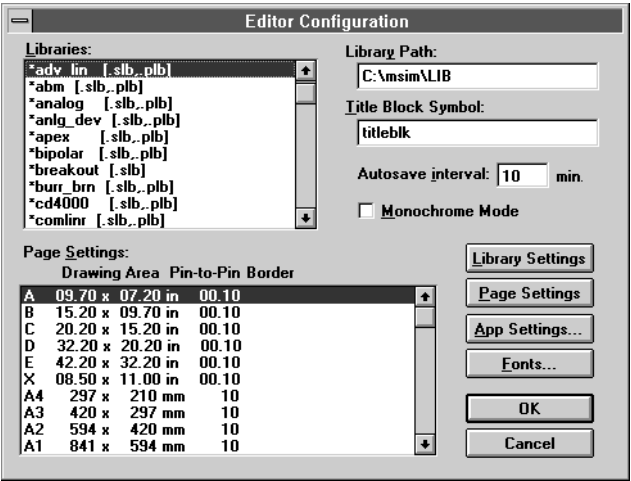
A library can be added anywhere to the list of configured libraries either as a global library or a local library for the currently active schematic.

## Adding a library

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box.

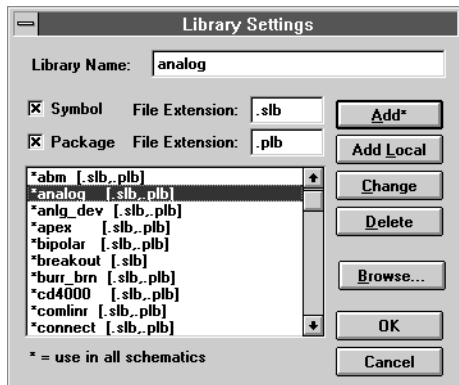
### Options Menu

Options	Analysis	Tools
Display Options...		
Page Size...		
Auto-Repeat...		
Auto-Naming...		
Set Display Level...		
Editor Configuration...		



- 2 Click Library Settings to display the Library Settings dialog box.





- 3 In the list of libraries, select the location for the new library.  
The new library will be added directly *above* the library you select.
- 4 If the library you are adding is a symbol library, click in the Symbol check box. If the library you are adding has an associated package library, click in the Package check box.
- 5 Type the name of the library in the Library Name text box.  
Do not type a file extension; the file extension is appended automatically.
- 6 Click Add\* for a global library, or click Add Local for a local library.
- 7 Click OK.
- 8 In the Editor Configuration dialog box, click OK.

When removing a library, it is only removed from the configured libraries list, it is not deleted.

### Options Menu

Options	Analysis	Tools
Display Options...		
Page Size...		
Auto-Repeat...		
Auto-Naming...		
Set Display Level...		
Editor Configuration...		



## Removing libraries

If you no longer need a library in the list of configured libraries, you can remove the library from the list of configured libraries.

### Removing a library

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on [2-12](#)).
- 2 Click Library Settings to display the Library Settings dialog box (shown on [2-13](#)).
- 3 In the list of libraries, select the library that you want to remove.

The name of the selected library displays in the Library Name text box.

- 4 Click Delete.
- 5 Click OK.
- 6 In the Editor Configuration dialog box, click OK.

## Correcting library names

If you type a library name incorrectly, you can change the name as it appears in the list of configured libraries.

### Correcting a library name

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on [2-12](#)).
- 2 Click Library Settings to display the Library Settings dialog box (shown on [2-13](#)).
- 3 In the list of libraries, select the library that you want to change.

The name of the selected library displays in the Library Name text box.

- 4 Type a new name for the library in the Library Name text box.

Do not type a file extension; the file extension is appended automatically.

- 5 Click Change.
- 6 Click OK.
- 7 In the Editor Configuration dialog box, click OK.

## Changing the search order

The order in which Schematics searches libraries for a symbol follows the order of the libraries in the configured libraries list. You can change the position of a library in the list.

## Changing a library position

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on 2-12).
- 2 Click Library Settings to display the Library Settings dialog box (shown on 2-13).
- 3 In the list of libraries, select the library that you want to move.  
  
The name of the selected library displays in the Library Name text box.
- 4 Click Delete.
- 5 In the list of libraries, select the new location for the library.  
  
The library is placed directly *above* the library you select.
- 6 Type the name of the library in the Library Name text box.  
  
Do not type a file extension; the file extension is appended automatically.
- 7 If the library is a symbol library, click in the Symbol check box. If the library has an associated package library, click in the Package check box.
- 8 Click Add\* for a global library or click Add Local for a local library.
- 9 Click OK.

### Options Menu

Options	Analysis	Tools
Display Options...		
Page Size...		
Auto-Repeat...		
Auto-Naming...		
Set Display Level...		
Editor Configuration...		



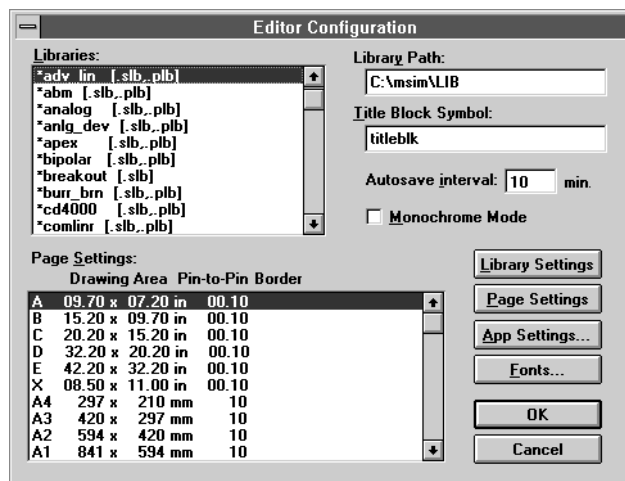
**10** In the Editor Configuration dialog box, click OK.

## Changing the search path

Schematics looks for a library according to the path(s) specified by the Library Path in the Editor Configuration dialog box.

## Changing the library search path

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box.



- 2 Type a new path or add to the existing path in the Library Path text box.

Specify multiple directories by separating them with a semicolon:

```
c:\msim\lib
c:\msim\lib;c:\project\lib
```

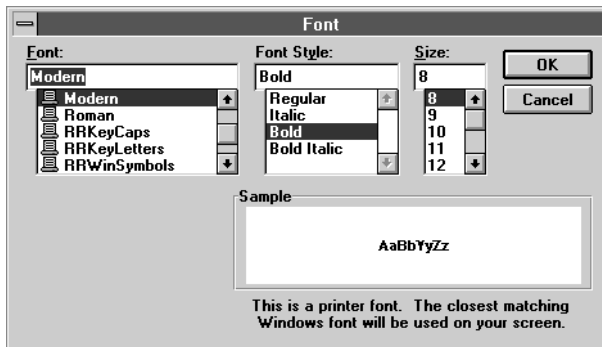
- 3 Click OK.

## Changing Fonts

To change the font used by Schematics to display and print text, use the Editor Configuration function to select any available TrueType font.

### Selecting a font and style

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on 2-12).
- 2 Click Fonts to display the Font dialog box.



- 3 Select a font from the Font list box. Select a style and size from their respective list boxes. A sample of the selected font is shown in the Sample box.
- 4 Click OK.

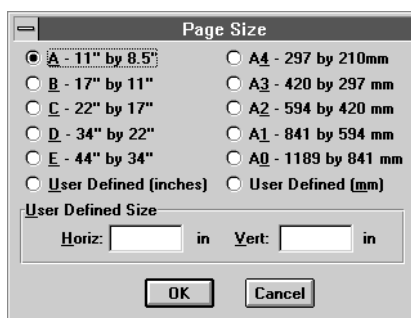
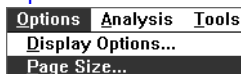
## Changing Page Size

Schematics supports standard page sizes A through E and A0 through A4. It also allows you to specify a user defined page size.

### Changing the page size

- 1 Select Page Size from the Options menu to display the Page Size dialog box.

#### Options Menu



- 2 Click one of the radio buttons to select a pre-defined page size, or indicate a User Defined Size by typing the page dimensions in the Horiz: and Vert: text boxes.
- 3 Click OK to change the page size of the currently active page and to establish the default page size for all later pages.

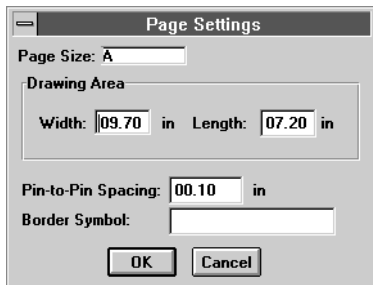
## Changing Page Settings

For each of the pre-determined page sizes, you can change the drawing area, the pin-to-pin spacing and the default border symbol name.

### Drawing area

#### Changing the drawing area

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 In the Page Settings list box, select the page size that you want to change.
- 3 Click Page Settings to display the Page Settings dialog box.



- 4 Type the drawing area dimensions for the page size in the Width and Length text boxes.
- 5 Click OK.
- 6 In the Editor Configuration dialog box, click OK.

## Pin-to-pin spacing

You can scale symbols so they will appear larger on the schematic. You do this by changing the pin-to-pin spacing for a given page size.

### Changing the pin-to-pin spacing

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 In the Page Settings list box, select the page size that you want to change.
- 3 Click Page Settings to display the Page Settings dialog box (shown on **2-19**).
- 4 Type a new value in the Pin-to-Pin Spacing text box.
- 5 Click OK.



## Border symbol

The default border type is a single line around the drawing area. Use the symbol editor to create your own border symbols. You can choose a different border symbol if you have previously created one and added it to a symbol library.

### Changing the current border symbol

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 In the Page Settings list box, select the page size that you want to change.
- 3 Click Page Settings to display the Page Settings dialog box (shown on **2-19**).
- 4 Type the name of the new custom border symbol in the Border Symbol text box.
- 5 Click OK.



## Changing Grid and Gravity

The grid and gravity functions of Schematics eases your drawing tasks and can help make your schematic more precise.

### Grid On

When Grid On is enabled, the grid is displayed in the drawing area of the schematic editor window.

### Enabling or disabling the grid display

- 1 Select Display Options from the Options menu to display the Display Options dialog box.

- 2 Click the Grid On check box to enable or disable the grid display.

An “X” in the check box indicates that the grid is “ON” or enabled.

- 3 Click OK.

### Snap-to-grid

Snap-to-grid causes components to snap to the nearest grid point when you place them; before placing, objects may be moved freely. If Snap-to-grid is disabled, movement is by grid units rather than a smooth motion.

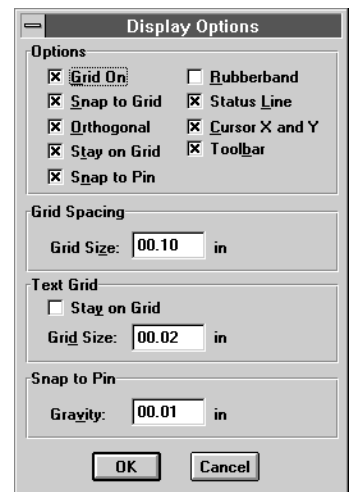
### Enabling or disabling snap-to-grid

- 1 Select Display Options from the Options menu to display the Display Options dialog box.

- 2 Click the Snap-to-Grid check box to enable or disable snap-to-grid.

An “X” in the check box indicates that snap-to-grid is “ON” or enabled.

- 3 Click OK.



### Stay-on-grid

Stay-on-grid places components strictly on the grid. Components always stay on the grid lines, even while being moved.

### Enabling or disabling stay-on-grid

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **2-21**).
- 2 Click the Stay-on-Grid check box to enable or disable stay-on-grid.  
  
An “X” in the check box indicates that stay-on-grid is “ON” or enabled.
- 3 Click OK.

### Snap-to-pin

Snap-to-pin, when enabled, causes the endpoint of a wire or bus segment to snap to the nearest pin if one is found inside the radius defined by the Gravity setting.

### Enabling or disabling snap-to-pin

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **2-21**).
- 2 Click the Snap-to-Pin check box to enable or disable snap-to-pin.  
  
An “X” in the check box indicates that snap-to-pin is “ON” or enabled.
- 3 Click OK.

## Grid spacing

Grid Spacing defines the horizontal and vertical grid spacing on your drawing area. The default spacing is 10 units. This corresponds to (and is displayed as) 0.10 inches for US-standard page sizes, and 2.5 millimeters for metric page sizes. The minimum grid spacing allowed is 0.01 inches (US sizes).

## Specifying grid spacing

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **2-21**).
- 2 In the Grid Spacing area, type the grid spacing value in the Grid Size text box.
- 3 Click OK.

## Gravity

The gravity setting specifies how close an object must be to a pin to snap to it. Gravity is only functional when snap-to-pin is enabled.

## Specifying gravity

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **2-21**).
- 2 Type the snap-to-pin gravity value in the Gravity text box.
- 3 Click OK.

## Text grid

Text Grid allows you to set the grid spacing for text separately from the normal grid spacing. The text grid is usually set to some smaller percentage of the regular drawing grid. This allows you to align text along smaller increments of the regular grid.

### Enabling text grid and specifying text grid size

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **2-21**).
- 2 In the Text Grid area, click the Stay-on-Grid check box to enable the text grid.  
  
An “X” in the check box indicates that the text grid is “ON” or enabled.
- 3 In the Text Grid field, type the text grid spacing value in the Grid Size text box.
- 4 Click OK.

When Autosave is enabled, Schematics creates a temporary file with the same name as the current working file, and an extension ending in ‘v’ (for example, “.scv,” “.slv,” “.plv”). If you have a power outage or system failure, you can retrieve your work from these files.

The temporary files are deleted each time a schematic or library is successfully closed or saved. When you open a file, Schematics compares that file to the autosave file, if one is present. If the autosave file is more recent than the requested file, Schematics provides a warning and allows you to restore the file as the current schematic or library.

While an autosave is in progress, a message appears in the status line and the hourglass symbol displays in place of the pointer. Wait until the autosave is finished before continuing.

### Setting the Autosave Interval

The autosave interval specifies the time interval, in minutes, in which Schematics saves any modified schematics or libraries.

#### Setting the Autosave interval

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 In the Autosave interval text box, type the number of minutes for the autosave interval.
- 3 Click OK.

#### Disabling Autosave

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 Type 0 in the Autosave interval text box.
- 3 Click OK.

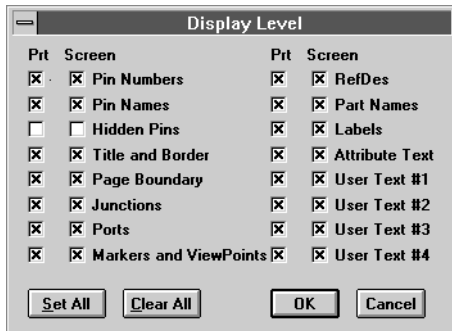
## Controlling the Display in Schematics

Schematics allows you to determine which elements of a design are to display and print. The elements that you can either display or not display (and print or not print) are:

- Pin Numbers
- Pin Names
- Hidden Pins
- Title and Border
- Page Boundary
- Junctions
- Ports
- Markers and Viewpoints
- Reference Designators
- Part Names
- Labels
- Attribute Text
- User-specified text

### Setting display levels

- 1 Select Set Display Level from the Options menu to display the Display Level dialog box.



#### Options Menu

Options	Analysis	Tools
Display Options...		
Page Size...		
Auto-Repeat...		
Auto-Naming...		
Set Display Level...		

- 2 Click to select or de-select the items you want, either under the Prt (printer) column, the Screen column or both.
- 3 To select all items to print and display, click Set All.
- 4 To deselect all items, click Clear All.
- 5 Click OK.

## Configuring Colors

The configuration file “msim.ini” is created during the installation process and is located in the Windows directory. Most of the settings in the “msim.ini” file are controlled from Options menu selections. Colors, however, are not.

Change the colors used for drawing and displaying objects in the schematic editor by using any text editor to edit the “msim.ini” file.

Colors for all items are specified as *<item name>=<color>*. The item names and what they represent are listed in Table 2-3.

You can specify a color name or an RGB value. Available colors are:

black	blue	brown
brightwhite	cyan	darkblue
darkcyan	darkgray	darkgreen
darkmagenta	darkred	green
lightgray	magenta	red
yellow		

If you do not specify a color for any item, the color for that item defaults to the color specified for FOREGROUND.

### Editing the "msim.ini" file

**Note** Prior to editing the “msim.ini” file, be sure that *Schematics* is not open.

- 1 Using a text editor, such as MicroSim’s TEXTEDIT, open the “msim.ini” file.
- 2 Scroll to the [SCHEMATICS COLORS] section of the file.
- 3 Find the name of the item you want to change and type a new color value.
- 4 Save the file.

**Table 2-3** *Schematics Colors Items*

Item Name	Description	Default*
ATTRIBUTES	Specifies the color of attribute text.	
BACKGROUND	Specifies the color of window background.	BRIGHTWHITE
BORDER	Specifies the color of the Schematics border.	
BUS	Specifies the color of buses.	
GRID	Specifies the color of the grid dots.	
FOREGROUND	Specifies the default color for items not explicitly specified.	BLUE
HIDDENPINS	Specifies the color of hidden pins.	
JUNCTION	Specifies the color of junctions.	
MARKER	Specifies the color of marker and viewpoint symbols.	
PAGEBOUNDARY	Specifies the color of the page bounding box.	
PIN	Specifies the colors of symbol pins.	
PINNAME	Specifies the color of pin name text.	DARKGREEN
PINNO	Specifies the color of the pin number text.	
PLSYN	Specifies the color of PLSyn parts and blocks (that is, those parts with IMPL=PLSYN, and blocks with a DSL view).	
PORT	Specifies the color of port symbols.	
PORTLABEL	Specifies the color of port labels.	
REFDES	Specifies the color of the REFDES attribute.	
SELECTION	Specifies the color of selected objects.	RED
STIMULUS	Specifies the color of StmEd stimulus parts (that is, those with a STIMULUS attribute).	
SYMBOLS	Specifies the color of symbol graphics.	DARKGREEN

**Table 2-3**    *Schematics Colors Items*

Item Name	Description	Default*
SYMNames	Specifies the color of PART attribute.	
TEXT1	Specifies the color of text placed with Draw/Text.	
SYMTEXT	Specifies the color of symbol text.	
WIRE	Specifies the color of wires.	DARKGREEN
WIRELABEL	Specifies the color of wire labels.	

\* If a default setting is not specified for any color item, the color for that item defaults to the color specified for FOREGROUND. The default color for FOREGROUND is BLUE.

## Changing the Select Part List Size

The Select Part list box on the toolbar contains a scroll-down list of recently placed parts. The default length of this list is ten items.

To change the length of the list, use a text editor to edit the MRPLISTSIZE item in the [SCHEMATICS] section of the “msim.ini” file.

MRPLISTSIZE=<length of list>

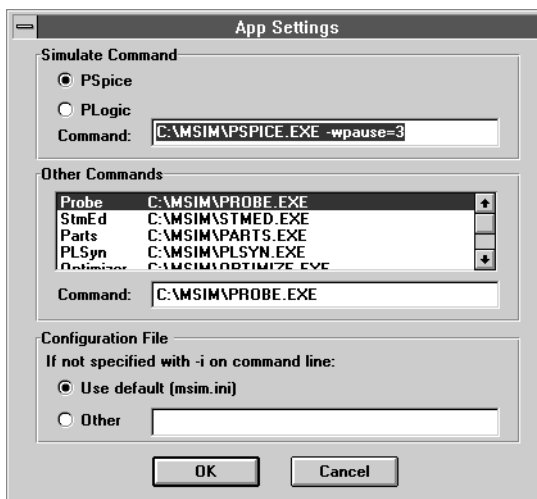


## Changing Application Settings

You have the option to change the location of the .exe files for the MicroSim programs that Schematics interfaces with. You can also configure a different text editor (besides MicroSim's TEXTEDIT) and specify an initialization file other than the installed default initialization file.

### Changing where to find programs

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on 2-12).
- 2 Click App Settings to display the App Settings dialog box.



The Simulate Command area shows the path that Schematics uses to execute the PSpice or PLogic program.

- 3 To change one of the simulate commands, click either the PSpice or PLogic radio button in the Simulate Command area and type a new command in the Command text box.
- 4 Similarly, to change any of the other command lines, click to select the command in the Other Commands list box and type a new command in the Command text box.
- 5 Click OK.

- 6 In the Editor Configuration dialog box, click OK.

## Changing the Configuration File

To change the configuration file for Schematics:

- 1 In the Windows Program Manager, select the Schematics icon.
- 2 Select Properties from the File menu.
- 3 Append -i<configuration file name> to the Command Line.
- 4 Click OK.

A configuration file other than the default "msim.ini" file can be used for any of the MicroSim programs that Schematics interfaces with.

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 Click App Settings to display the App Settings dialog box (shown on **2-29**).
- 3 In the Configuration File area, click the Other radio button.
- 4 Type the file name of the configuration file in the text box.
- 5 Click OK.
- 6 In the Editor Configuration dialog box, click OK.

## Specifying a different text editor

Editing text in Schematics is done using a program called TEXTEDIT. You can specify and use a different text editor.

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on 2-12).
- 2 Click App Settings to display the App Settings dialog box (shown on 2-29).
- 3 Select the “Text Editor” command in the Other Commands list box.
- 4 Type a new command in the Command text box to specify the path and name of the text editor you want to use.
- 5 Click OK.
- 6 In the Editor Configuration dialog box, click OK.



# Zooming and Panning in Schematics

## Zooming

When working on a design, you can zoom in (enlarge the view) or zoom out (reduce the view) to view a larger or smaller portion of the schematic window. Zooming in reduces the area viewed and enlarges the objects viewed. Zooming out increases the area viewed and reduces the size of the objects viewed.

### Zooming in

Shortcut: press **Ctrl**+**I** .



- 1 Select In from the View menu.
- 2 Move the pointer to the desired center of the zoom and click.

### Zooming in about the center of the window

Click the Zoom In icon.

### Zooming out

Shortcut: press **Ctrl**+**O** .



- 1 Select Out from the View menu.
- 2 Move the cursor to the desired center of the zoom and click.

### Zooming out about the center of the window

Click the Zoom Out icon.



### Zooming in on a selected area of the page

- 1 Click the Zoom Area icon, or select Area from the View menu.
- 2 Click and drag the pointer outline to select the desired display area.

Shortcut: press **Ctrl**+**A** .

### Zooming out to view the full schematic page

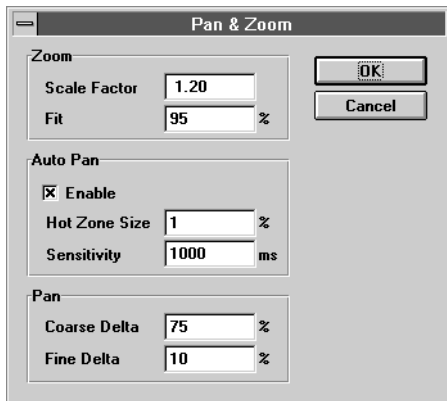
Select Entire Page from the View menu.

# Setting Zoom Parameters

## Setting Scale Factor and Fit

The Zoom parameters tailor how the work area will be magnified or reduced when you make selections from the View menu, or click any of the zoom icons.

- 1 Select Pan & Zoom from the Options menu to display the Pan & Zoom dialog box.



- 2 Type a value in the Scale Factor text box.

This value defines the factor by which the screen is magnified or reduced when you select Zoom In or Zoom Out. A Scale Factor of 2 will double (or halve) the size of objects viewed.

- 3 Type a value in the Fit text box.

This value defines the percentage of the work area to be filled with the complete schematic when you select View Fit. Choose a decimal value between 50 and 100. A typical value is 90.

- 4 Click OK.

## Fitting to a Page

All of the parts, wires and text within the current window (excluding the title block) are displayed by fitting the view to the page.

### Fitting the view to the page

Click the View Fit icon, or select Fit from the View menu.



Shortcut: press **Ctrl+N**.

## Panning

Panning allows you to select a new window centering point. The current zoom scale remains the same. When you select the new center point, the schematic is panned until the selected point is in the center of the window.

### Panning to a new center

- 1 Select Pan—New Center from the View menu.
- 2 Move the cursor to the desired window center and click.

#### View Menu

View	Options	Analysis
Fit		Ctrl+N
In		Ctrl+I
Out		Ctrl+O
Area		Ctrl+A
Previous		
Entire Page		
Redraw		Ctrl+L
Pan - New Center		

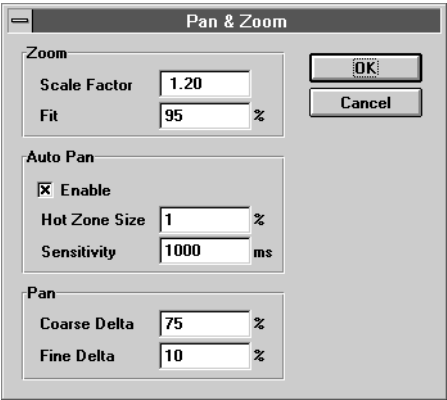
### Automatic Panning

If Auto Pan is enabled, the pointer turns to a solid black arrow when you move it to the edge of the window. If you leave the arrow at the edge of the window for a few moments, the view pans in the direction of the arrow. You can pan up, down, left and right using this method.



### Enabling Automatic Panning

- 1 Select Pan & Zoom from the Options menu to display the Pan & Zoom dialog box.



#### Options Menu



- 2 Select the Enable Auto Pan check box.
- 3 Click OK.

### Setting Hot Zone Size

The Hot Zone Size determines the width of the zone in which the pointer has to rest to trigger the Auto Pan function.

- 1 Select Pan & Zoom from the Options menu to display the Pan & Zoom dialog box.

- 2 Type a value in the Hot Zone Size text box.

Values must be in the range from 1 to 10. If the value is set to 1 (default value), the hot zone is 1% of the dimensions of the screen.

- 3 Click OK.

## Setting Auto Pan Sensitivity

The Auto Pan sensitivity setting determines how long the pointer must remain on the window border before the panning takes place.

- 1 Select Pan & Zoom from the Options menu to display the Pan & Zoom dialog box (shown on **2-35**).
- 2 Type a value in the Sensitivity text box in the Auto Pan area.  
The value in the text box is the time delay in milliseconds. The default is 1000 milliseconds (1 second) which is relatively slow.
- 3 Click OK.

## Setting Pan Coarse Delta and Fine Delta

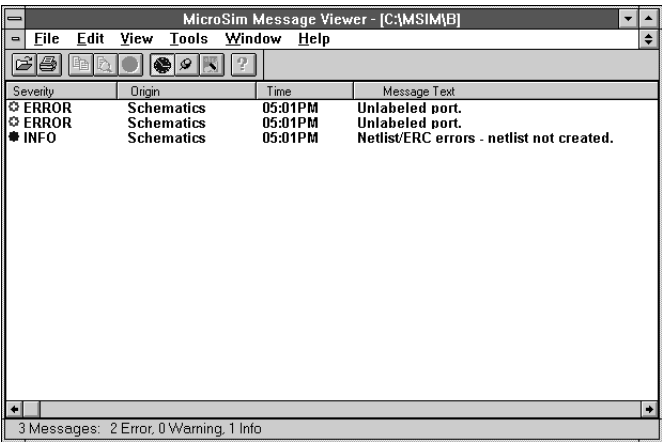
The Pan parameters determine the degree of movement of the work area when you use the scroll bars.

- 1 Select Pan & Zoom from the Options menu to display the Pan & Zoom dialog box (shown on **2-35**).
- 2 Type a value in the Coarse Delta text box.  
The value in the text box is the degree of movement of the design when you click in a scroll bar on either side of the slider. The value must be between 1 and 50 and represents a percentage of the visible work area.
- 3 Type a value in the Fine Delta text box.  
Fine Delta defines the percentage of movement of the design when you pan by clicking on one of the scroll bar arrows. The value must be between 1 and 5 and represents a percentage of the visible work area.
- 4 Click OK.



# Using the Message Viewer

The Message Viewer displays text describing a condition, status or other information concerning the operation of MicroSim applications.



The Message Viewer appears when any condition generates a message that requires you to be informed. For example, warnings and error messages that occur during netlisting will appear in the message viewer.

If you have more than one design open and close one design, the messages pertaining to the design you close no longer display. When you close the last design, the message viewer closes. Also, the message viewer closes when all MicroSim applications are closed or when you explicitly close it.

The Message Viewer uses all standard Windows controls for scrolling, sizing and selecting.

## On-line Help

Another way to view on-line help is:

- 1 Right-click in the message area to display a menu.



- 2 Click Help On to view a context-sensitive help message.

The Message Viewer has an on-line help feature that allows you to view a help message directly relating to the currently selected message.

To view a context-sensitive help message:

- 1 Select the message in the message viewer window.
- 2 Press **F1**.

## Locating the Source of a Message

Another way to locate the message source:

- 1 Right-click the message line to display a menu.




- 2 Click Find In Design.

Many messages displayed in the Message Viewer contain a hypertext link that points to the source of the message. This allows you to go to the location in the design that caused the message to be generated.

To locate the source of a message, double-click the message in the Message Viewer window.

## Indicated Severity

Each message is preceded by a  marker. The color of the marker indicates the severity of the message.

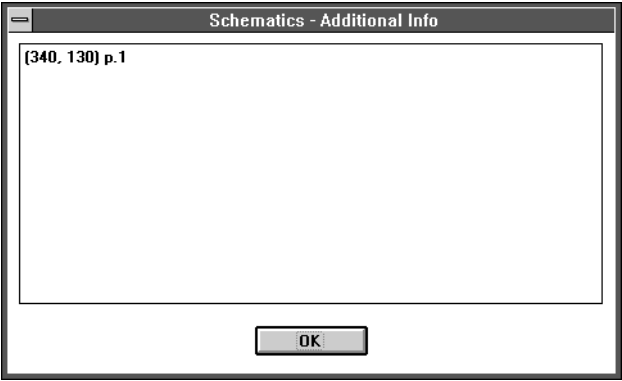
Color	Indication
Blue	Informational message. No user action is required.
Yellow	Warning message. May require some user action.
Red	Error message. Must be corrected before continuing.
Black	Fatal error message. Indicates a non-recoverable error condition.

# Additional Information

Some messages contain additional text. That is, the message contains several lines of information while only one line is displayed. Lines containing additional information are indicated by a plus sign in the severity marker preceding the message text.

When the Message Viewer contains any messages with additional information, the More Info icon on the tool bar is active.

- 1 Click either the plus sign in the line of text or the More Info icon to view the Additional Info dialog box.



Another way to view additional information:

- 1 Right-click the message line to display a menu.



- 2 Click Additional Info to display the Additional Info dialog box.

- 2 Click OK to dismiss the dialog box.

# Closing the Message Viewer

To close the Message Viewer, do one of the following:

- a Select Exit from the File menu.
- b Double-click on the Control-menu box in the upper left corner of the window.

## File Menu

File	Edit	View	Tool
Open...			Ctrl+O
Close			
Print...			Ctrl+P
Print Setup...			
Exit			



---

# Creating and Editing Designs

---

## 3

### Overview

This chapter contains the step-by-step procedures for creating, editing and printing a schematic which includes:

*Starting the Schematic Editor on page 3-3* describes how to start the schematic editor and how to open a new or existing file.

*Finding Parts on page 3-4* describes how to find parts by name, by description and by searching the symbol libraries.

*Placing and Editing Parts on page 3-9* describes the detailed steps for placing parts, changing the orientation of parts prior to placing them, editing part attributes, placing multiple instances of a part and automatically assigning reference designators.

*Placing Power and Ground Symbols on page 3-27* describes how to place and edit power and ground symbols.

*Using Wires and Buses on page 3-29* describes drawing and labeling of wires and buses and describes the drawing options that affect the placement of wires and buses.

*Using Ports on page 3-38* describes the use of off-page and global ports.

*Selecting and Moving Parts, Wires and Attributes on page 3-40* describes how to select and move parts, wires and attributes.

*Creating and Editing Title Blocks on page 3-46* describes how to create and edit the title block on your schematic.

*Creating and Editing Annotation Items on page 3-49* describes how to create and edit annotation items (non-electrical information).

*Creating and Editing Multi-sheet Designs on page 3-53* describes how to create and edit a multi-sheet design.

*Printing Your Design on page 3-57* describes how to print your design.

*Closing the Schematic Editor on page 3-63* describes how to save a schematic and how to close the schematic editor.

# Starting the Schematic Editor

Start the schematic editor by double-clicking on the Schematics icon in the MicroSim program group. An empty schematic page displays.



If you already have Schematics running with another schematic displayed, click the New File icon to start a new schematic.



## Opening a File

To open a new file, click the New File icon. An empty schematic page displays.

To open an existing file and display the schematic for editing, click the Open File icon. Previously opened schematics remain open until explicitly closed.



To close files or to close the schematic editor, see *Closing the Schematic Editor on page 3-63*.

## Finding Parts

### Simulation Checklist

When you are drawing a design for simulation, keep the following in mind:

- The symbols that you place must have corresponding simulation models associated with them.
- The design will need sources of stimulus.
- For any part that has an associated simulation model, unmodeled pins are indicated by a 'broken' pin.

Parts represent electrical devices such as resistors, operational amplifiers, diodes, voltage sources and digital gates that make up the circuit diagram.

The graphical representation of a part is a *symbol* stored in a symbol library.

For those parts with a simulation model available, the *model definition* is stored in a model library.

For parts applicable to PCB layout, the *package definition* is stored in the package library.


You can use symbols supplied with MicroSim Schematics, or you can create your own symbols and store them in user-defined symbol libraries. You can select a symbol from a library by name or by browsing the list of available parts.

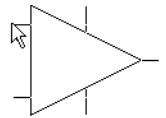
The available parts are only those contained in configured libraries. If you have a library of parts and they are not available, you need to add the library to the list of configured libraries. See *Configuring Symbol Libraries on page 2-12*.

## Getting Parts by Name

The Select Part list box on the toolbar contains a scroll-down list of recently used parts. You can also type a name in the Select Part list box to select a part.

### Selecting a part by name

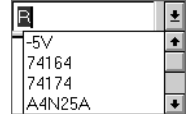
- 1 Type the name of the part you want to place in the Select Part list box.
- 2 Press **Enter** .
- 3 Move the outline of the selected symbol to any location on the schematic and click to place the part.
- 4 Right-click to stop placing parts.



### Placing a previously selected part

Once you select a part for placement, the part name is listed in the Select Part list box on the toolbar and can easily be recalled.

- 1 Click the Select Part list box arrow.  
A scroll list displays containing the names of the last ten parts that have been placed.
- 2 Click the name of the part you want to place.
- 3 Move the outline of the selected symbol to any location on the schematic and click to place the part.
- 4 Right-click to stop placing parts.





## Searching for Parts in the Libraries

The symbol libraries contain symbols for many parts. There are three methods for selecting parts from libraries:

- Search for the part by name.
- Search for the part by description.
- Browse through the symbol libraries.

Each of these methods are described below.



**Note** One of two Part Browser dialog boxes may display: the Part Browser Advanced or the Part Browser Basic. If the Part Browser Basic dialog box displays, click Advanced to display the Part Browser Advanced dialog box. If the Part Browser Advanced dialog box displays, click Basic to display the Part Browser Basic dialog box. The steps in finding a part by name are the same with either dialog box.

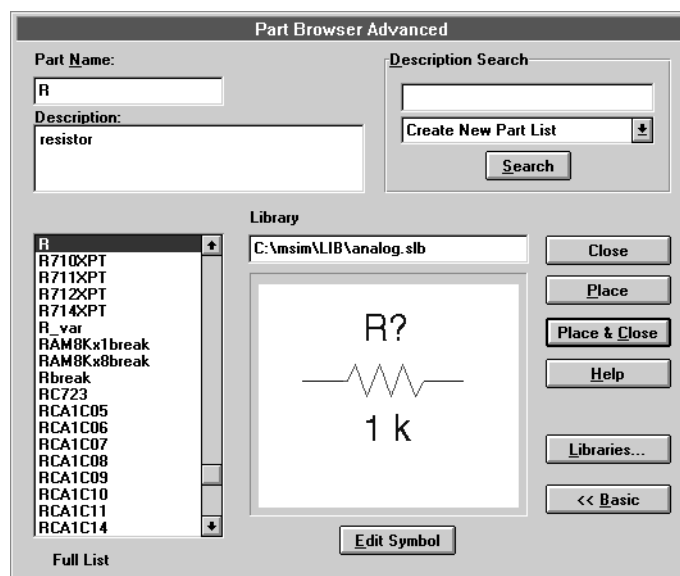
When typing a part name or a description, you can use the “\*” and “?” wildcard characters.

An “\*” is a wildcard that matches zero or more characters. For example, R\* matches R, R1 and R12.

A “?” is a wildcard that matches any single character. For example, R? matches R1 but not R or R12.

### Selecting a part by name

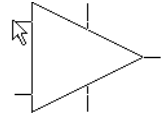
- 1 Click the Select Part icon to display a Part Browser dialog box.



- 2 Type the name of the part in the Part Name text box, or select the part name from the list of available parts at the left side of the dialog box.
- 3 Click Place to place the part (with the browser remaining open) or click Place & Close (to place the part and close the browser).



- 4 Move the outline of the selected symbol to any location on the schematic and click to place the part.
- 5 Right-click to stop placing parts.



## Selecting a part by description

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on 3-6).



**Note** One of two Part Browser dialog boxes may be displayed: the Part Browser Advanced or the Part Browser Basic. If the Part Browser Basic dialog box displays, click Advanced to display the Part Browser Advanced dialog box. You can only use the Part Browser Advanced dialog box to search for a part by description.

- 2 Type a description of the part in the Description Search text box.
- 3 Select one of the options in the scroll list in the Description Search area:
  - a Select Create New Part List to create a new (sub)list of parts in the parts list box.
  - b Select Add to Part List to add to the set of parts listed in the parts list box.
  - c Select Search within Part List to restrict the search to the (partial) list of parts in the parts list box.
- 4 Click Search.

The number of items found in the search is given beneath the parts list box.

The search function searches all configured symbol libraries for parts whose descriptions match the description you entered. When the search is complete, all parts whose descriptions match are listed in the partial list at the left of the dialog box.

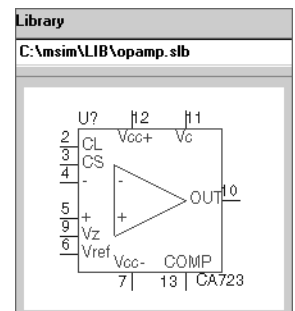
- 5 Select the part from the parts list box.

You can preview a part symbol by clicking the part name in the parts list box. A preview of the part displays at the right of the part list.

When typing a description, you can use the “\*” and “?” wildcard characters.

An “\*” is a wildcard that matches zero or more characters. For example, R\* matches R, R1 and R12.

A “?” is a wildcard that matches any single character. For example, R? matches R1 but not R or R12.



- 6 Click Place to place the part (with the browser remaining open) or click Place & Close (to place the part and close the browser).
- 7 Move the outline of the selected symbol to any location on the schematic and click to place the part.
- 8 Right-click to stop placing parts.

#### Browsing symbol libraries to select a part



- 1 Click the Select Part icon to display the Part Browser dialog box (shown on 3-6).

**Note** One of two Part Browser dialog boxes may display: the Part Browser Advanced and the Part Browser Basic. If the Part Browser Basic dialog box displays, click Advanced to display the Part Browser Advanced dialog box. If the Part Browser Advanced dialog box displays, click Basic to display the Part Browser Basic dialog box. The steps in finding a part by browsing symbol libraries are the same with either dialog box.



- 2 Click Libraries to display the Library Browser dialog box (shown on 2-13).

Use the Library Browser dialog box to select a library and view a list of parts contained in each library. If you select a part from the Library Browser dialog box, the part name displays in the Part Name text box in the Part Browser dialog box and the part is selected for placing.



- 3 Click OK.
- 4 Click Place to place the part (with the browser remaining open) or click Place & Close (to place the part and close the browser).
- 5 Move the outline of the selected symbol to any location on the schematic and click to place the part.
- 6 Right-click to stop placing parts.

