

MicroSim PSpice & Basics

Circuit Analysis Software

User's Guide


MicroSim[®]
MicroSim Corporation
20 Fairbanks
Irvine, California 92618
(714) 770-3022

Version 8.0, June, 1997.

Copyright 1997, MicroSim Corporation. All rights reserved.
Printed in the United States of America.

TradeMarks

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

Additional marks of MicroSim include: "StmEd," "Stimulus Editor," "Probe," "Parts," "Monte Carlo," "Analog Behavioral Modeling," "Device Equations," "Digital Simulation," "Digital Files," "Filter Designer," "Schematics," "PLogic," "PCBoards," "PSpice Optimizer," and "PLSyn" 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.

Mathcad copyright © 1991-1997 by Mathsoft, Inc.

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. Corporate Program Licensing and multiple copy discounts are available.

Technical Support

Internet Tech.Support@MicroSim.com

Phone (714) 837-0790

FAX (714) 455-0554

WWW <http://www.microsim.com>

Customer Service

Internet Sales@MicroSim.com

Phone (714) 770-3022

Contents

Before You Begin

Welcome to MicroSim	xxiii
MicroSim PSpice Overview	xxiv
How to Use this Guide	xxv
Typographical Conventions	xxv
Related Documentation	xxvi
Online Help	xxvii
If You Have PSpice Basics and Not the Standard Package	xxviii
What's New	xxx

Part One Simulation Primer

Chapter 1 Things You Need to Know

Chapter Overview	1-1
What is PSpice?	1-2
Analyses You Can Run with PSpice	1-3
Basic Analyses	1-3
DC sweep & other DC calculations	1-3
AC sweep and noise	1-4
Advanced Multi-Run Analyses	1-5
Parametric and temperature	1-5
Monte Carlo and sensitivity/worst-case	1-6
Using PSpice with Other MicroSim Programs	1-7
Using Schematics to Prepare for Simulation	1-8
What is the Stimulus Editor?	1-8
What is the Parts Utility?	1-9
What is Probe?	1-9
Files Needed for Simulation	1-10
Files That Schematics Generates	1-10
Netlist file	1-11

Circuit file	1-11
Other Files That You Can Configure for Simulation	1-11
Model library	1-12
Stimulus file	1-13
Include file	1-13
Configuring model library, stimulus, and include files	1-13
Files That PSpice Generates	1-14
Probe data file	1-14
PSpice output file	1-15

Chapter 2 Simulation Examples

Chapter Overview	2-1
Example Circuit Creation	2-2
Finding Out More about Setting Up Your Schematic	2-5
Bias Point Analysis	2-6
Running PSpice	2-6
Using the Bias Information Display	2-7
Using the Simulation Output File	2-9
Finding Out More about Bias Point Calculations	2-10
DC Sweep Analysis	2-10
Setting Up and Running a DC Sweep Analysis	2-10
Displaying DC Analysis Results in Probe	2-11
Finding Out More about DC Sweep Analysis	2-15
Transient Analysis	2-16
Finding Out More about Transient Analysis	2-19
AC Sweep Analysis	2-20
Setting Up and Running an AC Sweep Analysis	2-20
AC Sweep Analysis Results	2-22
Finding Out More about AC Sweep and Noise Analysis	2-23
Parametric Analysis	2-24
Setting Up and Running the Parametric Analysis	2-25
Analyzing Waveform Families in Probe	2-27
Finding Out More about Parametric Analysis	2-29
Probe Performance Analysis	2-30
Finding Out More about Performance Analysis	2-32

Part Two Design Entry

Chapter 3 Preparing a Schematic for Simulation

Chapter Overview	3-1
Checklist for Simulation Setup	3-2
Typical Simulation Setup Steps	3-2
.	3-3
Advanced Design Entry and Simulation Setup Steps	3-4
When Netlisting Fails or the Simulation Does Not Start	3-4
Things to check in your schematic	3-5
Things to check in your system configuration	3-5
Using Parts That You Can Simulate	3-6
Vendor-Supplied Parts	3-7
Part naming conventions	3-7
Finding the part that you want	3-8
Passive Parts	3-10
Breakout Parts	3-11
Behavioral Parts	3-12
Using Global Parameters and Expressions for Values	3-12
Global Parameters	3-12
Declaring and using a global parameter	3-13
Expressions	3-14
Specifying expressions	3-14
Defining Power Supplies	3-18
Defining Stimuli	3-19
Using VSTIM and ISTIM	3-20
If you want to specify multiple stimulus types	3-20
Using VSRC or ISRC symbols	3-21
Things to Watch For	3-22
Unmodeled Parts	3-22
Do this if the part in question is from the MicroSim libraries	3-22
Check for this if the part in question is custom-built	3-24
Unconfigured Model, Stimulus, or Include Files	3-24
Check for this	3-25
Unmodeled Pins	3-25
Check for this	3-26
Missing Ground	3-26
Check for this	3-26
Missing DC Path to Ground	3-27
Check for this	3-27