**Note** Appendix E contains a list of symbol libraries supplied with MicroSim Schematics and device types.

# Placing and Editing Parts

Once you have selected a part, you can place one or more instances of the part on the schematic. When the part is selected, an outline of the selected part is displayed attached to the pointer.

## Placing a symbol on the schematic

- 1 Use Select Part to select the part from a symbol library.
- 2 Move the symbol outline to where you want the symbol placed and click.

You can place as many instances of the symbol as you want, by moving to another location and clicking again. Each time you point and click, another instance of the part is placed on the page.

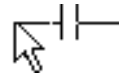
## Stopping placement of this symbol

To stop placing a symbol, do one of the following:

- a Double-click to place the last instance of the symbol and stop placing the symbol.
- b Right-click to stop placing the symbol without placing an additional symbol.

The outline changes back to a pointer.

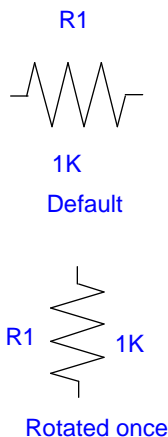
If you type `c` in the Part Name text box to select a capacitor, you will see an outline of the capacitor symbol attached to the pointer (as shown below).



See also *Repeating Part Placements on page 3-20* and *Global Editing of Attributes on page 3-18*.

## Rotating and Flipping Parts

Parts being placed on the schematic, parts already placed and entire areas of a schematic can be rotated and flipped (mirrored).



**Figure 3-1** *Rotating a Part*

### Rotating a part before placing it on the schematic

- 1 Select the part to be placed.
- 2 Press **Ctrl**+**R** to rotate it.

Note that the symbol outline rotates 90 degrees counterclockwise. Each time you press **Ctrl**+**R** the image again rotates 90 degrees in the counterclockwise direction.

### Rotating an already-drawn object

- 1 Select the part to be placed.
- 2 Press **Ctrl**+**R** to rotate it.

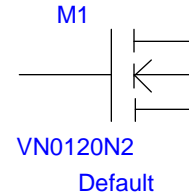
### Rotating an area of the schematic

- 1 Drag with the mouse to outline and select the area to be rotated.
- 2 Press **Ctrl**+**R** to rotate the area. The selected area rotates 90 degrees counterclockwise about the center point of the selected area.

## Flipping a part before placing it on the schematic

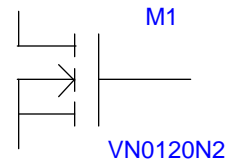
- 1 Select the part to be placed.
- 2 Press **Ctrl**+F to flip it.

Note that the symbol outline is a mirror image of what it was before. Each time you press **Ctrl**+F the image is flipped.



## Flipping an already-drawn object

- 1 Select the part.
- 2 Press **Ctrl**+F to flip it.



Flipped once

## Flipping an area of the schematic

- 1 Drag with the mouse to outline and select the area to be flipped.
- 2 Press **Ctrl**+F to flip the area about the vertical axis.

**Figure 3-2** *Flipping a Part*

# Editing Part Attributes

Parts, ports, wires (nets), buses and most other symbols have associated attributes. An attribute consists of a name and an associated value. (See *Attributes on page 2-4*.)

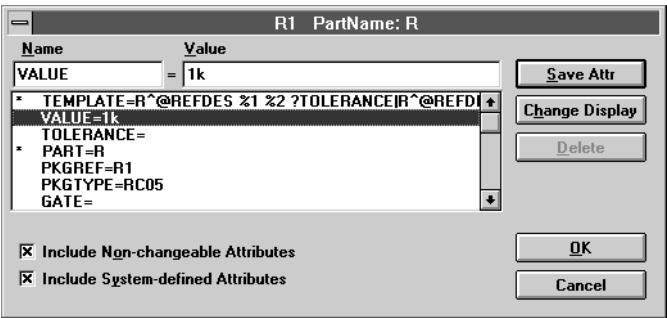
You can create new attributes or edit existing attributes of a part on the schematic. These functions are performed in the Attribute Editing dialog box.

## Editing attributes

### Editing an attribute

**Note** You can quickly change the value of a displayed attribute, such as a resistor value, by double-clicking it.

- 1 Double-click the part to display the Attribute Editing dialog box.



**Note** If you double-click when selecting the attribute, the pointer is placed on the current value in the Value text box.



- 2 In the list of attributes and values, select the attribute to be edited.  
The attribute name displays in the Name text box and the current value in the Value text box.
- 3 Edit the value in the Value text box.
- 4 Click Save Attr.
- 5 Click OK.

An attribute with an '\*' next to it indicates that the attribute cannot be changed or deleted in the schematic editor because the attribute was made an intrinsic property when the symbol was created. These attributes can only be modified in the symbol editor.

An 'a' indicates that the attribute has been annotated as a result of back annotation or has been assigned by the packager.

Any changes you make to the part attributes are made to the individual part instance you selected. The original part contained in the symbol library remains unchanged.

Attribute names can contain any alphanumeric characters (A–Z, 0–9) and the underscore character. Attributes cannot be self-referencing.

The two check boxes at the bottom of the dialog allow you to control whether or not non-changeable and/or system defined attributes are included in the display.

System defined attributes have reserved attribute names. MicroSim Schematics uses these attributes for specific purposes, primarily during netlisting and packaging. System defined attributes are:

PART  
MODEL  
PKGREF  
REFDES  
GATE  
GATETYPE  
COMPONENT  
SWAP  
SIMULATIONONLY  
IMPL  
TEMPLATE  
BIASVALUE  
LABEL  
PAGENO  
PAGECOUNT  
PAGESIZE  
PAGETITLE  
NODE  
MARKERTYPE  
PROBEVAR



### Adding attributes

#### Adding a new attribute

- 1 Double-click the part to display the Attribute Editing dialog box (shown on **3-12**).
- 2 Double-click in the Name text box and type the new attribute name.
- 3 Press **Tab** and type the new attribute value in the Value text box.
- 4 Click Save Attr.
- 5 Click OK.

The new attribute and its value apply only to the part instance you are editing on the current schematic. The attribute and value are saved only with the schematic; they are not saved in the symbol library.



### Deleting attributes

#### Deleting an attribute

- 1 Double-click the part to display the Attribute Editing dialog box (shown on **3-12**).
- 2 Select the attribute to delete.
- 3 Click Delete.
- 4 Click OK.

**Note** *You cannot delete non-changeable or system-defined attributes.*

## Changing the display of attributes

You can change which attributes are displayed on the schematic.

### Turning the attribute display on and off

- 1 Double-click the part to display the Attribute Editing dialog box (shown on 3-12).
- 2 Select the attribute whose display you want to enable or disable.
- 3 Click Change Display to display the Change Attribute dialog box.

**Note** You can not change the display of non-changeable attributes.


A dialog box titled "Change Attribute". It has a "Name:" field with "VALUE" and a "Value:" field with "1k". Below these is a "What to Display" section with five radio button options: "Value only" (selected), "Name only", "Both name and value", "Both name and value only if value defined", and "None". Below that is a "Display Characteristics" section with "Orient:" set to "horizontal", "Hjust:" set to "left", "Layer:" set to "Attribute Text", "Vjust:" set to "normal", and "Size:" set to "100 %". At the bottom are two checkboxes: "Changeable in schematic" (checked) and "Keep relative orientation" (unchecked), followed by "OK" and "Cancel" buttons.

- 4 Click one of the check boxes in the What to Display area.  
With many attributes such as the package reference and reference designator, only the value displays. With others, such as package type, neither the name nor the value displays.
- 5 Click OK to close the Change Attribute dialog box.
- 6 In the Attribute Editing dialog box, click OK.

Your choices are:

- Display the value of the attribute only
- Display the name of the attribute only
- Display both the name and the value of the attribute
- Display both the name and value of the attribute only if the attribute is defined

**Changing other display characteristics of the attributes of a part instance**



- 1 Double-click the part to display the Attribute Editing dialog box (shown on 3-12).
- 2 Select the attribute whose display characteristics you want to change.
- 3 Click Change Display to display the Change Attribute dialog box.  
  
The name of the attribute and the current attribute value display.
- 4 Select or type a value for any of the Display Characteristics.  
  
You can change any of the characteristics as described in Table 3-1.
- 5 Click OK to close the Change Attribute dialog box.
- 6 Click OK to close the Attribute Editing dialog box.

This procedure only changes the display characteristics for the attributes of the one instance of this part on the current schematic. To change display characteristics for the attributes of a part for every instance placed on every schematic, you have to change the global characteristics of the symbol. See *Editing the Default Attributes of a Symbol on page 3-19*.

**Table 3-1**    *Text Characteristics*

Characteristic	Explanation
Orient:	Allows you to position the text horizontally, vertically, upside down, or down in relation to the defining point of the text string.

**Table 3-1** *Text Characteristics*

Characteristic	Explanation
Layer:	Specifies a text display level as defined by the Set Display Level function under the Options menu. Defaults to Attribute Text Layer. You can specify a user-defined layer.
Size:	Determines the size of the text of a displayed text item. The size is expressed as a percentage of the default (100%) size.
Hjust:	Sets the horizontal justification for the placement of text items (left, center, or right).
Vjust:	Sets the vertical justification for placing text items (top, normal, or bottom).

## Global Editing of Attributes

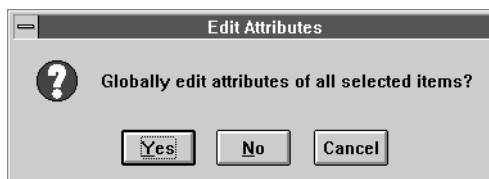
MicroSim Schematics allows you to change an attribute on multiple parts at the same time.

### Assigning the same attribute value to multiple parts

- 1 Select more than one part, or select an area of the drawing enclosing the parts.
- 2 Select the Edit Attributes icon.



A confirmation dialog box displays asking if you want to globally edit attributes of all selected items.



- 3 Click Yes to display the Global Edit Attributes dialog box.



**Note** Click Browse to view a list of attributes for the selected items. If the value of an attribute is the same for all selected items, the value displays. Otherwise, no value displays

- 4 Type the name of the attribute in the Attribute Name text box.
- 5 Type a value for the attribute in the Value text box.
- 6 Click OK.

The named attribute is changed to the specified value for all selected parts having that attribute.

## Editing the Default Attributes of a Symbol

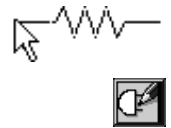
When placing parts, you might want to change the value of an attribute for all parts of a certain type, such as a resistor. For example, you might want to change the default value for all resistors being placed from a value of 1 Kohm to 10 Kohm.

### Changing the default value of a resistor

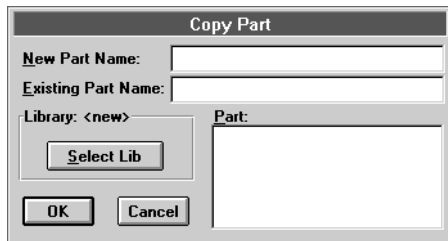
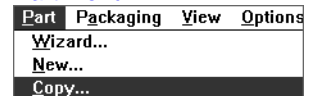
- 1 Select a resistor symbol on the schematic.
- 2 Click the Edit Symbol icon to display the resistor symbol in the symbol editor window.

Note the name of the symbol library `analog.slb` in the title bar of the symbol editor window.

- 3 Click the New File icon to display a new (blank) symbol editor window.
- 4 Select Copy from the Part menu to display the Copy Part dialog box.



#### Part Menu



- 5 Click Select Lib to display the Open dialog box.
- 6 Select `analog.slb` from the scroll list.
- 7 Click OK.
- 8 Double-click R in the Part list box.
- 9 Select Close from the File menu.  
You are prompted to “Save changes to part R?”
- 10 Click Yes.

You are prompted for a library file name.

Any custom symbol changes that you make should be saved in your own custom library.

If you save symbol changes in the MicroSim libraries, you will lose the changes when you update your software.

**11** Type a name in the File Name text box.

**12** Click OK.

You are prompted to add the library to the list of configured libraries.

**13** Click Yes.

## Repeating Part Placements

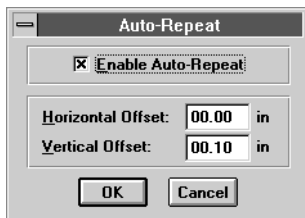
If you are placing parts in line with each other and evenly spaced, use the Auto-Repeat function.

### Automatically repeating part placements

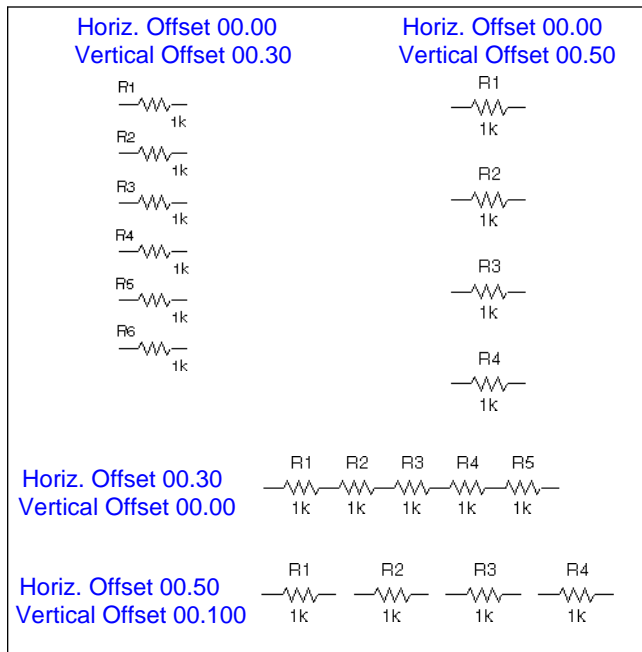
Before selecting the part for placement, enable the Auto-Repeat function and set the offset spacing.

- 1** Select Auto-Repeat from the Options menu to display the Auto-Repeat dialog box.
- 2** Click to place an X in the Enable Auto-Repeat check box.
- 3** Specify a horizontal and/or vertical offset for the part placements.

Figure 3-3 illustrates parts that are placed at various vertical and horizontal offsets.



By default the Enable Auto-Repeat check box is disabled, the Horizontal Offset spacing is set to 00.00 and the Vertical Offset is set to 00.10.



**Figure 3-3** *Placing Resistors with Various Vertical and Horizontal Offsets*

- 4 Select the part from the symbol library.
- 5 Place the first instance of the part.
- 6 Press **[Space]** once to place each subsequent instance of the same part.

### Manually repeating part placements

- 1 If Auto-Repeat is not enabled press **[Space]** to repeat the last action.

If the last action was placing a part, pressing **[Space]** changes the pointer to the symbol outline of the last part placed.

- 2 Move the symbol outline to any location on the schematic and click to place the part.
- 3 To stop placing the part, do one of the following:
  - a Double-click to place the last instance of the part and stop placing the part.

**Note** Use **[F8]** and **[Shift][F8]** to enable and disable Auto-Repeat. If you do not need to change the offsets, this is a convenient way to quickly place arrays of parts and/or wires.



- b** Right-click to stop placing the part without placing an additional part.

The outline changes back to a pointer.

## Automatically Assigning Reference Designators

The Auto-Naming function is useful for assigning reference designators to parts as they are placed. The default starting reference designator number is 1. When placing resistors, the first one placed is R1, the second R2, etc. You can set the starting reference designator to any number, depending on the way you package or organize your design.

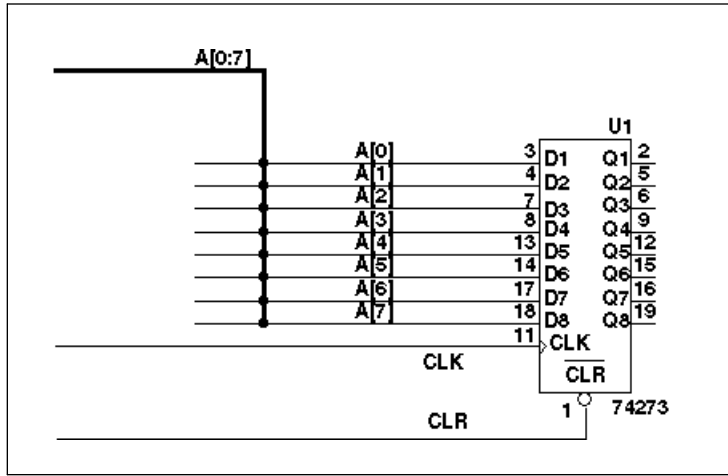
### Enabling Auto-Naming of reference designators

- 1** Select Auto-Naming from the Options menu to display the Auto-Naming dialog box.
- 2** Click to place an X in the Enable Auto-Naming check box.
- 3** Type a starting designator number if you want the numbering to start at a number other than one.
- 4** Click OK.



## Example—Using Auto-Repeat and Auto-Naming

Use the following procedure to create part of the drawing shown in Figure 3-4 using the Auto-Repeat and Auto-Naming functions.



**Figure 3-4** *Auto-Naming for Bus Labels*

### Placing the bus and part

- 1 Click the Draw Bus icon.
- 2 Move the pencil-shaped pointer to the location of one end of the first bus segment. Click to start drawing the bus.
- 3 Click at each vertex of the bus. Double-click at the end of the bus.
- 4 Type 74273 in the Select Part list box on the toolbar.
- 5 Press .
- 6 Move the outline pointer to the location of U1 and double-click to place a single instance of the 74273 part.





### Drawing the first wire segment connecting the part to the bus

- 1 Click the Draw Wire icon.
- 2 Move the pencil-shaped pointer to a point on the bus where wire segment A[0] attaches to the bus. Click to start drawing the wire.
- 3 Move to pin D1 on U1 and double-click.

### Using Auto-Repeat to create the remaining wire segments

- 1 Enable Auto-Repeat (see *Automatically repeating part placements on page 3-20*).
- 2 Set the horizontal offset to 00.00 and the vertical offset to 00.10.
- 3 Press **[Space]** seven times to draw seven additional wire segments.

### Using Auto-Naming to quickly label the wire segments



Shortcut: press **[Ctrl]+E** .

- 1 Select Auto-Naming from the Options menu to display the Auto-Naming dialog box.
- 2 In the Wire/Port Labels area, click to place an X in the Enable Auto-Increment check box.
- 3 Type A[0] in the Label Template text box.
- 4 Click OK.
- 5 Select the first wire to be labeled.
- 6 Select Label from the Edit menu to label the wire.
- 7 To label each of the remaining wire segments:
  - a Select the wire.
  - b Press **[Space]**.

## Replacing Parts

A single part on a schematic is easily be replaced, as well as all parts of a given type on a page or on all pages of a multi-page design. Instead of having to delete one part, find another in a library and place the new part, you can replace the old with the new in one operation.

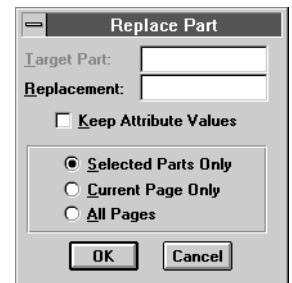
### Replacing a single part

- 1 Select the part to be replaced.
- 2 Select Replace from the Edit menu to display the Replace Part dialog box.

Note that Target Part is grayed out. This is because you have already selected a part and don't need to specify one in the dialog box.

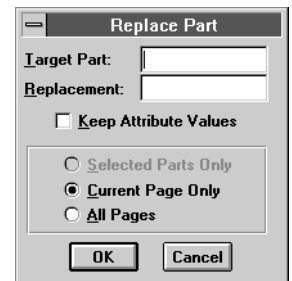
- 3 In the Replacement text box, type the name of the replacement part.
- 4 If you want the attribute values of the part being replaced applied to the replacement part, click the Keep Attribute Values check box.
- 5 Click OK.

The selected part is replaced.



### Replacing multiple parts of the same name

- 1 Select Replace from the Edit menu to display the Replace Part dialog box.
- 2 In the Target Part text box, type the name of the part(s) to be replaced.
- 3 In the Replacement text box, type the name of the replacement part(s).
- 4 If you want the attribute values of the parts being replaced applied to the replacement parts, click the Keep Attribute Values check box.



- 5 Click the Current Page Only radio button to replace all target parts on the current schematic page or click the All Pages radio button to replace all target parts on all pages of a multi-page design.
- 6 Click OK.

All of the named target parts on the page (or design) are replaced.

### Replacing multiple parts in a selected set

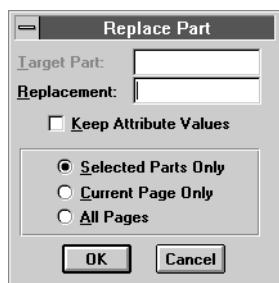
- 1 Select the group of parts to be replaced. (See *Selecting more than one object on page 3-40*.)

- 2 Select Replace from the Edit menu to display the Replace Part dialog box.

Note that the Selected Parts Only radio button is checked.

- 3 In the Replacement text box, type the name of the replacement part(s).
- 4 If you want the attribute values of the parts being replaced applied to the replacement parts, click the Keep Attribute Values check box.
- 5 Click OK.

All of the selected parts on the page (or design) are replaced.



# Placing Power and Ground Symbols

In MicroSim Schematics, power and ground symbols are a type of global port symbol. The label on the port defines the name of the power supply.

## Placing Power and Ground Symbols

### Placing a symbol on the schematic

- 1 Use Select Part (see *Finding Parts on page 3-4*) to select a port symbol.
- 2 Move the outline pointer to where you want the part located and click.

Place several instances of the symbol by moving to another location and clicking again. Each time you point and click, a single instance of the symbol is placed on the page.

### Stopping part placement

To stop placing the symbol, do one of the following:

- a Double-click to place the last instance of the symbol and stop placing the symbol.
- b Right-click to stop placing the symbol without placing an additional symbol.

The outline changes back to a pointer.

Placing and editing power and ground symbols is the same as placing and editing other part symbols with the following considerations:

- Power and ground symbols are contained in the “port.slb” symbol library.
- You can use the symbol editor to create your own custom power and ground symbols.

The power and ground symbols contained in “port.slb” library and available for placing on a schematic are:

AGND  
EGND  
+5V  
-5V  
BUBBLE  
GND\_ANALOG  
GND\_EARTH

## Creating Custom Power and Ground Symbols

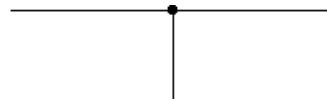
Since power and ground symbols are just like any other symbols, you can use the symbol editor to create your own custom power and ground symbols.

See **Chapter 5**, *Creating and Editing Symbols*, *Drawing Symbol Graphics on page 5-11*.

# Using Wires and Buses

Parts and ports contain one or more pins to which connections can be made. Electrical connections are formed by joining pins of parts and ports with wires and buses.

A junction dot appears where three items are joined.



## Drawing and Labeling Wires

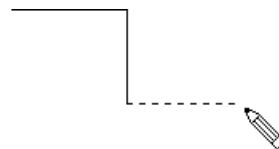
### Drawing a wire

- 1 Click the Draw Wire icon to change the pointer to a pencil shape.
- 2 Click to start the wire.
- 3 Click at each vertex of the wire.
- 4 Click on a pin, another wire or a bus to end the wire (or double-click to end at any point).



If you terminate the wire on a pin, another wire or a bus and do not double-click, draw-wire mode remains active allowing you to start and draw additional wires.

- 5 To stop drawing the wire, do one of the following:
  - a Double-click to place the last segment of the wire.
  - b Right-click to stop drawing the wire without drawing an additional segment.



The outline changes back to a pointer.

If a wire segment ends on the end of another wire segment, they become part of the same wire. If the wires are colinear (they lie on the same vertical or horizontal line), the two wire segments are merged into one. No junction is created. This principle also applies to bus segments.

If a wire segment is added so that its end-point intersects another wire segment at a point other than its endpoints, the second segment is split into two segments at that point. All three segments become part of the same wire.



You can place a label on selected wires, bus segments or ports. Wire and bus segments may display multiple labels.

A wire connected to a bus must be labeled with one of the signals on the bus.

## Rewiring

Shortcut: press **Ctrl**+**D**.

The Rewire function reroutes a selected wire or bus segment without disconnecting its end points. The results of rewiring depend on the Rubberbanding setting.

- 1 Select Rewire from the Draw menu to change the pointer to a cross symbol.
- 2 Click a wire segment to move.
- 3 Click to place a vertex.
- 4 Double-click to place the last vertex and stop rewiring.

## Labeling a wire

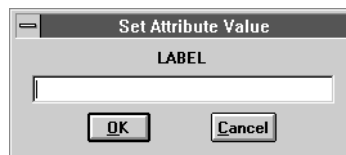
By default, wires with the same label are not treated as connected. If you want to connect them, you must connect each wire to an off-page port with the same label.

To change the default:

- 1 Select Restricted Operations from the Options menu.
- 2 Select the Connectivity via wire labels check box.
- 3 Click OK.

You can assign labels to wires for clarity. Labels are not required except on wires that are connected to buses.

- 1 Double-click the wire segment that you want to label, and to display the Set Attribute Value dialog box.



- 2 Type the label in the LABEL text box.
- 3 Click OK.

**Note** Use the following procedure to edit existing labels.

## Editing a wire label

- 1 Double-click the wire (or the label) to display the Set Attribute Value dialog box with the existing label displayed in the LABEL text box.
- 2 Edit the existing label or delete it and type in a new label.

## Drawing and Labeling Buses

The connectivity of buses and bus segments in MicroSim Schematics is controlled by labeling. The rules of connectivity are:

- A bus label specifies the signals it carries and the order of the signals.
- A bus can connect to another bus only if one is a subset of the other (i.e., A[0-31] and A[16-31]).
- A bus electrically connects to a pin of a part or port if the pin name indicates the same number of signals. Connection is in the order specified; for example, a bus labeled A[31-0] connected to a port labeled Addr[32-63] will electrically connect A[31] with Addr[32], A[30] with Addr[33], etc.
- For a wire to be connected to a bus, the wire must be labeled with one of the signals on the bus.
- Valid syntaxes for labeling a bus are:

CLK[0-15]

CLK[0:15]

CLK[0..15]

CLK1, CLK2, data1, data2, input,...

In the latter form, each and every signal in the bus must be included in the series. The signals are separated by commas.

**Note** *Buses must be labeled.*

### Drawing a bus

- 1 Click the Draw Bus icon to change the pointer to a pencil shape.
- 2 Click to start the bus.
- 3 Click at each vertex of the bus.
- 4 Right-click to end the bus and change the pencil back to a pointer.



### Labeling a bus

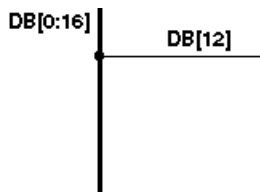
- 1 Double-click the bus segment to display the Set Attribute Value dialog box (shown on **3-30**).
- 2 Type the label in the LABEL text box.
- 3 Click OK.

### Editing a bus label

- 1 Double-click the bus (or the label) to display the Set Attribute Value dialog box (shown on **3-30**) and the existing label in the LABEL text box.
- 2 Edit the existing label or delete it and type in a new label.

### Connecting wires to buses

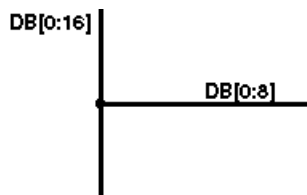
- 1 Draw a wire so that it ends on the bus.
- 2 Label the wire with one of the signals on the bus. For example, you can label the wire DB[12] or DB12 if the bus is labeled DB[0:16].



Connecting a bus segment to the middle of another bus segment creates a junction. The bus segments become part of the same bus unless labeled differently. This allows sub-buses to be taken off a main bus, for example A[0-7] from A[0-31].

### Splitting buses

- 1 Draw a bus segment and end it on the main bus.
- 2 Label the bus segment with a subset of the signals on the main bus.



For example, you can label the bus segment DB[0:8] if the main bus is labeled DB[0:16].

## Automatically Labeling Wires and Buses

Use the Auto-Naming function to set up the labeling of wires and ports. The syntax specified in the Label Template text box allows you to name a uniform collection of wires.

### Naming a collection of wires connected to a bus

- 1 Select Auto-Naming from the Options menu to display the Auto-Naming dialog box.
- 2 In the Wire/Port Labels area, click to place an X in the Enable Auto-Increment check box.
- 3 In the Label Template text box, type the label for the first wire in the series, for example, CLK[0]. Wires will be labeled incrementally in the order selected, as CLK[0], CLK[1], CLK[2]...
- 4 Click OK.
- 5 Select the first wire to be labeled.
- 6 Select Label from the Edit menu to label the wire.
- 7 To label each of the remaining wires in the series:
  - a Select the wire.
  - b Press Space.



Shortcut: press Ctrl+E .

## Specifying Drawing Options

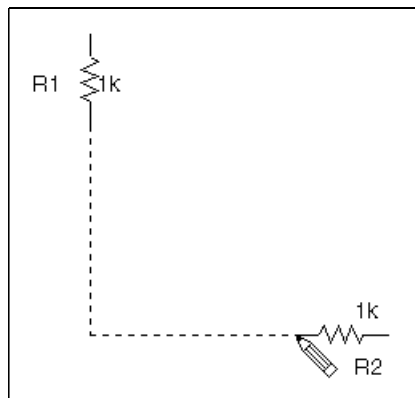
Several options aid in drawing wires and buses and in placing parts.

- The Orthogonal option constrains wires and buses to vertical and horizontal lines.
- The Snap-to-Grid option keeps parts, wires, and buses aligned to grid lines.
- The Snap-to-Pin option constrains wire and bus placements to the nearest pin.
- The Rubberband option maintains connectivity between parts when they are moved.

### Orthogonality

The Orthogonal option allows wires and buses to be drawn only as horizontal and vertical lines. The default setting for Orthogonal is enabled.

Figure 3-5 illustrates two resistors connected by a wire drawn with Orthogonal enabled. The wire was drawn by clicking at the bottom of R1 and moving directly to R2 and clicking again. The wire was drawn by vertical and horizontal lines even though the movement of the pointer was diagonal.



**Figure 3-5** *Orthogonal Wire Drawing*

## Enabling orthogonal drawing

- 1 Select Display Options from the Options menu to display the Display Options dialog box.
- 2 In the Options section, click to place an X in the Orthogonal check box.
- 3 Click OK.

## Snap-to-Grid

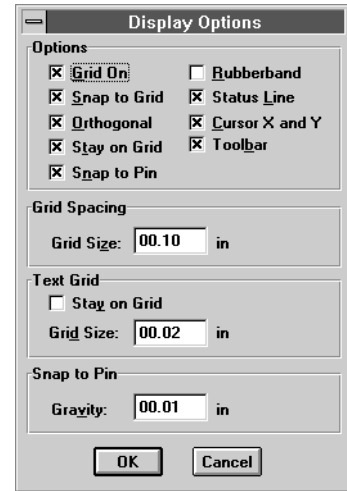
The Snap-to-Grid option causes components to *snap* to the nearest grid point when placing them. When Snap-to-Grid is disabled and Stay-on-Grid is enabled, objects move by grid bar increments. The default setting for Snap-to-Grid is enabled. The default size of the grid spacing is set to 0.10”.

## Enabling Snap-to-Grid

- 1 Select Display Options from the Options menu to display the Display Options dialog box.
- 2 In the Options section, click to place an X in the Snap-to-Grid check box.
- 3 To change the size of the grid, specify a new value in the Grid Size text box, in the Grid Spacing area.
- 4 Click OK.

If Snap-to-Grid and Stay-on-Grid are both disabled, you can place objects at any point in the drawing. However, doing so may make it somewhat more difficult to make subsequent wire and bus connections and is highly discouraged.

### Options Menu



## Snap-to-Pin

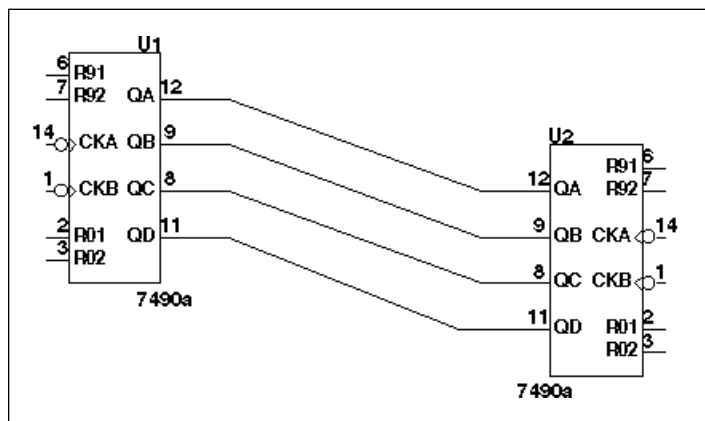
The Snap-to-Pin function causes the endpoint of a wire or bus segment to snap to the nearest pin if one is found within the distance specified by the Gravity setting. The default setting for Snap-to-Pin is enabled with the Gravity distance set to 0.01”.

## Enabling Snap-to-Pin

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **3-35**).
- 2 In the Options section, click to place an X in the Snap-to-Pin check box.
- 3 To change the Gravity, specify a different Gravity setting in the Gravity text box in the Snap-to-Pin section.
- 4 Click OK.

## Rubberbanding

The Rubberband option allows you to move one or more selected objects to a desired location while maintaining connectivity. Figure 3-6 illustrates two devices connected by rubberbanded wires. The default setting for Rubberband is disabled.



**Figure 3-6** *Rubberbanding of Wires*

## Enabling Rubberbanding

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on 3-35).
- 2 Click to place an X in the Rubberband check box.
- 3 Click OK.



# Using Ports

Signals can be connected without using wires or buses by connecting them to global or off-page ports and labeling the ports with the same name.

A third type of port, interface port, provides connections between the pins of a hierarchical block or symbol and the underlying schematic. See **Chapter 6, *Creating and Editing Hierarchical Designs, Using Interface Ports on page 6-12.***

## Off-page ports

Off-page ports connect to other off-page ports with the same name on the same page or on other pages within the same schematic. If you are working on a schematic and you need to connect signals between pages, use off-page ports.

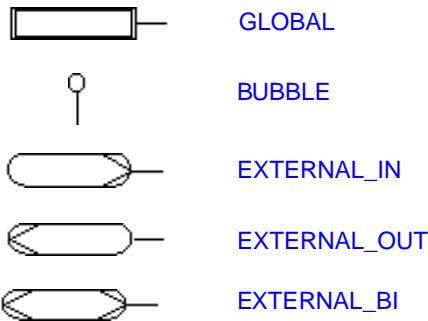


**Figure 3-7** *Off-page Port*

## Global ports

Global ports connect to other global ports of the same name anywhere in the schematic hierarchy.

The symbol library “port.slb” contains several port symbols. You can also create your own port symbols using the symbol editor.



**Figure 3-8** *Global Ports*

## Placing a global port

- 1 Click the Select Part icon to display a Part Browser dialog box, (shown on 3-6).
- 2 Click Libraries to display the Library Browser dialog box (shown on 2-13).
- 3 In the Library list box, select “port.slb.”
- 4 In the Part list box, double-click on “GLOBAL.”
- 5 Click Place to place the global port, or click Place & Close to close the dialog box and place the global port.



**Note** A quick way to place the global port is to type “global” in the Select Part list box on the toolbar.

## Labeling a global port

- 1 Double-click on the port to display the Set Attribute Value dialog box (shown on 3-30).
- 2 Type the label in the LABEL text box.
- 3 Click OK.

# Selecting and Moving Parts, Wires and Attributes

Before performing any operation on a schematic object, you have to select the object. You can make multiple selections or select whole areas of the schematic.

Once you select an object, you can move, copy, delete, edit, cut and paste that object.

## Selecting


### Selecting an object (a part, wire or bus on the schematic)

Point to the object with the mouse and click to select it.

The object color change (the default is red) indicates it is selected.

Once the object is selected, you are ready to perform an action. Selecting a new object causes any previously selected objects to be deselected.

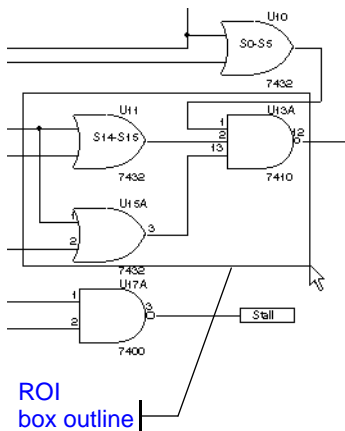
### Selecting more than one object

Hold down  while clicking and selecting a group of objects.

The objects color change indicates they are selected.

### Selecting all objects within a given area of your schematic

Select the area by holding down the mouse button while dragging the mouse across the area.



**Figure 3-9** *Region of Interest Box*

A Region of Interest box (ROI box) appears to indicate that all objects within it are selected. **Only objects entirely contained within the box are selected.**

### Selecting an attribute of an object

Point to the attribute and click.

A rectangle is drawn around the attribute; a dotted bounding box also appears around the object to which the attribute belongs.

### De-selecting selected objects

Click to select an object other than the selected object, or click in a blank area of the schematic.

## Moving

### Moving an object

- 1 Select an object (or group of objects).
- 2 Put the mouse pointer on the object, or in the area designated by the ROI box. While holding down the mouse button, drag the object(s) to the desired location.
- 3 Release the mouse button.

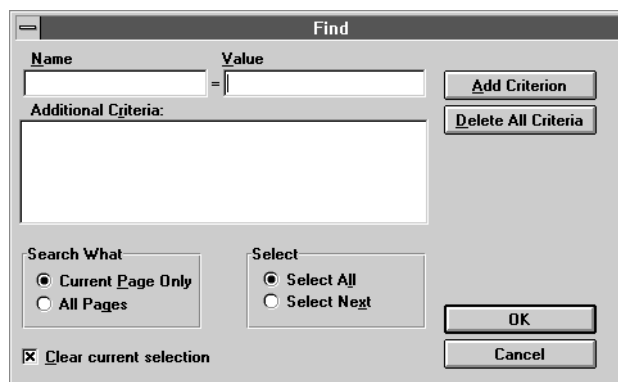
## Searching for and Selecting Parts

Sometimes, to select an object, you first have to find it. This can take time in a sizeable design. MicroSim Schematics allows you to search for objects and to specify search criteria. Any objects located in the search are selected on the schematic.

The search is done using the Find selection from the Edit menu. You can search for any parts, wires and buses that contain attributes.

### Finding a part

- 1 Select Find from the Edit menu to display the Find dialog box.



When typing an attribute name and value, you can specify an exact value or use wildcards.

An “\*” is a wildcard that matches zero or more characters. For example, R\* matches R, R1 and R12.

A “?” is a wildcard that matches any single character. For example, R? matches R1 but not R or R12.

- 2 Specify the search criteria:
  - a Type an attribute name in the Name text box.
  - b Type the attribute value in the Value text box.
  - c Click Add Criterion to add the criterion to the Additional Criteria text box.
  - d Repeat steps a through c as many times as necessary to add more search criteria.
- 3 Click the Current Page Only radio button to search only on the current page, or click the All Pages radio button to search all pages in a multi-page design.
- 4 Click the Select All radio button to have all items meeting the search criteria selected, or click the Select Next radio button to have only the next item meeting the search criteria selected.
- 5 Click OK to begin the search.

The status line displays the number of items found and selected.

## Cutting, Copying and Pasting

MicroSim Schematics contains several common editing features that allow you to cut, copy, paste, copy to clipboard, delete and undelete selected objects. All of these functions are available under the Edit menu. Most can be activated from the keyboard.

The cut, copy, copy to clipboard and delete functions only apply when an object is selected. To learn how to select single and multiple objects as well as objects within a given area, see *Selecting on page 3-40*.

### Cutting

The Cut function deletes the selected object (or group of objects) from the schematic and copies it to a buffer for use with the Paste function. The buffer retains only the last object that was cut.

#### Cutting a selected object

Select Cut from the Edit menu.

Shortcut: press **Ctrl**+X.

### Deleting

The delete function deletes an object. A deleted object cannot be copied or pasted.

#### Deleting a selected object

Press **Delete**.

The object is deleted and can only be recovered with the undelete function (see *Undeleting on page 3-44*).

### Copying

The Copy function makes a copy of the selected object for pasting. The selected object remains on the schematic and a copy is placed in a buffer.

#### Copying a selected object

Select Copy from the Edit menu.

Shortcut: press **Ctrl**+C.

### Pasting

The Paste function places one or more copies of the last object stored in the buffer (from a Cut or Copy operation) onto the schematic.

### Pasting an object

Shortcut: press **Ctrl**+V.

With Auto-Repeat enabled (see *Repeating Part Placements on page 3-20*), use **Space** to place repeated copies of items from the buffer.

- 1 Select Paste from the Edit menu to change the pointer to the shape of the object last cut or copied.
- 2 Click to place the object at the current pointer location.  
  
Continue moving the pointer to various locations and clicking to place additional copies of the object. Right-click to stop pasting.

### Undeleting

Undelete restores the object last cut or deleted from the schematic. Undelete returns the object to the exact position on the schematic from where it was cut or deleted.

### Undeleting an object

Shortcut: press **Ctrl**+U.

Select Undelete from the Edit menu.

### Copying to the Clipboard

The Copy to Clipboard function copies objects within the rectangular area to the Microsoft Windows Clipboard for use in other Windows programs. Electrical or connectivity information is not copied to the clipboard. This function is useful if you want to make a copy of your schematic to include in another type of file, such as a word processor file.

If the grid is enabled, the grid dots are copied to the clipboard along with the schematic. If you don't want the grid dots copied, disable the grid before copying. See *Grid On on page 2-21*.

**Copying an area of the schematic to the Windows Clipboard**

- 1 Select the area to be copied.
- 2 Select Copy to Clipboard from the Edit Menu.

The area is copied to the Windows Clipboard. The copied area remains unchanged on the schematic.

Edit Menu

Edit	Draw	Navigate	View
Undo			Ctrl+Z
Redo			Ctrl+Y
Undelete			Ctrl+U
Cut			Ctrl+X
Copy			Ctrl+C
Paste			Ctrl+V
Copy to Clipboard			

**Importing a schematic into Microsoft Word**

Before selecting the area to be copied, disable the grid. If the grid is enabled, the grid dots will be copied into the Microsoft Word document.

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **3-35**).
- 2 Click the Grid On check box to disable the grid.
- 3 Copy an area of the schematic to the clipboard, as explained in *Copying an area of the schematic to the Windows Clipboard*.
- 4 In Microsoft Word, place a frame where you want the schematic to be placed.
- 5 Press **Ctrl+V** to paste the contents of the clipboard.



# Creating and Editing Title Blocks

Each new schematic is created with a title block in the lower right corner of the page. The title block is treated as an annotation symbol and each text field is an attribute. As such, you can edit the attributes of the title block much the same as you would the attributes of other objects. You can enter information into the title block in the default format, or you can create a custom title block.

## Editing Page Title

The page title, when specified, displays in the title block.

Navigate Menu

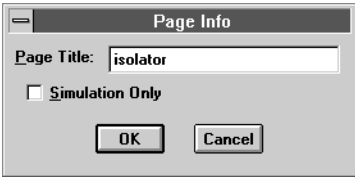


### Changing the page title

- 1 Select Edit Page Info from the Navigate menu to display the Page Info dialog box.
- 2 Type a page title in the Page Title text box.

The Simulation Only check box, when checked, indicates that the entire page is to be ignored for purposes other than simulation (e.g., for PCB layout).

- 3 Click OK.



## Entering Information into the Title Block

Entering information into the existing title block can be done in one of two ways: (1) by editing the attributes of the title block, in which case you can enter information into any or all fields of the title block, or (2) by editing an individual attribute of the title block.

MicroSim Corporation 20 Fairbanks Irvine, CA 92718 714-770-3022	Page Size:
Example Schematic	
Revision: 1.001.1a	January 31, 2009
Page 1 of 1	

## Entering information into various attributes of the title block

- 1 Double-click the title block; or select the title block, then select Attributes from the Edit menu to display the Attribute Editing dialog box.

Name	Value
Page Size	User
Page Size=User	
Page Title=Example Schematic	
Page NO= 1	
Page Count=1	
Revision=1.001.1a	
Date=January 31, 2009	
Page Text=Page of	

☒ Include Non-changeable Attributes  
☒ Include System-defined Attributes

- 2 In the attribute list, double-click the attribute.
- 3 Type the information in the Value text box.
- 4 Click Save Attr.
- 5 Select another attribute or click OK.

### Editing one attribute of the title block

- 1 Double-click the attribute of the title block to display the Set Attribute Value dialog box (shown on **3-30**).
- 2 Type or correct the information in the text box.
- 3 Click OK.

## Creating a Custom Title Block

Since the title block is treated as a symbol, you can use the symbol editor to create your own custom title block or edit the existing title block to suit your requirements. See **Chapter 5, *Creating and Editing Symbols***.

Instead of creating a title block each time you start a new schematic, you can copy the TITLEBLK symbol from the “special.slb” symbol library to your own custom symbol library and modify it to suit your needs. You must configure your custom symbol library in order for your custom title block symbol to be available for use.

### Using a custom title block symbol

Once you have created a custom title block, you have to specify that block in order to use it in the current schematic.

### Specifying a new title block symbol

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on **2-12**).
- 2 Type the name of the title block symbol in the Title Block Symbol text box.
- 3 Click OK.

# Creating and Editing Annotation Items

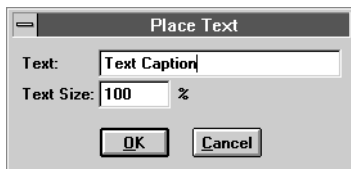
Annotation symbols record non-electrical information on the schematic, such as comments and tables. Annotation symbols may consist of text and/or graphics. Title blocks and page borders are considered annotations.

There are three ways to add text annotation to a design:

- Draw the text on the design. Using this method, you can place the text anywhere on the drawing and move it as a separate object.
- Add the text as an attribute to an object. This method is useful when you want to describe the functionality of an object, such as a block.
- Use the symbol editor to create annotation (non-electrical) symbols that you can place on the schematic.

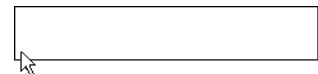
## Adding text to the drawing

- 1 Click the Draw Text icon to display the Place Text dialog box.



- 2 Type the text in the Text text box.
- 3 Type a value for Text Size (if you want other than 100%).
- 4 Click OK.

An outline box follows the pointer and indicates the outline of the text string.



- 5 Move the outline to the desired location and click to place the text.

## Text Caption

The outline box remains on the screen. You can move and click to place the same text string in several locations.

- 6 To stop placing the text string, do one of the following:
  - a Double-click to place the last instance of the text string and stop placing the text string.
  - b Right-click to stop placing the text string without placing an additional text string.

The outline changes back to a pointer.

### Editing annotation text

- 1 Double-click the text to display a dialog box.

The name of the dialog box differs depending on whether the text is drawn on the schematic or added as an attribute.

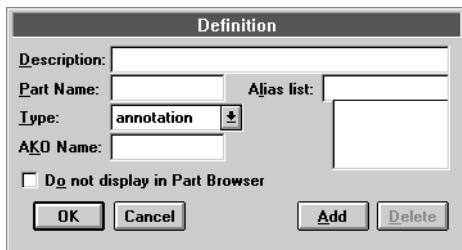
- 2 In either case, edit the text in the text box.
- 3 Click OK.

# Defining and Adding Annotation Symbols

Creating annotation symbols and adding them to your own custom library allows you to re-use them easily in later designs.

## Creating annotation symbols

- 1 Select the Edit Symbol icon to start the symbol editor.
- 2 Select Open from the File menu.  
Choose an existing library to open. This is where the annotation symbol will be saved.
- 3 Select New from the Part menu to display the Definition dialog box.



- a Type a name for the symbol in the Part Name text box.
  - b Select Annotation from the Type scroll list.
  - c Click OK.
- 4 Use the symbol editor drawing tools to add graphics to the symbol. (See **Chapter 5, Creating and Editing Symbols, Drawing Symbol Graphics on page 5-11.**)

Since annotation symbols are non-electrical, pins should not be used.

- 5** Select Attributes from the Part menu to display the Attributes dialog box (shown on **4-14**).
  - a** Add any attributes to contain custom information for later placing on a schematic.
  - b** Click OK.
- 6** Select Save from the File menu.

# Creating and Editing Multi-sheet Designs

A schematic can contain one or more pages. As a schematic grows beyond a single page, ports are used to establish connectivity. Off-page ports provide connectivity between pages of the same schematic. Global ports provide connectivity across schematic pages to other global ports of the same name, anywhere in the schematic hierarchy. Off-page and global ports name the nets to which they are connected.

The Navigate menu allows you to move between pages in multi-sheet designs and provides the means to create new pages, copy pages from other schematics and delete pages.

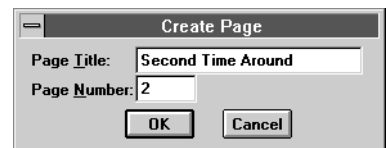
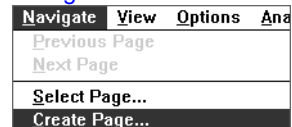
## Adding a Page to a Design

There are two ways to add additional pages to your schematic: (1) by creating a new page or (2) by copying a page from the current schematic or another schematic.

### Creating a new page

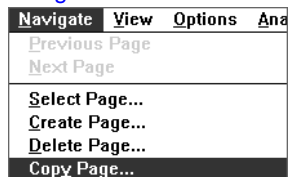
- 1 Select Create Page from the Navigate menu to display the Create Page dialog box.
- 2 Type a title for the new page in the Page Title text box.
- 3 You can accept the next sequential number for the page number of the new page or type in a different number.
- 4 Click OK to add the new page.

Navigate Menu



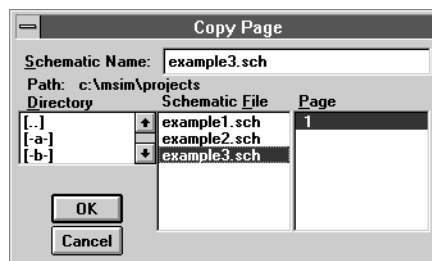


### Navigate Menu



## Copying a page

- 1 Select Copy Page from the Navigate menu to display the Copy Page dialog box.



- 2 Select the schematic file from the dialog.
- 3 Select a page number, if the page to be copied is part of a multi-page schematic.
- 4 Click OK to add the page to the current schematic after the current page and renumber all further pages.

## Creating Connections between Pages

Use off-page ports to create connections between pages. Off-page ports can either be labeled or unlabeled. If an off-page port is unlabeled, it must be connected to a labeled wire or bus.

### Connecting a signal between pages



- 1 Place an off-page port (“OFFPAGE”) on one schematic page.
- 2 Connect a labeled wire or a bus signal to the off-page port.
- 3 Repeat steps 1 and 2, using the same signal name, on the other schematic page(s).

## Viewing Multiple Pages

To view pages in a multi-page design, use the Previous Page, Next Page and Select Page selections under the Navigate menu.

### Viewing the previous page

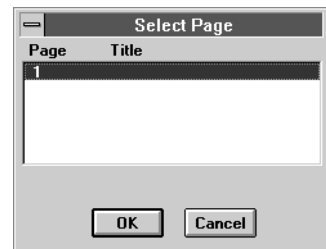
Select Previous Page from the Navigate menu.

### Viewing the next page

Select Next Page from the Navigate menu.

### Viewing a particular page

- 1 Select Select Page from the Navigate Menu to display the Select Page dialog box.
- 2 Double-click the desired page number and title.
- 3 Click OK.



### Viewing multiple pages at the same time

- 1 Select New from the Windows menu.
- 2 Select Previous Page, Next Page or Select Page from the Navigate menu to display the previous page, the next page or the selected page, respectively.

## Cutting, Copying and Pasting between Pages

Cutting and pasting or copying from one page to another in a multi-page design is done in almost the same manner as on a single sheet design. See *Cutting, Copying and Pasting on page 3-43*.

- 1 Cut or copy the object.
- 2 Navigate to the page where the object is to be placed (see *Viewing Multiple Pages*).
- 3 Paste the object.

## Deleting a Page

To delete a page from a multi-page design, use Delete Page under the Navigate menu.

### Deleting a page

- 1 Navigate to the page you want to delete.
- 2 Select Delete Page from the Navigate menu to display a Delete Page confirmation dialog box.
- 3 Click OK to delete the page.



# Printing Your Design

Printing options allow you to print one or more pages or a selected area of a schematic.

## Printing the current page of the current schematic

Click the Print icon.



The page is immediately printed on the current (default) printer.

## Printing a selected area of the current page

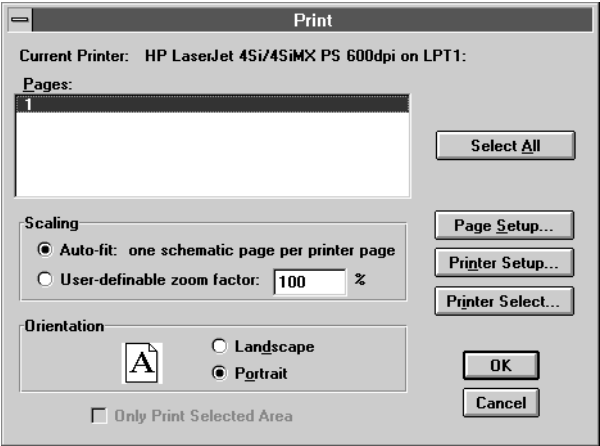
- 1 Select an area of the schematic. (See *Selecting on page 3-40.*)
- 2 Click the Print icon.  
The selected area is immediately printed on the default printer.

## Printing selectively

- 1 Select Print from the File menu to display the Print dialog box.

### File Menu

File	Edit	Draw	Navigate	View	O
New					
Open...					
Close					
Import...					
Save					Ctrl+S
Save As...					
Print...					



- 2 Select the desired page(s) from the Pages list, or click Select All to print all of the pages of the current schematic.

- 3 Select one of the scaling options. See *Scaling*.
- 4 Select an Orientation, either Landscape or Portrait.

Most schematics are in landscape format. Landscape is the schematic editor default format.

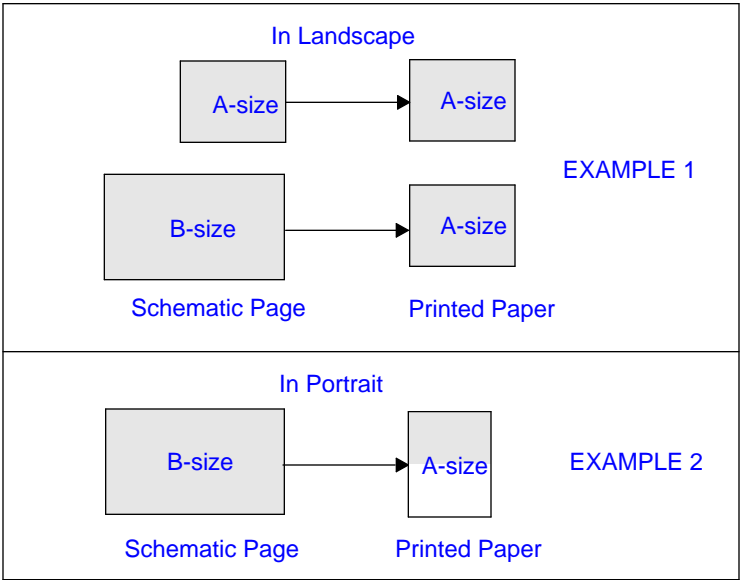
- 5 Click OK.

## Scaling

Scaling options allow you to control the size of the printout.

### Auto-fit

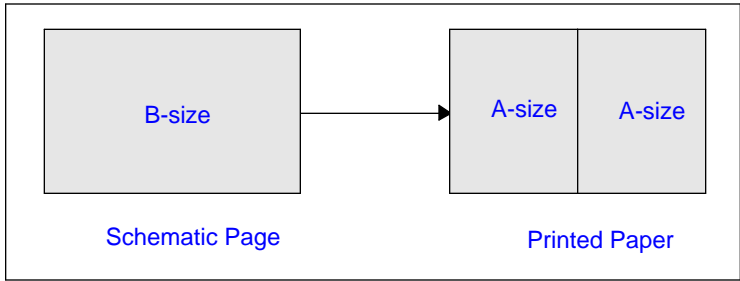
Auto-fit scales the size of the page to print one schematic page per sheet of printer paper. For example, if the schematic page (set through the Page Size selection in the Options menu) is B-size and your printer paper is A-size, Auto-fit automatically sets a zoom factor of 50% so that the B-size drawing fits on the A-size paper as Example 1 of Figure 3-10 shows. If the orientation is set to Portrait, as in Example 2, the zoom factor would be automatically set to a smaller percentage to fit the entire schematic on the page.



**Figure 3-10** *Printing with Auto-Fit Enabled*

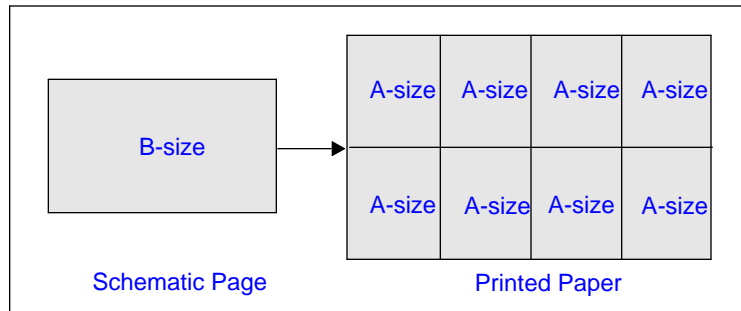
**User-definable zoom factor**

User-definable zoom factor allows you to set a custom zoom factor. For example, with the zoom factor set at 100%, a B-size schematic will print on two A-size sheets of paper when the printer is configured in portrait mode, as shown in Figure 3-11.



**Figure 3-11** *Zoom Factor Set to 100% with Printer Configured in Portrait Mode*

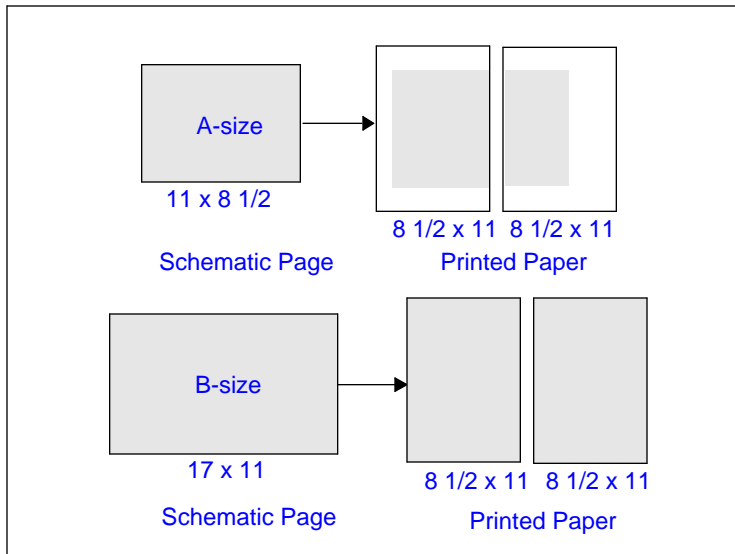
With the zoom factor set to 200%, a B-size drawing will print on eight sheets of paper as shown in Figure 3-12. Doubling the zoom factor quadruples the number of printer pages needed to print a schematic.



**Figure 3-12** *Zoom Factor Set to 200% with Printer Configured in Portrait Mode*

With user-definable zoom enabled, the printer configured in portrait mode and a 100% zoom factor, as shown in Figure 3-13:

- An A-size schematic will print on two sheets of A-size paper.
- A B-size schematic will print on two sheets of A-size paper.

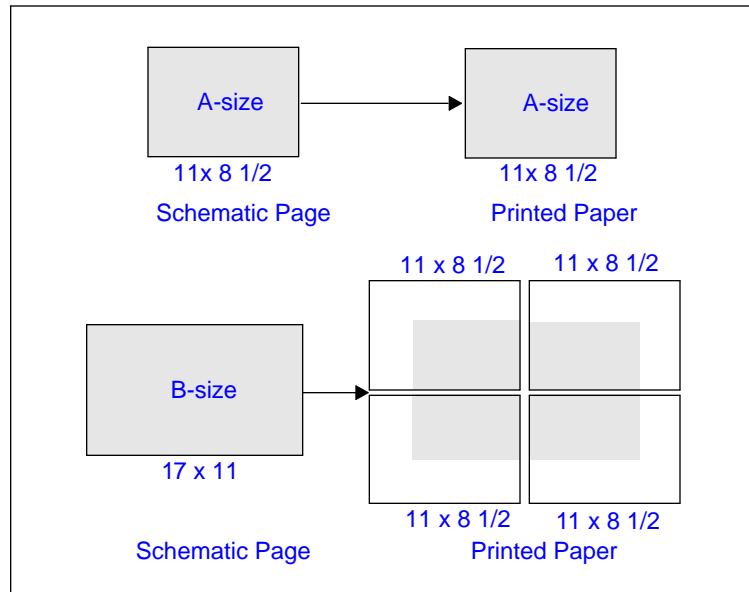


**Figure 3-13** *User-definable Zoom Enabled in Portrait Mode*

In landscape mode, using a 100% zoom factor, as shown in Figure 3-14:

- An A-size schematic will print on one sheet of A-size paper.
- A B-size drawing will print on four sheets of A-size paper.





**Figure 3-14** *User-definable Zoom Enabled in Landscape Mode*

# Closing the Schematic Editor

You can close the schematic editor, thereby closing any open schematics. You can close an open schematic without exiting the schematic editor.

## Closing the schematic editor

To exit the schematic editor and close all currently open schematics, choose one of the two following methods:

- Select Exit from the File menu.
- Double-click the Control-menu box in the upper-left corner of the schematic editor window.

You will be prompted to save any unsaved open schematics.

## Closing a schematic

To close a schematic without closing the schematic editor, select Close from the File menu.

If the current schematic has not been saved since last being edited, you will be prompted to save it.

File Menu

File	Edit	Draw	Navigate
New			
Open...			
Close			
Import...			
Save			Ctrl+S
Save As...			
Print...			
Printer Select...			
Edit Library			
Symbolize...			
Reports...			
Current Errors..			F10
Exit			



File Menu

File	Edit	Draw
New		
Open...		
Close		

---

# Using the Symbol Editor

---

# 4

## Overview

The symbol editor allows you to perform the following tasks:

- Create and edit symbols for use in the schematic editor.
- Edit existing libraries.
- Create new libraries.

This chapter provides background information about the symbol editor, which includes:

*Starting the Symbol Editor on page 4-5* describes procedures for starting and closing the symbol editor.

*Symbol Editor Window on page 4-8* describes the use of menus, the Toolbar and toolbar icons, the status line and the keyboard.

*Changing Text Characteristics on page 4-12* describes procedures for changing the text characteristics of attribute text, pin name and number display, and free-standing text.

*Changing Grid and Gravity on page 4-16* describes enabling and disabling grid, setting grid spacing, setting gravity and using text grid.

*Zooming and Panning on page 4-20* references the zoom and pan features of the symbol editor.

*Printing Symbols on page 4-21* describes how to print the symbols created with the symbol editor.

# Components

A component or device has several aspects associated with it:

- symbol—the graphical representation used in drawing schematics
- packaging information—defines the names of the package types (footprints) in which the component is available, the pin number assignments for those package types, and the number of gates (for multi-gate components)
- footprint(s)—used for board layout
- simulation model—if the component can be simulated with PSpice A/D or PLogic

## Symbols

Symbols are created and modified with the symbol editor. Symbols are stored in symbol libraries (.slb). Symbols consist of graphics, pins (for electrical symbols), and attributes.

## Packaging Information

Packaging information for a component is closely related to the symbol but is kept separately in a package definition. Package definitions are stored in package libraries (.plb). The association of the symbol and the packaging definition is by name. Generally, you will create a symbol with a given name and a package definition with the same name. If you use the Symbol Creation Wizard in Schematics to create symbols, it will automatically create a package definition for you.

Package definitions are created and modified with the Schematics symbol editor. They are also used, and can be created and modified, in the MicroSim PCBoards PCB layout editor.

## Footprints

The footprint for a component is the definition of its mechanical outline, pad pattern, identifiers, and physical extent (boundary). The packaging information for a symbol defines the names of the footprints (package types) in which it is available. For each footprint, the packaging information defines the physical pin numbers assignments for the pins. When a symbol is placed on a schematic, the PKGTYPE attribute defines the name of the footprint to be used in the layout.

Footprints themselves are created and maintained with the MicroSim PCBoards PCB layout editor. Refer to *Creating Footprints using the Footprint Editor* in the *MicroSim PCBoards User's Guide*.

## Simulation Models

If a component can be simulated, it will have an associated simulation model. The MODEL attribute on a symbol defines the name of the simulation model. Simulation models are stored in model libraries (.lib). You can create new simulation models with Parts or with a text editor. Refer to *Creating Models* in your PSpice user's guide.

# Starting the Symbol Editor

## Starting the symbol editor

In the schematic editor, click the Edit Symbol icon to create a new symbol editor document window if one does not already exist.



If you already have a symbol editor window open, you will be prompted to save any unsaved changes to the current symbol. You can only have one symbol editor window open at a given time.

When you save the symbol library, any open schematics are updated with the changes made in the symbol editor.

## Loading a Library for Editing

To edit or create symbols or package definitions in an existing library, you must first load the library for editing. You can also create a new library to contain the symbols or package definitions that you create.

### Opening an existing library

- 1 Click the Open File icon on the toolbar.
- 2 Type the name of the library in the Open dialog box.
- 3 Click OK.



### Creating a new library

Click the New File icon on the toolbar.



Any symbol or package definition you create will be saved in the new library.

You are prompted to name the library when you save the first symbol.

A quick way to edit a symbol for a part used on a schematic is:

- 1 Click on the part on the schematic to select it.
- 2 Click the Edit Symbol icon on the toolbar.

The symbol editor is started, the library containing the symbol is loaded, and the symbol is displayed for editing.

## Saving your Changes



To save newly created symbols or changes to existing symbols:

Click the File Save icon on the toolbar.

If the library is not configured for use in the schematic editor, you will be asked if you want to configure the library. Answer YES to make the symbols in the library available for use in schematics.

If the library is already configured, any schematics using symbols you have changed will be updated to use the new symbol.



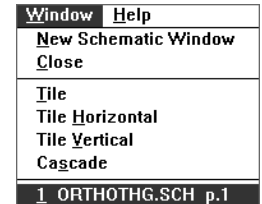
## Returning to the Schematic Editor

To return to the schematic editor and keep the symbol editor window open for additional symbol editing:

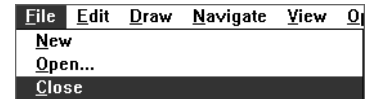
- Minimize the symbol editor window, or
- Click in the schematic editor window, or
- Select the schematic editor window from the Window menu.

When you are finished with the symbol editor close the symbol editor window by double-clicking the Control-menu box in the upper-left corner, or selecting Close from the File menu.

Window Menu



File Menu



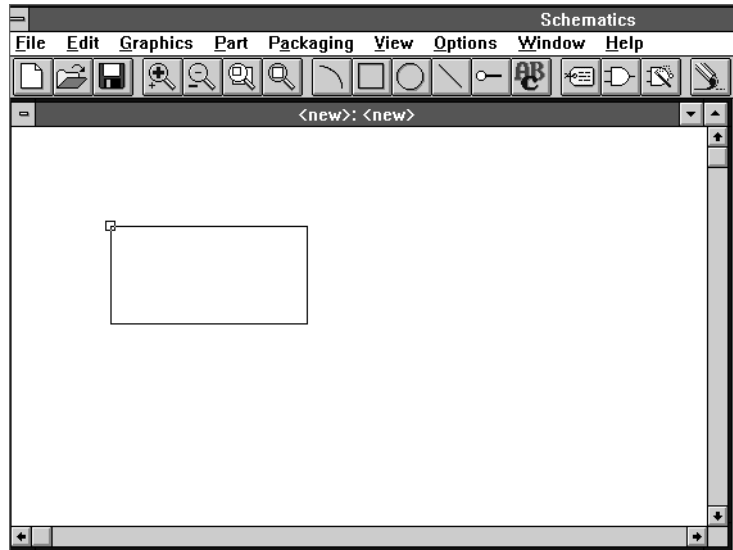
## Starting Automatically

If you are going to use the symbol editor more than the schematic editor, you can have the symbol editor start automatically when you start Schematics. Add the -sym option to the Command Line in the Windows Program Item Properties dialog box for the Schematics icon. For example:

```
C:\MSIM\psched.exe -sym
```

# Symbol Editor Window

When you start the symbol editor, the symbol editor window displays.



**Note** *You can only open one symbol editor window at a time and you can only edit one symbol at a time.*

## Refreshing the Screen



To clean up and refresh the screen, click the Redraw icon on the toolbar.

## Menus

There are a series of menus from which you can select the function you want to perform.

The display and operation of the menus and submenus follows a standard Windows layout and operation.

Schematics provides different menus for the schematic editor and for the symbol editor. The menu changes as you change active windows.











## Toolbar

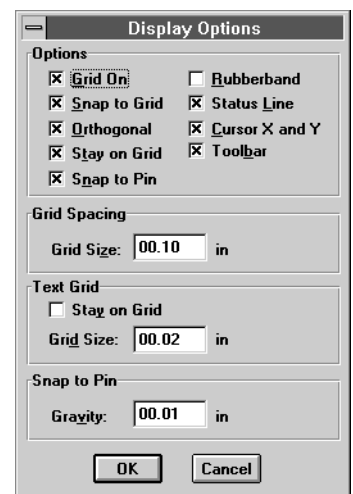
Toolbar buttons provide shortcuts for initiating functions.

To enable or disable the Toolbar display:








- 1 Select Display Options from the Options menu to display the Display Options dialog box.
- 2 Click the Toolbar check box.

**Table 4-1** *Symbol Editor Toolbar Icons*

Icon	Name	Function	Page
	New File	creates a new symbol library	4-5
	Open File	opens an existing symbol library	4-5
	Save File	saves the current symbol library	4-6
	Zoom In	views smaller area of the symbol	2-32
	Zoom Out	views a larger area of the symbol	2-32
	View Area	views a selected area of the symbol	2-32
	View Fit	fits the symbol view to the page	2-34
	Draw Arc	draws an arc shape on the symbol	5-11
	Draw Box	draws a box on the symbol	5-11
	Draw Circle	draws a circle on the symbol	5-12



**Table 4-1** *Symbol Editor Toolbar Icons*

Icon	Name	Function	Page
	Draw Line	draws a line on the symbol	<b>5-12</b>
	Place Pins	places pins on the symbol	<b>5-14</b>
	Draw Text	places a text string on the symbol	<b>5-13</b>
	Edit Attributes	edit the attributes of a symbol	<b>5-37</b>
	Get New Part	gets a symbol from a symbol library for editing	<b>5-10</b>
	Start Wizard	starts the Symbol Creation Wizard	<b>5-3</b>
	Redraw	refreshes the symbol editor screen display	<b>4-8</b>

## Title Bar

The symbol editor window title bar displays the name of the symbol library and the symbol currently being edited. For example:


```
[ C : \MSIM\LIB\PORT . SLB : GLOBAL ]
```

When you open a symbol editor window and have not specified a symbol for editing, the title bar displays:








```
<new> : <new>
```

This indicates that you are editing a new symbol in a new library.

## Keyboard

Table 4-2 lists the function keys you can use instead of menu selections to enable or disable certain functions. For those functions that toggle, pressing the function key enables the feature, and pressing  Shift plus the function key disables the feature.

**Table 4-2** *Symbol Editor Function Keys*

Key	Action	Menu	Selection
	help	Help	
	grid on	Options	Display Options
	text stay-on-grid	Options	Display Options
	auto-scroll	Options	Display Options
	stay-on-grid	Options	Display Options
	auto-repeat	Options	Auto-Repeat
	current errors	File	Current Errors

# Changing Text Characteristics

For any text placed on your symbol, such as free standing text, pin names and attribute names and values, there are options to set the desired text size, orientation, horizontal justification and vertical justification.

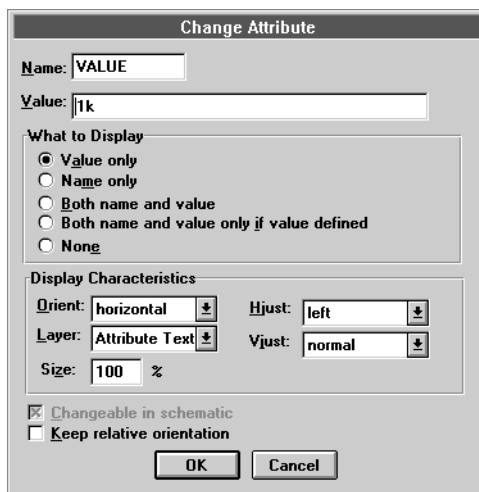
## Attribute Text

You can change the text characteristics of any of the displayed attributes of the symbol. The text characteristic changes you make are only applied to the attribute that you are currently editing.

### Changing attribute text characteristics

- 1 Double-click the text to display the Change Attribute dialog box.

Change any of the characteristics of the text in the Display Characteristics area of the dialog box, as shown in Table 4-3.



The image shows a 'Change Attribute' dialog box. It has a title bar 'Change Attribute'. Inside, there are two text input fields: 'Name: VALUE' and 'Value: 1k'. Below these is a section 'What to Display' with five radio button options: 'Value only' (selected), 'Name only', 'Both name and value', 'Both name and value only if value defined', and 'None'. Below that is a section 'Display Characteristics' with four dropdown menus: 'Orient: horizontal', 'Hjust: left', 'Layer: Attribute Text', and 'Vjust: normal'. There is also a 'Size: 100 %' field. At the bottom, there are two checkboxes: 'Changeable in schematic' (checked) and 'Keep relative orientation' (unchecked). At the very bottom are 'OK' and 'Cancel' buttons.

2 Click OK.

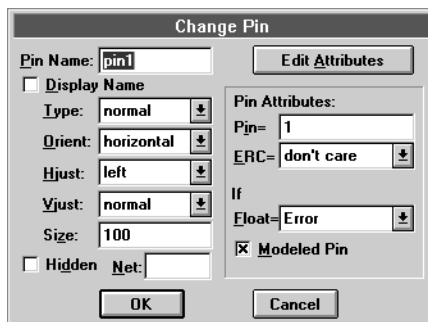
**Table 4-3** *Display Characteristics*

Characteristic	Explanation
Orient:	Allows you to position the text horizontally, vertically, upside down, or down in relation to the defining point of the text string.
Layer:	Specifies a text display level as defined by the Set Display Level function under the Options menu. Defaults to Attribute Text Layer. You can specify a user defined layer.
Size:	Determines the size of the text of a displayed text item. The size is expressed as a percentage of the default (100%) size.
Hjust:	Sets the horizontal justification for the placement of text items (left, center, or right).
Vjust:	Sets the vertical justification for placing text items (top, normal, or bottom).

## Pin Name and Number

### Changing pin name text characteristics

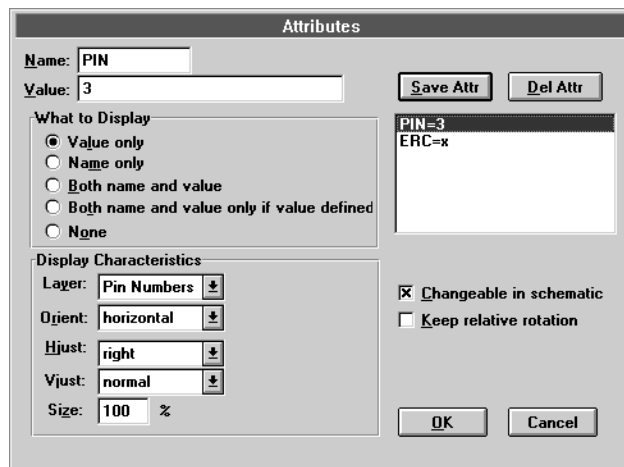
- 1 Double-click the pin name or pin number to display the Change Pin dialog box.



- 2 Change any of the text characteristics as shown in Table 4-3.
- 3 Click OK.

### Changing pin number text characteristics

- 1 Double-click the pin name or pin number to display the Change Pin dialog box.
- 2 Click Edit Attributes to display the Attributes dialog box.



- 3 Click to select an item in the list box.



Change any of the characteristics of the text in the Display Characteristics area of the dialog box, as shown in Table 4-3.

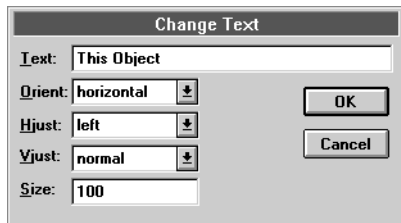
- 4 Click OK.
- 5 In the Change Pin dialog box, click OK.

## Free-Standing Text

You can change the text characteristics of any of the free-standing text that you have placed on the symbol. The changes you make are only applied to the text item you are currently editing.

### Changing free-standing text characteristics

- 1 Double-click the text to display the Change Text dialog box.



Change the orientation, justification or size, as shown in Table 4-3 on **4-13**.

- 2 Click OK.

# Changing Grid and Gravity

The symbol editor has grid spacing and gravity settings independent of the grid spacing and gravity settings of the schematic editor. You can turn the grid on or off, and change the settings.

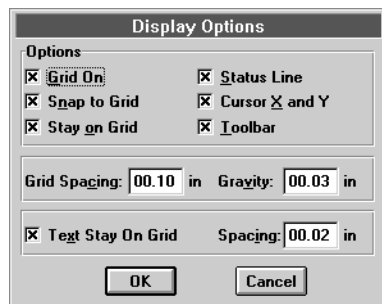
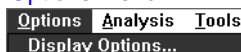
## Grid On

When Grid On is enabled, the grid displays in the drawing area of the symbol editor window.

### Enabling or disabling Grid On

- 1 Select Display Options from the Options menu to display the Display Options dialog box.
- 2 Click the Grid On check box to enable or disable the grid display.  
  
An “X” in the check box indicates that the grid is “ON” or enabled.
- 3 Click OK.

#### Options Menu



## Snap-to-Grid

Snap-to-Grid causes elements to snap to the nearest grid point when you place them.

When moving a part outline to place a part, you can move the part outline freely. If Snap-to-Grid is disabled, the part outline movement is by grid units rather than a smooth motion.

## Enabling or disabling Snap-to-Grid

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown above).
- 2 Click the Snap-to-Grid check box to enable or disable Snap-to-Grid.  
  
An “X” in the check box indicates that Snap-to-Grid is enabled.
- 3 Click OK.

## Stay-on-Grid

Stay-on-Grid causes elements to be placed strictly on the grid.

When moving a part outline to place the part, if Stay-on-Grid is enabled, the part outline moves from grid line to grid line instead of moving in a smooth motion.

## Enabling or disabling Stay-on-Grid

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **4-16**).
- 2 Click the Stay-on-Grid check box to enable or disable Stay-on-Grid.  
  
An “X” in the check box indicates that Stay-on-Grid is enabled.
- 3 Click OK.

## Grid Spacing

Grid Spacing defines the horizontal and vertical grid spacing on the drawing area. The default spacing is 0.10 inches for US-standard page sizes, and 2.5 millimeters for metric page sizes. The minimum grid spacing allowed is 0.01 (US sizes).

The default grid spacing of 0.1" is the required pin-to-pin spacing. If you try to save a symbol that has pins at intervals other than 0.1", you will be warned that such pins are considered to be off grid and need to be on-grid for proper connectivity.

### Setting grid spacing

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **4-16**).
- 2 Type the grid spacing value in the Grid Size text box.
- 3 Click OK.

## Gravity

The Gravity setting determines how close the pointer must be to an object in order for the object to be selected when you click the mouse. The default is .03" (or .75mm).

### Changing the gravity

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **4-16**).
- 2 Change the value displayed in the Gravity text box.
- 3 Click OK.