Chapter 4 Creating and Editing Models

Chapter Overview	4-1
What Are Models?	4-3
Models defined as model parameter sets	4-3
Models defined as subcircuit netlists	4-3
How Are Models Organized?	4-4
Model Libraries	4-4
Model Library Configuration	4-5
Global vs. Local Models and Libraries	4-5
Nested Model Libraries	4-6
MicroSim-Provided Models	4-6
Tools to Create and Edit Models	4-7
Ways to Create and Edit Models	4-8
Using the Parts Utility to	
Edit Models	4-10
Ways to Use the Parts Utility	4-11
Parts-Supported Device Types	4-12
Ways To Characterize Models	4-13
Creating models from data sheet information	4-13
Analyzing the effect of model parameters on device characteristics	4-14
How to Fit Models	4-14
Running the Parts Utility Alone	4-16
Starting the Parts utility	4-16
Enabling and disabling automatic symbol creation	4-16
Saving global models (and symbols)	4-17
Running the Parts Utility from the Symbol Editor	4-18
Starting the Parts utility	4-18
Saving global models	4-19
Running the Parts Utility from the Schematic Editor	4-20
What is an instance model?	4-20
Starting the Parts utility	4-21
Saving local models	4-21
What happens if you don't save the instance model	4-22
The Parts Utility Tutorial	4-23
Creating the half-wave rectifier schematic	4-23
Starting the Parts utility for the D1 diode	4-24
Entering data sheet information	4-24
Extracting model parameters	4-27
Adding curves for more than one temperature	4-28
Completing the model definition	4-29
Using the Model Editor	4-29

Changing Model Properties	4-30
Editing .MODEL definitions	4-30
Editing .SUBCKT definitions	4-31
Changing the model name	4-31
Running the Model Editor from the Symbol Editor	4-31
Starting the model editor	4-31
Saving global models	4-32
Running the Model Editor from the Schematic Editor	4-33
What is an instance model?	4-33
Starting the model editor	4-34
Saving local models	4-34
Example: Editing a Q2N2222 Instance Model	4-35
Starting the model editor	4-35
Editing the Q2N2222-X model instance	4-35
Saving the edits and updating the schematic	4-36
Using the Create Subcircuit Command	4-37
Changing the Model Reference to an Existing Model Definition	4-38
Reusing Instance Models	4-39
Reusing Instance Models in the Same Schematic	4-40
Making Instance Models Available To All Schematics	4-40
Configuring Model Libraries	4-41
The Library and Include Files dialog box	4-41
How PSpice Uses Model Libraries	4-43
Search order	4-43
Handling duplicate model names	4-43
Adding Model Libraries to the Configuration	4-44
Changing Local and Global Scope	4-45
Changing Model Library Search Order	4-45
Changing the Library Search Path	4-46

Chapter 5 Creating Symbols for Models

Chapter Overview	5-1
What's Different About Symbols Used for Simulation?	5-3
Ways to Create Symbols	
for Models	5-4
Preparing Your Models for Symbol Creation	5-5
Using the Symbol Wizard	5-6
How to Start the Symbol Wizard	5-6
How the Symbol Wizard Works	5-7
Creating AKO Symbols	5-8
What Are Base vs. AKO Symbols?	5-8

Base and AKO Symbols in Symbol Libraries	5-8
How to Create AKO Symbols	5-9
Completing the Configuration of Your Part	5-11
Using the Parts Utility to Create Symbols	5-11
Starting the Parts Utility	5-12
Setting Up Automatic Symbol Creation	5-12
Basing New Symbols On a Custom Set of Symbols	5-13
Editing Symbol Graphics	5-15
How Schematics Places Symbols	5-15
Defining Important Symbol Elements	5-16
Origin	5-16
Bounding box	5-16
Grid spacing for graphics	5-17
Grid spacing for pins	5-17
Defining Symbol Attributes Needed for Simulation	5-18
MODEL	5-18
SIMULATION ONLY	5-19
TEMPLATE	5-19
TEMPLATE syntax	5-20
TEMPLATE examples	5-23

Chapter 6 Analog Behavioral Modeling

Chapter Overview	6-1
Overview of Analog Behavioral Modeling	6-2
The abm.slb Symbol Library File	6-3
Placing and Specifying ABM Parts	6-4
Net Names and Device Names in ABM Expressions	6-4
Forcing the Use of a Global Definition	6-5
ABM Part Templates	6-6
Control System Parts	6-7
Basic Components	6-9
Limiters	6-10
Chebyshev Filters	6-11
Integrator and Differentiator	6-14
Table Look-Up Parts	6-14
Laplace Transform Part	6-18
Math Functions	6-21
ABM Expression Parts	6-21
An Instantaneous Device Example: Modeling a Triode	6-25
PSpice A/D-Equivalent Parts	6-28
Implementation of PSpice A/D-Equivalent Parts	6-29

Modeling Mathematical or Instantaneous Relationships	6-30
EVALUE and GVALUE parts	6-30
EMULT, GMULT, ESUM, and GSUM	6-32
Lookup Tables (ETABLE and GTABLE)	6-33
Frequency-Domain Device Models	6-35
Laplace Transforms (LAPLACE)	6-35
Frequency Response Tables (EFREQ and GFREQ)	6-37
Cautions and Recommendations for Simulation and Analysis	6-40
Instantaneous Device Modeling	6-40
Frequency-Domain Parts	6-41
Laplace Transforms	6-41
Non-causality and Laplace transforms	6-42
Chebyshev filters	6-43
Frequency tables	6-44
Trading Off Computer Resources For Accuracy	6-45
Basic Controlled Sources	6-46
Creating Custom ABM Parts	6-46