## Text Stay-on-Grid

Text Stay-on-Grid allows you to set the grid spacing for text separately from the normal grid spacing. The text grid is usually set to some smaller percentage of the regular drawing grid. This allows you to align text along smaller increments of the regular grid.

### Enabling text grid and specifying text grid size

- 1 Select Display Options from the Options menu to display the Display Options dialog box (shown on **4-16**).
- 2 In the Text Stay-on-Grid field, click the Text Stay-on-Grid check box to enable the text grid.

An “X” in the check box indicates that the text grid is enabled.

- 3 Type the text grid spacing value in the Spacing text box.
- 4 Click OK.

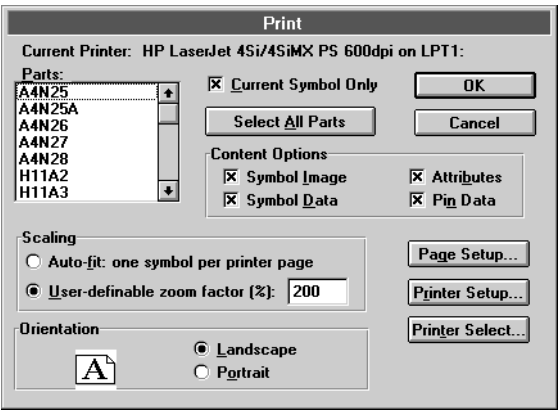
# Zooming and Panning

The zoom and pan features in the symbol editor are the same as they are in the schematic editor. See *Zooming and Panning in Schematics on page 2-32*.

# Printing Symbols

## Printing a symbol

- 1 Select Print from the File menu to display the Print dialog box.



File Menu

File	Edit	Draw	Navigate	View	O
New					
Open...					
Close					
Import...					
Save					Ctrl+S
Save As...					
Print...					

- 2 Select the part or parts to be printed.
  - a Click the Current Symbol Only check box to print the symbol currently being edited, or
  - b Select one or more parts from the list of parts in the current library from the Parts list, or
  - c Click Select All Parts to print all parts in the current library.
- 3 In the Content Options area, click the check box to select the information you would like to see printed. The Content Options are described in Table 4-4.
- 4 Select one of the Scaling options.

Auto-fit expands the symbol to full page size. User-definable allows you to specify the zoom factor for the size of the symbol.
- 5 Select an Orientation: Landscape or Portrait.
- 6 Click OK.

**Table 4-4**    *Content Options*

Option	Description
Symbol Image	specifies printing the graphics of the selected symbol
Attributes	specifies printing the attributes and the attribute values of the selected symbol
Symbol Data	specifies printing the description, type, Bbox dimensions and origin position of the selected symbol
Pin Data	specifies printing the pin data of the selected symbol



---

# Creating and Editing Symbols

---

## 5

### Overview

This chapter describes how to use the symbol editor to copy, create and edit symbols and has the following sections:

*Creating New Symbols on page 5-3* describes the four essential methods of creating a new symbol.

*Editing Existing Symbols on page 5-10* describes the procedures for creating symbols for simulation models.

*Drawing Symbol Graphics on page 5-11* describes the assortment of drawing tools provided for creating and editing a symbol.

*Defining and Editing Pins on page 5-19* describes the editing features for defining and editing pins, and for defining and editing packaging definitions.

*Editing Symbol Attributes on page 5-24* describes how to add and edit the properties of a symbol.

*Using Symbol Aliases on page 5-26* describes how to give a symbol an alternate name.

*Specifying Part Packaging Information on page 5-27* describes procedures for defining packaging information.

*Configuring Custom Libraries on page 5-41* describes the procedure for making a custom library accessible to Schematics.

# Creating New Symbols

There are four methods for creating a new symbol:

- Use the Symbol Creation Wizard. The wizard prompts you through the steps of creating a symbol and also creates packaging information for the symbol.
- Make a copy of an existing symbol under another name and modify the copy.
- Import a symbol definition exported by another Schematics user.
- Create an A Kind Of (AKO) symbol which is a reference of any existing symbol.

## Using the Symbol Creation Wizard

The Symbol Creation Wizard helps you to create new symbols. Some of the features and benefits of using the Symbol Creation Wizard are:

- Eases the creation of symbols by guiding you through each step of the process.
- Provides simple point-and-click dialog boxes.
- Provides feedback as you make decisions.
- Avoids using complicated mouse/keyboard interactions.
- Allows you to navigate through the symbol creation process by backing up to repeat a step and moving forward.

When you start the Symbol Creation Wizard, you are first asked for a symbol name and for a description. You are then taken through a progression of screens that provide you with information, ask you questions and present you with choices.

If you are creating a symbol for an existing simulation model, see **Chapter 7, *Preparing Your Design for Simulation, Creating Symbols for Existing Simulation Models*** on page 7-6.



**Note** *Symbol names can not contain spaces.*

### Starting the Symbol Creation Wizard

- 1 With the symbol editor window active, click the Start Wizard icon.
- 2 Follow the instructions that appear on the screen.

Create New Symbol

Please enter the name of the symbol to be created and a short description.

New Symbol Name:

New Symbol Description:

Help Cancel < Back Next > Finish

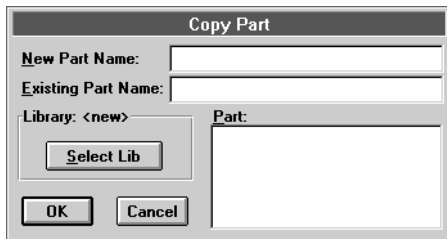
### Creating a Symbol by Copying Another Symbol

An easy way to create a symbol is to make a copy of a similar symbol and modify the copy.

# Making a Copy of a Symbol

## Copying a symbol from another library

- 1 Select Copy from the Part menu to display the Copy Part dialog box.



Part Menu

Part	Packaging	View	Options
Wizard...			
New...			
Copy...			

- 2 Click Select Lib and select a library from the Open dialog box.
- 3 Type the name of the part to be copied in the Existing Part Name text box, or select it from the Part list box.
- 4 Type a new name for the part in the New Part Name text box.
- 5 Click OK.

You can now edit the symbol. When you finish editing, use the Save selection from the File menu to save your changes.

To create the symbol in the *current* library, click the File Save icon.



## Creating a new symbol in a *different* library

- 1 Select Save to Library from the Part menu.
- 2 Type the name of the library (.slb) where the symbol is to be saved.

If there is packaging information associated with the symbol, use the Copy selection from the Packaging menu to similarly copy the package definition.

# Importing a symbol definition

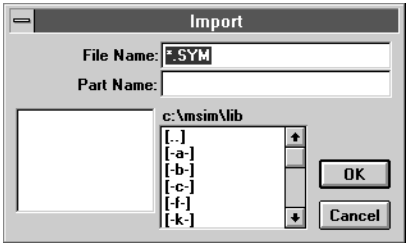
The Import function allows you to import a symbol that has been previously exported as an individual file and incorporate it into a symbol library file.

## Importing a symbol

Part Menu

Part	Packaging	View	Options
Wizard...			
New...			
Copy...			
Get...			Ctrl+G
Remove...			
Save to Library...			
Attributes...			
Definition...			Ctrl+D
Pin List...			Ctrl+P
Get Symbol Graphics...			
Export...			
Import...			

- 1    Select Import from the Part menu to display the Import dialog box.



- 2    In the File Name text box, type the name of the file to be imported, or select the file name from the file selection list box.
- 3    In the Part Name text box, type the name to be given to the imported symbol.
- 4    Click OK.

The Export function allows you to write a symbol definition from the current symbol library to a text file. This function allows you to transfer symbols from one library to another, or from one platform to another without having to transfer the entire symbol library.

## Exporting a symbol

- 1 Select Export from the Part menu to display the Export Parts dialog box.



Part Menu

Part	Packaging	View	Options
Wizard...			
New...			
Copy...			
Get...			Ctrl+G
Remove...			
Save to Library...			
Attributes...			
Definition...			Ctrl+D
Pin List...			Ctrl+P
Get Symbol Graphics...			
Export...			

- 2 In the Part Name text box, type the name of the symbol to be exported, or select it from the list box.
- 3 In the File Name text box, type the name of the file to which the part definition is to be written.
- 4 Click OK.

## Using AKO Symbols

Some of the MicroSim symbol libraries are made up of a few base symbols and several AKO (A Kind Of) symbols. In the “bipolar.slb” symbol library, for example, there are several parts, but only two real symbols: qnpn and qpnp. Every other symbol references one of these two base symbols. Said another way, every other part is *A Kind Of* one of the base symbols.

The AKO mechanism in Schematics allows you to draw one symbol (i.e. the *base* symbol), and then reference that symbol for all other symbols of the same type. For example, to create the bipolar symbol library, we created one symbol for an npn transistor (qnpn) and one symbol for a pnp transistor (qpnp). Every other bipolar transistor symbol is an AKO of either qnpn or qpnp.

A base symbol must be contained in the same library as the symbols which reference it. Base symbols do not need to be displayed in the Part Browser. If you copy an AKO symbol from another library (see *Creating a Symbol by Copying Another Symbol on page 5-4*), you must also copy its base symbol.

### Creating a base symbol in a custom symbol library

- 1 In the symbol editor, select New from the Part menu.
- 2 Type a name for the part in the Part Name text box. (E.g., TestCase)
- 3 Type a description of the part in the Description text box.
- 4 Leave the AKO Name text box blank, and select the Do not display in Part Browser check box.
- 5 Click OK.

### Saving the symbol to a library

- 1 Select Save from the File menu.
- 2 Type the name of the library in the File Name text box.
- 3 Click OK.
- 4 In the Configure dialog box, answer Yes to Add to list of Schematics configured libraries?"

Now you can draw a symbol using the procedures given in “Drawing Symbol Graphics” and “Defining and Editing Pins” in Chapter 5, Creating and Editing Symbols, on pages 5-12 through 5-23.

You have defined this to be a base symbol by leaving the AKO Name text box blank and selecting the Do not display in Part Browser check box.

Once you have created the base symbol, you can create other symbols which reference the base symbol (i.e., AKO symbols).



## Creating an AKO symbol

- 1 In the symbol editor, select New from the Part menu.
- 2 Type a name for the part in the Part Name text box.
- 3 Type a description of the part in the Description text box.
- 4 Type the name of the base symbol in the AKO Name text box. (E.g., AKOTest)
- 5 Click OK.

The symbol graphics of the base symbol displays in the symbol editor window.

Select Save from the File menu to save the custom symbol library.

# Editing Existing Symbols

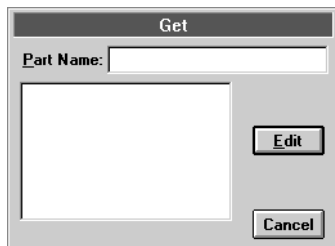
To edit an existing symbol, you must first load the library in which the symbol is stored.

## Loading a symbol library

- 1 Click the File Open icon on the toolbar.
- 2 Type a library name in the Open dialog box.
- 3 Click OK.

## Selecting a part for editing

- 1 Click the Get New Part icon on the toolbar to display the Get dialog box.



- 2 Select a part from the list of names.
- 3 Click Edit.

## Editing the packaging information for the symbol in the current package library

- 1 Select Edit from the Packaging menu.
- 2 See *Specifying Part Packaging Information on page 5-27* for more information.

# Drawing Symbol Graphics

There are several graphic tools available for drawing symbols. These tools allow you to draw circles, lines, arcs and boxes. You can also place pins and text on your symbol.

## Elements of a Symbol

Drawing a symbol consists of placing various combinations of arcs, boxes, lines, circles, text and pins in the appropriate locations.

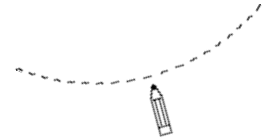
### Arc

#### Drawing an arc

- 1 Click the Draw Arc icon to change the pointer to a pencil shape.
- 2 Click to establish the end-point for the arc.
- 3 Click again to establish the other end-point for the arc.

A straight dotted line connects the two end-points.

- 4 As you move the pointer out from the last end point, the dotted line takes on the shape of an arc. When the arc reaches the desired shape, click to fix the arc at that location.

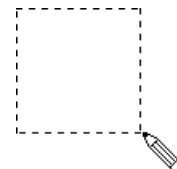


### Box

#### Drawing a box

- 1 Click the Draw Box icon to change the pointer to a pencil shape.
- 2 Click at the location for the upper left corner of the box.
- 3 Move the pointer down and to the right.

A dotted box outline follows the pointer.

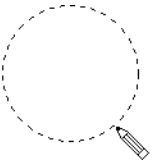


- 4 Click at the location of the lower right corner of the box.

### Circle

#### Drawing a circle

- 1 Click the Draw Circle icon to change the pointer to a pencil shape.
- 2 Click the location of the center of the circle.
- 3 Move outward from the center of the circle.  
A dotted circle outline follows the pointer.
- 4 Click to complete the circle.



### Line

#### Drawing a line

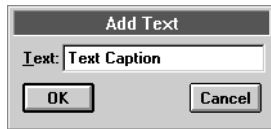
- 1 Click the Draw Line icon to change the pointer to a pencil shape.
- 2 Click to establish the beginning point of the line.  
As you move the pointer, a dotted line follows.
- 3 Click to place one or more vertices.
- 4 Double-click to establish the end point of the line, or right-click at any point to stop line drawing.



## Text

### Adding a text string to the symbol

- 1 Click the Draw Text icon to display the Add Text dialog box.

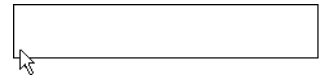


- 2 Type the text string in the text box.
- 3 Click OK.

An outline box follows the pointer and indicates the outline of the text string.
- 4 Move the outline to the location where you want to place the text.
- 5 Click to place the text string.

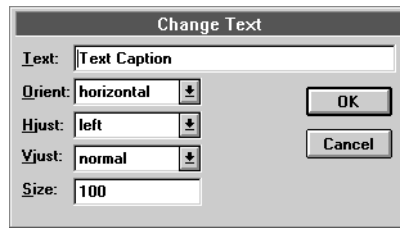
The outline box remains on the screen. You can move and click to place the same text string in several locations.
- 6 To stop placing text, do one of the following:
  - a Double-click to place the last instance of the text and stop placing text.
  - b Right-click to stop placing text without placing any additional text.

The outline changes back to a pointer.



### Revising a text string or changing text characteristics

- 1 Double-click the text string to display the Change Text dialog box.



- 2 Make the desired changes to the text string and its characteristics.

Instructions for setting text characteristics are included in the previous chapter. See *Changing Text Characteristics on page 4-12*.

- 3 Click OK.

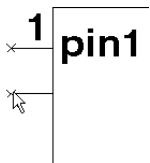
### Pins

#### Adding pins to a symbol

- 1 Click the Place Pins icon to change the pointer to a pin outline.
- 2 Move the pin outline to the desired location and click to place the pin.

A small 'x' appears on one end marking the connectivity point of the pin.

- 3 Click at each location to place additional pins.
- 4 Do one of the following:
  - a Double-click to place the last pin and stop placing pins.
  - b Right-click to stop placing pins without placing an additional pin.



**Note** When placing pins, the pin type defaults to the type that was last placed, or to the type last specified.

The procedures for defining and editing pins are explained in a following section beginning on

## Rotating and Flipping

Elements being drawn, elements already drawn and entire areas of a drawing can be rotated and flipped. A rotated element is rotated 90° counter-clockwise. A flipped element is mirrored about the Y axis.

### Rotating elements

#### Rotating an element before placing it on the drawing

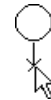
- 1 Select the drawing element.
- 2 Press **[Ctrl]+R** to rotate it 90° counterclockwise.



Default

#### Rotating an object already drawn

- 1 Select the object.
- 2 Press **[Ctrl]+R** to rotate it 90° counterclockwise.



Rotated once

#### Rotating an area of the drawing

- 1 Click and drag with the mouse to select and outline the area to be rotated.
- 2 Press **[Ctrl]+R** to rotate the area about its center point.

**Figure 5-1** *Rotating a*

### Flipping elements

#### Flipping an element before placing it on the drawing

- 1 Select the drawing element.
- 2 Press **[Ctrl]+F** to flip it.



Default

#### Flipping an object already drawn

- 1 Select the part.
- 2 Press **[Ctrl]+F** to flip it.



Flipped once

**Figure 5-2** *Flipping a*

## Flipping an area of the drawing

- 1 Click and drag with the mouse to select and outline the area to be flipped.
- 2 Press **Ctrl+F** to flip the area about its vertical axis.

## Selecting

### Selecting an element of the drawing

Point to the item with the mouse and click to select it.

The item color (the default is red) indicates it is selected.

Once the item is selected, you are ready to perform an action. Selecting a new item causes any previously selected items to be deselected.

### Selecting more than one element

Hold down **Shift** while clicking and selecting.

The elements change color which indicates they are selected.

### Selecting all elements within a given area of the drawing

Select the area by holding down the mouse button while dragging the mouse across the desired area. A Region of Interest box (ROI box) appears to indicate all items within it are selected. **Only items entirely contained within the box are selected.**



## Moving

### Moving an object

- 1 Select an object (or group of objects).
- 2 Put the mouse pointer on the object or in the area designated by the ROI box. While holding down the mouse button, drag the object(s) to the desired location.
- 3 Release the mouse button.

## Cutting, Copying and Pasting

The symbol editor has several common editing functions that allow you to cut, copy, paste, repeat, delete and undelete selected objects. All of these functions are available under the Edit menu. Most can be activated from the keyboard.

The cut, copy and delete functions only apply when an object is selected. (See *Selecting on page 5-16* to learn how to select single and multiple objects as well as objects within a given area.

### Cutting

The Cut function deletes the selected object (or group of objects) from the drawing and copies it to a buffer for use with the Paste function. The buffer retains only the last object that was cut.

### Cutting a selected object

Select Cut from the Edit menu.

Shortcut: press **Ctrl**+X.

### Deleting

The delete function deletes an object (or set of objects). A deleted object cannot be copied or pasted.

### Deleting a selected object

Press **Delete**.

### Undeleting

Undelete restores the object last cut or deleted from the drawing. Undelete returns the object to the exact position on the drawing from which it was cut or deleted.

### Undeleting an object

Shortcut: press **Ctrl**+U .

Select Undelete from the Edit menu.

### Copying

The Copy function makes a copy of the selected object for pasting. The selected object remains on the schematic and a copy is placed in the buffer.

### Copying a selected object

Shortcut: press **Ctrl**+C .

Select Copy from the Edit menu.

### Pasting

The Paste function places one or more copies of the last object stored in the buffer (from a Cut or Copy operation) onto the drawing.

### Pasting an object

Shortcut: press **Ctrl**+V .

- 1 Select Paste from the Edit menu to change the pointer to shape of the object last cut or copied.

- 2 Click to place the object at the current pointer location.

Continue moving the pointer to various locations and clicking to place additional copies of the object.

- 3 To stop pasting the object, do one of the following:

- a Double-click to paste the last instance of the object and stop pasting.

- b Right-click to stop pasting without pasting another object.

With Auto-Repeat enabled, use **Space** to place repeated copies of items from the buffer without using the Paste function.

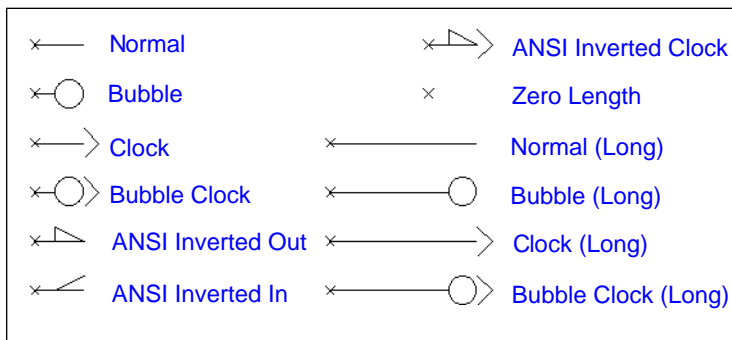
# Defining and Editing Pins

Pins establish the input and output terminals into and out of symbols. For a pin you can:

- choose the type of graphic to display
- specify a pin name
- specify a pin number
- choose whether the name, the number, or both should be displayed

## Specifying Pin Types

Figure 5-3 shows the twelve types of pins that you can place using Schematics.



**Figure 5-3** Pin Types

## Selecting a pin type for pins to be placed

**Note** To specify the default pin type, do not select any pins when performing the following procedure. If a pin is selected, the pin type will only apply to that pin.

- 1 Select Pin Type from the Edit menu to display the Pin Type dialog box.



You can also select the pin, then select Pin Type from the Edit menu.

Shortcut: press **Ctrl**+**T** .

The Float= and Modeled Pin fields in the Change Pin dialog are only relevant for symbols that are going to be simulated with PSpice or PLogic. For additional information, refer to your PSpice user's guide.

To create pin names with overbars, use the `\` character, e.g., `\CLK\`

Use the Pin List selection from the Part menu to view and edit the list of all pins for the symbol.

- 2 Click the appropriate radio button to select the pin type.
- 3 Click OK.

### Changing the type of a placed pin

- 1 Double-click the pin to display the Change Pin dialog box.



- 2 In the Type list box, select a pin type.
- 3 Click OK.

### Changing the pin name

As you placed pins, default names were assigned. To change the name of a pin:

- 1 Double-click on the pin or pin name.
- 2 In the Change Pin dialog, type in the pin name in the Pin Name text box.
- 3 Click OK.

**Note** Pin names *MUST* be unique, i.e., you cannot have two pins with the same name.

## Changing the pin number

As you placed pins, default pin numbers were assigned. To change the pin number for a pin:

- 1 Double-click the pin or pin number of the displayed pin.
- 2 In the Change Pin dialog box, type the pin number in the Pin text box.
- 3 Click OK.

## Turning the display of the pin name on or off

By default, pins you place on symbols will have their pin names displayed. To turn display of the pin name off:

- 1 Double-click the pin or pin name.
- 2 In the Change Pin dialog box, click the Display Name check box to disable the name display.
- 3 Click OK.

## Turning the display of the pin number on or off

By default, pins you place on symbols will have their pin names displayed. To turn display of the pin name off:

- 1 Double-click the pin or pin number.
- 2 In the Change Pin dialog box, click Edit Attributes.
- 3 In the Edit Attributes dialog box, select the PIN= entry in the list box.
- 4 In the What to Display area, check None to turn display of the pin number off.
- 5 Click Save Attr.
- 6 Click OK.
- 7 In the Change Pin dialog box, click OK

## Defining and Editing Hidden Power and Ground Pins

The symbol editor allows you to set a pin to be invisible. If you set the visibility off, you must supply the name of a connecting net (normally a global net like \$G\_DPWR or \$G\_DGND) for the pin in the Net text box. The net is recorded as a symbol attribute (not a pin attribute). The `IPIN(<pinname>)=<net name>` attribute conveys the net name.

You can also select the pin and select Change from the Edit menu.

When you place the part on a schematic, you can change the power or ground net to which the part is connected by changing the value of the attribute.

### Defining a hidden pin

- 1 Double-click the pin to display the Change Pin dialog box.
- 2 Click to place an X in the Hidden check box.
- 3 In the Net text box, type the name of the net to which the hidden pin is to be connected.
- 4 Click OK.

## Specifying Symbol Origin and Bounding Box

The origin is designated for placing a part, and is the point about which the part is rotated. By default, the origin is at (0,0). It is maintained as a point of reference on the schematic.

The bounding box defines the selection area of the symbol when placed on a schematic. After drawing a symbol, all of the elements of the symbol must be enclosed in the bounding box.

### Origin

You can change the origin to be at another point on the symbol. By default, part symbols in the symbol libraries have the origin on the hotspot of the upper left-most pin. The hot-spot is the point of connection (to a wire or to another pin).

If you change the origin of a symbol in the symbol editor, (thus changing the location of the symbol graphics relative to that

point) the symbol graphics relocate accordingly in the schematic editor whenever you edit previously created schematics.

## Editing a part origin

- 1 Select Origin from the Graphics menu to change the pointer to a pencil shape.
- 2 Move the pointer to the point on the object where you want to place the origin and double-click to fix the origin at that point.

### Graphics Menu

Graphics	Part
Arc	
Box	
Circle	
Line	
Pin	
Text...	
Bbox	
Origin	

## Bounding Box

The bounding box is the rectangular dotted line surrounding the symbol. When you click a part from within the schematic editor, the area in which you can click and have that part be selected is defined by the bounding box of the symbol.

## Resizing the bounding box

- 1 Select Bbox from the Graphics menu to change the pointer to a pencil shape.
- 2 Click to begin sizing the bounding box.
- 3 Move the pointer down and to the right. A dotted box outline follows the pointer.
- 4 Click at the location of the lower right corner of the bounding box.

- All pins *must* be contained within the bounding box for proper connections to be made in the schematic editor.
- Hidden pins, like those found on digital parts, do not have to be, and in most cases are not, contained within the bounding box.
- Attributes do not need to be contained within the bounding box.

# Editing Symbol Attributes

You can add attributes (properties) to a symbol. When you add an attribute, you specify a name and a default value. This value can be changed when the symbol is used on a schematic. You can specify whether the attribute should be displayed or not displayed.

There are two attributes that are automatically added to symbols that are created.

- The REFDES attribute, whose default value is U?, specifies the reference designator pattern to use in the schematic editor.
- The PART attribute displays the symbol's name.

**Note** When the symbol is placed on the schematic, Schematics automatically fills in the value of the PART attribute to be the name you used to place the symbol, i.e., if the symbol has several aliases, it fills in the alias that you used. Therefore, you can assign a value in the symbol editor and use it to place the text on the symbol, but when the symbol is used, the value will be re-assigned.

## Adding an attribute

- 1 Click the Attributes icon on the toolbar to display the Attributes dialog box.

**Attributes**

Name: VALUE  
Value: 1k

Save Attr Del Attr

What to Display

- ☒ Value only
- ☐ Name only
- ☐ Both name and value
- ☐ Both name and value only if value defined
- ☐ None

Display Characteristics

Layer: Attribute Text  
Orient: horizontal  
Hjust: left  
Vjust: normal  
Size: 100 %

TEMPLATE=R\*@REFDES %1  
REFDES=R?  
VALUE=1k  
PART=R  
TOLERANCE=

☒ Changeable in schematic  
☐ Keep relative rotation

OK Cancel



- 2 Type the name of the attribute in the Name text box.
- 3 Type in the default value, optionally, in the Value text box.
- 4 By default, the attribute value displays on the symbol. To turn display off, choose None in the What to Display area.
- 5 By default, the attribute value can be changed in the schematic editor on an instance-by-instance basis. To prevent changes to the attribute value, clear the Changeable in schematic check box.
- 6 By default, attributes that are displayed do not have their text rotated if the symbol is rotated on the schematic (to make it more readable). To have it rotate with the symbol, select the Keep relative rotation check box.
- 7 Click Save Attr
- 8 Click OK

### Editing a displayed attribute

If the attribute is displayed, double-click on it.

To edit an undisplayed attribute, or to make multiple changes, click the Attributes icon on the toolbar.

## Using Symbol Aliases

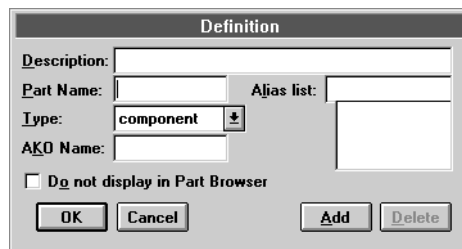
A symbol has a name. It can also have one or more aliases associated with it. Aliases are other names by which the device represented by the part is known. For example, you can have a symbol named 74AC269 which has as one of its aliases HD74AC269P.

When defining an alias, keep in mind that the aliased device will share the same graphics, pins and attributes as the primary symbol. When you place it on the schematic, however, it is treated as a separate part type. The name displayed on the schematic is that of the alias. Each alias requires its own packaging information.

### Adding an alias for a symbol

The PART attribute you define on a symbol will have its value filled in when you place it on the schematic. The name that you call it up with will get filled in as the PART attribute's value.

- 1 Select Definition from the Part menu to display the Definition dialog box.



- 2 Type in the name of the alias in the Alias list text box.
- 3 Click Add.
- 4 Click OK.

# Specifying Part Packaging Information

If you are going to use a symbol for PCB layout, you will need to specify package or device information.

Package information consists of:

- the number of gates per package
- list of package types (footprints) in which the device is available
- one or more pin assignment lists
- functionally equivalent pins that can be swapped in layout

Package information is used by Schematics to package together gates and to generate layout netlists. It is also used by MicroSim PCBoards.

## Pin Assignment Lists

A pin assignment list is a list of physical pin number assignments for each package type in which a device is available. Since a device may be available in several package types (DIP14, LCC20, etc.) and since each may have different pin number assignments, a single package definition can contain more than one pin assignment list. Each pin assignment list is associated with a list of package types (footprints) for which the pin number assignments are valid.

## Packaging Definitions

Packaging information is kept in a package definition, separate from the symbol definition. Both are maintained using the symbol editor. By default, the name of the package definition for a symbol corresponds to the symbol name. (This can be overridden by explicitly adding a COMPONENT attribute to the symbol. Such an attribute is generally used for devices which have non-standard part names such as BJTs whose names begin with a 'Q,' or for those with more than one type of gate.)

Package definitions are stored in package libraries. These libraries normally have the same name as the corresponding symbol libraries, but a different extension (".plb"). Package libraries are similar to symbol libraries in that they must be configured into the schematic editor's list of libraries.

The Packaging menu in the symbol editor allows you to create and edit package definitions.

## Creating a New Package Definition

You can quickly create a package definition for an existing symbol.

### Creating a package definition for an existing symbol

Shortcut: press **Ctrl**+E .

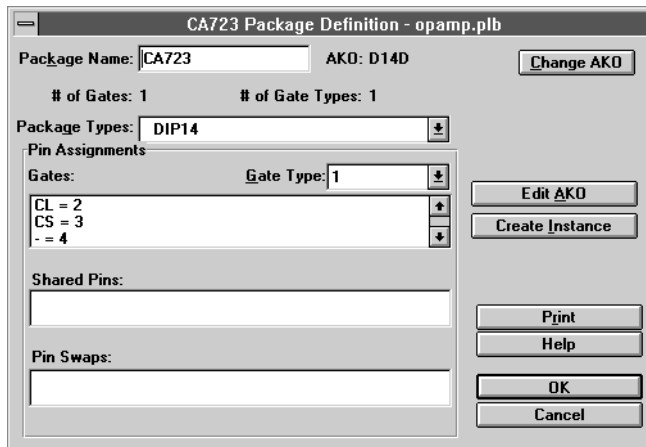
- 1 Load the symbol in the symbol editor.
- 2 Select Edit from the Packaging menu to display the Edit Package Definition dialog box.

The dialog box displays the message, "No package definition exists. Create?"

- 3 Click OK.

The dialog box displays the message, "Use symbol information to create default pin assignments?"

- 4 Click Yes to display the Package Definition dialog box.



The image shows a dialog box titled "CA723 Package Definition - opamp.plb". It contains the following fields and controls:

- Package Name:** A text box containing "CA723".
- AKO:** A text box containing "D14D" with a "Change AKO" button to its right.
- # of Gates:** A text box containing "1".
- # of Gate Types:** A text box containing "1".
- Package Types:** A dropdown menu showing "DIP14".
- Pin Assignments:** A section containing:
  - Gates:** A text box containing "CL = 2", "CS = 3", and "- = 4".
  - Gate Type:** A dropdown menu showing "1".
- Shared Pins:** An empty text box.
- Pin Swaps:** An empty text box.
- Buttons:** "Edit AKO", "Create Instance", "Print", "Help", "OK", and "Cancel".

- 5 Type a name for the package in the Package Name text box. Accept the default values for all other fields.
- 6 Click OK.

A simple single-gate package definition is created using the pin numbers specified by the symbol and defaulting to a DIP14 package type. You can edit the package definition if necessary. See *Editing a Package Definition* on page 5-31.

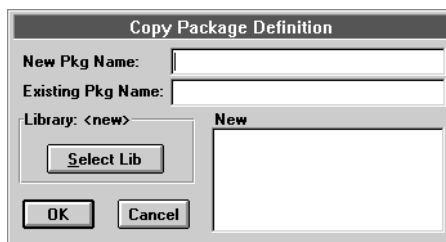
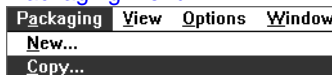
## Copying a Package Definition

The Copy function from the Packaging menu allows you to create a new package definition from an existing one. As with the Copy function under the Parts menu, the definition may be copied from the current library or a different library. See *Making a Copy of a Symbol* on page 5-5.

### Copying a package definition

- 1 Select Copy from the Packaging menu to display the Copy Package Definition dialog box.

#### Packaging Menu



- 2 Type the name of the package to be copied in the Existing Pkg Name text box. To select a package definition from another library, click Select Lib and select a library from the File Open dialog box.
- 3 Type a new name for the package in the New Pkg Name text box.
- 4 Click OK.

## Editing a Package Definition

You can edit a package definition for the current symbol or for any package definition in the current package library.

### Editing the package definition for the current symbol

- 1 Select Edit from the Package menu to display the Package definition dialog box (shown on 5-29).

The options within the dialog box are discussed in the following sections.

- 2 When you are finished with the dialog box, click OK.

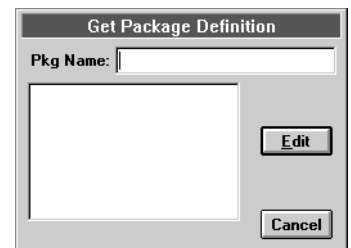
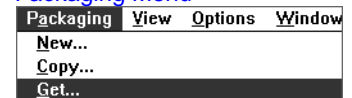
### Editing a package definition in the current package library

- 1 Select Get from the Packaging menu to display the Get Package Definition dialog box.
- 2 Type the name of the package in the Pkg Name text box or select a name from the list box.
- 3 Click Edit to display the Package Definition dialog box with the values for the requested package listed.

The options within the dialog box are discussed in the following sections.

- 4 When you are finished with the Package Definition dialog box, click OK.

Packaging Menu



### Editing Package Types

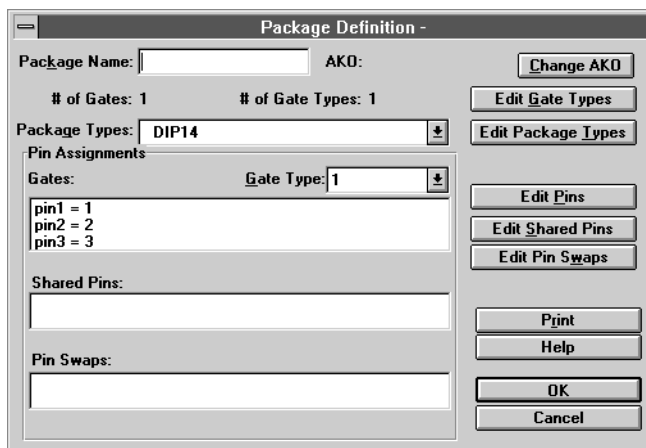
When you are editing a package definition, you can specify the package types in which a component is available. The package type name defines the footprint name to be used in layout.

## Adding a package type for a component

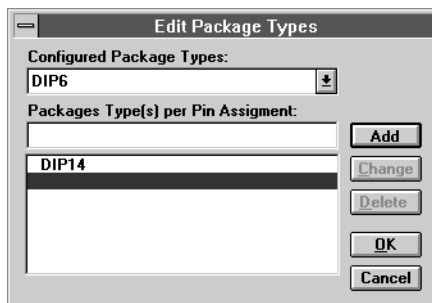
To add a new package type to the list of available package types for the component:

Shortcut: press **Ctrl+E**.

- 1 Select Edit from the Packaging menu to display the Package Definition dialog box with the values for the current symbol listed.



- 2 Click Edit Package Types to display the Edit Package Types dialog box.



**Note** The Configured Package Types List is a list of commonly used package types; it is not an exhaustive list.

- 3 In the Package Type(s) per Pin Assignment text box, type the name of the package (e.g., DIP14) or choose a type from the Configured Package Types scroll list.
- 4 Click Add.
- 5 Click OK.
- 6 In the Package Definition dialog box, click OK.



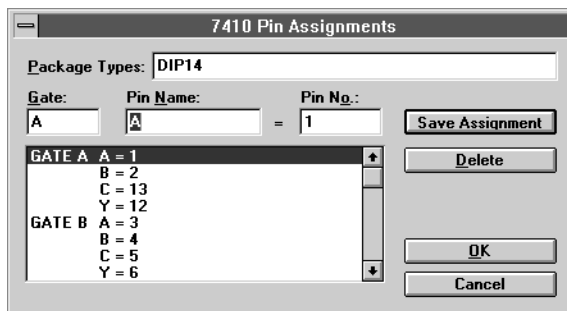
## Specifying Physical Pin Numbers

For each package type (or group of package types that share the same pin-out) the physical pin numbers for each pin must be defined. The Pin Assignments area in the Package Definition dialog box shows the pin numbers currently assigned for each logical pin on the symbol (for the package type currently displayed).

**Note** The pin name **must** match that used in the symbol. For components only available in a single package type, this is usually the case. If the pin name does not match that used on the symbol, or you need to make a change, use the following procedure.

## Editing Pin Numbers

- 1 Select Edit from the Packaging menu to display the Package Definition dialog box (shown on 5-32).
- 2 In the Package Types list box, select the package type to be edited.
- 3 Click Edit Pins to display the Pin Assignments dialog box.



- 4 Click on the pin you want to edit in the list in the lower left of the dialog box.  
The pin name and pin number appear in their respective boxes directly above the list.
- 5 Make your changes in the Pin Name and Pin No. text boxes.
- 6 Click Save Assignment.

**Note** Pin numbers can be alphanumeric.

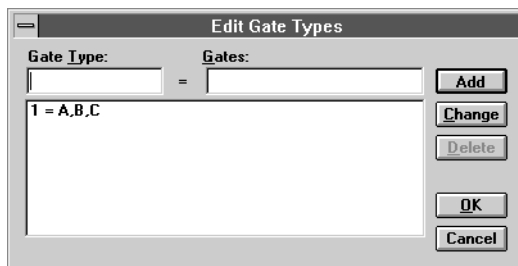
Any changes you make to a pin assignment are not effective until you select Save Assignment. If you make a change to a pin and then select another pin from the list without saving, the changes are not implemented.

- 7 When you are finished editing pins, click OK.
- 8 In the Package Definition dialog box, click OK.

## Specifying information for multi-gate components

### Defining the number of gates and their gate names

- 1 Select Edit from the Packaging menu to display the Package Definition dialog box (shown on 5-32).
- 2 Click on Edit Gate Types to display the Edit Gate Types dialog box.



Parts in which all gates are the same have only one type of gate (gate 1 by default). By defining the *names* of each gate, you also define the *number* of gates.

- 3 Select the entry in the list box labeled 1.
- 4 In the Gates text box, type the names of the gates separated by commas (e.g., A,B,C,D).
- 5 Click on Change.
- 6 Click on OK.
- 7 In the Package Definition dialog box, click OK.

## Defining pin number assignments

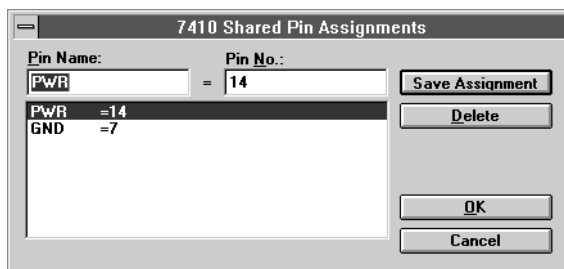
- 1 Select Edit from the Packaging menu to display the Package Definition dialog box (shown on 5-32).
- 2 Click on Edit Pins to display the Pin Assignments dialog box (shown on 5-33).
- 3 In the Pin No. text box, type a pin number for each pin for each gate defined in the previous procedure.
- 4 Click Save Assignment.
- 5 Click on OK.
- 6 In the Package Definition dialog box, click OK.