Part Three Setting Up and Running Analyses

Chapter 7 Setting Up Analyses and Starting Simulation

Chapter Overview	7-1
Analysis Types	7-2
Setting Up Analyses	7-3
Execution Order for Standard Analyses	7-3
Output Variables	7-4
Modifiers	7-6
Starting Simulation	7-10
Starting Simulation from Schematics	7-11
Starting Simulation Outside of Schematics	7-11
Setting Up Batch Simulations	7-12
Multiple simulation setups within one circuit file	7-12
Running simulations with multiple circuit files	7-12
The Simulation Status Window	7-14

Chapter 8 DC Analyses

Chapter Overview	8-1
DC Sweep	8-2
Minimum Requirements to Run a DC Sweep Analysis	8-2
Overview of DC Sweep	8-3

Setting Up a DC Stimulus	8-4
Nested DC Sweeps	8-5
Curve Families for DC Sweeps	8-6
Bias Point Detail	8-8
Minimum Requirements to Run a Bias Point Detail Analysis	8-8
Overview of Bias Point Detail	8-8
Small-Signal DC Transfer	8-10
Minimum Requirements to Run a Small-Signal DC Transfer Analysis	8-10
Overview of Small-Signal DC Transfer	8-11
DC Sensitivity	8-12
Minimum Requirements to Run a DC Sensitivity Analysis	8-12
Overview of DC Sensitivity	8-12

Chapter 9 AC Analyses

Chapter Overview	9-1
AC Sweep Analysis	9-2
What You Need to Do to Run an AC Sweep	9-2
What is AC Sweep?	9-2
Setting Up an AC Stimulus	9-3
Setting Up an AC Analysis	9-5
AC Sweep Setup in “example.sch”	9-6
How PSpice Treats Nonlinear Devices	9-7
What’s required to linearize a device	9-7
What PSpice does	9-7
Example: Nonlinear behavioral modeling block	9-7
Noise Analysis	9-9
What You Need to Do to Run a Noise Analysis	9-9
What is Noise Analysis?	9-10
How PSpice calculates total output and input noise	9-10
Setting Up a Noise Analysis	9-11
Analyzing Noise in Probe	9-12
About noise units	9-13
Example	9-13

Chapter 10 Transient Analysis

Chapter Overview	10-1
Overview of Transient Analysis	10-2
Minimum Requirements to Run a Transient Analysis	10-2
Minimum circuit design requirements	10-2
Minimum program setup requirements	10-2

Defining a Time-Based Stimulus	10-3
Overview of Stimulus Generation	10-3
The Stimulus Editor Utility	10-5
Stimulus Files	10-5
Configuring Stimulus Files	10-6
Starting the Stimulus Editor	10-6
Defining Stimuli	10-8
Example: piecewise linear stimulus	10-8
Example: sine wave sweep	10-9
Creating New Stimulus Symbols	10-10
Editing a Stimulus	10-12
To edit an existing stimulus	10-12
To edit a PWL stimulus	10-12
To select a time and value scale factor for PWL stimuli	10-12
Deleting and Removing Traces	10-13
Manual Stimulus Configuration	10-13
To manually configure a stimulus	10-13
Transient (Time) Response	10-15
Internal Time Steps in Transient Analyses	10-17
Switching Circuits in Transient Analyses	10-17
Plotting Hysteresis Curves	10-18
Fourier Components	10-20

Chapter 11 Parametric and Temperature Analysis

Chapter Overview	11-1
Parametric Analysis	11-2
Minimum Requirements to Run a Parametric Analysis	11-2
Overview of Parametric Analysis	11-3
Example: RLC Filter	11-3
Entering the schematic	11-3
Running the simulation	11-4
Using performance analysis to plot overshoot and rise time	11-5
Example: Frequency Response vs. Arbitrary Parameter	11-8
Setting up the circuit	11-8
Displaying results in Probe	11-9
Temperature Analysis	11-11
Minimum Requirements to Run a Temperature Analysis	11-11
Overview of Temperature Analysis	11-11

Chapter 12 Monte Carlo and Sensitivity/Worst-Case Analyses

Chapter Overview	12-1
----------------------------	------

Statistical Analyses	12-2
Overview of Statistical Analyses	12-2
Output Control for Statistical Analyses	12-3
Model Parameter Values Reports	12-3
Waveform Reports	12-4
Collating Functions	12-4
Temperature Considerations in Statistical Analyses	12-6
Monte Carlo Analysis	12-7
Tutorial: Monte Carlo Analysis of a Pressure Sensor	12-10
Drawing the schematic	12-10
Defining component values	12-11
Setting up the parameters	12-12
Using resistors with models	12-13
Saving the schematic	12-14
Defining tolerances for the resistor models	12-14
Setting up the analyses	12-17
Running the analysis and viewing the results	12-18
Monte Carlo Histograms	12-19
Chebyshev filter example	12-19
Creating models for Monte Carlo analysis	12-19
Setting up the analysis	12-20
Creating histograms	12-20
Worst-Case Analysis	12-25
Overview of Worst-Case Analysis	12-25
Inputs	12-25
Procedure	12-26
Outputs	12-26
An important condition for correct worst-case analysis	12-27
Worst-Case Analysis Example	12-28
Hints and Other Useful Information	12-32
VARY BOTH, VARY DEV, and VARY LOT	12-32
Gaussian distributions	12-33
YMAX collating function	12-33
RELTOL	12-33
Sensitivity analysis	12-33
Manual optimization	12-34
Monte Carlo analysis	12-34

Part Four Viewing Results

Analyzing Waveforms Chapter 13 in Probe

Chapter Overview	13-1
Overview of Probe	13-2
Elements of a Plot Window	13-3
Managing Multiple Plot Windows	13-4
Printing multiple windows	13-4
Setting Up Probe	13-5
Configuring Probe Colors	13-5
Editing display and print colors in the msim.ini file	13-5
Configuring trace color schemes	13-7
Customizing the Probe Command Line	13-8
Configuring Update Intervals	13-8
Running Probe	13-9
Starting Probe	13-9
Other Ways to Run Probe	13-11
Starting Probe during a simulation	13-11
Pausing a simulation and then running Probe	13-11
Interacting with Probe while in monitor mode	13-12
Using Schematic Markers to Add Traces	13-12
Limiting Probe Data File Size	13-15
Limiting file size using markers	13-15
Limiting file size by suppressing the first part of simulation output	13-16
Using Simulation Data from Multiple Files	13-17
Setting up Probe for automatic loading of data files	13-17
Appending data files	13-18
Adding traces from specific loaded data files	13-19
Saving Simulation Results in ASCII Format	13-20
Example	13-21
Running the Simulation	13-21
Displaying voltages on nets and currents into pins	13-23
User Interface Features	13-24
Zoom Regions	13-24
Scrolling Traces	13-25
Modifying Trace Expressions and Labels	13-26
Moving and Copying Trace Names and Expressions	13-27
Copying and Moving Labels	13-28
Tabulating Trace Data Values	13-28
Cursors	13-29

Probe Trace Expressions	13-32
Basic Output Variable Form	13-33
Output Variable Form for Device Terminals	13-34
Analog Trace Expressions	13-40
Trace expression aliases	13-40
Arithmetic functions	13-40
Rules for numeric values suffixes	13-42

Viewing Results

Chapter 14 on the Schematic

Chapter Overview	14-1
Viewing Bias Point Voltages	
and Currents	14-2
How it works	14-2
If you run more than one analysis type	14-2
The Bias Information Toolbar Buttons	14-3
The Enable Display buttons	14-3
The Show/Hide buttons	14-3
Showing Voltages	14-4
Clearing and adding selected voltage values	14-4
Showing Currents	14-6
Clearing and adding selected current values	14-6
Changing the Precision of Displayed Data	14-7
Moving Voltage and Current Labels	14-7
Verifying Label Associations	14-8
Changing Display Colors	14-9
If you want obsolete voltage and current labels to change appearance	14-10
If You Have Hierarchical Symbols or Blocks	
on Your Schematic	14-10
Other Ways to View	
Bias Point Values	14-11
Using the VIEWPOINT Symbol to Display Voltage	14-11
Using the IPROBE Symbol to Display Current	14-11
.	14-12

Chapter 15 Other Output Options

Chapter Overview	15-1
Viewing Analog Results in the PSpice Window	15-2
Writing Additional Results to the PSpice Output File	15-3
Generating Plots of Voltage and Current Values	15-3
Generating Tables of Voltage and Current Values	15-4

Appendix A Setting Initial State

Appendix Overview	A-1
Save and Load Bias Point	A-2
Save Bias Point	A-2
Load Bias Point	A-3
Setpoints	A-4
Setting Initial Conditions	A-6

Appendix B Convergence and “Time Step Too Small Errors”

Appendix Overview	B-1
Introduction	B-2
Newton-Raphson Requirements	B-2
Is There a Solution?	B-3
Are the Equations Continuous?	B-4
Are the derivatives correct?	B-4
Is the Initial Approximation Close Enough?	B-5
Bias Point and DC Sweep	B-7
Semiconductors	B-7
Switches	B-8
Behavioral Modeling Expressions	B-9
Transient Analysis	B-10
Skipping the Bias Point	B-11
The Dynamic Range of TIME	B-11
Failure at the First Time Step	B-12
Parasitic Capacitances	B-13
Inductors and Transformers	B-13
Bipolar Transistors Substrate Junction	B-14
Diagnostics	B-15

Index

Figures

Figure 1-1	Simulation Design Flow	1-7
Figure 1-2	Schematics-Generated Data Files That PSpice Reads	1-10
Figure 1-3	User-Configurable Data Files That PSpice Reads	1-11
Figure 1-4	Data Files That PSpice Creates	1-14
Figure 2-1	Diode Clipper Circuit	2-2
Figure 2-2	Connection Points	2-4
Figure 2-3	PSpice Simulation Status Window	2-6
Figure 2-4	Clipper Circuit with Bias Point Voltages Displayed	2-7
Figure 2-5	Simulation Output File	2-9
Figure 2-6	DC Sweep Dialog Box	2-11
Figure 2-7	Probe Plot	2-12
Figure 2-8	Clipper Circuit with Voltage Marker on Net Out	2-12
Figure 2-9	Voltage at In, Mid, and Out	2-13
Figure 2-10	Trace Legend with Cursors Activated	2-13
Figure 2-11	Trace Legend with V(Mid) Symbol Outlined	2-14
Figure 2-12	Voltage Difference at V(In) = 4 Volts	2-15
Figure 2-13	Diode Clipper Circuit with a Voltage Stimulus	2-16
Figure 2-14	Stimulus Editor Window	2-17
Figure 2-15	Transient Analysis Dialog Box	2-18
Figure 2-16	Sinusoidal Input and Clipped Output Waveforms	2-19
Figure 2-17	Clipper Circuit with AC Stimulus	2-20
Figure 2-18	AC Sweep and Noise Analysis Dialog Box	2-21
Figure 2-19	dB Magnitude Curves for “Gain” at Mid and Out	2-22
Figure 2-20	Bode Plot of Clipper’s Frequency Response	2-23
Figure 2-21	Clipper Circuit with Global Parameter Rval	2-24
Figure 2-22	Parametric Dialog Box	2-26
Figure 2-23	Small Signal Response as R1 is Varied from 100Ω to 10 kΩ	2-27
Figure 2-24	Comparison of Small Signal Frequency Response at 100 and 10 kΩ Input Resistance	2-29
Figure 2-25	Performance Analysis Plots of Bandwidth and Gain vs. Rval	2-31
Figure 4-1	Relationship of Parts Utility to Schematics and PSpice	4-10