Once you have defined the names of the gates, you must define pin numbers for each pin in each gate.

## Defining shared power and ground pins

- 1 Select Edit from the Packaging menu to display the Package Definition dialog box (shown on 5-32).
- 2 Click on Edit Shared Pins to display the Shared Pin Assignments dialog box.

On the symbol for the gate defined above, if there are any shared power pins, ground pins or both, you have to define them as hidden pins.



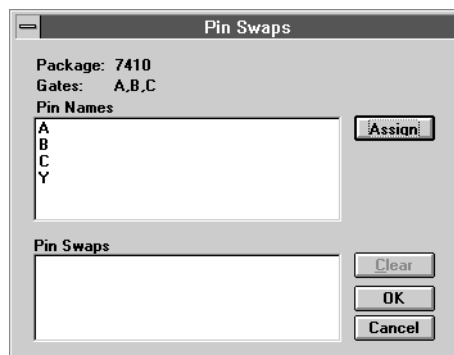
- 3 Type the name of the pin (as defined on the symbol) in the Pin Name text box.
- 4 Type the physical pin number in the Pin No. text box.
- 5 Click Add.
- 6 Click Save Assignment.
- 7 Click on OK.
- 8 In the Package Definition dialog box, click OK.

## Specifying which pins can be swapped

Pins within a gate that are logically equivalent to one another can be swapped. Pin swapping is usually done during layout to minimize the complexities of circuit routing.

## Enabling pin swapping

- 1 Select Edit from the Packaging menu to display the Package Definition dialog box (shown on [5-32](#)).
- 2 Click on Edit Pin Swaps to display the Pin Swaps dialog box.



- 3 In the Pin Names list box, select two or more pins that you want to swap.
- 4 Click Assign.  
The pin numbers appear in the Pin Swaps list box separated by commas.
- 5 Repeat steps 3 and 4 for any other pin number combinations for which you want to swap.
- 6 Click on OK.
- 7 In the Package Definition dialog box, click OK.

## Creating components with multiple gate types

Some components consist of two or more different types of gates (e.g., ECL devices). Each type of gate will have a different logical symbol with a unique name but reference the same package definition. For these types of components, you have to perform several additional steps in defining the package.

## Associating more than one symbol with a component

For each symbol:

- 1 Click the Edit Attributes icon to display the Attributes dialog box.



The 'Attributes' dialog box is shown. It has a title bar 'Attributes'. Inside, there are two text boxes: 'Name:' and 'Value:'. To the right of the 'Value:' box are two buttons: 'Save Attr' and 'Del Attr'. Below these is a section 'What to Display' with four radio buttons: 'Value only', 'Name only', 'Both name and value', and 'None' (which is selected). To the right of this section is a large empty rectangular box. Below 'What to Display' is a section 'Display Characteristics' with several controls: 'Layer:' with a dropdown menu showing 'Attribute Text', 'Orient:' with a dropdown menu showing 'horizontal', 'Hjust:' with a dropdown menu showing 'left', 'Vjust:' with a dropdown menu showing 'bottom', and 'Size:' with a text box showing '100' and a '%' symbol. To the right of these are two checkboxes: 'Changeable in schematic' (checked) and 'Keep relative rotation' (unchecked). At the bottom are two buttons: 'OK' and 'Cancel'.

- 2 Type a new attribute name COMPONENT in the Name text box.
- 3 Type a value in the Value text box that explicitly specifies the package definition name.

Example: For a 10102 package, you could have two symbols: 10102NOR and 10102ORNOR. Both symbols could have the attribute COMPONENT = 10102.

- 4 Click Save Attr.
- 5 Type a new attribute name GATETYPE in the Name text box.



- 6 Type a value in the Value text box corresponding to one of the gate type(s) specified in the package definition.

Example: For the 10102NOR symbol, GATETYPE = 1; for the 10102ORNOR symbol, GATETYPE = 2.

- 7 Click Save Attr. Click OK to exit the dialog box.

- 8 In the procedure for specifying information for multi-gate components (see *Specifying information for multi-gate components on page 5-34*), define the different gate types comprising the package.

Example: Type 1 is the NOR gate and type 2 is the ORNOR gate.

- 9 For each gate type, define the gates and the pin assignments for each gate (see *Specifying information for multi-gate components on page 5-34*).

The pin assignments define the pin numbers for each gate that correspond to each pin name.

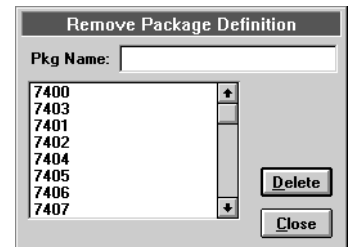
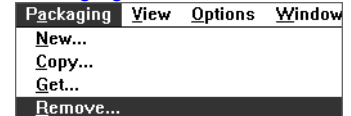
## Deleting a Package Definition

The Remove function allows you to delete one or more package definitions from the current library.

### Deleting a package definition

- 1 Select Remove from the Packaging menu to display the Remove Package Definition dialog box.
- 2 Type the name of the package to be deleted, or click to select a package from the list of packages.
- 3 Click Delete to delete the selected item.
- 4 Click Close.

Packaging Menu



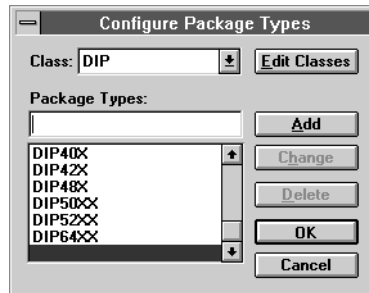
## Configuring Package Types

When you create package definitions and specify package types for a device, you can pick from a list of commonly used package type names or enter one of your own. To add to the list of commonly used package type names that are presented, use the Configure Package Types selection from the Packaging menu. Also use this selection to configure package types into the package classes that are used when you package a design.

When you package a design, you can assign priorities to use when deciding which package type to assign to devices that are available in more than one type. For example, all DIP package types (DIP8, DIP14, etc.) are assigned to the DIP class. You can indicate that you want to use DIP package types whenever possible. Or you might change the priorities to assign SMT package types if possible.

## Adding a package type

- 1 Select Configure Package Types from the Packaging menu to display the Configure Package Types dialog box.



- 2 In the Package Types text box type in the new package type name.
- 3 In the Class list box, choose one of the existing classes for the new package type. If you need to create a class:
  - a Click Edit Classes.
  - b Type in the name of the new class in the Package Class text box.
  - c Click Add.
  - d Click OK.
- 4 Click Add.
- 5 Click OK.

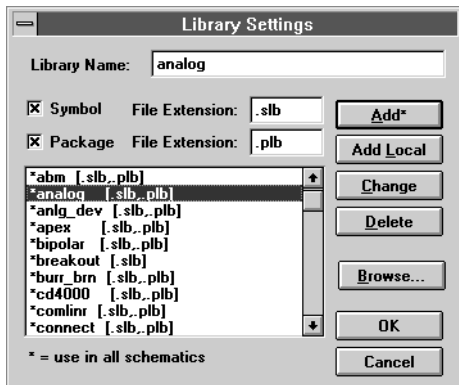


# Configuring Custom Libraries

When you create a library, whether it is a library of symbols or a library of packaging information, the symbols and packaging information are not available for use in the schematic editor until the library is configured. Configuration consists of adding the library file to the list of configured files.

## Making a symbol library accessible to Schematics

- 1 From the schematic editor, select Editor Configuration from the Options menu to display the Editor Configuration dialog box (shown on 2-12).
- 2 Click Library Settings to display the Library Settings dialog box.



- 3 Type the name of the new symbol library in the Library Name text box (without the .slb extension).
- 4 Check the Symbol check box to indicate that the new library is a symbol library. Symbol libraries are searched in the order in which they are listed in the Library Settings dialog box.

**Note** When you save changes to a library, you will be asked if you want to add the library to the list of configured libraries. Answer yes if you want to make the library available to all

- 5 To add a symbol library at a specific point in the list:
  - a Click the library name above where you want to include the one you are adding to the list.
  - b Type the name of the one you are adding in the Library Name text box. Be sure that the appropriate check boxes are checked to indicate whether you are configuring just the symbol library, or the symbol and package library.
- 6 Click Add.
- 7 Click OK.

**Note** *You may need to modify the Library Path (in the upper right corner of the Editor Configuration dialog box) to include any directory paths that contain library files you added in the previous dialog.*

When exiting the Editor Configuration dialog box, Schematics reloads all of the symbol libraries in the list, making the symbols contained in them immediately accessible in the schematic editor.

---

# Creating and Editing Hierarchical Designs

---

## 6

### Overview

This chapter explains the procedures for creating and editing a hierarchical design. Many of the procedures used for creating and editing a hierarchical design are the same as those for creating and editing a design as explained in **Chapter 3, *Creating and Editing Designs***.

This chapter has the following sections that explain the procedures that are unique to hierarchical designs:

*Creating and Editing Hierarchical Blocks on page 6-4* describes how to create and edit hierarchical blocks placed on a schematic.

*Creating and Editing Hierarchical Symbols on page 6-9* describes how to use the symbol editor to create hierarchical symbols.

*Using Interface Ports on page 6-12* describes how to specify connections to lower-level schematics.

*Setting Up Multiple Views on page 6-13* describes how to set up and use alternate representations for a hierarchical block or symbol.

*Navigating through Hierarchical Designs on page 6-15*

describes how to move between pages in a hierarchical design.

*Assigning Instance-Specific Part Values on page 6-17* describes how to assign instance-specific parts values.

*Passing Information between Levels of Hierarchy on page 6-18* describes how to define the parameters of hierarchical blocks and symbols without concern for how deeply their contents are nested.

*Example—Creating a Hierarchical Design on page 6-20*

provides the step-by-step procedures for creating the top-level schematic with the block symbol representing the lower-level schematic and creating the lower-level schematic.

# Hierarchical Design Methods

You can create a hierarchical drawing in either of two ways:

- Create a block and later assign a schematic to the block (top-down method).
- Create a schematic and turn it into a symbol to be used in a higher level design (bottom-up method).

## Top-down method

By creating one or more blocks and wiring them together, you can establish a functional block diagram. The block diagram can be used as a top-level sketch for your design.

Once you have mapped out the block circuitry, you can push into each block and start drawing a new schematic, or assign an existing schematic to the block.

You can also set the *view* that each schematic will represent (i.e., PCB, transistor, etc.).

## Bottom-up method

If you already have a schematic that you would like to use in larger designs, you can create a hierarchical symbol to represent the schematic. The hierarchical symbol can then be electrically connected in another design.

Hierarchical design is a useful way to structure large projects, especially those starting from a block diagram and those with multiple occurrences of common circuitry. Use the method of design that best fits your design needs for each circuit you create.

## Creating and Editing Hierarchical Blocks

A hierarchical block represents a collection of circuitry in the form of one or more lower-level schematics. The block appears on a schematic as a rectangle with two or more input and output ports.

You can place one or more instances of a hierarchical block on a schematic. Once you place a block, you can stretch it, reshape it and move it. You can create a schematic to be represented by the block or associate an existing schematic with the block.

Wires and buses that end at any of the edges of the block automatically connect to the block. Pins are created where these connections occur. A default pin name appears within the block; this pin name can be changed.

### Creating a hierarchical block

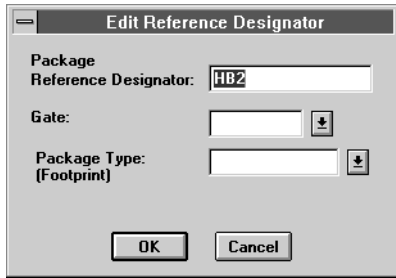
- 1 Click the Draw Block icon to change the pointer to a rectangle representing the block.
- 2 Move the block to the desired location and click to place it.
- 3 Right-click to stop placing blocks.

The block is assigned a reference designator of  $HBn$  (where  $n$  is a sequential number beginning with 1). You can change the reference designator of the block.



## Changing the reference designator of the hierarchical block

- 1 Double-click the HB $n$  reference designator to display the Edit Reference Designator dialog box.



- 2 Type the reference designator in the Package Reference Designator text box.
- 3 Click OK to close the dialog box.

The block, as placed, is a standard size, orientation and shape. You can stretch and reshape the block.

## Resizing a hierarchical block

- 1 Select the block.
- 2 Place the pointer anywhere inside the block. Shift-right-click and move the mouse to resize and reshape the block.
- 3 Release the mouse button when the block is the desired size and shape.

## Creating a schematic for a hierarchical block

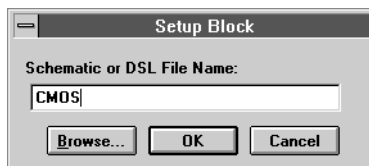
- 1 Select the block.



Shortcut: press **F2**.

- 2 Select Push from the Navigate menu.  
If the block is new, the Setup Block dialog box displays.

You can also double-click the block to achieve the same results as steps 1 and 2.



**3** Type the new schematic name.

**4** Click OK.

**Note** *Interface input and output ports are created automatically only the first time you push into the block. Thereafter, you must manually add any additional interface input and output ports.*

A new schematic displays and contains interface input and output ports corresponding to the pins connected to the block. The input ports correspond to the pins connected to the left side of the block. The output ports correspond to the pins connected to the right side of the block.



You can move the interface port symbols in the same way that you move other symbols.

You can also associate an existing schematic with a hierarchical block. See *Associating an Existing Schematic on page 6-8*.

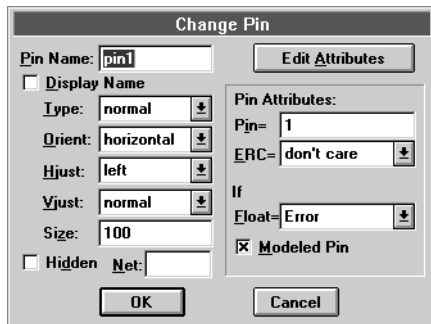
### Editing a pin name on a hierarchical block

**1** Select the pin on the hierarchical block.

**2** Click the Edit Attributes icon to display the Change Pin dialog box.







- 3 Type the desired pin name in the Pin Name text box.
- 4 Click OK.

Double-click the pin to achieve the same results as steps 1 and 2.

### Deleting a pin on a hierarchical block

- 1 Select the pin.
- 2 Press **Delete**.

## Associating an Existing Schematic

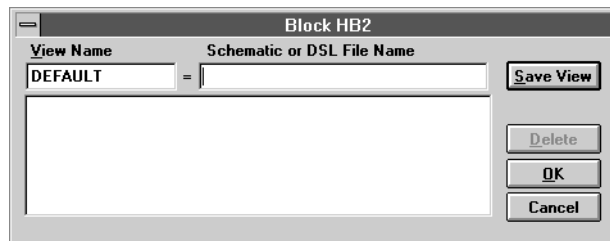
Instead of pushing into the block to create a schematic (as described on 6-5), you can associate an existing schematic with a hierarchical block.

### Associating an existing schematic with a hierarchical block

- 1 Select the hierarchical block.
- 2 Select Views from the Edit menu to display the Block View dialog box.

#### Edit Menu

Edit	Draw	Navigate	View
Undo			Ctrl+U
Cut			Ctrl+X
Copy			Ctrl+C
Paste			Ctrl+V
Copy to Clipboard			
Delete			DEL
Attributes...			
Label...			Ctrl+E
Model...			
Stimulus			
Symbol			
Views...			



- 3 In the Schematic Name text box, type the name of the schematic to be associated with this block.
- 4 Click OK.

When you create a schematic by *pushing* into the block, Schematics automatically places one interface port on the new schematic for each wire or bus connected to the block.

When you *associate* an existing schematic with a block, you have to create and connect the necessary interface ports.

# Creating and Editing Hierarchical Symbols

Schematics uses two basic types of symbols: primitive and hierarchical.

## Primitive symbols

- Are the lowest level symbols explicitly containing all of the information required by the netlister.
- Can be modified by editing their graphics, pins, and/or attribute lists in the symbol editor.

Most of the symbols currently provided in the Schematics symbol libraries are primitive. Note that a symbol, for example a flip-flop, may be primitive for a PCB netlister, but hierarchical for PSpice or PLogic.

## Hierarchical symbols

- May have the same appearance as primitive symbols in Schematics.
- Contain one or more levels of schematics inside them while primitive symbols do not.
- May be modified by pushing into them from within the schematic editor or symbol editor and editing the associated schematics.

There is no built-in limit to the number of levels of nesting allowed in a symbol. Schematics allows the nesting of hierarchical symbols or blocks within other hierarchical symbols or blocks.

## Creating a Hierarchical Symbol

The Symbolize function automatically creates a symbol to represent a schematic. The symbol editor is then used to modify any portion of the resulting symbol (graphics, pins and attributes).

When preparing a schematic for symbolization, follow these guidelines:

- Place input and output interface ports (IF\_IN, IF\_OUT) at the inputs and outputs of the schematic. Interface ports are mapped to I/O pins placed on the left (input) and right (output) of the new symbol.

- Place global ports (GLOBAL or BUBBLE) to bring out global nets/connections as hidden pins. Global ports are mapped to hidden pins placed on the top and bottom of the new symbol. An IPIN<sub>(xxx)</sub> attribute, whose value is the name of the net to which it is to be connected, is created for each hidden pin. Hidden pins are especially useful for global power and ground on digital parts (\$G\_DPWR, \$G\_DGND).

File Menu

File	Edit	Draw	Navigate	View	Q
New					
Open...					
Close					
Import...					
Save					Ctrl+S
Save As...					
Print...					
Printer Select...					
Edit Library					
Symbolize...					

**Note** You shouldn't modify the symbol libraries that were shipped with your system. Create a new symbol library for the custom symbols you create.

### Symbolizing a schematic

- 1 Open the schematic.
- 2 Select Symbolize from the File menu to display the Save As dialog box.
- 3 Type the name of the symbol.
- 4 Click OK.  
  
A file selection dialog box prompts for a symbol library in which to save the symbol.
- 5 Select a library.
- 6 Click OK.

When you symbolize a schematic, the resulting symbol is hierarchical (i.e., it will have a schematic associated with it such that you can push into the symbol and view that schematic).

Once you have symbolized your schematic, you need to make the symbols in your new symbol library accessible to Schematics.

# Converting Hierarchical Blocks to Symbols

When you finish editing a hierarchical block, you have the option of turning the block into a symbol. By making the block a symbol, you make it accessible for use in other schematics.

## Converting a block to a symbol

- 1 Select the block.
- 2 Select Convert Block from the Edit menu to display the Save As dialog box.
- 3 Type a name for the symbol.
- 4 Click OK to display the Open dialog box.
- 5 Select a library.
- 6 Click OK.

Schematics creates a rectangular symbol to represent the block. One pin is placed on the new symbol for each wire or bus that was connected to the block. The width of the block shrinks one unit on each side to accommodate symbol pins without requiring rewiring.

Edit Menu

Edit	Draw	Navigate	View
Undo			Ctrl+U
Cut			Ctrl+X
Copy			Ctrl+C
Paste			Ctrl+V
Copy to Clipboard			
Delete			DEL
Attributes...			
Label...			Ctrl+E
Model...			
Stimulus			
Symbol			
Views...			
Convert Block...			

**Note** Converting a block to a symbol is a one-way process. Once you convert a block into a symbol, you cannot change that symbol back into a block.

## Using Interface Ports

When you use a block or symbol to represent an underlying schematic, connections to the underlying schematic are made via the pins on the block or symbol. The pins on the block or symbol must correspond to interface ports placed on the underlying schematic, i.e., for each pin there must be a corresponding interface port with the same name as the pin.

If a bus is connected to the block or symbol, the pin name must indicate the number of signals, e.g., CLK[0:3]. The interface port would have the same name, e.g., CLK[0:3].

If you make changes to the pins on a block or symbol, you must make the corresponding changes on any underlying schematics it represents.

If you:

Add a pin

Delete a pin

Change the name  
of a pin

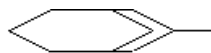
You Must:

Add an interface port with the  
same name as the pin

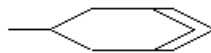
Delete the corresponding  
interface port

Change the label of the  
corresponding interface port

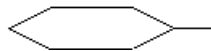
There are three interface port symbols available in “port.slb”:



IF\_IN



IF\_OUT



INTERFACE

You can use the symbol editor to create custom interface ports.

# Setting Up Multiple Views

A view is an underlying representation of a hierarchical block or symbol. A block can have more than one underlying representation by having multiple views.

For example, you can define a part that has a transistor-level schematic as one view and a behavioral model schematic as another view.

**Note** *There are no restrictions as to how many views a part can have, as well as on what the views are.*

Hierarchical symbols have one or more views. Every hierarchical block or symbol always has a default view, which is initialized as the first schematic assigned to it. You can change the default view. You can create and associate additional views at any time. You can modify, delete or rename views. Each view is associated with a schematic, and multiple views can point to the same schematic.

If you are also using PLSyn for programmable logic synthesis, you may also configure a view to be the name of a DSL file. Refer to the *MicroSim PLSyn User's Guide* for more information.

## Translators

To take different views of a design, configure a *translator* to look for separate view attributes. A translator produces an alternate representation for a schematic. For example, the Schematics netlister is a translator that operates on a schematic to produce a PSpice/PLogic netlist. A translator typically looks at information carried by the symbols on a schematic and may or may not also use the implicit connectivity.

### Options Menu



### Setting up an associated view for the Translator

- 1 Select Translators from the Options menu to display the Translators dialog box.



- 2 Select a Translator from the list or type a name in the Translator text box.
- 3 Type the name of the view in the View text box.
- 4 Click Apply.
- 5 Click OK.





# Navigating through Hierarchical Designs

The Navigate menu has functions that allow you to move between pages, create new pages, delete pages and copy pages.

You can move within a hierarchical design using functions from the Navigate menu. You can push into a block from the schematic, move up and down in hierarchical levels and identify the hierarchical path of a selected symbol.

## Moving down in a hierarchy

- 1 Select the hierarchical block or symbol.
- 2 Select Push from the Navigate menu.
  - a If the selected item is represented by only one lower-level schematic, the schematic is displayed for editing.
  - b If the selected item represents more than one schematic (i.e., has multiple views), a dialog box displays from which you can select the view to be edited.

Shortcut: press **[F2]**.

Double-clicking the hierarchical block or symbol gives the same results as steps 1 and 2.

## Moving up in a hierarchy

Select Pop from the Navigate menu.

If you have made any changes to the current level in the hierarchy, you are prompted to save the modifications or to move up to the next higher level without saving changes.

Shortcut: press **[F3]**.

## Moving to the top in a hierarchy

Select Top from the Navigate menu.

The top-level schematic displays in the active window.

Navigate Menu

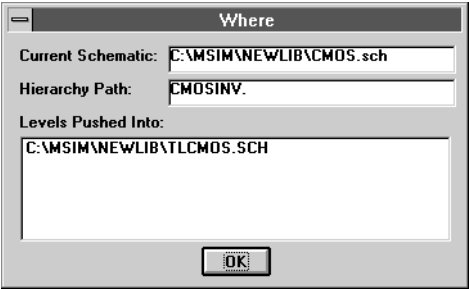
Navigate	View	Options	And
Previous Page			
Next Page			
Select Page...			
Create Page...			
Delete Page...			
Copy Page...			
Edit Page Info...			
Edit Schematic Instance			
✓ Edit Schematic Definition			
Push			F2
Pop			F3
Top			

Finding where current schematic fits in a hierarchy

Navigate Menu

Navigate	View	Options	Ana
Previous Page			
Next Page			
Select Page...			
Create Page...			
Delete Page...			
Copy Page...			
Edit Page Info...			
Edit Schematic Instance			
✓ Edit Schematic Definition			
Push		F2	
Pop		F3	
Top			
Where...			

- 1 Select Where from the Navigate menu to display the Where dialog box.



The dialog box shows where the current schematic fits in the hierarchy of the current design.

- 2 Click OK.

# Assigning Instance-Specific Part Values

The Edit Schematic Instance function allows you to view and edit the instance specific attributes associated with the instance of the block or hierarchical symbol into which you are pushed. You can only add, change or delete attributes when this function is activated. Any changes only apply to this instance of the hierarchical block or symbol into which you are pushed.

## Editing an instance of the schematic

- 1 Select Push from the Navigate menu to push into the hierarchical block or symbol.
- 2 Select Edit Schematic Instance from the Navigate menu.  
A check mark appears next to the menu item to show that it has been selected.

**Note** Any changes you make affect only this instance of the schematic. To make changes to the schematic itself, use the Edit Schematic Definition function.

Shortcut: press [F2].

Navigate Menu

Navigate	View	Options	Anal
Previous Page			
Next Page			
Select Page...			
Create Page...			
Delete Page...			
Copy Page...			
Edit Page Info...			
Edit Schematic Instance			

## Editing the schematic definition

- 1 Select Push from the Navigate menu to push into the hierarchical block or symbol.
- 2 Select Edit Schematic Definition from the Navigate menu.  
A check mark appears next to the menu item to show that it has been selected.  
You can now edit the schematic into which you are pushed. Any changes that you make affect all instances of hierarchical blocks and symbols that reference this schematic.

**Note** Edit Schematic Instance and Edit Schematic Definition are mutually exclusive functions.

Shortcut: press [F2].

Navigate Menu

Navigate	View	Options	Anal
Previous Page			
Next Page			
Select Page...			
Create Page...			
Delete Page...			
Copy Page...			
Edit Page Info...			
Edit Schematic Instance			
✓ Edit Schematic Definition			

## Passing Information between Levels of Hierarchy

With Schematics, you can create a lower-level schematic such that different instances of it will have different component values. For instance, a lower-level schematic contains a certain resistor. The hierarchical block or symbol representing the lower-level schematic defines the value of the resistor. The following procedure shows how you can place one instance of a block and define the resistor value to be 10K and another instance and have the resistor value be 20k.

- 1 In the lower-level schematic, double-click on the resistor value to display the Set Attribute Value dialog box (shown on **3-30**).
- 2 In the Value text box type `{@RESISTORVALUE}`.
- 3 Click OK.
- 4 Save the lower-level schematic.
- 5 Place a block representing the lower-level schematic on the top-level (or higher-level) schematic (see *Creating and Editing Hierarchical Blocks on page 6-4*).
- 6 Select the block.
- 7 Select Attributes from the Edit menu to display the Attribute Editing dialog box (shown on **3-12**).
- 8 Add an attribute called RESISTORVALUE with a value of 10k.
- 9 Click OK.
- 10 Place another block representing that same lower-level schematic on the top-level schematic.

Shortcut: Press **Ctrl**+S .

When you netlist the top-level schematic, the two instances of the lower-level schematic will have different resistor values. This is due to the way that attributes are evaluated in Schematics.

- Schematics first searches for an attribute at the current level of the hierarchy. If the attribute is not found at that level, Schematics then searches the parent level. It continues up the hierarchy until it either finds a definition or until it reaches the top of the hierarchy.
- When Schematics finds an attribute, it evaluates the attribute at the level where it is found. If the attribute value refers to other attributes, those other attributes must exist at the current level or higher in the hierarchy.

For example, hierarchical symbol A defines two attributes:  $X=@Y$  and  $Y=10$ . Symbol A contains an instance of a symbol B; B contains an expression referring to the attribute X ( $\{ @X \}$ ) and defines the value of attribute Y to be 20 ( $Y=20$ ).

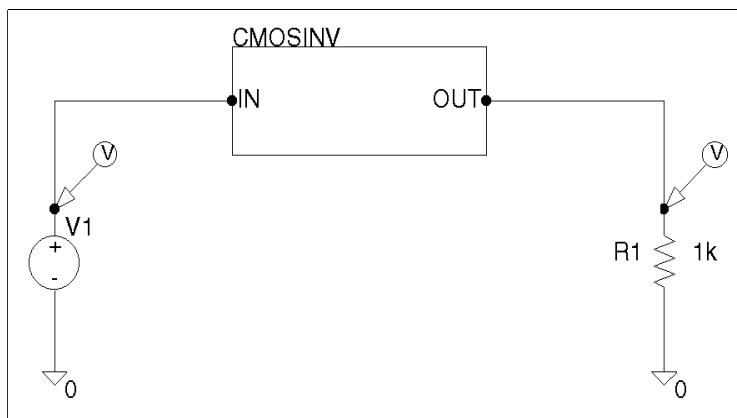
The evaluation of the expression  $\{ @X \}$  is:

- X is searched for on the current level.
- There is no X attribute at this level, so the parent environment (symbol A) is searched.
- An attribute named x is found at this level. This attribute is evaluated in the environment supplied by A.
- The first stage of this evaluation delivers the result @Y. This is then processed to yield the result 10.
- The final result is to make the result of the expression in B be  $\{ 10 \}$  ( $\{ @X \} = \{ 10 \}$ ).
- Note that the definition for Y in the environment supplied by symbol B is not used when evaluating X in A's environment.

## Example—Creating a Hierarchical Design

This example shows you how to create schematics from the top level down. The design consists of a simple schematic with a block representing a CMOS inverter and a lower-level schematic for the inverter.

Follow this example to create the top-level circuit shown in Figure 6-1 and the inverter schematic shown in Figure 6-2.



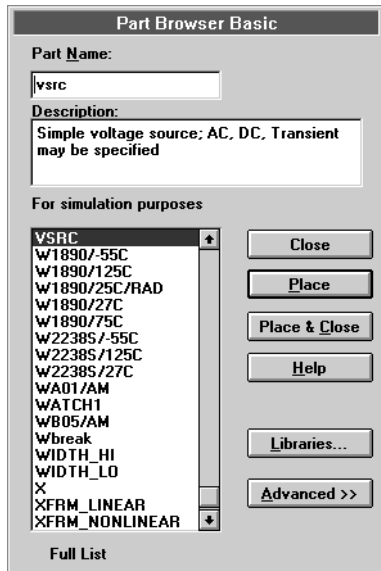
**Figure 6-1** *Top-level Schematic Drawing for CMOS Inverter*

## Drawing the Top-Level Schematic

To create the top-level schematic, start by placing a VSRC power supply connected as an input to a block representing a CMOS inverter. Draw the block, place a resistor and two ground symbols, and connect the components.

## Placing the voltage source

- 1 Click the Select Part icon to display the Part Browser dialog box.



**Note** One of two Part Browser dialog boxes may be displayed: the Part Browser Advanced and the Part Browser Basic. The advanced browser contains many features that you don't need to use for this example. If the Part Browser Advanced dialog box displays, click <<Basic to display the Part Browser Basic dialog box.

- 2 Type VSRC in the Part field.
- 3 Click Place & Close.
- 4 Move the part symbol to the desired location and click to place the symbol.
- 5 Right-click to stop placing parts.

## Creating the block representing the CMOS inverter

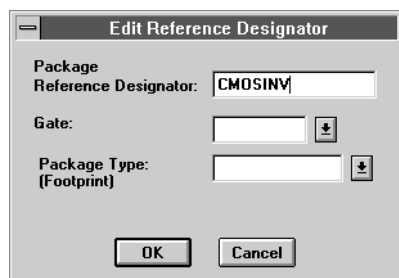
- 1 Click the Draw Block icon to change the pointer to a rectangle.

The rectangle represents the block to be drawn.



- 2 Press **Ctrl+R** to rotate the rectangle.
- 3 Move the pointer to the desired location and click to place the block symbol.
- 4 Right-click to stop placing blocks.

- 5 Double-click the HB1 reference designator to display the Edit Reference Designator dialog box.



- 6 Type CMOSINV in the text box. This changes the value of the REFDES attribute from HB1 to CMOSINV.
- 7 Click OK.

### Drawing the output load resistor

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on 6-21).
- 2 Type R in the Part field.
- 3 Click Place & Close.
- 4 Press **Ctrl**+R to rotate the resistor symbol.
- 5 Move the resistor symbol to the desired location. Click to place the symbol.
- 6 Right-click to stop placing parts.



### Placing the analog ground symbols

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on 6-21).
- 2 Type AGND in the Part field.
- 3 Click Place & Close.

You can place the two grounds so that they connect to the power source and the load resistor. This negates having to draw wires between the symbols.





- 4 Move the ground symbol to the desired locations. Click to place each symbol.
- 5 Right-click to stop placing parts.

## Wiring the symbols

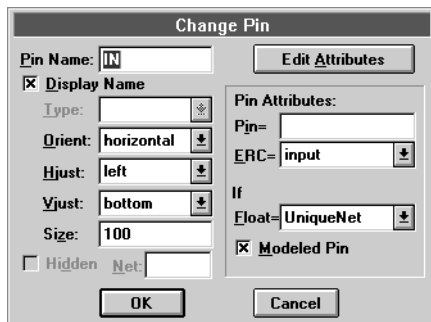
Now that you have placed all of the symbols, wire the symbols to look like the schematic shown in Figure 6-1.

- 1 Click the Draw Wire icon to change the pointer to a pencil.
- 2 Click the top of V1. Click at the location of the wire vertex (where it turns from the vertical to the horizontal). Click the left side of the CMOS block. The wire is complete when it shows connection on both ends.
- 3 Repeat step 2 to connect a wire from the right side of the CMOS block to the top of the load resistor.
- 4 Right-click to stop wire drawing.



## Changing the names of the pins on the block

- 1 Double-click the pin labeled P1 to display the Change Pin dialog box.



- 2 Type IN in the Pin Name text box.
- 3 Click OK.
- 4 Double-click the pin labeled P2 to display the Change Pin dialog box.
- 5 Type OUT in the Pin Name text box.

- 6 Click OK.

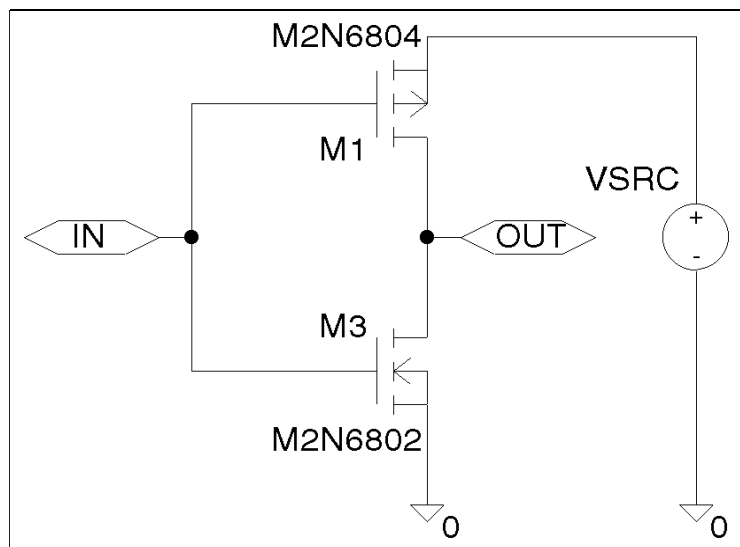
### Saving your work as a top-level schematic



- 1 Click the Save File icon.
- 2 Type `t1cmos` as the name of the file (the `.sch` file extension is assigned by default).
- 3 Click OK.

## Drawing the Lower-Level Schematic

The top-level design is complete. Now you can create the inner schematic of the CMOS inverter. To do so, select the block and use the Push selection from the Navigate menu to push to a lower level. Because you haven't defined the lower-level yet, you are presented with a Setup Block dialog box to name the new schematic.



**Figure 6-2** Schematic of CMOS Inverter

## Selecting the block and naming the new schematic

- 1 Click the CMOSINV block to select it.
- 2 Select Push from the Navigate menu.

Shortcut: press **[F2]**.

Since the block is new, the Setup Block dialog box displays.



Double-clicking the block gives the same results as steps 1 and 2.

- 3 Type in the new schematic name, `cmos`.
- 4 Click OK.
- 5 Move the interface port symbols in the same way you move other symbols:
  - a Click to select it.
  - b Click and drag it to the desired location.
  - c Release to complete the move.

You are presented with a new schematic containing one interface input port and one interface output port. (They represent the block you drew on the top-level schematic.) If you had connected more pins to the block, more interface ports would appear after pushing into the lower-level schematic.

Now draw the schematic for the CMOSINV block as shown in Figure 6-2.

## Drawing the two MOSFET devices

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on **6-21**).
- 2 Type M2N6804 in the Part text box.
- 3 Click Place & Close.
- 4 Press **[Ctrl]+R** , **[Ctrl]+R** and **[Ctrl]+F** to rotate the symbol twice and flip it once so that the source and bulk pins appear at the top. (To check M1, see Figure 6-2 on page 6-24.)
- 5 Move the part symbol to the desired location of M1 and click to place the part.
- 6 Right-click to stop placing parts.





- 7** Click the Select Part icon to display the Part Browser dialog box (shown on **6-21**).
- 8** Type M2N6802 in the Part text box.
- 9** Click Place & Close.
- 10** Move the part symbol to the desired location of M2 and click to place the part.
- 11** Right-click to stop placing parts.

If you want to clean-up your schematic, click each of the MOSFET device names (M2N6804 and M2N6802) and move them slightly so that the label does not overlap on one of the pins.

### Drawing the voltage source and specifying the DC voltage attribute



- 1** Click the Select Part icon to display the Part Browser dialog box (shown on **6-21**).
- 2** Type VSRC in the Part text box.
- 3** Click Place & Close.
- 4** Move the symbol to the desired location and click to place it.
- 5** Right-click to stop placing parts.

## Drawing the two analog ground symbols

- 1 Click the Select Part icon to display the Part Browser dialog box (shown on **6-21**).
- 2 Type AGND in the Part text box.
- 3 Click OK.
- 4 Move the symbol to the desired location and click to place it. Repeat for the second ground symbol.
- 5 Right-click to stop placing parts.



## Drawing the wires

Click the Draw Wire icon and draw wires to connect parts and symbols as shown in Figure 6-2.



## Saving the file

Click the Save File icon to save the schematic. You are not prompted for a file name because the schematic was named when you pushed into it from the top-level schematic.



---

# Preparing Your Design for Simulation

---

# 7

## Overview

This chapter provides guidelines for preparing your schematic for simulation and references further information contained in your PSpice user's guide.

A design that is targeted for simulation will have:

- parts for which there are simulation models available and configured (Refer to *Linking a Symbol to a Model or Subcircuit Definition* in your PSpice user's guide.)
- sources of stimulus to the circuit (Refer to *Minimum circuit design requirements* tables in the list of tables in your PSpice user's guide.)

This chapter has the following sections:

*Creating Designs for Simulation and Board Layout on page 7-3*

*Specifying Simulation Model Libraries on page 7-5*

*Creating Symbols for Existing Simulation Models on page 7-6*

*Editing Simulation Models from Schematics on page 7-8*

*Adding and Defining Stimulus on page 7-9*

*Starting the Simulator on page 7-10*

*Viewing Results on page 7-11*

# Creating Designs for Simulation and Board Layout

When creating designs for both simulation and printed circuit board layout, some of the parts you use will be for simulation only (e.g., simulation stimulus parts like voltage sources), and some of the parts you use will have simulation models that only model some of the pins of the real device.

Those parts which are to be used for simulation, but not for board layout, will have a `SIMULATIONONLY` attribute. To see an example of this, place and double-click on a VDC voltage source to bring up the attribute editing dialog box.

You can add this (or any) attribute(s) to your own custom symbols.

## Specifying Part Attributes

- 1 In the symbol editor, select Attributes from the Part menu to display the Attribute Editing dialog box.
- 2 Double-click in the Name text box and type `SIMULATIONONLY`.
- 3 Click Save Attr.
- 4 Click OK.

For more information on defining attributes on part symbols, Refer to *Defining Part Symbol Attributes* in the *Creating Symbols* chapter of your PSpice user's guide.



## Handling Unmodeled Pins

For those parts that have some pins that are not modeled, the pins when placed on the schematic appear broken. To see an example of this, place an instance of the PM-741 part from the “opamp.slb” symbol library. The OS1 and OS2 pins are not modeled, so only the +, -, V+, V-, and OUT pins are netlisted for simulation.

The OS1 and OS2 pins are displayed as “broken” so that you know not to connect any wires to these pins. If you do connect wires to these broken pins and nowhere else, you will get a floating node message from the simulator.