Figure 4-2	Process and Data Flow for the Parts Utility	4-13
Figure 4-3	Parts Utility Window with Data for a Bipolar Transistor	4-14
Figure 4-4	Schematic for a Half-Wave Rectifier	4-23
Figure 4-5	Diode Model Characteristics and Parameter Values for the Dbreak-X Instance Model	4-24
Figure 4-6	Assorted Device Characteristic Curves for a Diode	4-27
Figure 4-7	Forward Current Device Curve at Two Temperatures	4-28
Figure 4-8	AKO Model Definition Before and After Flattening	4-30
Figure 4-9	Model Editor Showing Q2N2222 with a DEV Tolerance Set on Rb	4-36
Figure 5-1	Rules for Pin Callout in Subcircuit Templates	5-25
Figure 6-1	LOPASS Filter Example	6-11
Figure 6-2	HIPASS Filter Part Example	6-12
Figure 6-3	BANDPASS Filter Part Example	6-12
Figure 6-4	BANDREJ Filter Part Example	6-13
Figure 6-5	FTABLE Part Example	6-16
Figure 6-6	LAPLACE Part Example 1	6-19
Figure 6-7	Lossy Integrator Example: Viewing Gain and Phase Characteristics with Probe	6-19
Figure 6-8	LAPLACE Part Example 2	6-19
Figure 6-9	ABM Expression Part Example 1	6-22
Figure 6-10	ABM Expression Part Example 2	6-23
Figure 6-11	ABM Expression Part Example 3	6-24
Figure 6-12	ABM Expression Part Example 4	6-24
Figure 6-13	Triode Circuit	6-25
Figure 6-14	Triode Subcircuit Producing a Family of I-V Curves	6-27
Figure 6-15	EVALUE Part Example	6-31
Figure 6-16	GVALUE Part Example	6-31
Figure 6-17	EMULT Part Example	6-32
Figure 6-18	GMULT Part Example	6-32
Figure 6-19	EFREQ Part Example	6-38
Figure 6-20	Voltage Multiplier Circuit (Mixer)	6-40
Figure 7-1	PSpice Status Window	7-14
Figure 8-1	DC Sweep Setup Example	8-2
Figure 8-2	Example Schematic example.sch	8-3
Figure 8-3	Curve Family Example Schematic	8-6
Figure 8-4	Device Curve Family	8-7
Figure 8-5	Operating Point Determination for Each Member of the Curve Family	8-7
Figure 9-1	AC Analysis Setup for example.sch	9-6
Figure 9-2	Device and Total Noise Traces for “example.sch”	9-14
Figure 10-1	Relationship of Stimulus Editor with Schematics and PSpice	10-5
Figure 10-2	Transient Analysis Setup for example.sch	10-15

Figure 10-3	Example Schematic example.sch	10-16
Figure 10-4	ECL Compatible Schmitt Trigger	10-18
Figure 10-5	Netlist for Schmitt Trigger Circuit	10-18
Figure 10-6	Hysteresis Curve Example: Schmitt Trigger	10-19
Figure 11-1	Passive Filter Schematic	11-3
Figure 11-2	Current of L1 when R1 is 1.5 Ohms	11-5
Figure 11-3	Rise Time and Overshoot vs. Damping Resistance	11-6
Figure 11-4	Inductor Waveform Data Viewed with Derived Rise Time and Overshoot Data	11-7
Figure 11-5	RLC Filter Example Circuit	11-8
Figure 11-6	Probe Plot of Capacitance vs. Bias Voltage	11-10
Figure 11-7	Example Schematic example.sch	11-12
Figure 12-1	Example Schematic example.sch	12-6
Figure 12-2	Monte Carlo Analysis Setup for example.sch	12-7
Figure 12-3	Summary of Monte Carlo Runs for example.sch	12-8
Figure 12-4	Parameter Values for Monte Carlo Pass 3	12-9
Figure 12-5	Pressure Sensor Circuit	12-10
Figure 12-6	Model Definition for RMonte1	12-15
Figure 12-7	Pressure Sensor Circuit with RMonte1 and RTherm Model Definitions . . .	12-16
Figure 12-8	Chebyshev Filter	12-20
Figure 12-9	Monte Carlo Analysis Setup Example	12-20
Figure 12-10	1 dB Bandwidth Histogram	12-22
Figure 12-11	Center Frequency Histogram	12-24
Figure 12-12	Simple Biased BJT Amplifier	12-28
Figure 12-13	YatX Goal Function	12-29
Figure 12-14	Amplifier Netlist and Circuit File	12-30
Figure 12-15	Correct Worst-Case Results	12-31
Figure 12-16	Incorrect Worst-Case Results	12-31
Figure 12-17	Schematic Demonstrating Use of VARY BOTH	12-32
Figure 12-18	Circuit File Demonstrating Use of VARY BOTH	12-32
Figure 13-1	Probe Window with Two Plot Windows	13-3
Figure 13-2	Trace Legend Symbols	13-19
Figure 13-3	Section Information Message Box	13-20
Figure 13-4	Example Schematic Example.sch	13-21
Figure 13-5	Probe Main Window with Loaded Example.dat and Open Plot Menu	13-22
Figure 13-6	Output from Transient Analysis: Voltage at OUT1 and OUT2	13-23
Figure A-1	Setpoints	A-4