Double-click the part to display the Attribute Editing dialog box. Note that the TEMPLATE attribute for the part only calls out the +, -, V+, V-, and OUT pins. The OS1 and OS2 pins are not called out in the TEMPLATE because those two pins are not modeled in the simulation model for the PM-741 part. You can view the simulation model definition for the PM-741 part from Schematics.

### Viewing a part’s simulation model

- 1 Click the part to select it.
- 2 Select Model from the Edit menu.
- 3 Click Edit Instance Model (Text) to display the Schematics Model Editor and view an instance of the simulation model definition.
- 4 Click Cancel to exit the Model Editor without saving.

# Specifying Simulation Model Libraries

Refer to the *Creating Models* chapter of your PSpice user's guide for information about creating and configuring simulation model libraries. Each part that you intend to simulate must have a simulation model defined.

## Checking whether a part has a simulation model defined

Double-click the part on the schematic to display the Attribute Editing dialog box. If a simulation model is available for a part, the part will have:

- 1 a TEMPLATE attribute specifying the PSpice simulation netlisting syntax for the part
- 2 a MODEL attribute specifying the name of the model or subcircuit

The TEMPLATE contains “@MODEL” somewhere along the line.

The simulation model specified by the MODEL attribute must be contained in a model library that is configured.

## Checking whether a simulation model library is configured

Select Library and Include Files from the Analysis menu to bring up the Library and Include Files dialog box.

The set of simulation model libraries configured are listed in the Library Files area.

For information on configuring simulation model libraries, refer to *Configuring the Library* section in the *Creating Models* chapter in your PSpice user's guide.

# Creating Symbols for Existing Simulation Models

If you have a simulation model for a part and you want to create a symbol to represent that part, perform the following procedure.

- 1 Note the PSpice model type. If it is a .subckt, then the type is X.
- 2 If it is a subcircuit definition, make a note of the pin order on the .subckt line. If it is a model, refer to the *MicroSim PSpice A/D Reference Manual* for pin order.
- 3 Make a note of the EXACT model name. Case is not important.
- 4 Click the Edit Symbol icon on the toolbar to enter the symbol editor.



If you want to add symbols to an existing library, select Open from the File menu and browse for the desired library.

- 5 In the symbol editor either select New from the Part menu to draw new graphics or select Copy from the Part menu to copy a symbol of the desired type from the breakout library “breakout.slb”.
- 6 If you are placing new pins, then take note of the names you give them, or of the default names (PIN1, PIN2, etc.) since you will need these when constructing the PSpice netlisting template.
- 7 Once you have the symbol graphics and pins placed and defined, click the Edit Attributes icon to edit the PART, MODEL, and TEMPLATE attributes.
- 8 PART and MODEL should have their values set to the model or subcircuit name. Except for case, it must match exactly.
- 9 Check or edit the TEMPLATE attribute. If you are working with a subcircuit definition, then it must start with X. Otherwise, it will begin with the PSpice device prefix



For further information on editing symbol attributes, see *Editing Symbol Attributes* on page 5-24.

Refer to the *Defining Part Symbol Attributes* section in the *Creating Symbols* chapter of your PSpice user's guide.

specific to your device type (e.g., Q for bipolars, M for MOSFETs, etc.)

The PSpice device type letter is followed by “^@REFDES” and then a space.

A list of the pin names follows. Each pin name is preceded by a “%” and followed by a space. The pin names called out in the TEMPLATE must match those on the symbol exactly. For example, if you have pins named PIN1 and PIN2 on the symbol, the the TEMPLATE will call out %PIN1 %PIN2.

Pins in the TEMPLATE must appear in the same order as they do in the model or subcircuit definition. The link between the pins in the model or subcircuit definition and the symbol is completely order dependent. They are not required to have the same name.

- 10 Save the library. If it is a new library you will be prompted for a name and given the opportunity to configure it for use in Schematics. It is recommended that you do configure it at this point.
- 11 Select Close from the File menu to return to the schematic editor.

See *Saving your Changes on page 4-6*.

# Editing Simulation Models from Schematics

You can define and edit simulation models directly from Schematics.

Models can be defined using the Parts utility or the text editor (sometimes called the Model Editor).

The Parts utility is useful for characterizing specific models from data sheet curves. The text editor is useful if model parameters are already defined (such as for models from a vendor) or if the model is not supported by the Parts utility.

Refer to the *Using the Parts Utility* and *Using the Model Editor (Text Editor)* sections of the *Creating Models* chapter in your PSpice user's guide.

# Adding and Defining Stimulus

The Stimulus Editor is a utility that allows you to set up and verify the input waveforms for a transient analysis. You can create and edit voltage sources, current sources, and digital stimuli for your circuit. Menu prompts guide you to provide the necessary parameters, such as the rise time, fall time, and period of an analog repeating pulse, or the complex timing relations with repeating segments of a digital stimulus. Graphical feedback allows you to verify the waveform.

## Placing Stimulus Sources

Stimulus sources come from the “source.slb” symbol library and are one of

- VSTIM - voltage stimulus source for transient analysis
- ISTIM - current stimulus source for transient analysis
- DIGSTIM - digital stimulus source

You can place any of these sources by typing the name of the source in the Select Part list box on the toolbar. The AC and DC sources are VAC and VDC, and can be placed similarly.

- 1 Double-click in the Select Part drop-down list box and type the name of the source.
- 2 Press  and click to place the source.
- 3 Right-click to stop placing sources.

## Using the Stimulus Editor

For information on using the Stimulus Editor, refer to *Stimulus Editor Utility* in the *Transient Analysis* chapter in your PSpice user's guide.

## Setting Up Analyses

Refer to your PSpice user's guide for information about setting up and running the many different analysis types support by PSpice A/D.

## Starting the Simulator

You can start the simulator directly from Schematics by clicking on the Simulate icon on the toolbar, or selecting Simulate from the Analysis menu.

For more information, refer to the *Starting Simulation* section of the *Setting Up Analyses and Starting Simulation* chapter in your PSpice user's guide.



# Viewing Results

You can use Probe to view and perform waveform analysis of the simulation results. For more information, refer to the *Waveform Analysis* chapter of your PSpice user's guide.

## Viewing Results as You Simulate

You can configure Probe to run automatically when the simulation has finished, or to monitor waveforms as the simulation progresses.

These procedures are outlined in the *What You Need to Know to Run Probe* section of the *Waveform Analysis* chapter in your PSpice user's guide.

## Using Markers

You can place markers on your schematic to indicate the points for which you want to see waveforms displayed in Probe.

For more information on markers, refer to the *Schematic Markers* section of the *Waveform Analysis* chapter in your PSpice user's guide.



## Configuring Probe Display of Simulation Results

To configure what Probe displays when it is started, select Probe Setup from the Analysis menu. You are given the following choices:

- **Restore Last Probe Session**—This restores the display characteristics from the last session of Probe.
- **Show All Markers**—This displays the waveforms at the points on the schematic which have been marked by markers.
- **Show Selected Markers**—This displays the waveforms only for those points on the schematic where the markers have been selected.
- **None**—This displays a blank Probe window, ready for you to select which traces you want to add.

---

# Targeting Your Design for Programmable Logic

---

## 8

### Overview

Schematics running with PLSyn provides an integrated environment wherein you can design by combining graphical and language based design definitions for Programmable Logic Devices (PLDs).

All or any part of a schematic can be programmed into PLD parts. You can define programmable logic using logic symbols, such as gates and flip-flops, or DSL (Design Synthesis Language) blocks or both. Programmable logic symbols and DSL blocks can be placed anywhere on your schematic—on any page, and at any level of hierarchy.

This chapter contains the following sections:

*Targeting Parts for Programmable Logic on page 8-3.*

*Creating and Editing DSL Blocks on page 8-4.*

*Simulating a Programmable Logic Design from Schematics on page 8-5.*

*Using PLSyn on page 8-6.*

*Updating the Schematic with the PLD(s) on page 8-7.*

# Targeting Parts for Programmable Logic

Programmable logic is defined on a schematic by placing generic logic symbols, such as NAND4 or JKFF, or specific 74xx series symbols, such as 74LS04 or 74HC107 and setting the value of their IMPL attributes to PLSyn. The “dig\_prim.slb” symbol library contains over 100 ready-to-use programmable logic symbols. Refer to the *Programmable Logic Symbols* section of the *MicroSim PLSyn User’s Guide* for more detail.

IMPL is short for “implementation”.

## Creating and Editing DSL Blocks

Design Synthesis Language (DSL) blocks are hierarchical blocks that have a language-based definition instead of a symbolic definition. When you place a block, instead of associating it with another schematic, you associate a “.dsl” file containing the procedural definition for the block.

Refer to the *DSL Blocks* and *Creating a DSL Block* sections of *MicroSim PLSyn User's Guide*.

# Simulating a Programmable Logic Design from Schematics

Once you have entered a design which includes programmable logic, you may simulate it at any time, both before and after you have chosen a physical implementation. You can specify the simulation parameters by selecting Setup from the Analysis menu.

For additional details, refer to the *Setting up and Starting Simulations* section of *MicroSim PLSyn User's Guide* .

## Using PLSyn

After you have described your design in Schematics, you use PLSyn to create the physical implementation of your programmable logic.

To start PLSyn from Schematics, select Run PLSyn from the Tools menu.

**Note** *This command only appears if you have PLSyn installed.*

Refer to the *Physical Implementation* chapter in *MicroSim PLSyn User's Guide*.

# Updating the Schematic with the PLD(s)

After you have created a physical implementation from the programmable logic symbols and DSL blocks on the schematic, the PLD(s) can be back annotated from PLSyn to the schematic. Then you can generate a netlist for PCB layout.

Refer to the *Updating the Schematic* chapter in *MicroSim PLSyn User's Guide* .



---

# Preparing Your Design for Board Layout

---

## 9

### Overview

This chapter describes how to prepare your design for use with a board layout program and has the following sections:

*Connectors on page 9-3* describes placing connectors to provide the interface between the PCB and the rest of the system. This section also describes how to create connector symbols.

*Packaging the Parts in Your Design on page 9-6* describes the process of collecting individual gates into physical packages and reassigning reference designators and gate names to reflect how they are packaged.

*Generating a Bill of Materials Report on page 9-12* describes how to generate a report listing the quantities of each component type used in the design along with corresponding reference designators.

*Swapping Pins on page 9-17* describes how to swap pins on a given gate.

*Interfacing to MicroSim PCBoards on page 9-18* describes the procedures for using Schematics with MicroSim PCBoards.

*Interfacing to Other Board Layout Products on page 9-23*  
describes the procedures for using Schematics with the board layout products from other vendors.

# Connectors

Connectors provide the interface between a PCB and the rest of a system.

The distinction between connectors and ports on a schematic is important and is shown in Table 9-1. Off-page ports are not physical connectors so you cannot use an off-page port as connector or vice versa. You may use them together if you want to have both connectivity and a physical part by attaching an off-page port to the pin of the connector.

During simulation, connectors are largely ignored except that you can attach a marker to a connector pin to view waveforms in Probe. You can also connect stimuli to connector pins to simulate the external interface to the circuit.

**Table 9-1** *Distinctions between Connectors and Ports*

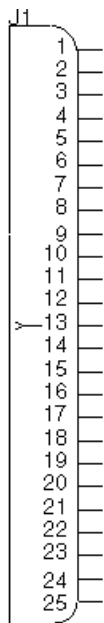
Connectors	Ports
define physical connection points on the PCB.	define logical connection point on the schematic
are not included in layout netlist	are included in layout netlist
cannot be used to create connectivity on the schematic	are used to create connectivity on the schematic
are not logical devices	are not physical connectors

## Placing Connectors

Connectors are added by placing connector symbols on the schematic. You can use the connector symbols shipped with Schematics (found in the “connect.slb” symbol library) or you can create your own using the symbol editor. (See **Chapter 5, Creating and Editing Symbols.**)

There are two styles of connector symbols:

- those representing the entire connector
- those representing a single pin of a connector



**Figure 9-1** *Entire Connector Symbol*

### Using connector symbols that represent the entire connector

These symbols will have as many pins as the physical connector they represent. You can wire signals directly to the pins or connect labeled off-page (or global) ports to each pin. The label indicates the signal name that will be connected to the pin. Any off-page ports in the design with that same signal name will be connected to that connector pin.

Two connector symbols that represent an entire connector are DB25F-B and EDGE40M-B.

### Using connector symbols that represent a single pin of a connector

In cases where a connector has a large number of pins, you may want to use a symbol that represents a single pin of the connector so you can attach connector pins to nets spread over multiple pages.

When an instance of such a connector symbol is placed on the schematic, it is assigned an arbitrary reference designator and gate. The reference designator indicates which physical connector instance the connector pin is part of (e.g. P1), and the gate indicates which physical pin (e.g., 1, 2, etc.). Thus, the entire connector is considered a multi-gate package with each gate having a single pin. All connector pin instances with the same reference designator are a part of the same physical connector.

Normally, you would assign the reference designator and gate manually. (Or, you could automatically package the pins. However, this will result in an arbitrary grouping of signals which is not usually desired.) To change the reference designator, double-click on the reference designator on the schematic. To change the pin number (gate), double-click on the pin number.

Two connector symbols that represent single pins of a connector are DB25 and EDGE40.



**Figure 9-2** *Single Pin Symbol*

## Creating Single-pin Connector Symbols

When creating a connector pin symbol, you must correctly define the connector package in order for the layout netlist to be correct. For example, in creating a 62-pin edge connector, instead of creating a single symbol for a 62-pin edge connector with all 62 pins, you can create a symbol of a single connector pin and attach to it PKGREF and GATE attributes (created and assigned when the symbol is placed). You would then assign the attribute values for each instance of the pin to make the correct pin assignment to the connector. Each pin in the connector is the equivalent of a single gate in a multi-gate package. Therefore, by assigning to each connector pin instance a specific combination of PKGREF and GATE attribute values, you can define the wiring of the connector in the layout.

## Packaging the Parts in Your Design

Symbols used in Schematics represent either an individual gate of a packaged device, or a complete device. When a symbol representing a single gate is placed on a schematic, it is assigned a unique reference designator (if Auto-Naming is enabled), and by default, is made the first gate in the package. *Packaging* is the process of collecting these individual gates into physical packages and reassigning reference designators and gate names to reflect how they are packaged.

The packager uses the package definitions for devices that are in the package libraries. Package definitions contain information such as the number of gates, gate names and pin number assignments. Package definitions are created and maintained using the symbol editor. See *Specifying Part Packaging Information on page 5-27* for more information.

The packager assigns reference designators, gates and package type attributes to parts on the schematic.

- The PKGREF attribute is the reference designator for the package.
- The GATE attribute contains the gate identifier, if any.
- The PKGTYPE attribute contains the name of the physical package (footprint) to be used (e.g., DIP14, LCC20).
- The REFDES attribute is the reference designator normally displayed on the schematic. It is a combination of the PKGREF and GATE attributes.

For example, if PKGREF=U31 and GATE=a, the PKGREF will be U31a.

The REFDES attribute cannot be edited directly.

To change the REFDES, change either the PKGREF or the GATE attribute. The REFDES attribute will be automatically updated to reflect the change.

Pin numbers for devices with package definitions are determined from the package definition rather than from the symbol.

- Pin numbers are dependent on the gate (for multi-gate parts) and package type (for devices with alternative pin assignments based on package type).

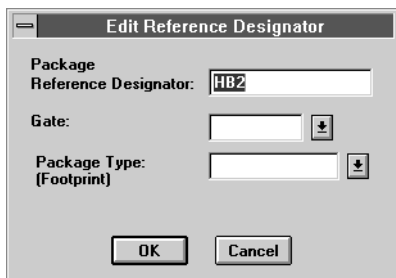
Until both GATE and PKGTYPE attributes are assigned values, no pin numbers are shown.

- For single gate packages with no gate name (e.g., blank instead of A) no GATE attribute value is required.
- The pin number visibility and location information from the symbol is used to determine whether, and if so, where, the pin numbers from the package definition are to be shown on the schematic.
- If a device has no package definition, then the pin number information is determined by the symbol definition.

## Assigning Reference Designators Manually

### Assigning Reference Designator

- 1 Double-click the displayed reference designator to display the Edit Reference Designator dialog box.



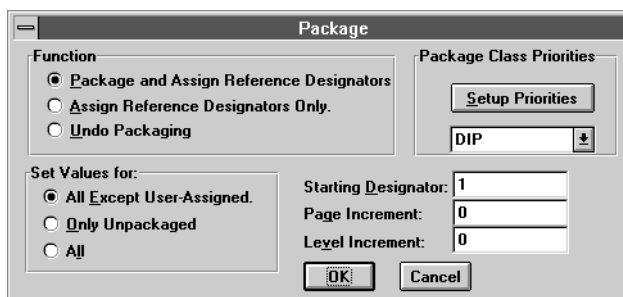
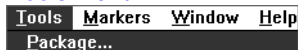
- 2 Type a new value in the Package Reference Designator text box.
- 3 Click OK.

If you have other parts that you want to automatically package together, use the All Except User Assigned option when you package the design.

### Automatically packaging at a later time

- 1 Select Package from the Tools menu to display the Package dialog box.

#### Tools Menu



- 2 In the Set Values for area, click to select the All Except User-Assigned radio button.

Any manually assigned reference designator values and gates will be kept.

- 3 Click OK.



# Assigning Reference Designators Automatically

Use the Package selection from the Tools menu to package individual parts into physical packages.

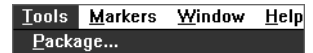
## Packaging and assigning reference designators

- 1 Select Package from the Tools menu to display the Package dialog box (shown on 9-8).
- 2 In the Function area, click the Package and Assigns Reference Designators radio button.
- 3 In the Set Values for area, click one of the three radio buttons to specify which parts will have reference designator and/or package information assigned.
  - a Choose All Except User-Assigned to restrict the function to those attributes that you have not manually assigned.
  - b Choose Only Unassigned to restrict the function to values for reference designator, gate and package type attributes that have not been assigned.
  - c Choose All to allow the function unrestricted access to all parts, overriding user-assigned attribute values.

If you want to change the package class priorities, see the procedure in the following section.

- 4 If you want to specify the number of the first reference designator to be assigned other than the default value of 1, type a value in the Starting Designator text box.
- 5 If you want to specify the amount to add to the starting designator between pages of the schematic, type a value in the Page Increment text box.
- 6 If you want to specify the amount to add to the starting designator between levels of hierarchy, type a value in the Level Increment text box.
- 7 Click OK.

### Tools Menu



## Setting Package Class Priorities

Priorities can be set for the packager to use in determining which package type to assign when a part is available in more than one type. For example, you could specify that a DIP package type be used. If the part is not available in DIP, then it could assign SMT, and so forth.

For details on adding package types and classes, see *Configuring Package Types on page 5-39*.

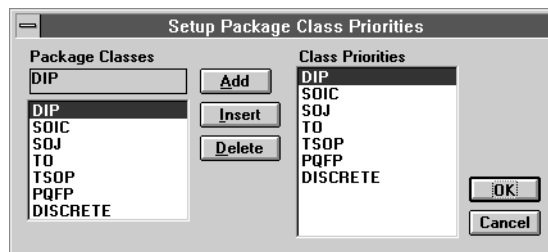
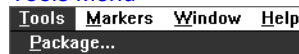
This is done by grouping commonly used package types into classes. For example, all sizes of DIP packages (DIP8, DIP16, etc.) belong to the DIP class.

For each device to be assigned a package type, the packager will go through the package classes in the order listed, and assign the first package type defined for that device which belongs to that class. However, if only one package type is defined for the package, it will be used, whether it is in the list or not.

### Setting up priorities

- 1 Select Package from the Tools menu to display the Package dialog box (shown on 9-8).
- 2 Click Setup Priorities to display the Setup Package Class Priorities dialog box.

#### Tools Menu



- 3 If you want to add a package class to the Class Priorities list, select a class from the Package Classes list on the left side of the dialog box and click Add.

The package class is added to the end of the Class Priorities list.

- 4 If you want to delete a class priority, select a class from the Class Priorities list, then click Delete.

- 5 If you want to insert a class into the Class Priorities list, first select a class from the Package Classes list, then select an item in the Class Priorities list and click Insert.

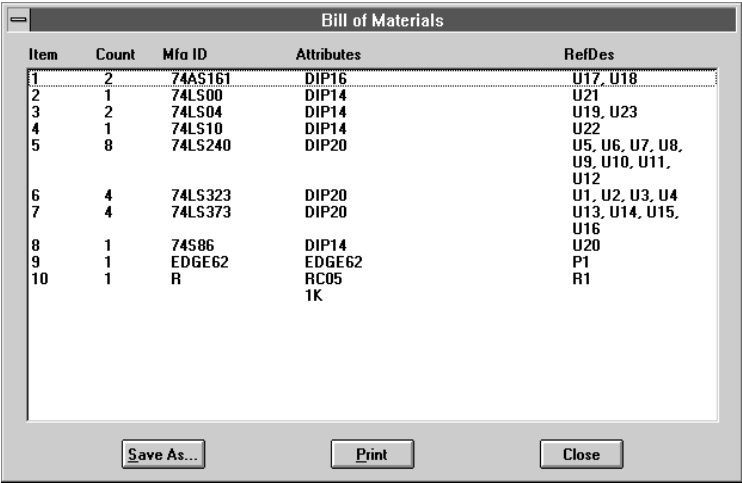
The package class is added to the list before the item selected in the Class Priorities list.

- 6 Click OK.
- 7 In the Package dialog box, click OK.

# Generating a Bill of Materials Report

A Bill of Materials report lists the quantities of each component type used in the design along with corresponding reference designators. You can also include information such as values for part instance attributes (VALUE and TOLERANCE) and user defined attributes contained in a *component description file*.

Figure 9-3 is an example of a Bill of Materials report generated for the PCBEX schematic shipped with your software. The only optional attribute chosen for display is the part instance VALUE attribute.



Item	Count	Mfg ID	Attributes	RefDes
1	2	74AS161	DIP16	U17, U18
2	1	74LS00	DIP14	U21
3	2	74LS04	DIP14	U19, U23
4	1	74LS10	DIP14	U22
5	8	74LS240	DIP20	U5, U6, U7, U8, U9, U10, U11, U12
6	4	74LS323	DIP20	U1, U2, U3, U4
7	4	74LS373	DIP20	U13, U14, U15, U16
8	1	74S86	DIP14	U20
9	1	EDGE62	EDGE62	P1
10	1	R	RC05 1K	R1

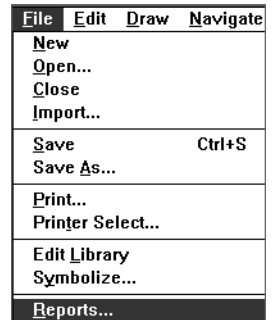
**Figure 9-3**    *Bill of Materials Report*

## Generating a Bill of Materials Report

- 1 Select Reports from the File menu to display the Reports dialog box.



### Edit Menu



- 2 Click Display.

The Bill of Materials dialog box again displays and you can print, display or save the report.

## Closing the Reports dialog box

Click Close.

## Printing and Saving the Report

### Printing a Bill of Materials report

- 1 If not already in the Reports dialog box, select Reports from the File menu.
- 2 In the Reports dialog box, click Print.



### Writing the Bill of Materials report to a file

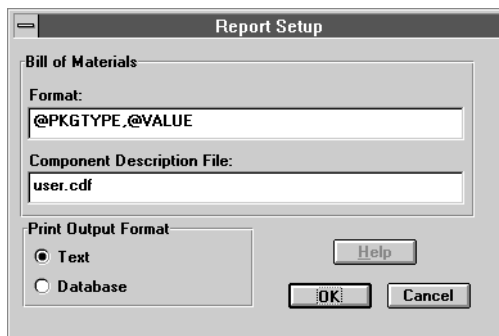
- 1 If not already in the Reports dialog box, select Reports from the File menu.
- 2 In the Reports dialog box, click Save As to display a standard Save dialog box.



## Customizing the Format of the Report



- 1 Click Setup to display the Report Setup dialog box.



- 2 In the Format text box, type the attributes to be displayed in the report according to the following syntax:

*[descriptive text]*@<attribute name>

where the '@' sign indicates *value* substitution for the named attribute. Specify multiple attributes by using the above syntax in a comma-separated list. For example, you could specify that the part instance *VALUE* attribute, and your own user defined attributes, *COST* and *ADDR*, be reported by entering the following into the Format text box:

value = @VALUE, cost = @COST, address = @ADDR

- 3 In the Component Description File text box, type the name of the component description file (".cdf") to be used.
- 4 Choose a Print Output format.  
Choose Text to format the Bill of Materials report in ASCII format with one entry per component type.
- 5 Click OK.

## User Defined Component Information

You can display user-specific component information (e.g., costs and in-house part numbers) in the Bill of Materials report. The Bill of Materials report will take a component description file as input.

The component description file (“*.cdf*” extension) is a user-created and maintained text file that contains component information such as cost, supplier name and in-house order numbers. To facilitate extraction of this information from an external component database, each file entry must be in comma-separated format as follows:

```
<component name>, <footprint name>,  
<manufacturing ID>, <attribute name>,  
<attribute value>
```

When you specify more than one user-defined property for a given component type, you must give each entry identical *<component name>*, *<footprint name>* and *<manufacturing ID>* values. For example, two entries for the LM124 component might appear as:

```
LM124, DIP14, LM124J-ND, COST, $4.05  
LM124, DIP14, LM124J-ND, SUPPLIER, National
```

Each Schematics software installation is shipped with a “*user.cdf*” file that you can edit to create a custom component description file.

### Specifying user-defined component attribute descriptions

- 1 Select Reports from the File menu to display the Reports dialog box (shown on **9-13**).
- 2 Click Setup to display the Report Setup dialog box (shown on **9-14**).
- 3 In the Component Description File text box, type the name of the user defined file that you want to use.
- 4 Click OK.
- 5 In the Reports dialog box, click OK.



## Exporting to a Spreadsheet or Database Program

The report can be output in a database format so you can use the report in a spreadsheet application.

### Specifying the format of the Bill of Materials report

- 1 Select Reports from the File menu to display the Reports dialog box (shown on **9-13**).
- 2 Click Setup to display the Report Setup dialog box (shown on **9-14**).
- 3 In the Print Output Format area, click Database.  
Choose Database to format the Bill of Materials report with one attribute name/value pair per entry. This results in multiple entries for component types with multiple attributes.
- 4 Click OK.
- 5 In the Reports dialog box, click OK.





# Swapping Pins

To swap pins on a given gate, add a SWAP attribute whose value is the pin names of the two pins to be swapped. For example:

```
SWAP=A B
```

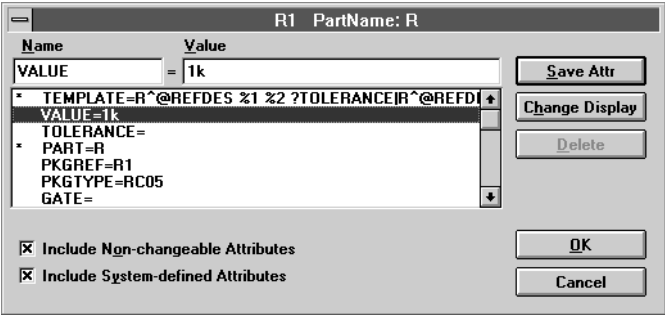
will swap pin A with pin B.

## Swapping pins

- 1 Select Attributes from the Edit menu to display the Attribute Editing dialog box.

Edit Menu

Edit	Draw	Navigate	View
Undo			Ctrl+U
Cut			Ctrl+X
Copy			Ctrl+C
Paste			Ctrl+V
Copy to Clipboard			
Delete			DEL
Attributes...			



- 2 In the Name text box, type SWAP.
- 3 In the Value text box, type A B.
- 4 Click Save Attr.
- 5 Click OK.

**Note** A and B must be pin names, not pin numbers.



# Interfacing to MicroSim PCBboards

## Creating a Layout Netlist for MicroSim PCBboards Checklist:

- 1 Check that PCBboards is the configured layout editor using the Configure Layout Editor selection from the Tools menu.
- 2 Select Package from the Tools menu to package the schematic and assign distinct reference designators.
- 3 Define any special trace width, trace clearance for nets or placement attributes for parts.
- 4 Generate a Bill of Materials report and check the package types are correct for the parts.
- 5 Create a layout netlist using the Create Layout Netlist selection from the Tools menu.
- 6 Select Run PCBboards from the Tools menu to start MicroSim PCBboards .

**Note** *The Run PCBboards menu item appears on the Tools menu only if you have MicroSim PCBboards as the configured layout editor.*

MicroSim PCBboards is a PCB layout editor that allows you to interactively specify printed circuit board structure as well as the components, metal and graphics required for fabrication. Designs entered with Schematics can be easily transferred to PCBboards for layout. Placement and trace properties are specified on the schematic for use by PCBboards. Schematic and/or board layout changes are automatically tracked; forward and backward annotation capabilities help maintain consistency between the schematic and the layout.

## Specifying Trace Properties

You can specify trace widths, clearances and padstacks to be used for vias for routing. You do this by adding the attributes shown in Table 9-2 to wires on the schematic.

**Table 9-2**    *Trace Properties Attributes*

Attribute	Description
NET_TRACE_WIDTH	Width (in mils or mm) of trace segments in this net. If units are not specified, the value is interpreted as inches.
NET_CLEARANCE	Edge-to-edge space (in mils or mm) between the trace segments and other layout objects. If units are not specified, the value in interpreted as inches.
NET_PADSTACK	Name of the padstack definition to be used for vias when routing traces.

To specify one or more of the above trace properties for a net:

- 1 Click on any segment of a wire that is part of the net to select it.
- 2 Click the Edit Attributes icon to display the Attribute Editing dialog box.
- 3 Type one of the above attribute names listed in Table 9-2 in the Name text box.
- 4 In the value text box, type the width, clearance or padstack value.
- 5 Click Save Attr.
- 6 Click OK.

## Specifying Component Locations

You can specify the position and orientation to use in layout for parts on the schematic. You do this by adding one or more of the attributes shown in Table 9-3 to the parts.

**Table 9-3** *Component Location Attributes*

Attribute	Description
COMP_LAYER	Name of the PCBoards layer representing the surface on which the component is to be positioned. By default, "Solder" is the bottom surface and "Component" is the top surface.
COMP_X	X-axis coordinate position (in mils or mm) of the part (at its origin) in the board layout. If units are specified, the value is interpreted as inches.

**Table 9-3**    *Component Location Attributes*

Attribute	Description
COMP_Y	X-axis coordinate position (in mils or mm) of the part (at its origin) in the board layout. If units are specified, the value is interpreted as inches.
COMP_ANGLE	Orientation of the part specified as an angle in degrees.
COMP_FIXED	If YES, designates the part is immovable; COMP_X and COMP_Y must be specified. This attribute defaults to NO.

To specify one of the properties shown in Table 9-3 for a part on the schematic:

- 1 Double-click the part to display the Edit Attributes dialog box.
- 2 In the Name text box, type the property name, e.g., COMP\_X.
- 3 In the Value text box, type the property value.
- 4 Click Save Attr.
- 5 Click OK.

## Cross-Probing

You can also cross probe the schematic from MicroSim PCBoards. Refer to the *Cross Probing* section of the *Working with the PCBoards User Interface* chapter in your *MicroSim PCBoards User's Guide*.

You can select a part or wire in Schematics and highlight the corresponding component or trace in the PCBoards layout.

### Highlighting the component in the layout for a part on the schematic

- 1 Select Cross-Probe Layout from the Tools menu in Schematics.
- 2 Click the part whose component you wish to see in the layout.

## Highlighting the trace in the layout for a net on the schematic

- 1 Select Cross-Probe Layout from the Tools menu in Schematics.
- 2 Click any segment of any wire that is part of the net.

## Highlighting multiple components or traces at once

- 1 Select the parts or wires in the schematic
- 2 Select Cross-Probe Layout from the Tools menu.

## Applying Backward ECOs

Each time a change is made to the layout, such as placement of a new component, deletion of an existing component, change to a reference designator, addition of a net, or addition/deletion of pins from nets, MicroSim PCBoards notes the change. When you save the layout, MicroSim PCBoards writes all changes to a backward ECO file (“`.bco`”). The next time you load the design into Schematics, the backward ECO file is checked and the backward ECO process begins.

Changes can be applied selectively to the schematic. Some changes are handled automatically by Schematics (gate swaps, pin swaps, reference designator changes); others must be manually applied (deletion/addition of components, deleting/adding pins to a net, addition/deletion of nets). An audit trail of ECO decisions—whether changes are pending, ignored, or require manual insertion—is maintained in the backward ECO log file (“`.blg`”).

Refer to the *Applying Backward ECOs* section of the *MicroSim PCBoards User's Guide* for more details.

## Applying Forward ECOs

Changes made to the schematic can be communicated to MicroSim PCBoards and automatically applied to the layout. Refer to the *Applying Forward ECOs* section in the *MicroSim PCBoards User's Guide* for details.

# Interfacing to Other Board Layout Products

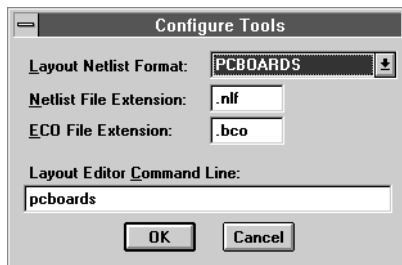
Schematics creates layout netlists in the formats shown in Table 9-4.

**Table 9-4** *Supported Layout Packages and File Formats*

Package	Layout Netlist	ECO File
PCBoards	.nlf	.ecf (forward) .ecb (backward)
PADS	.pad	.eco
P-CAD	.alt	(none)
Protel	.pro	.eco
Tango	.tan	.eco
CADSTAR	.cdn	.rin
SCICARDS	.upl	.sif
EDIF 2 0 0	.edf	(none)

## Selecting a layout format

- 1 Select Configure Layout Editor from the Tools menu to display the Configure Tools dialog box.



- 2 Select a format from the Layout Netlist drop-down list.

The layout netlist will be created with the name  
<schematic>.<netlist file extension>

- 3 Click OK.

## Creating a Layout Netlist

Select Create Layout Netlist from the Tools menu.

## Layout Mapping Files

When creating layout netlists, Schematics uses mapping files. These files let you customize the handling by the layout netlister of part, net and package type names. Mapping files are text files that you can edit with any text editor. Schematics is shipped with mapping files containing defaults and sample entries.

Mapping files exist for each of the supported layout formats.

- <layout format>.xmp  
Contains rules for creating entries in the parts list section of the netlist. If the part names used in Schematics are different than the names used in the layout package, you can specify the substitution to be done.
- <layout format>.xnt  
Contains name mappings for nets. For example, in Schematics the default power net on some digital parts is named \$G\_DPWR while in most layout systems it is +12V.
- <layout format>.xpk  
Contains name mappings for package types. For example, in Schematics a package type is called TO33 whereas it is TO-33 in the layout editor.



## Common syntax

Each file (“*.xmp*,” “*.xnt*” and “*.xpk*”) consists of a number of lines. Empty lines and those starting with the ‘#’ character are ignored. Otherwise a line consists of one or more comma-separated identifiers followed by either an *AKO specification* or a *replacement string*.

An AKO (A Kind Of) specifier consists of the keyword AKO followed by an identifier. Schematics looks up AKO definitions until it finds a definition that is not an AKO, then uses this new definition (circular AKO chains are not allowed).

The replacement string is processed further by Schematics and then becomes an entry for the part in the Partlist section of the layout netlist.

Examples:

```
DIODEDIODE,@PART
2N2220,2N2221,2N2222 TRNPN
1N914,1N915 AKO DIODE
```

The first example is a replacement string rule. This says that the string *DIODE* is to be replaced by the string *DIODE,@PART*. The second example shows how more than one identifier can reference the same replacement string. The third example shows an AKO specification. In this case, the string *1N914* references the *DIODE* rule. This is a replacement (i.e., not another AKO), so the net result is that Schematics will replace *1N914* by

*DIODE,@PART*.

A target identifier may end with the ‘\*’ character to indicate a match with a pattern containing the same leading characters up to the ‘\*.’ For example, *LCC\** matches *LCC20*, *LCC28*, etc.

Rules are tested in the order found in the file. A default rule specified anywhere in the file will be used when no matching pattern is found. Rules can be empty (i.e., a line may consist of a pattern or patterns, only).

## Parts List Mapping (.xmp)

Once Schematics has found a matching rule in the map file for the COMPONENT or PART attribute of a part, it further processes the replacement string. This processing is similar to the processing of the TEMPLATE attribute of a part when a simulation netlist is created. Identifiers in the string prefixed by one of the characters '@,' '?,' '~,' '#,' and `` are treated as part attribute names. A simple example would be a string such as @PART—this is replaced by the value of the PART attribute. An error occurs if the PART attribute is not defined.

When the `` (backquote) character precedes a '@,' '?,' '~,' or '#' character, it acts as a modifier. It causes the mapped value of an attribute (looked up in the “.xpk” file) to be used instead of the attribute value itself. For example,

`@PKG

would be replaced by the value of the PKG attribute, mapped by any matching rule in the “.xpk” file.

### Examples:

#### 1 Capacitors

We need to be able to provide for a generic capacitor (where the designer has not provided any information beyond the capacitor’s value and possibly a tolerance), and also for a more specialized capacitor (where the designer supplies the exact package type as well as the component value and tolerance).

To support the simple case, a rule of the following form will be required in the “.xpk” file:

CAPCAP,@value?tolerance|,@tolerance|

C AKO CAP

These rules will match a part with COMPONENT or PART attribute whose value is CAP or C. They will produce entries in the Part list like:

C101CAP,10uF

or

C102 CAP,10uF,20%

depending on whether a TOLERANCE attribute has been specified. The VALUE attribute must be defined - Schematics will issue an error message if a capacitor has no assigned value.

To support the case where the designer wishes to specify a particular capacitor type (for example, CAP\CR08\5G from the PADS library), the designer places an instance of a capacitor and then sets the COMPONENT attribute to CAP\CR08\5G.

The following rule in the “.xpk” file will support this:

```
CAP\* ?component|@component|@part|
@value?tolerance|@tolerance|
```

This tells Schematics to use the value of the COMPONENT attribute if that is set, or else to use the value of the PART attribute. This is followed by the value (required) and tolerance (optional).

**Note** *The general form of the rule will work for all similar types of capacitors such as CAP\CR20\3G, etc.*

## 2 TTL Devices

In this case, a rule is needed that passes the COMPONENT or PART attribute (e.g., 74LS04) through to the Part list, appending a package specifier for chip carrier devices. This “.xmp” rule will work:

```
74* ?component|@component|@part|&`pkg_type
```

This outputs the value of COMPONENT or PART, then tests the PKGTYPE attribute to see if it is defined. If so, its value is applied to the set of rules defined in the “.xpk” file. The translated value is then output.

Consider handling package types DIP14, DIP16, etc.; and LCC20, LCC28, etc. For DIP packages, the Partlist item should have no suffix (in other words, a DIP version of a 74LS04 is just a 74LS04). For the chip carrier packages, the item should have suffix -CC.

The following rules in the “.xpk” file will implement this:

```
DIP*
CC*      -CC
```

Note that the DIP\* rule is empty; it matches package classes such as DIP14 but there is no resulting replacement string.

The LCC\* rule matches all strings that start with LCC; so it will match package classes LCC20, LCC28, etc. It appends the string -CC to the COMPONENT (or PART) name.

## Back Annotation

During the course of layout generation, a design can undergo changes that make the design information in the PCB layout database inconsistent with that in the schematic database. When this occurs, the schematic must be back annotated with the design changes made during layout to resynchronize the schematic and layout data.

Design changes are usually documented as Engineering Change Orders (ECOs). Design changes from the layout to the schematic are called “Backward ECOs” because the direction of the change is opposite to the normal flow of design data.

Schematics supports the following types of backward ECOs:

- Changing the reference designator of a part.
- Swapping two gates.
- Swapping two pins.