Tables

Table 1-1	DC Analysis Types	1-3
Table 1-2	AC Analysis Types	1-4
Table 1-3	Time-Based Analysis Types	1-5
Table 1-4	Parametric and Temperature Analysis Types	1-6
Table 1-5	Statistical Analysis Types	1-6
Table 2-1	Association of Probe Cursors with Mouse Buttons	2-13
Table 3-1	Operators in Expressions	3-15
Table 3-2	Functions in Arithmetic Expressions	3-16
Table 3-3	System Variables	3-18
Table 4-1	Models Supported in the Parts Utility	4-12
Table 4-2	Sample Diode Data Sheet Values	4-25
Table 5-1	Symbol Names for Custom Symbol Generation	5-13
Table 6-1	Control System Parts	6-7
Table 6-2	ABM Math Function Parts	6-21
Table 6-3	ABM Expression Parts	6-22
Table 6-4	PSpice A/D-Equivalent Parts	6-28
Table 6-5	Basic Controlled Sources in analog.slb	6-46
Table 7-1	Classes of PSpice Analyses	7-2
Table 7-2	Execution Order for Standard Analyses	7-4
Table 7-3	PSpice Output Variable Formats	7-7
Table 7-4	Element Definitions for 2-Terminal Devices	7-8
Table 7-5	Element Definitions for 3- or 4-Terminal Devices	7-9
Table 7-6	Element Definitions for Transmission Line Devices	7-9
Table 7-7	Element Definitions for AC Analysis Specific Elements	7-10
Table 8-1	DC Sweep Circuit Design Requirements	8-2
Table 8-2	Curve Family Example Setup	8-6
Table 10-1	Stimulus Symbols for Time-Based Input Signals	10-3
Table 11-1	Parametric Analysis Circuit Design Requirements	11-2
Table 12-1	Collating Functions Used in Statistical Analyses	12-4
Table 13-1	Default Probe Item Colors	13-6
Table 13-2	Mouse Actions for Cursor Control	13-30
Table 13-3	Key Combinations for Cursor Control	13-31

Table 13-4	Probe Output Variable Formats	13-35
Table 13-5	Examples of Probe Output Variable Formats	13-36
Table 13-6	Output Variable AC Suffixes	13-37
Table 13-7	Device Names for Two-Terminal Device Types	13-37
Table 13-8	Terminal IDs by Three & Four-Terminal Device Type.	13-38
Table 13-9	Noise Types by Device Type	13-39
Table 13-10	Probe Arithmetic Functions	13-40
Table 13-11	Output Units Recognized by Probe	13-42

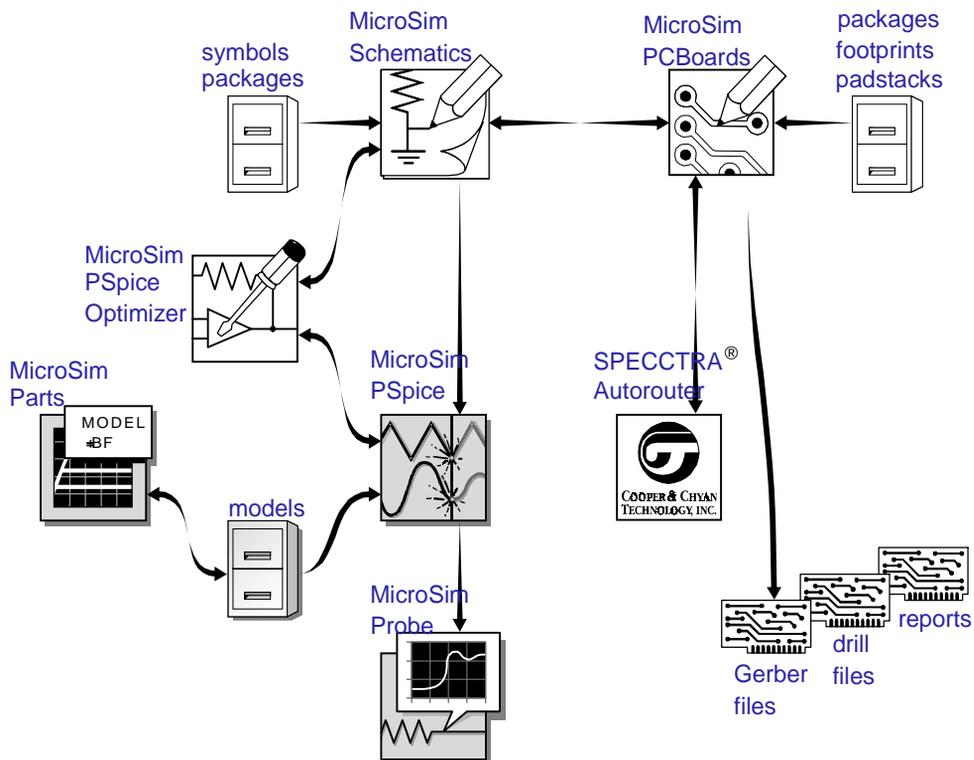
Before You Begin

Welcome to MicroSim

Welcome to the MicroSim family of products. Whichever programs you have purchased, we are confident that you will find that 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 PSpice Overview

MicroSim PSpice can simulate analog circuits. Once you prepare a schematic for simulation, MicroSim Schematics generates a circuit file set. The circuit file set, containing the circuit netlist and analysis commands, is read by PSpice for simulation. The results are formulated into meaningful graphical traces in Probe which can be marked for display directly from your schematic.



How to Use this Guide

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

This guide assumes that you are familiar with Microsoft Windows (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*.

Typographical Conventions

Before using PSpice, 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
	Press 	A specific key or key stroke on the keyboard.
monospace font	Type VAC... clipper.sch	Commands/text entered from the keyboard; library and file names.
	 To improve accuracy...	Tip providing advice or different ways to do things.
	 Be careful...	Important note or cautionary message.

Related Documentation

Documentation for MicroSim products is available in both hard copy and online. To access an online manual instantly, you can select it from the Help menu in its respective program (for example, access the Schematics User's Guide from the Help menu in Schematics).

Note *The documentation you receive depends on the software configuration you have purchased.*

The following table provides a brief description of those manuals available in both hard copy and online.

This manual...	Provides information about how to use...
MicroSim Schematics User's Guide	MicroSim Schematics, which is a schematic capture front-end program with a direct interface to other MicroSim programs and options.
MicroSim PCBoards User's Guide	MicroSim PCBoards, which is a PCB layout editor that lets you specify printed circuit board structure, as well as the components, metal, and graphics required for fabrication.
MicroSim PSpice A/D & Basics+ User's Guide	PSpice A/D, Probe, the Stimulus Editor, and the Parts utility, which are circuit analysis programs that let you create, simulate, and test analog and digital circuit designs. It provides examples on how to specify simulation parameters, analyze simulation results, edit input signals, and create models.
MicroSim PSpice & Basics User's Guide	MicroSim PSpice & MicroSim PSpice Basics, which are circuit analysis programs that let you create, simulate, and test analog-only circuit designs.
MicroSim PSpice Optimizer User's Guide	MicroSim PSpice Optimizer, which is an analog performance optimization program that lets you fine tune your analog circuit designs.
MicroSim PLSyn User's Guide	MicroSim PLSyn, which is a programmable logic synthesis program that lets you synthesize PLDs and CPLDs from a schematic or hardware description language.
MicroSim FPGA User's Guide	MicroSim FPGA—the interface between MicroSim Schematics and XACTstep—with MicroSim PSpice A/D to enter designs that include Xilinx field programmable gate array devices.
MicroSim Filter Designer User's Guide	MicroSim Filter Designer, which is a filter synthesis program that lets you design electronic frequency selective filters.

The following table provides a brief description of those manuals available online *only*.

This online manual...	Provides this...
MicroSim PSpice A/D Online Reference Manual	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 Online Manual	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.
Online Library List	A complete list of the analog and digital parts in the model and symbol libraries.
MicroSim PCBoards Online Reference Manual	Reference information for MicroSim PCBoards, such as: file name extensions, padstack naming conventions and standards, footprint naming conventions, the netlist file format, the layout file format, and library expansion and compression utilities.
MicroSim PCBoards Autorouter Online User's Guide	Information on the integrated interface to Cooper & Chyan Technology's (CCT) SPECCTRA autorouter in MicroSim PCBoards.

Online Help

Selecting Search for Help On from the Help menu brings up an extensive online help system.

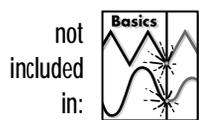
The online help from these programs includes:

- step-by-step instructions on how to set up PSpice simulations and analyze simulation results
- reference information about PSpice
- Technical Support information

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

If You Have PSpice Basics and Not the Standard Package

PSpice Basics provides the basic functionality needed for analog design without the advanced features in the full PSpice package. Because this guide is for both PSpice Basics and PSpice users, there are some features described here that are not available to PSpice Basics users.



The Basics icon (shown in the sidebar) is used throughout this user guide to mark each section or paragraph which describes a feature *not available* to PSpice Basics users. If an entire section describes a “non-Basics” feature, the icon is placed next to the section title. If an individual paragraph describes a “non-Basics” feature, the icon is placed next to the paragraph.

The following table identifies which features are included with PSpice and PSpice Basics.