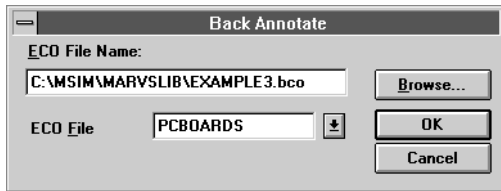
You can automatically apply backward ECOs from ECO files generated by:

PADS  
CADSTAR  
Tango  
Protel  
SCICARDS

If any of the other ECO operations are present, Schematics logs the warning and displays them as “Unsupported function” messages. You need to manually make the changes for any of the unsupported functions listed in the back annotation log. Back annotation messages are logged in the same manner as other Schematics errors and warnings.

## Using back annotation

- 1 Select Back Annotate from the Tools menu to display the Back Annotate dialog box.



### Tools Menu



- 2 Type the name of the ECO file generated by the layout package in the ECO File Name text box.
- 3 Select an ECO file format from the ECO File Format list box.
- 4 Click OK.

If, in step 2, if you don't know the file name, click Browse and select a file using the standard open file dialog box.

---

# Importing OrCAD SDT Schematics

---

A

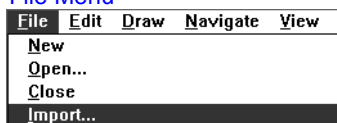
## Importing OrCAD Files

A schematic created with OrCAD SDT can be loaded into Schematics for editing. The schematic and any symbols it uses are translated from the OrCAD SDT format into the Schematics format. This is a *one-way process*. You can also translate individual OrCAD SDT libraries with this command.

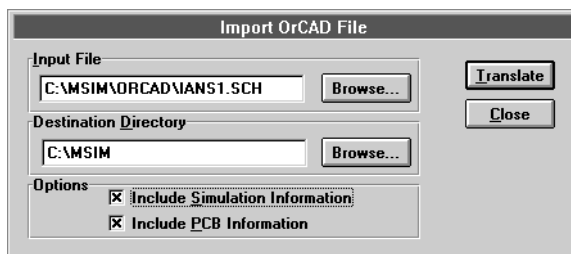
When translating OrCAD SDT schematics, Schematics uses the OrCAD SDT configuration file, “sdt.cfg,” to determine which OrCAD SDT symbol libraries to use to find symbols. Symbols referenced by the schematic are translated and put in a Schematics local symbol library. The “sdt.cfg” file must be either in the same directory as the OrCAD SDT schematic, or in the TEMPLATE subdirectory under the directory pointed to by the environment variable ORCADPROJ.

### Importing an OrCAD file

#### File Menu



- 1 Select Import from the File menu to display the Input OrCAD File dialog box.



- 2 Type the name of the file to be imported in the Input File text box, or click the Browse button to the right of the Input File text box to select a file.
- 3 Type the name of the directory where the translated schematic is to be saved in the Destination Directory text box, or click the Browse button to select a file.

The translated schematic file is given the same name as the original OrCAD SDT schematic file (“*.sch*” extension). A translated symbol library is given the same file name, but with a “*.slb*” extension instead of “*.sch*.”

- 4 To select either of the options, click to place an X in the check box for the options you want to select. The two options are explained below.
- 5 Click Translate.

# Import Options

## Include simulation information

Select the Include Simulation Information option to add simulation attributes to symbols in the library. This option also converts part field data (part properties) on the OrCAD SDT schematic into information that can be used by the Schematics simulation netlister.

The default mapping between OrCAD SDT part field data and Schematics attributes is contained in the files “devmap.ini” and “orc\_map.txt.” This information is based on the standard OrCAD SDT libraries. You can change or add additional parts to these files with any text editor.

This option should be enabled if you plan to simulate your designs using PSpice or PLogic. If enabled, the PSpice Simulation Device Types Dialog will be displayed (see below).

## Include PCB Information

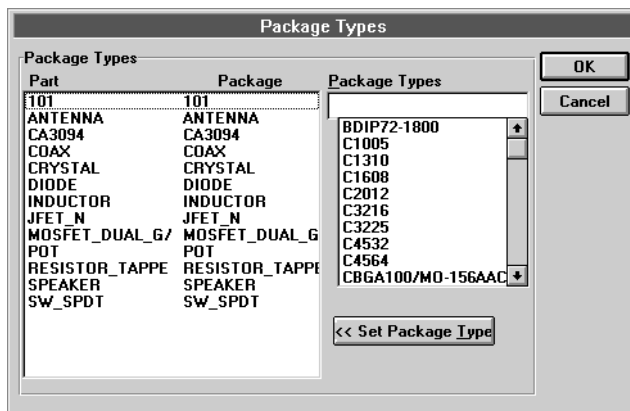
Select this option to create package libraries in addition to symbol libraries during translation.

Enable this option if you plan to netlist the schematic for layout. When you enable this option, the Package Types dialog box displays.



## Package Types Dialog Box

When you enable the Include PCB Information option in the Import OrCAD File dialog box, the Package Types dialog box displays.



To assign package types (footprints) and create packaging information for the parts on your schematic, Schematics uses the file “orc\_map.txt” to search for default package types. A list is created of all parts for which no entry was found. The Package Types dialog box lets you assign package types for these parts. The default is to make the package type (footprint) name the same as the part name.

### Completing the Package Types dialog box

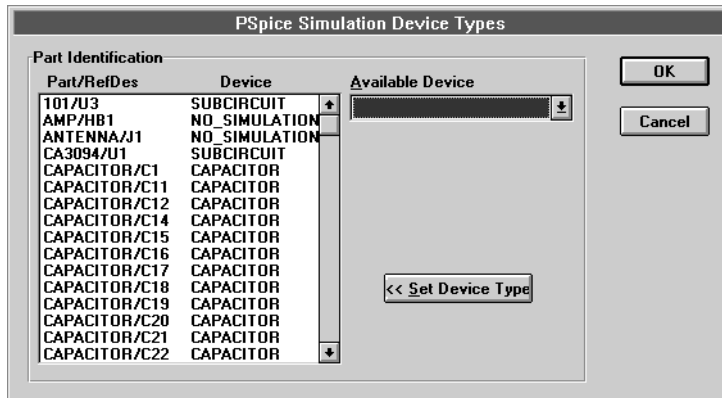
The Package Type section lists the current package types to be assigned to the corresponding parts. The initial default is to assign a footprint with the same name as the part.

The Parts section lists the parts for which no default package information was available.

Package Types defines the name of the package type to assign to the currently selected part(s). You can enter a name or choose one from the list.

Set Package Type assigns the package type in the Package Types text box to the currently selected part(s).

# PSpice Simulation Device Types Dialog Box



To convert OrCAD SDT part field data needed for simulation, Schematics must know the PSpice device type for each part placed on the schematic. An initial guess is made during translation, based on the first letter of the reference designator. The PSpice Simulation Device Types dialog displays the list of parts and the translator's initial assignment of device type. Use this dialog to review the list and make any necessary corrections.

Part field data conversion is determined by the "devmap.ini" mapping file. For each PSpice device type, the map file lists the OrCAD SDT part field corresponding to the simulation device parameter(s) for that device type. "Devmap.ini" is a text file which you may need to modify depending on how you used OrCAD SDT part fields.

## **Completing the PSpice simulation device types dialog box**

Part/RefDes lists each part's symbol name and reference designator on the schematic.

Device lists the current PSpice device types to be assigned to the corresponding parts.

Available Device lists all of the available PSpice simulation device types.

Set Device Type assigns the device type in the Available Device text box to the currently selected part(s).

## **Translating Multi-Page Schematics**

To translate a multi-page schematic, translate the root page (the page containing the |LINK text that lists the files of the other pages). All pages referenced by the |LINK text are translated. Schematics keeps all pages of a multi-page schematic in a single file; thus, there will only be one resulting schematic file—the root—which contains all the pages.

## **Translating Hierarchical Schematics**

To translate a hierarchical schematic, translate the topmost sheet. Any lower-level schematic referenced by sheet symbols will also be translated.

# Translating Large Designs

Large designs may take a very long time to translate. To facilitate the translating of large designs, a standalone version of the translator, “orctrans.exe,” is provided.

## Text Size

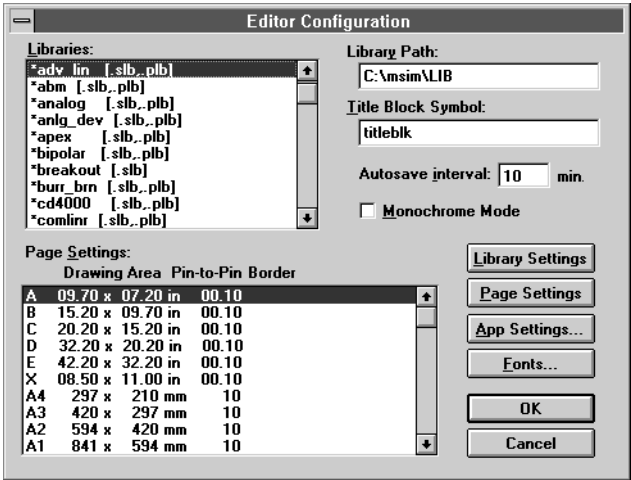
Text sizes on translated schematics/symbols are based on an 8-point font size. You can change the font that Schematics uses for text display.

### Changing display font size

- 1 Select Editor Configuration from the Options menu to display the Editor Configuration dialog box.

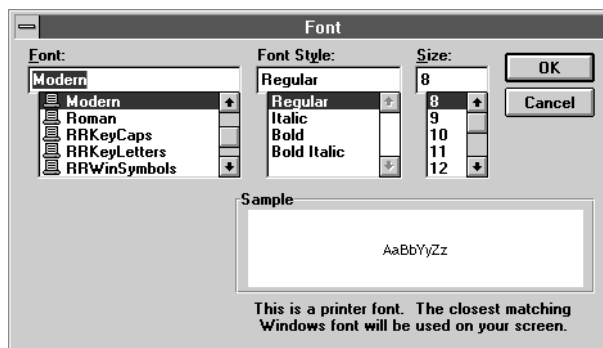
Options Menu

Options	Analysis	Tools
Display Options...		
Page Size...		
Auto-Repeat...		
Auto-Naming...		
Set Display Level...		
Editor Configuration...		



- 2 Click the Fonts button to display a standard Windows font selection dialog box.





- 3 Select a new font size in the Font scroll box.
- 4 Click OK.
- 5 In the Editor Configuration dialog box, click OK.

## Connecting Signal via Labels

In OrCAD, if two wires have the same label, the netlister will connect them. In Schematics, by default, this is not the case and you must connect each wire to a port. Use the following procedure to change the default behavior to emulate OrCAD.

### Enabling connectivity via labels

- 1 Select Restricted Operations from the Options menu to display the Restricted Operations dialog box.



- 2 Click the Connectivity via wire labels check box.
- 3 Click OK.

# Differences between OrCAD SDT and Schematics

Although Schematics makes every attempt to translate the schematic completely, there may be discrepancies in the resulting schematic depending on how certain features of OrCAD SDT were used. You should always check any errors and warnings that occur during the translation, as well as reviewing the resulting schematic.

Here are the differences you should consider when translating your OrCAD SDT schematic:

- Schematics has fixed size ports; i.e., they do not resize based on the label. When an OrCAD SDT schematic is translated, port symbols are created for each port size.
- In OrCAD SDT, you can have ports with multiple connections. This is translated as two ports overlaid with pins facing opposite directions.
- In Schematics, wires connect directly to buses without using special symbols such as bus entries. Buses *must* be labeled with the set of signals they represent. Wires connected to buses must be labeled with the signal name within the bus to which they correspond.

Therefore, during translation,

- OrCAD SDT “Bus Entry” objects are converted into regular wires or buses.
- Prior to translation, when drawing “bus rippers,” you must use the OrCAD SDT “Bus Entry” object. If you draw a normal wire at an angle to look like the regular bus ripper/entry object, the wire will not be connected to the bus.
- *Simulation* attributes that are added to symbols are based on information from the OrCAD SDT standard libraries. Therefore, pin name references in these attributes can be incorrect if a part (e.g., user-drawn) is being used that has

the same name as another part (e.g., from the standard libraries) on the schematic.

- Wires that cross pins exactly at the pin hotspot are translated as connected.
- Unlabeled wires connected to buses are translated as unconnected wires, i.e., they are not connected to the bus.
- The OrCAD SDT feature—pins with a pin number of zero are not drawn—is not supported. Therefore some symbols are drawn with unexpected pins or with missing pins. Only the first representation of the symbol (GATE A) is translated.
- Text with a vertical orientation is translated into rotated text.
- Dashed annotation lines are not included in the translated schematic.
- De-Morgan equivalents (convert symbols) are not supported. You get the non-converted version on your schematic.
- Fill patterns used in parts are not translated.
- “Stimulus,” “Trace,” and “Layout” symbols on the OrCAD SDT schematic are not translated.
- Wires and buses in Schematics can only have a single label. Any connections, bus mapping or splitting that use the OrCAD multiple label feature will not translate correctly. In Schematics, you will need to manually reconnect these wires and buses.

In Schematics, you can split a bus directly by labeling the subbus with a subset of the signals on the main bus, e.g., if the main bus is data[0..15], you can connect a bus to it and label it data[0..7] and connect this directly to a port or pin with a name of the same width, e.g. inputData[0..7]. (See *Splitting buses on page 3-32*.)

- By default, Schematics does not consider wires with the same label to be connected unless they are drawn as connected. You must connect them to offpage ports or enable the Connectivity via labels option. To change the default, see *Enabling connectivity via labels on page A-8*).

---

# Library Expansion and Compression Utility

---

## B

A Library Expansion and Compression utility is provided with all software packages that include Schematics and/or MicroSim PCBoards. This utility works with the symbol (Schematics), package (Schematics and MicroSim PCBoards) and footprint (MicroSim PCBoards) libraries.

It can be used for:

- Maintaining library files.
- Reorganizing a library file.
- Batch library file creation.
- Salvaging a corrupted library file. Expansion, Compression and the List File

The Library Expansion and Compression utility expands a library by reading it line by line, checking against the offset and size in the library file index, writing each definition out as a separate text file and reporting any discrepancies. It produces the “.lst” file as it processes library file. A list file (“.lst” extension) contains a mapping of component names to text file names for the individual definitions. Its format is outlined in Table B-1.



**Table B-1** *List File Format*

<b>Symbol, Package or Footprint File Name</b>	<b>Symbol, Package or Footprint</b>
x.sym, x.pkg or x.fpd	x
yy.sym, yy.pkg or yy.fpd	yy
z_1.sym, z_1.pkg or z_1.fpd	z/1

Compressing a library re-builds a library by reading the “.lst” file (ignoring any further text on the line) and extracting the definitions from the appropriate component file (“.sym,” “.pkg,” or “.fpd”). The extracted definitions are then written to the specified library file in the order read. Thus, symbol, package and footprint libraries are built from text files generated by the Export function from the Part menu function, the Library Expansion utility or both.

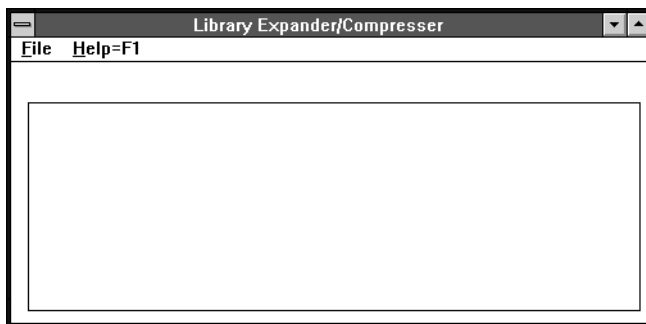
# Running the Utility

To start the Library Expansion and Compression Utility:

- 1 In the Windows File Manager, select Run from the File menu.
- 2 Type the following on the Command Line:

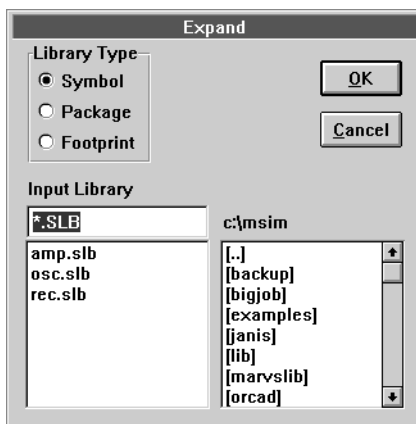
`C:\msim\lxcwin.exe`

The Library Expander/Compressor window displays.



## Expanding a library

- 1 Select Expand from the File menu to display the Expand dialog box.



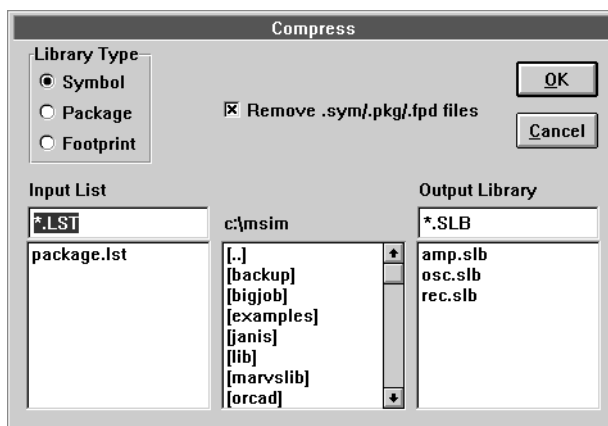
- 2 Select the Library Type: Symbol, Package or Footprint.

- 3 Select a library file in the Input Library list.
- 4 Click OK.

A set of either symbol (“`.sym`”), package (“`.pkg`”) or footprint (“`.fpd`”) files is created for each definition in the selected Input Library file. A list file (“`.lst`”) is also created.

### Compressing a library

- 1 Select Compress from the File menu to display the Compress dialog box.



- 2 Select the Library Type (defaults to the last type selected in this dialog or by selecting Expand from the File menu).
- 3 Select the appropriate list file (“`.lst`”) from the Input List.
- 4 Select a library file for output from the Output Library list, or enter the name of a new file in the text box.
- 5 If you would like to delete the selected input files when the compression is complete, enable the “Remove `.sym/.pkg/.fpd` files” check box.
- 6 Click OK.

The Output Library file is created from the individual component files listed in the list file.

---

# Advanced Netlisting Configuration Items

---

C

## Specifying PSpice Node Name Netlisting Preferences

By default, the PSpice netlister assigns names such as \$N\_001 to nodes that are not explicitly labeled.

You can change the format the netlister uses to create these names by using a text editor and editing the “msim.ini” file in the Windows directory. Add a line to the [SCHEMATICS] section in the form:

```
nltemplate=<prefix> %[minimum width]ld
```

The default value is:

```
nltemplate=$N_%04ld
```

The term ld must be in lowercase.

If [*minimum width*] begins with a zero, leading 0's are added if the number of characters in the node number is less than the minimum width. For example, `n%ld` would create node numbers N1, N2, N3 instead of `$N_001`, `$N_002`, `$N_003`.

# Specifying Board Layout Node Name Netlisting Preferences

To change any of these settings, use a text editor and edit the “msim.ini” file in the Windows directory.

PCBHIERPATHSEP is the separator character to use when creating hierarchal net names in layout netlisting.

The PCBTEMPLATE item specifies the form that the layout netlister uses for creating node names.

The default value is:

```
PCBTEMPLATE=NN_%04LD
```

## Customizing EDIF Netlists

You can change the amount each level in the netlist is indented by changing the `EDIFINDENT` item in the [SCHEMATICS] section of the “msim.ini” initialization file. Use a text editor to edit the “msim.ini” file in the Windows directory.

`EDIFINDENT` specifies the character to use to indent each level in an EDIF netlist.

The default is:

```
EDIFINDENT="  "
```

---

# Attribute List

---

## D

This appendix is a list of attribute names used by Schematics and descriptions of each of those attributes.



Attribute	Description	See Notes
COMPONENT	The name of the package definition to be used for a part. If the name of the package definition is the same as the part name, then the COMPONENT attribute is not necessary.	2 5
GATE	<p>The gate within the package to which a particular part instance is assigned. For example, if a part is one of four gates in a package (e.g., A, B, C, D), this attribute specifies which gate the part is assigned to. Valid values for the GATE attribute (A, B, C, D in this example) are specified as part of the package definition for the part (given in the appropriate “.plb” Package Library file). Packages with only one gate do not require a gate name. This terminology applies not only to digital parts, but also to analog parts with multiple <i>gates</i> in a package.</p> <p>When a part is placed, the GATE attribute is assigned the value of the first gate defined for that part, or nothing at all if there is only one gate in the package. The value of the GATE attribute will be reassigned when the schematic is packaged. You can edit the value of the GATE attribute, to manually assign a gate, by selecting Attributes from the Edit menu or double-clicking on the REFDES attribute. See REFDES below.</p>	2 4
GATETYPE	The name of the gate type of a part. If a package contains more than one type of gate (e.g., a package that contains an AND gate and a NOR gate), then there will be different symbols to represent each type. Each of these symbols must have a GATETYPE attribute, whose value is used during packaging to identify the correct pin assignments within the package definition. For multi-part packages, the package definition will contain a set of pin assignments for each gate type. The identifiers used in the package definition for each gate type must match the GATETYPE attributes on the symbols.	2 5
MODEL	<p>The name of the model referenced for simulation. This name should match the name of the .model or .subckt definition of the simulation model as it appears in the Model Library file (“.lib”). For example, if your design includes a 2N2222 bipolar transistor, for which the .model name is Q2N2222, then the MODEL attribute on the symbol for that part should be Q2N2222. This MODEL attribute can then be referenced in the TEMPLATE attribute for netlisting:</p> <p>TEMPLATE=Q^@REFDES %c %b %e @MODEL</p> <p>where REFDES=Q1 and MODEL=Q2N2222, could result in a netlist entry of Q_Q1 4 6 8 Q2N2222.</p>	1 3 5

---

Attribute	Description	See Notes
MODEL (con't.)	In the schematic editor Edit Attributes dialog, the MODEL attribute is marked with an asterisk. This means that the attribute is not changeable using this dialog. You must select Model from the Edit menu and use the Edit Model dialog box to either change the model reference or to create an instance model. To edit the underlying model definition of a part, select Model from the Edit menu in the symbol editor, not in the schematic editor.	
PART	The name of the part that was retrieved from the Symbol Library and placed. When you use Get New Part, to select and place a part, the PART attribute reflects the name of the part that you selected. The value of this attribute will not change, for instance, when you change the MODEL attribute. You can always see what part you placed by referring to the value of this attribute, which is usually displayed on the schematic for all devices (for breakout devices, the MODEL attribute is the one visible on the schematic). The PART attribute is only changeable in the symbol editor.	1 2 5
PKGREF	The Package Reference Designator. If there are four gates in a package (e.g., A, B, C and D), then the PKGREF for all four parts would be the same (e.g., U1) and the GATE attribute distinguishes them (e.g., U1A, U1B, U1C, U1D). The PKGREF is the first component of the REFDES attribute. See REFDES below.	2 4
PKGTYPE	The physical carrier type to be used for the part. (Examples: DIP14, LCC20, DIP8). If the package definition for the part has only one available package type defined, then the PKGTYPE attribute will be assigned this value. You can manually assign the package type by editing/creating this attribute, or you may have the PKGTYPE attribute assigned during packaging.	2 4
REFDES	The Reference Designator of a part. The value of the REFDES attribute is a combination of the Package Reference Designator (PKGREF) and the gate (GATE) attributes. For example, if your PKGREF is U1 and your GATE is A, then your REFDES will be U1A (and will appear as such on your schematic). REFDES cannot be edited directly in the schematic editor. You must edit the PKGREF and/or GATE attributes instead. When you double-click the REFDES of a part within the schematic editor, the dialog which appears has two edit controls: one for the Package Reference Designator and one for the Gate.	1 2 3 4 5

Attribute	Description	See Notes
SIMULATION-ONLY	If present, this attribute indicates that the part only has meaning for simulation. There will be a netlist entry for parts with this attribute, but no layout netlist entry. The SIMULATIONONLY attribute identifies parts such as voltage and current sources, breakout devices (found in “breakout.slb”) and <i>special</i> symbols (found in “special.slb”).	2 5
TEMPLATE	The recipe for creating a netlist entry for simulation. The pin names specified in the TEMPLATE must match the pin names on the symbol. The number and order of the pins listed in the TEMPLATE must match those appropriate for the associated .model/.subckt definition referenced for simulation. The TEMPLATE attribute is only changeable in the symbol editor.	1 3 5

- Notes:**
- 1 These attributes are not changeable within the schematic editor. These are the attributes marked by an asterisk in the Edit Attributes dialog.
  - 2 These attributes pertain to packaging and/or board layout.
  - 3 These attributes pertain to PSpice and PLogic.
  - 4 These attributes are automatically provided by Schematics when a part is placed in the schematic editor.
  - 5 These attributes must be provided by the user when creating or modifying a symbol in the symbol editor.

---

When you use the symbol editor to create a new symbol, the PART, MODEL, REFDES and TEMPLATE attributes are provided as a default set. You can provide any other attributes as needed.

---

# Symbol Libraries

---

## E

Symbols are stored in symbol libraries. The symbol library files have a “.slb” extension and contain graphical representations and attributes of parts.

The contents of the symbol libraries provided with Schematics are listed in Table E-1.

Parts from libraries marked with <sup>†</sup> do not have corresponding simulation models.

**Table E-1** *Symbol Libraries*

Symbol Library File Name	Contents
7400.slb	7400-series TTL
74ac.slb	Advanced CMOS
74act.slb	TTL-compatible, Advanced CMOS
74als.slb	Advanced low-power Schottky TTL
74as.slb	Advanced Schottky TTL
74f.slb	FAST

**Table E-1** *Symbol Libraries (continued)*

<b>Symbol Library File Name</b>	<b>Contents</b>
74h.slb	High-speed TTL
74hc.slb	High-speed CMOS
74hct.slb	TTL-compatible, high-speed CMOS
74l.slb	Low-power TTL
74ls.slb	Low-power Schottky TTL
74s.slb	Schottky TTL
abm.slb	Behavioral modeling blocks
adv_lin.slb	Advanced Linear Devices: operational amplifiers
analog.slb	Passive and semiconductor primitives
analog_p.slb	Same devices as “analog.slb” with visible pin numbers for R, L, C, R_VAR and C_VAR devices
anlg.slb <sup>†</sup>	Multiplexers, DAC, ADC, voltage-to-frequency converters
anlg1.slb <sup>†</sup>	DAC, ADC, sample-and-hold amplifiers
anlg2.slb <sup>†</sup>	voltage references, voltage regulators, PWM, DAC, ADC, transceivers
anlg_dev.slb	Analog Devices Inc.: operational amplifiers, transistor arrays, buffers, voltage references, analog multipliers, analog switches
apex.slb	Apex Microtechnology Corporation: operational amplifiers
atmel.slb <sup>†</sup>	Atmel Corporation: EEPROM, PROM, SRAM, PLD
bipolar.slb	Bipolar transistors
breakout.slb	Parameterized devices for model purposes
broktree.slb <sup>†</sup>	Brooktree Corporation: DAC, delay lines, comparators
burr_bm.slb	Burr-Brown Corporation: operational amplifiers
cd4000.slb	CD4000 digital devices

**Table E-1** *Symbol Libraries (continued)*

<b>Symbol Library File Name</b>	<b>Contents</b>
chips.slb <sup>†</sup>	Chips and Technologies, Inc.: CPU
cmos.slb <sup>†</sup>	counters, shift registers, PLL, buffers, modulators, gates, adders, switches, multipliers, display drivers, timers, flip-flops, latches
comlinr.slb	Comlinear Corporation: operational amplifiers
connect.slb	Connectors
dallas.slb <sup>†</sup>	Dallas Semiconductor: delay lines, SRAM, transceivers, timers, FIFO, microcontrollers
dataconv.slb <sup>†</sup>	ADC, DAC
dig_ecl.slb	Emitter coupled logic devices
dig_gal.slb	Generic array logic devices
dig_misc.slb	Miscellaneous digital devices
dig_pal.slb	Programmable array logic devices
dig_prim.slb	Digital primitives for use with PLSyn as well as general simulation purposes
diode.slb	Diodes, Zener diodes, current regulator diodes, varactors
ebipolar.slb	European bipolar transistors
ecl.slb <sup>†</sup>	Motorola Corp., National Semiconductor Inc.: DRAM, gates, multiplexers, level translators, prescalers, error correction/detection
ediode.slb	European diodes and rectifiers
elantec.slb	Elantec Inc.: operational amplifiers, transistor arrays
epwrbjt.slb	European power bipolar transistors
exel.slb <sup>†</sup>	Exel Microelectronics Inc.: EEPROM
filtsub.slb	Filters
fujitsu.slb <sup>†</sup>	Fujitsu Limited: PROM, DRAM, SRAM, EEPROM
fwbell.slb	F.W. Bell: Hall effect devices

**Table E-1** *Symbol Libraries (continued)*

<b>Symbol Library File Name</b>	<b>Contents</b>
goldstar.slb <sup>†</sup>	Goldstar Semiconductor Group: ROM, DRAM, SRAM
har_dig.slb	Harris Semiconductor Corp.: PROM, microprocessors, VART, interface, transceivers, controllers, SRAM
harris.slb	Harris Semiconductor Corp.: operational amplifiers, MCT, bridge drivers, transistor arrays, power MOSFET
hughes.slb <sup>†</sup>	Hughes Microelectronic Center: display drivers, CPU, SRAM
hyundai.slb <sup>†</sup>	Hyundai Electronic Inc. Ltd.: PLD, DRAM, SRAM
intel.slb <sup>†</sup>	Intel Corp.: EPROM, CPU, math co-processors, microcontrollers, SRAM, network processors
jbipolar.slb	Japanese bipolar transistors
jdiode.slb	Japanese diodes, rectifiers, Zener diodes, varactors, Schottky diodes
jfet.slb	Junction field-effect transistors
jjfet.slb	Japanese junction field-effect transistors
jopamp.slb	Japanese operational amplifiers
jpwrbjt.slb	Japanese power bipolar transistors
jpwrmos.slb	Japanese power MOSFETs
lin_tech.slb	Linear Technology Corporation: operational amplifiers
magnetic.slb	Magnetic cores, inductor coupling devices
marker.slb	Probe markers (this file is automatically accessed by Schematics and should not be included in the list of configured library files)
mcpwrsys.slb <sup>†</sup>	Micro Power Systems: ADC, DAC, data acquisition, voltage references
mcrndram.slb <sup>†</sup>	Micron Semiconductor, Inc.: DRAM

**Table E-1** *Symbol Libraries (continued)*

<b>Symbol Library File Name</b>	<b>Contents</b>
mcmram2.slb <sup>†</sup>	Micron Semiconductor, Inc.: DRAM
mcmsram.slb <sup>†</sup>	Micron Semiconductor, Inc.: SRAM
memory.slb <sup>†</sup>	EPROM, PROM, SRAM, PAL, DRAM, EEPROM
misc.slb	Timers, CMOS transistor arrays, variable admittance, variable impedance, three-phase transformers, relays, DC motor, time-dependent switches
mitmem.slb <sup>†</sup>	Mitsubishi Electric Corporation: EEPROM, PROM, DRAM, SRAM
mitram.slb <sup>†</sup>	Mitsubishi Electric Corporation: DRAM, SRAM
mitrom.slb <sup>†</sup>	Mitsubishi Electric Corporation: EPROM
mosel.slb <sup>†</sup>	Mosel-Vitolic Inc.: SRAM, FIFO
moto.slb <sup>†</sup>	Motorola Corp.: CPU, microcontrollers
moto7.slb <sup>†</sup>	Motorola Corp.: SCR, triac
motoramp.slb	Motorola Corp.: operational amplifiers
motormos.slb	Motorola Corp.: power MOSFET
motor_rf.slb	Motorola Corp.: RF bipolar transistors
nat_semi.slb	National Semiconductor Inc.: operational amplifiers
nsclnapp.slb <sup>†</sup>	National Semiconductor Inc.: video sync generators, power drivers, level translators, display drivers, PLL, switches, noise reduction processors, power amplifiers, timers
nsdram.slb <sup>†</sup>	National Semiconductor Inc.: error detection/ correction, DAC, memory controllers
nsnetwk.slb <sup>†</sup>	National Semiconductor Inc.: interface controllers, network interface
nsucont.slb <sup>†</sup>	National Semiconductor Inc.: microcontrollers



**Table E-1** *Symbol Libraries (continued)*

<b>Symbol Library File Name</b>	<b>Contents</b>
oki.slb <sup>†</sup>	OKI Semiconductor: display drivers, DRAM, EEPROM, EPROM, DRAM, SRAM, microcontrollers, clock, speech synthesis, recorders, CODEC, modems
opamp.slb	Operational amplifiers, voltage comparators, voltage regulators, voltage references
opto.slb	Opto couplers
pansonc.slb <sup>†</sup>	Panasonic Industrial Group: ROM, DRAM, SRAM, FIFO
polyfet.slb	PolyFet RF Devices: RF MOSFET
port.slb	Global ports, off-page ports, interface ports, ground symbols
pwrbjt.slb	Power bipolar transistors
pwrmos.slb	Power MOSFET
seeq.slb <sup>†</sup>	Seeq Technology Inc.: data link controllers, Manchester code converters
sgsthom.slb <sup>†</sup>	SGS_Thompson Microelectronics: EPROM, SRAM, FIFO, EEPROM
sipex.slb <sup>†</sup>	Sipex Corporation: ADC, DAC, voltage references, sample-and-hold amplifiers, data acquisition
smos.slb <sup>†</sup>	S-MOS Systems: DC/DC converters, voltage regulators, display drivers, SRAM
sony.slb <sup>†</sup>	Sony Corporation: ADC, gates, ALU, multiplexers, SRAM
source.slb	Voltage and current stimulus devices
special.slb	Simulation pseudocomponents (IC, NODESET, etc.)
swit_rav.slb	Averaged switched-mode power supply blocks
swit_reg.slb	Switched-mode regulators

**Table E-1** *Symbol Libraries (continued)*

<b>Symbol Library File Name</b>	<b>Contents</b>
tex_inst.slb	Texas Instruments Inc.: operational amplifier, voltage comparators
thyristr.slb	SCR, triac, UJT
ti1.slb <sup>†</sup>	Texas Instruments Inc.: line drivers, transceivers, display drivers, ADC, switches
ti2.slb <sup>†</sup>	Texas Instruments Inc.: SRAM, EPROM, DRAM, PROM, memory controllers
tilsi.slb <sup>†</sup>	Texas Instruments Inc.: FIFO, error detection/correction, multipliers, pipeline registers, flip-flops, bus transceivers, memory controllers
tline.slb	Transmission lines
ttl.slb <sup>†</sup>	Multiplexers, counters, flip-flops, bus transceivers, gates, monostable multivibrators, encoders, FIFO, buffers, adders, decoders
vlsitec.slb <sup>†</sup>	VLSI Technology Inc.: ALU, UART, memory controllers, CPU, display drivers
weitek.slb <sup>†</sup>	Weitek Corp.: math co-processors
wsi.slb <sup>†</sup>	WaferScale Integration Inc.: PROM, EEPROM, CPU, multipliers
xicor.slb <sup>†</sup>	XICOR Inc.: SRAM, EEPROM, potentiometers
xtal.slb	Quartz crystals
zilog.slb <sup>†</sup>	Zilog Inc.: I/O controllers, CPU, counters

---

# Glossary

---

<b>ABM</b>	analog behavioral model. A view of a hierarchical schematic used for analysis. See also <i>View</i> .
<b>AKO</b>	“A Kind Of” symbol. Symbols must either contain graphics or refer to an AKO symbol. The AKO defines the symbol in terms of the graphics and pins of another part. Both must exist in the same symbol library file.
<b>alias</b>	an exact electrical equivalent that can be used to reference a symbol.
<b>annotation</b>	a means by which parts are labeled when they are placed, either automatically or manually.
<b>annotation symbol</b>	a symbol with no electrical significance, used to clarify, point out or define items on the schematic.
<b>attribute</b>	special characteristics (a name and an associated value) contained in a part instance or definition. For example, a MOSFET may contain specific length and width parameters which are represented as attributes on the symbol or part. Attributes may be changed through the schematic editor, the symbol editor or both.
<b>back annotation</b>	annotation of a schematic using an ECO file from the selected layout editor.
<b>block</b>	a user defined rectangle placed on a schematic. It is used to represent or hold the place for a collection of circuitry. The block is treated as a “black box” by Schematics. Schematics is aware of the connections going into and out of the block, but ignores the contents of the block until netlisting.

<b>bounding box</b>	a rectangular dotted line containing the graphics for a symbol and all visible pin connection points. In terms of the schematic editor, the position of the bounding box determines whether a point falls on a part when selecting parts, or whether it falls on the pin of a part when checking for electrical connections. <code>box</code> defines the selection area of the symbol when placed on a schematic
<b>bundle</b>	a collection of named wires or buses of the same type or purpose.
<b>bus</b>	a collection of homogeneously named signals.
<b>circuit</b>	a configuration of electrically connected components or devices.
<b>component</b>	See <i>package</i> .
<b>connector</b>	a physical device that is used for external connections to a circuit board. A connector has no electrical significance until it is connected on a PCB.
<b>current sensor</b>	displays the bias point current flow in a given direction.
<b>design</b>	a schematic drawing or set of drawings representing a circuit or PCB.
<b>device</b>	See <i>package</i> .
<b>display map</b>	a portion of the initialization file that specifies which schematic items are turned on and off for display and/or printing.
<b>ECO</b>	engineering change order. A design change usually requiring back annotation of the schematic.
<b>ERC</b>	electrical rules check, a process performed before generating a netlist or running the simulator. The ERC performs a preliminary connectivity check on the schematic. If the schematic is part of a hierarchical design, the check is performed only for the current hierarchical level.
<b>fileset</b>	the set of files required to perform a certain function.
<b>flat schematic</b>	a flat, single-level schematic containing only primitive symbols from the component libraries. A flat schematic can be either single or multiple page.
<b>gate</b>	a subset of a package, and corresponds to a part instance.
<b>global editing</b>	editing of a symbol, attribute or attribute value, saved in a library, and applied to all designs using that particular symbol.

<b>global port</b>	provides a connection to another global port of the same name anywhere in the schematic.
<b>gravity</b>	the property of a drawing object to snap to the nearest grid or pin when being placed on a drawing or moved about a drawing.
<b>gravity radius</b>	the distance between the cursor and an object on the schematic in which the object can be selected.
<b>grid</b>	a pattern of horizontal and vertical lines that aid in placing objects on a schematic or symbol drawing.
<b>hidden pins</b>	pins that are not connected by wires and buses, but through an attribute that names the net to which they belong.
<b>hierarchical design</b>	a design of more than one level wherein a portion of the design (lower-level schematic) is represented by a block or symbol on a higher-level schematic.
<b>hotspot</b>	the point at the end of a pin that forms a junction when it intersects with a wire or bus segment
<b>instance name</b>	a unique name for a part instance.
<b>interface port</b>	a port providing connectivity to the pins of hierarchical symbols or blocks.
<b>junction</b>	a graphical indication that wires, buses and/or pins are electrically connected.
<b>marker</b>	notations placed on a schematic to identify locations for observing voltage, current or digital signal levels or waveform behavior when the circuit is analyzed.
<b>message</b>	a character string generated by an application, describing some kind of condition, status or other information and displayed by the Message Viewer.
<b>model definition</b>	an underlying description of the electrical behavior of a part using a set of variable parameters. Used by PSpice and/or PLogic.
<b>msim.ini</b>	the initialization file, usually contained in the Windows directory, containing start-up and configuration information for MicroSim programs, including Schematics.
<b>navigation</b>	the process of moving between pages in a multi-sheet schematic or between levels in a hierarchical design.

<b>net</b>	a set of electrically connected part pins. A net may be <i>anonymous</i> or <i>named</i> . An anonymous net might be the junction of two resistors. A named net could be a wire labeled <code>CLOCK</code> connecting two digital parts.
<b>netlist</b>	a list providing the circuit definition and connectivity information in simulation netlist format.
<b>nodeset</b>	a symbol containing one or two pins, permitting you to initialize a node voltage for simulation.
<b>off-page port</b>	a port connecting pages of a schematic. Off-page ports may or may not contain a <code>LABEL</code> attribute.
<b>origin</b>	the point on a symbol designated for placing a part. When a symbol is rotated on the schematic, it is rotated about this point.
<b>package</b>	a physical device consisting of one or more gates.
<b>package library definition</b>	the use of a mathematical model to represent the physical operation of a circuit design.
<b>package type</b>	an attribute specifying the type of physical package that the actual circuit board will use. For example, DIP14, chip carrier, surface mount.
<b>package type class</b>	an attribute specifying grouping of similar package types. For example, DIP would be the class for all sizes of dip package types (DIP14, etc.).
<b>page</b>	one sheet of a multiple-sheet schematic. A page may contain both parts (represented by symbols), port instances, connectors and annotation symbols. A page may or may not have a title. Each schematic page represents a single page of a circuit design.
<b>part</b>	an electrical component which is represented by a schematic symbol. A part refers to the logical rather than the physical component.
<b>part definition</b>	See <i>symbol definition</i> .
<b>part description</b>	describes the symbol in terms of its <code>symbol type</code> , such as “2-input NAND.”
<b>part instance</b>	refers to an occurrence of a symbol in a schematic.
<b>part outline</b>	consists of the symbol for a part (graphics and pins), minus any text.

<b>pin</b>	contained in parts, ports and off-page connectors. Parts can contain multiple pins. Each part contains specific pin names associated with the part. Pins may connect to a wire, a bus or another pin.
<b>pin definition</b>	provides the pin number, the location of each pin relative to the symbol origin and the electrical attributes of the pin.
<b>pin name</b>	a name that uniquely identifies the pin on a part.
<b>pin number</b>	the physical device pin number.
<b>pin-to-pin spacing</b>	determines the size of the symbols as they appear on the printed page. The distance between pins is set during the initial installation, but may be changed.
<b>port</b>	provides connectivity across schematic pages. A port provides the anchor for a single pin. Ports are chosen from library files, placed, moved and deleted in the same way as are parts. Ports may have multiple connections. Ports consist of three types: global, interface and off-page (defined in this Appendix).
<b>primitive symbol</b>	a symbol that is an indivisible component for a specific netlist. That is, it is completely specified electrically for the purpose it is required to perform.
<b>reference designator</b>	an attribute used as a unique name on a given schematic level. For example, a resistor with the reference designator "R5," would indicate that it is the 5th instance of the resistor (R) on a particular schematic. For package parts, it consists of concatenation of the package's reference designator followed by the gate name assigned to the part. The reference designator is used as a base for the simulation netlist. Reference designators can be either automatically or manually assigned. Reference designators represent a unique name used to reference a physical device. Parts with the same reference designator are packaged into the same physical device.
<b>schematic</b>	a drawing consisting of the following components: one or more pages, a set of symbols representing local part definitions or parts in a library file and/or text.
<b>selection area</b>	when drawing or editing a schematic or symbol, the area identified and enclosed by a region-of-interest box for the purpose of performing some operation on the objects within the area.
<b>setpoint</b>	a special symbol used to specify initial node voltages during simulation.
<b>simulation</b>	the use of a mathematical model to represent the physical operation of a circuit design.

<b>stimulus</b>	symbols placed on a schematic to identify digital and analog voltage and signal sources used during simulation.
<b>symbol</b>	consists of the graphical representation of a logical or physical electronic part on the schematic. A symbol may have one or more associated attributes.
<b>symbol definition</b>	consists of the data from which the netlist is generated. A symbol or part definition consists of the following: the part name and any aliases, its attributes, primitive definition (also called the circuit definition) and pin definitions.
<b>symbolize</b>	creating a symbol to represent a schematic.
<b>translator</b>	(1) another name for a netlister (2) the process of reading a schematic created by another design program and converting it to a Schematics file.
<b>view</b>	a mechanism for allowing hierarchical symbols to have more than one underlying representation, for example, defining a flip-flop to have a transistor view or a gate view.
<b>viewpoint</b>	a special symbol used to display bias point voltages and currents during simulation.
<b>voltage viewpoint</b>	displays the bias point voltage at a pin. Any pin on part or a port may have a viewpoint attached.
<b>wire</b>	a graphical indication of a connection between pins, buses and other wires.



---

# Index

---

## A

a kind of, *See* AKO

ABM, [Glossary-1](#)

Add Text dialog box, [5-13](#)

adding

- library, [2-12](#)

- package type for a component, [5-32](#)

- page to design, [3-53](#)

- pins to symbol, [5-14](#)

- stimulus, [7-9](#)

- text to schematic, [3-49](#)

- text to symbol, [5-13](#)

Additional Info dialog box, [2-39](#)

advanced netlisting configuration items, [C-1](#)

AKO

- definition of, [Glossary-1](#)

- specifier, [9-25](#)

- symbols, [5-7](#)

AKO definition, [9-25](#)

alias

- definition of, [Glossary-1](#)

- using, [5-26](#)

analog behavioral model, *See* ABM

annotation

- back, [3-13](#), [9-28](#), [Glossary-1](#)

- definition of, [Glossary-1](#)

- page border, [2-4](#)

- symbol, [2-4](#), [3-49](#), [Glossary-1](#)

- title block, [2-4](#)

App Settings dialog box, [2-29](#), [2-31](#)

arc, drawing, [5-11](#)

assigning

- annotation, [3-13](#)

- attribute names, [D-2](#)

- attribute value, [3-18](#)

- instance-specific part values, [6-17](#)

- package types, [A-4](#)

- pin numbers, [5-35](#)

- pins, [5-27](#), [5-33](#)

- reference designator, [1-13](#), [3-22](#), [6-4](#), [9-7](#)

associating an existing schematic with a hierarchical block, [6-8](#)

attribute

- definition of, [Glossary-1](#)

- deleting, [3-14](#)

- editing, [3-12](#)

- enabling display, [3-15](#)

- global editing, [3-18](#)

- intrinsic property, [3-12](#)

- list, [D-1](#)

- non-changeable, [3-13](#)

- selecting, 3-41
- simulation, 7-3
- SWAP, 9-17
- system defined, 3-13
- text, 4-12
- value, 2-4, 3-12
- view, 6-13

Attribute Editing dialog box, 3-12

Attributes dialog box, 4-14, 5-37

auto-fit, 3-58, 4-21

automatic panning, 2-35

automatically assigning reference designators, 9-9

auto-naming, 1-12, 3-22, 3-23, 3-33

Auto-Naming dialog box, 1-12, 3-22

auto-repeat, 1-8, 3-20, 3-23, 5-18

Auto-Repeat dialog box, 3-20

autosave interval, 2-24

## B

Back Annotate dialog box, 9-29

back annotation, 3-13, 9-28, [Glossary-1](#)

backward ECO, applying, 9-21

base symbol, 5-8

behavioral model, 6-13

bill-of-materials report

- customizing, 9-14
- exporting, 9-16
- generating, 9-12
- printing, 9-13

block

- definition of, [Glossary-1](#)
- drawing, 2-7
- See also* hierarchical block

Block View dialog box, 6-8

board layout, preparing design for, 9-1

border symbol, changing, 2-20

bounding box, 5-23, [Glossary-2](#)

box, drawing, 5-11

breakout library, 7-6

browser

- library, 1-7, 1-15, 3-8
- part, 1-7, 1-15, 3-6

bundle, [Glossary-2](#)

bus

- definition of, [Glossary-2](#)
- drawing, 1-11, 2-7, 3-31
- labeling, 1-11, 3-31
- orthogonal, 3-34

- selecting, 3-40
- splitting, 3-32

## C

CADSTAR layout format, 9-23

Change Attribute dialog box, 3-15, 4-12

Change Pin dialog box, 4-14, 5-20, 6-7, 6-23

Change Text dialog box, 4-15, 5-13

changing

- application settings, 2-29
- attribute display, 3-15
- attribute value, 3-18
- border symbol, 2-20
- bounding box size, 5-23
- colors, 2-26
- default value, 3-19
- display characteristics, 3-16
- drawing area, 2-19
- free-standing text, 4-15
- gravity, 2-21, 3-36, 4-18
- grid, 2-21
- hierarchical block reference designator, 6-5
- library search order, 2-15
- page size, 2-18
- page title, 3-46
- part values, 1-13
- pin name text, 4-14
- pin number text, 4-14
- pin type, 5-20
- reference designators, 1-13
- search path, 2-16
- text characteristics, 4-12, 5-13
- text size, [A-7](#)
- value of attribute, 3-12

checking

- library configuration, 1-6
- library file index, [B-1](#)

checklist, simulation, 3-4

circle, drawing, 5-12

circuit, [Glossary-2](#)

clipboard

- copying to, 3-44
- pasting from, 3-45

closing

- message viewer, 2-39
- schematic editor, 3-63
- single schematic, 3-63
- symbol editor, 4-7

- colors, changing, 2-26
- component locations, 9-19
- component, *See* package
- components
  - aspects of, 4-3
  - description file, 9-15
  - multi-gate, 5-34
  - multiple gate types, 5-37
  - of design, 2-3
  - with more than one symbol, 5-37
- Configure Tools dialog box, 9-23
- configured package types list, 5-32
- configuring
  - application settings, 2-29
  - autosave interval, 2-24
  - colors, 2-26
  - display, 2-25
  - fonts, 2-17
  - library, 2-12
  - MicroSim Schematics, 2-11
  - msim.ini file, 2-30
  - netlist, C-1
  - page settings, 2-19
  - page size, 2-18
  - Select Part list size, 2-28
  - simulation library, 7-5
  - symbol libraries, 1-6, 2-12
- connecting
  - interface ports, 6-8
  - inter-page signals, 3-54
  - schematic pages, 2-4
  - signals via labels, A-8
  - wires to bus, 1-11, 3-32
- connections, 2-5, 3-31
- connectivity
  - implicit, 6-13
  - in multi-sheet design, 3-53
  - on-grid pins, 4-18
  - OrCAD schematics, A-8
  - orthogonal, 3-34
  - rubberbanding, 3-36
  - rules, 3-31
- connector
  - definition of, *Glossary-2*
  - how it differs from port, 9-3
  - placing, 9-3
  - symbol, 9-5
- conventions
  - mouse, xvii
  - typographical, xvii
- Copy Package Definition dialog box, 5-30
- Copy Page dialog box, 3-54
- Copy Part dialog box, 3-19, 5-5
- copying
  - between pages, 3-55
  - package definition, 5-30
  - page, 3-54
  - part, 3-19
  - selected object, 3-43, 5-18
  - symbol, 5-4
  - to clipboard, 3-44
- Create New Symbol dialog box, 5-4
- Create Page dialog box, 3-53
- creating
  - AKO symbol, 5-9
  - annotation items, 3-49
  - base symbol, 5-8
  - batch library files, B-1
  - connections between pages, 3-54
  - connector symbols, 9-5
  - custom title block, 3-48
  - design, 1-2
  - design for board layout, 7-3
  - design for simulation, 7-3
  - ground symbol, 3-28
  - hierarchical block, 6-4
  - hierarchical design, 1-2, 6-20
  - hierarchical symbols, 6-9
  - interface ports, 6-6, 6-8
  - layout format, 9-23
  - multiple-gate components, 5-37
  - multi-sheet design, 3-53
  - new attribute, 3-12
  - new page, 3-53
  - new symbol, 5-3
  - package definition, 5-28, A-4
  - power symbol, 3-28
  - schematic for hierarchical block, 6-5
  - symbol, 1-2
  - title block, 3-46
- cross probing, xxii, 9-20
- current sensor, *Glossary-2*
- current stimulus source, 7-9
- customizing bill-of-materials report, 9-14
- cutting
  - between pages, 3-55
  - selected object, 3-43, 5-17

## D

### default

- package types, [A-4](#)
- pin type, [5-14](#), [5-19](#)
- symbol attribute, [3-19](#)
- view, [6-13](#)

### defining

- connector package, [9-5](#)
- gate names, [5-34](#)
- hidden pins, [5-22](#)
- number of gates, [5-34](#)
- pin number assignments, [5-35](#)
- pin types, [5-19](#)
- shared power and ground pins, [5-35](#)
- stimulus, [7-9](#)

### definition

- AKO, [9-25](#)
- model, [3-4](#), [Glossary-3](#)
- package, [3-4](#), [9-6](#)
- package library, [Glossary-4](#)
- part, [Glossary-4](#)
- pin, [Glossary-5](#)

### deleting

- attribute, [3-14](#)
- package definition, [5-39](#)
- page, [3-56](#)
- pin on hierarchical block, [6-7](#)
- selected object, [3-43](#), [5-17](#)

### de-selecting objects, [3-41](#)

### design

- components of, [2-3](#)
- creating, [1-2](#)
- definition of, [Glossary-2](#)
- editing, [1-2](#)
- hierarchical, [6-3](#), [Glossary-3](#)
- methods, [6-3](#)
- multi-sheet, [3-53](#)
- packaging parts in, [9-6](#)
- printing, [3-57](#)
- starting new, [1-6](#)

design synthesis language block, *See* DSL block

device, *See* package

### dialog boxes, [6-5](#)

- Add Text, [5-13](#)
- Additional Info, [2-39](#)
- App Settings, [2-29](#), [2-31](#)
- Attribute Editing, [3-12](#)
- Attributes, [4-14](#), [5-37](#)
- Auto-Naming, [1-12](#), [3-22](#)

Auto-Repeat, [3-20](#)

Back Annotate, [9-29](#)

Block View, [6-8](#)

Change Attribute, [3-15](#), [4-12](#)

Change Pin, [4-14](#), [5-20](#), [6-7](#), [6-23](#)

Change Text, [4-15](#), [5-13](#)

Configure Tools, [9-23](#)

Copy Package Definition, [5-30](#)

Copy Page, [3-54](#)

Copy Part, [3-19](#), [5-5](#)

Create New Symbol, [5-4](#)

Create Page, [3-53](#)

Display Colors, [2-21](#)

Display Level, [2-25](#)

Display Options, [2-21](#), [3-35](#), [4-9](#)

Edit Attributes, [9-17](#)

Edit Gate Types, [5-34](#)

Edit Package Definition, [5-28](#)

Edit Package Types, [5-32](#)

Edit Reference Designator, [1-13](#), [6-5](#), [9-7](#)

Editor Configuration, [2-12](#), [5-41](#)

Export Parts, [5-7](#)

Find, [3-42](#)

Font, [2-17](#)

Get Package Definition, [5-31](#)

Global Edit Attributes, [3-18](#)

Import, [5-6](#)

Import OrCAD File, [A-2](#)

Library Browser, [1-7](#), [3-8](#), [3-39](#)

Library Settings, [2-12](#), [5-41](#)

Package, [9-8](#)

Package Definition, [5-28](#)

Package Types, [A-4](#)

Page Info, [3-46](#)

Page Settings, [2-19](#)

Page Size, [2-18](#)

Pan & Zoom, [2-35](#)

Part Browser, [1-7](#), [3-6](#), [6-21](#)

Pin Assignments, [5-33](#)

Pin Swaps, [5-36](#)

Pin Type, [5-19](#)

Place Text, [3-49](#)

Print, [3-57](#), [4-21](#)

Remove Package Definition, [5-39](#)

Replace Part, [3-25](#)

Report Setup, [9-14](#)

Reports, [9-13](#)

Select Page, [3-55](#)

Set Attribute Value, [1-10](#), [3-30](#), [6-18](#)

Set Up Block, [6-24](#)

- Setup Package Class Priorities, 9-10
- Shared Pin Assignments, 5-35
- Translators, 6-14
- Where, 6-16
- digital stimulus source, 7-9
- disabling
  - auto-repeat, 3-21
  - snap-to-grid, 2-21, 4-16
  - snap-to-pin, 2-22
  - stay-on-grid, 2-22, 4-17
  - toolbar display, 2-7, 4-9
- display characteristics, 4-12
- Display Colors dialog box, 2-21
- Display Level dialog box, 2-25
- display levels, setting, 2-25
- display map, Glossary-2
- Display Options dialog box, 2-21, 3-35, 4-9
- distinction between connectors and ports, 9-3
- Draw Arc icon, 5-11
- Draw Block icon, 6-4
- Draw Box icon, 5-11
- Draw Bus icon, 1-11, 3-23
- Draw Circle icon, 5-12
- Draw Line icon, 5-12
- Draw Text icon, 3-49, 5-13
- Draw Wire icon, 1-10, 3-24, 6-23
- drawing
  - additional pages, 3-53
  - arc, 5-11
  - area, 2-19
  - block, 2-7
  - box, 5-11
  - bus, 1-11, 2-7, 3-31
  - circle, 5-12
  - connections, 3-32
  - custom power and ground symbols, 3-28
  - line, 5-12
  - lower-level schematic, 6-24
  - options, 3-34
  - orthogonal wires and buses, 3-34
  - pins, 5-14
  - port, 3-39
  - symbol graphics, 5-11
  - text, 2-7, 3-49
  - top-level schematic, 6-20
  - wire, 1-10, 2-7, 3-29
- drawing area
  - border, 2-20
  - changing, 2-19
- DSL block, 8-4

## E

- ECO, 9-21, 9-28, Glossary-2
- ECO File formats, 9-23
- EDIF 2 0 0, 9-23
- Edit Attributes dialog box, 9-17
- Edit Attributes icon, 3-18, 5-37, 6-6
- Edit Gate Types dialog box, 5-34
- Edit Package Definition dialog box, 5-28
- Edit Package Types dialog box, 5-32
- Edit Reference Designator dialog box, 1-13, 6-5, 9-7
- Edit Symbol icon, 3-19
- editing
  - annotation items, 3-49
  - annotation text, 3-50
  - attribute, 3-12, 3-18
  - bounding box, 5-23
  - bus label, 3-32
  - design, 1-2
  - global, 3-18
  - hidden pins, 5-22
  - hierarchical block, 6-4
  - hierarchical design, 1-2
  - hierarchical symbols, 6-9
  - instance of schematic, 6-17
  - msim.ini file, 2-26
  - multi-sheet design, 3-53
  - package types, 5-31
  - page title, 3-46
  - part attribute, 3-12
  - part origin, 5-23
  - pin name on hierarchical block, 6-6
  - pin numbers, 5-33
  - pin types, 5-19
  - schematic definition, 6-17
  - simulation models, 7-8
  - symbol, 1-2
  - symbol default attribute, 3-19
  - title block, 3-46
- Editor Configuration dialog box, 2-12, 5-41
- electrical rules check, *See* ERC
- elements of a symbol, 5-11
- enabling
  - automatic panning, 2-35
  - auto-repeat, 3-20
  - grid display, 2-21, 4-16
  - orthogonal drawing, 3-35
  - pin swapping, 5-36
  - rubberbanding, 3-37
  - snap-to-grid, 2-21, 3-35, 4-16

- snap-to-pin, 2-22, 3-36
- stay-on-grid, 2-22, 4-17
- text grid, 2-23, 4-19
- toolbar display, 2-7, 4-9
- ERC, [Glossary-2](#)
- error message handling, [xxi](#)
- examples
  - creating hierarchical design, 6-20
  - drawing a schematic, 1-4
  - using auto-naming, 3-23
  - using auto-repeat, 3-23
- Export Parts dialog box, 5-7
- exporting
  - bill-of-materials report, 9-16
  - symbol, 5-7

## F

- file
  - opening, 2-7
  - saving, 2-7
- fileset, [Glossary-2](#)
- Find dialog box, 3-42
- finding
  - most recently placed part, [xxi](#), 3-5
  - part, 3-4, 3-42
- fitting view to page, 2-34
- flat schematic, [Glossary-2](#)
- flipping
  - already-drawn object, 3-11
  - area of schematic, 3-11
  - part, 3-10
  - symbol element, 5-15
- Font dialog box, 2-17
- fonts, changing, 2-17
- footprints, *See* package types
- forward ECO, applying, 9-22
- function keys, 2-9, 4-11

## G

- gate
  - definition of, [Glossary-2](#)
- Get Package Definition dialog box, 5-31
- global
  - port, [Glossary-3](#)
- Global Edit Attributes dialog box, 3-18
- global editing, [Glossary-2](#)
- global editing of attributes, 3-18

- global library, 1-6
- global port
  - placing, 3-38, 6-10
  - symbol, 3-27, 3-38
  - using, 3-38
- graphical representation, 3-4
- gravity
  - changing, 3-36, 4-18
  - definition of, [Glossary-3](#)
  - setting, 2-21
  - specifying, 2-23
- gravity radius, [Glossary-3](#)
- grid
  - definition of, [Glossary-3](#)
  - enabling, 3-35, 4-16
  - settings, 2-21
  - size, 3-35
  - snap to, 2-21, 3-35, 4-16
  - spacing, 2-23, 3-35, 4-18
  - stay on, 2-22, 3-35, 4-17
- ground symbols, placing, 1-16, 3-27

## H

- help, on-line, [xx](#)
- hidden pins, 2-5, 5-22, [Glossary-3](#)
- hierarchical block
  - converting to symbol, 6-11
  - creating, 6-4
  - creating a schematic for, 6-5
  - editing, 6-4
  - pin names, 6-6
  - reference designator, 6-4
  - resizing, 6-5
  - selecting, 6-15
- hierarchical design
  - creating, 1-2, 6-20
  - definition of, [Glossary-3](#)
  - editing, 1-2
  - methods, 6-3
  - navigating through, 6-15
  - passing information between levels, 6-18
- hierarchical parts, 2-3
- hierarchical symbol
  - creating, 6-9
  - selecting, 6-15
- horizontal offset, 3-21
- hotspot, [Glossary-3](#)
- HP users, [xvii](#)

**I**

## icons

- Analysis Setup, 2-8
- Analysis Simulate, 2-8
- Draw Arc, 4-9, 5-11
- Draw Block, 2-7, 6-4
- Draw Box, 4-9, 5-11
- Draw Bus, 1-11, 2-7, 3-23
- Draw Circle, 4-9, 5-12
- Draw Line, 4-10, 5-12
- Draw Text, 2-7, 3-49, 4-10, 5-13
- Draw Wire, 1-10, 2-7, 3-24, 6-23
- Edit Attributes, 2-8, 3-18, 4-10, 5-37, 6-6
- Edit Symbol, 2-8, 3-19
- Fit to Page, 4-9
- Get New Part, 4-10
- More Info, 2-39
- New File, 2-7, 3-3, 4-9
- Open File, 2-7, 3-3, 4-9
- Place Pins, 4-10, 5-14
- Print, 2-7, 3-57
- Redraw, 2-8, 2-10, 4-8, 4-10
- Save File, 2-7, 4-9, 6-24
- Select Part, 1-7, 2-8, 3-6, 6-21
- Start Wizard, 4-10, 5-4
- View Area, 2-7, 2-32, 4-9
- View Fit, 2-7, 2-34
- Zoom In, 2-7, 2-32, 4-9
- Zoom Out, 2-7, 2-32, 4-9

Import dialog box, 5-6

Import OrCAD File dialog box, A-2

## importing

- into Microsoft Word, 3-45
- OrCAD SDT file, A-2
- OrCAD SDT schematic, A-1

indicated severity of message, 2-38

input port, 6-6

instance name, Glossary-3

interface port, 3-38, 6-12, Glossary-3

## interfacing

- to MicroSim PC Boards, 9-18
- to other board layout products, 9-23

**J**

joining pins and wires, 2-5

junction, 2-5, Glossary-3

**K**

keyboard, 2-9, 4-11

**L**

label template, 3-33

## labeling

- bus, 1-11, 3-31
- global port, 3-39
- ports, 1-16
- wire, 1-10, 1-12, 3-33

## layout

- netlist, 9-24

## layout netlist

- creating, 9-24
- file formats, 9-23
- format, 9-23
- mapping files, 9-24

## library

- breakout, 7-6
- browser, 3-8
- configuring, 2-12
- custom, 3-19
- global, 2-12
- list file, B-1
- list of, E-1
- local, 2-12
- model, 3-4
- name, 5-42
- package, 3-4
- path, 5-42
- removing, 2-14
- salvaging corrupt, B-1
- simulation model, 7-5
- symbol, 2-12, 3-4, 5-5, E-1

Library Browser dialog box, 1-7, 3-8, 3-39

library expansion and compression utility, B-1

Library Settings dialog box, 2-12, 5-41

line, drawing, 5-12

## locating

- new library, 2-13
- pointer on schematic, 1-8, 2-9
- source of message, 2-38

## M

main functions of MicroSim Schematics, 1-2

main window, 2-6

mapping files, 9-24

marker

- attaching, 9-3

- definition of, [Glossary-3](#)

- using, 7-11

menus, 2-6, 4-8

message, [Glossary-3](#)

message viewer

- additional information, 2-39

- closing, 2-39

- using, 2-37

mirroring, *See* flipping

model

- behavioral, 6-13

- library, 3-4

- name, D-2

- simulation, 7-4, E-1

model definition, 3-4, [Glossary-3](#)

More Info icon, 2-39

mouse conventions, xvii

moving

- down in hierarchy, 6-15

- interface port symbols, 6-6

- object on schematic, 3-41

- parts, 1-14

- symbol element, 5-17

- text, 1-14

- to top of hierarchy, 6-15

- up in hierarchy, 6-15

- wires, 1-14

msim.ini

- definition of, [Glossary-3](#)

- editing, 2-30

- i option, 2-30

## N

navigating

- moving down in hierarchy, 6-15

- moving up in hierarchy, 6-15

- through hierarchical designs, 6-15

- to top of hierarchy, 6-15

navigation, [Glossary-3](#)

net, [Glossary-4](#)

netlist

- definition of, [Glossary-4](#)

- layout, 9-24

- simulation, 7-10

netlist names, [Glossary-1](#)

netlisting

- configuration items, C-1

- EDIF, C-4

- levels of schematics, 6-18

- preferences, C-3

new features of release, [xxi](#)

new file, 2-7

New File icon, 3-3

node names

- netlisting preferences, C-3

nodeset, [Glossary-4](#)

## O

off-page port

- definition of, [Glossary-4](#)

- symbol, 3-38

- using, 3-38

offset

- horizontal, 3-21

- vertical, 3-21

on-line help, [xx](#), 2-38

Open File icon, 3-3

opening

- existing file, 3-3

- file, 2-7

- new file, 3-3

options

- auto-fit, 3-58

- auto-naming, 1-12, 3-33

- auto-repeat, 1-8, 3-20

- configuration, 2-11

- drawing, 3-34

- editor configuration, 1-6

- OrCAD importing, A-3

- orthogonality, 3-34

- rubberbanding, 3-36

- scaling, 4-21

- snap-to-grid, 3-35, 4-16

- snap-to-pin, 3-36

- stay-on-grid, 3-35, 4-17

- text stay-on-grid, 4-19

OrCAD

- differences between OrCAD SDT and Schematics, [A-9](#)



- import options, [A-3](#)
- importing, [A-1](#)
- origin, [5-22](#), [Glossary-4](#)
- orthogonal drawing, [3-34](#)
- other MicroSim documents, [xix](#)
- output port, [6-6](#)

## P

- package
  - definition of, [Glossary-4](#)
  - information, [5-27](#)
  - library, [3-4](#)
- package class priorities, [9-10](#)
- package definition
  - contents, [9-6](#)
  - copying, [5-30](#)
  - creating, [5-28](#), [A-4](#)
  - current package library, [5-31](#)
  - current symbol, [5-31](#)
  - deleting, [5-39](#)
  - editing, [5-28](#)
  - how used, [9-6](#)
  - name, [D-2](#)
  - single-gate, [5-29](#)
  - where stored, [3-4](#)
- Package Definition dialog box, [5-28](#)
- Package dialog box, [9-8](#)
- package library definition, [Glossary-4](#)
- package type class, [Glossary-4](#)
- package types
  - assigning, [A-4](#)
  - definition of, [Glossary-4](#)
  - editing, [5-31](#)
  - list of, [5-27](#)
  - multiple, [5-32](#)
  - per pin assignment, [5-32](#)
  - symbol, [4-4](#)
- Package Types dialog box, [A-4](#)
- packager, [9-6](#)
- packaging
  - automatically, [9-8](#)
  - information, [4-3](#)
  - parts in design, [9-6](#)
  - reference designators, [9-9](#)
- PADS layout, [9-23](#)
- page
  - changing settings, [2-19](#)
  - changing size, [2-18](#)

- connecting, [3-54](#)
  - copying, [3-54](#)
  - creating new, [3-53](#)
  - definition of, [Glossary-4](#)
  - deleting, [3-56](#)
  - viewing multiple, [3-55](#)
- page border, [2-4](#)
- Page Info dialog box, [3-46](#)
- Page Settings dialog box, [2-19](#)
- Page Size dialog box, [2-18](#)
- page title, [3-46](#)
- Pan & Zoom dialog box, [2-35](#)
- panning
  - automatic, [2-35](#)
  - new center, [2-34](#)
- part
  - auto-repeat, [3-20](#)
  - copying, [3-19](#)
  - definition of, [Glossary-4](#)
  - finding, [3-4](#), [3-42](#)
  - flipping, [3-10](#)
  - graphical representation, [2-3](#)
  - instance, [3-14](#), [Glossary-4](#)
  - moving, [1-14](#), [3-41](#)
  - outline, [Glossary-4](#)
  - packaging, [5-27](#)
  - placing, [3-9](#)
  - previously selected, [3-5](#)
  - replacing, [3-25](#)
  - rotating, [3-10](#)
  - selecting, [3-5](#), [3-40](#)
- part browser, [3-6](#)
- Part Browser dialog box, [1-7](#), [3-6](#), [6-21](#)
- part definition, [Glossary-4](#)
- part description, [Glossary-4](#)
- part value, changing, [1-13](#)
- parts
  - hierarchical, [2-3](#)
  - primitive, [2-3](#)
- passing information between levels of hierarchy, [6-18](#)
- pasting
  - between pages, [3-55](#)
  - selected object, [3-44](#), [5-18](#)
- P-CAD layout, [9-23](#)
- pin
  - assignments, [5-32](#)
  - definition of, [Glossary-5](#)
  - drawing, [5-14](#)
  - hidden, [Glossary-3](#)
  - name, [5-33](#), [Glossary-5](#)

- name and number, 4-14
- number, [Glossary-5](#)
- shared power and ground, 5-35
- snap to, 2-22
- swapping, 5-36, 9-17
- text characteristics, 4-14
- pin assignment list, 5-27
- Pin Assignments dialog box, 5-33
- pin definition, [Glossary-5](#)
- pin numbers
  - assigning, 5-35
  - determining, 9-6
  - editing, 5-33
  - specifying physical, 5-33
- Pin Swaps dialog box, 5-36
- Pin Type dialog box, 5-19
- pin types
  - available, 5-19
  - changing, 5-20
  - defining, 5-19
- pins
  - broken, 7-4
  - hidden, 2-5
  - unmodeled, 3-4, 7-4
- pin-to-pin spacing, [Glossary-5](#)
- Place Pins icon, 5-14
- Place Text dialog box, 3-49
- placing
  - connectors, 9-3
  - global port, 3-39
  - global ports, 6-10
  - ground symbols, 1-16, 3-27
  - interface ports, 6-9
  - parts, 1-7, 3-9
  - ports, 1-15
  - power symbols, 1-16, 3-27
  - previously selected part, 3-5
  - repeating, 3-20
  - resistors, 1-8
  - stimulus sources, 7-9
  - symbol, 3-9
  - voltage source, 6-21
- PLD, 8-1
- popping, 6-15
- port
  - definition of, [Glossary-5](#)
  - global, 3-38, [Glossary-3](#)
  - how it differs from connector, 9-3
  - interface, 6-6, 6-12, [Glossary-3](#)
  - off-page, 3-38, 3-54, 9-3, [Glossary-4](#)

- where used, 2-4
- power symbols
  - custom, 3-28
  - placing, 1-16, 3-27
- prerequisite to drawing schematic, 1-2
- primitive parts, 2-3
- primitive symbol, 6-9, [Glossary-5](#)
- Print dialog box, 3-57, 4-21
- Print icon, 3-57
- printing
  - bill-of-materials report, 9-13
  - design, 3-57
  - schematic, 3-57
  - selected area, 3-57
  - symbol, 4-21
- programmable logic
  - simulating design of, 8-5
  - symbols, 8-7
  - targeting parts for, 8-3
- programmable logic devices, *See* PLD
- Protel layout format, 9-23
- pushing, 6-3, 6-15

## R

- Redraw icon, 2-10, 4-8
- reference designator
  - assigning, 9-7
  - auto-naming, 3-22
  - changing, 1-13
  - definition of, [Glossary-5](#)
  - value, [D-3](#)
- region of interest box, *See* ROI box
- related documentation, [xix](#)
- Remove Package Definition dialog box, 5-39
- removing
  - configured library, 2-14
  - package definition, 5-39
- repeating, part placement, 3-20
- Replace Part dialog box, 3-25
- replacing
  - multiple parts, 3-25
  - parts, 3-25
  - single part, 3-25
- Report Setup dialog box, 9-14
- Reports dialog box, 9-13
- revising, text string, 5-13
- rewiring, 3-30
- ROI box, 3-41

rotating  
     already-drawn object, 3-10  
     area of schematic, 3-10  
     part, 1-8, 3-10  
     symbol element, 5-15  
 rubberbanding, 3-36  
 rules of connectivity, 3-31

## S

salvaging a corrupted library file, B-1

Save File icon, 6-24

saving

    bill-of-materials report, 9-13  
     file, 2-7, 6-24, 6-27  
     schematic, 1-17  
     symbol, 5-8

scaling, 3-58, 4-21

schematic

    associating with hierarchical block, 6-8  
     definition of, [Glossary-5](#)  
     editing definition, 6-17  
     editing instance of, 6-17  
     locating in hierarchy, 6-16  
     symbolizing, 6-10  
     translating, 6-13

schematic editor

    closing, 3-63  
     using, 2-1

search capability of part browser, [xxi](#)

search criteria, 3-42

searching

    for parts, 3-6, 3-41  
     for schematic in hierarchy, 6-16

Select Page dialog box, 3-55

Select Part icon, 3-6, 6-21

Select Part list size, 2-28

selecting

    area of schematic, 3-40  
     area of symbol drawing, 5-16  
     area of view, 2-32  
     attribute of object, 3-41  
     auto-repeat, 3-20  
     element of drawing, 5-16  
     library, 3-8  
     multiple objects, 3-40  
     objects on schematic, 3-40  
     part by description, 3-7  
     part by name, 3-5

    part from symbol library, 3-8

selection area, [Glossary-5](#)

Set Attribute Level dialog box, 6-18

Set Attribute Value dialog box, 1-10, 3-30

Set Up Block, 6-5

Set Up Block dialog box, 6-5, 6-24

setpoint, [Glossary-5](#)

setting

    autosave interval, 2-24

    display levels, 2-25

    gravity, 4-18

    grid spacing, 3-35, 4-18

    package class priorities, 9-10

    text grid size, 4-19

Setup Package Class Priorities dialog box, 9-10

Shared Pin Assignments dialog box, 5-35

simulation

    attaching markers, 9-3

    attribute, 7-3

    checklist, 3-4

    definition of, [Glossary-5](#)

    include information, A-3

    models, 3-4, 4-4, 7-4, E-1

    of programmable logic design, 8-5

    parts designated for, D-4

snap-to-grid, 2-21, 4-16

snap-to-pin, 2-22, 3-36

source

    current stimulus, 7-9

    digital stimulus, 7-9

    voltage stimulus, 7-9

spacing, pin-to-pin, [Glossary-5](#)

splitting, bus, 3-32

Start Wizard icon, 5-4

starting

    library expansion and compression utility, B-3

    new design, 1-6

    schematic editor, 3-3

    simulator, 7-10

    symbol editor, 4-5

status bar, 2-9

stay-on-grid, 2-22, 3-35, 4-17

stimulus

    adding, 7-9

    defining, 7-9

    definition of, [Glossary-6](#)

    source, 3-4

Stimulus Editor, 7-10

subcircuit definition, 7-6

swapping pins, 9-17

### symbol

- adding pins to, 5-14
- annotation, 2-4, [Glossary-1](#)
- as element of a component, 4-3
- base, 5-8
- connector, 9-5
- converting from block, 6-11
- copying, 5-4
- creating, 1-2, 5-3
- default attribute, 3-19
- definition, [Glossary-6](#)
- definition of, [Glossary-6](#)
- editing, 1-2, 3-19
- elements of, 5-11
- exporting, 5-7
- for simulation model, 7-6
- global port, 3-27
- ground, 3-27
- hierarchical, 6-9
- library, 2-12, 3-4, 5-5, E-1
- placing, 3-9
- port, 2-4
- power, 3-27
- primitive, 6-9, [Glossary-5](#)
- printing, 4-21

### symbol attribute

- COMPONENT, D-2
- GATE, D-2
- GATETYPE, D-2
- MODEL, D-2
- PART, D-3
- PKGREF, D-3
- PKGTYPE, D-3
- REFDES, D-3
- SIMULATION- ONLY, D-4
- TEMPLATE, D-4

### Symbol Creation Wizard, 5-3

### symbol editor

- automatically starting, 4-7
- closing, 4-7
- starting, 4-5
- window, 4-8

### symbol libraries

- table of, E-1
- See also* library

### symbolize, 6-9, [Glossary-6](#)

### symbolizing a schematic, 6-10

### syntax

- bus labeling, 3-31

## T

### TangoPro layout format, 9-23

### targeting parts for programmable logic, 8-3

### text

- adding, 3-49
- adding to symbol, 5-13
- annotation, 3-49
- attribute, 4-12
- characteristics, 4-12
- drawing, 2-7
- free-standing, 4-15
- grid, 2-23
- moving, 1-14
- stay-on-grid, 4-19

### title bar, 4-10

### title block

- changing attributes, 3-47
- creating, 3-46
- creating a custom, 3-48
- editing, 3-46
- use, 2-4

### toolbar, 2-7, 4-9

### top, 6-15

### trace properties, 9-18

### translating

- hierarchical schematics, A-6
- large designs, A-7
- multi-page schematics, A-6
- OrCAD SDT to MicroSim Schematics, A-1
- schematic, 6-13
- setting up associated view, 6-14

### translator, 6-13, [Glossary-6](#)

### Translators dialog box, 6-14

### tutorial, *See* examples

### typographical conventions, xvii

## U

### undeleting, 3-44, 5-18

### UNIX users, xvii

### unmodeled pins, 3-4, 7-4

### updating schematic with PLDs, 8-6

### user defined component information, 9-15

### using

- AKO symbols, 5-7
- auto-naming, 3-23
- auto-repeat, 3-23
- back annotation, 9-29

- global ports, 3-38
- interface ports, 6-12
- marker, 7-11
- message viewer, 2-37
- MicroSim PLSyn, 8-6
- schematic editor, 2-1
- Stimulus Editor, 7-10
- symbol in schematic, 5-22
- this guide, xvii

#### utilities

- library compression, B-1
- library expansion, B-1
- running, B-3

## V

#### value

- instance-specific part, 6-17
- of attribute, 2-4

#### vertical offset, 3-21

#### view

- associated, 6-14
- behavioral model, 6-13
- definition of, Glossary-6
- setting up, 6-14
- setting up multiple, 6-13

#### View Area icon, 2-32

#### View Fit icon, 2-34

#### viewing

- multiple pages, 3-55
- page, 2-34
- selected area, 2-32
- simulation model, 7-4
- simulation results, 7-11

#### viewpoint, Glossary-6

#### voltage stimulus source, 7-9

#### voltage viewpoint, Glossary-6

- labeling, 1-10, 1-12, 3-33
- moving, 1-14
- orthogonal, 3-34
- rewiring segment, 3-30
- selecting, 3-40
- wizard, symbol creation, 5-3

## Z

#### zoom factor, 3-59

#### Zoom In icon, 2-32

#### Zoom Out icon, 2-32

#### zooming, 2-32

## W

#### Where dialog box, 6-16

#### where to find programs, 2-29

#### wildcard characters, 3-6, 3-7, 3-42

#### window

- main, 2-6
- symbol editor, 4-5, 4-8

#### wire

- definition of, Glossary-6
- drawing, 1-10, 2-7, 3-29