Feature	PSpice (Standard)	PSpice Basics
Benefits of integration with MicroSim Schematics		
graphical design entry (schematic capture)	yes	yes
simulation setup using dialog boxes	yes	yes
cross-probing	yes	yes
multi-window analysis of Probe data sets	yes	yes
marching waveforms in Probe	yes	yes
board layout package interfaces	yes	yes

Feature	PSpice (Standard)	PSpice Basics
Notable PSpice analysis and simulation features		
DC sweep, AC sweep, transient analysis	yes	yes
noise, Fourier, temperature analysis	yes	yes
parametric analysis	yes	no
Monte Carlo, sensitivity/worst-case analysis	yes	no
analog behavioral modeling (ABM)	yes	yes
Stimulus Editor	yes	no
Parts utility	yes	no
performance analysis (goal functions)	yes	no
save/load bias point	yes	no
Notable PSpice devices and library models		
GaAsFETs: Curtice, Statz, TriQuint, Parker-Skellern	all	Statz
MOSFETs: SPICE3 (1-3) with charge conservation, BSIM1, BSIM3 (version 3)	yes	yes
IGBTs	yes	no
JFETs, BJTs	yes	yes
resistor, capacitor, and inductor .MODEL support	yes	yes
ideal, non-ideal lossy transmission lines	all	ideal
coupled inductors	yes	yes
coupled transmission lines	yes	no
nonlinear magnetics	yes	no
voltage- and current-controlled switches	yes	yes
analog model library	10,200+	10,200+ *

Note For expert PSpice users, these are the PSpice circuit file commands that are not available in the Basics package:

- .STIMULUS
- .STIMLIB
- .SAVEBIAS
- .LOADBIAS

Feature	PSpice (Standard)	PSpice Basics
Purchase options		
MicroSim PCBoards	yes	yes
MicroSim PSpice Optimizer	yes	no
Device Equations	yes	no
network licensing	yes	no
Miscellaneous specifications		
unlimited circuit size	yes	no**

*. PSpice Basics package includes all libraries except IGBTs, SCRs, thyristors, PWMs, magnetic cores, and transmission lines.

**. Unlimited circuit size means you can have as many components in your design as can be accommodated by your computer's memory; rule-of-thumb is one Mb of RAM for every 400 transistors in the circuit. The PSpice Basics package is limited by the total device count: 100 large devices (all transistors and ABM devices) or 500 small devices.

What's New

To find out more, see [Chapter 14, Viewing Results on the Schematic](#).

Bias information display on your schematic After simulating, you can display bias point information on your schematic so you can quickly zero in on problem areas in your design. This means you can selectively display voltages on wire segments and currents on device pins.

To find out more, see [Using the Symbol Wizard on page 5-6](#).

Automatic symbol creation for existing device models using the symbol wizard The symbol wizard has been expanded to create symbols for entire model libraries. This is a fast way to create symbols for vendor models you have just received, or to supersede existing symbols with a new graphic standard.

To find out more, see [Using the Parts Utility to Create Symbols on page 5-11](#).

Automatic symbol creation for new device models using the Parts utility When extracting a simulation model using the Parts utility, you can now have Parts

automatically create a symbol for the model. After saving your work, the part is ready for use: just place and connect the symbol in your schematic and you're ready to simulate. The Parts utility handles all of the library configuration steps for you.

Device noise trace display in Probe When you run a noise analysis, PSpice now writes device noise contributions to the Probe data file. This means you can view device noise results as traces in Probe for each frequency in the corresponding AC analysis. (In earlier releases, you could find individual device contributions reported in the PSpice output file; that information is still available and reflects the same data you can now view in Probe.)

To find out more, see [Analyzing Noise in Probe on page 9-12](#).

BSIM3 version 3 MOSFET model The BSIM3 version 3 model is a deep submicron MOSFET model with the same physical basis as the BSIM3 version 2 model, but with several major enhancements. These enhancements include:

- A single I-V expression to describe current and output conductance in all regions of device operation.
- Better modeling of narrow width devices.
- A reformulated capacitance model to improve short and narrow geometry models.
- A new relaxation time model to improve transient modeling.
- Improved model fitting of various W/L ratios using one parameter set.

BSIM3 version 3 retains the extensive built-in dependencies of dimensional and processing parameters of BSIM3 version 2.

To find out more, refer to MOSFET devices in the *Analog Devices* chapter of the online *MicroSim PSpice A/D Reference Manual*.

Part One

Simulation Primer

Part One provides basic information about circuit simulation including examples of common analyses.

Chapter 1, Things You Need to Know, provides an overview of the circuit simulation process including what PSpice does, descriptions of analysis types, and descriptions of important files.

Chapter 2, Simulation Examples, presents examples of common analyses to introduce the methods and tools you'll need to enter, simulate, and analyze your design.

Things You Need to Know

1

Chapter Overview

This chapter introduces the purpose and function of the PSpice circuit simulator.

[What is PSpice? on page 1-2](#) describes PSpice capabilities.

[Analyses You Can Run with PSpice on page 1-3](#) introduces the different kinds of basic and advanced analyses that PSpice supports.

[Using PSpice with Other MicroSim Programs on page 1-7](#) presents the high-level simulation design flow.

[Files Needed for Simulation on page 1-10](#) describes the files used to pass information between MicroSim programs. This section also introduces the things you can do to customize where and how PSpice finds simulation information.

[Files That PSpice Generates on page 1-14](#) describes the files that contain simulation results.

What is PSpice?

MicroSim PSpice is a simulation program that models the behavior of an analog circuit. Used with MicroSim Schematics for design entry, you can think of PSpice as a software-based breadboard of your circuit that you can use to test and refine your design before ever touching a piece of hardware.

Run basic and advanced analyses PSpice can perform:

- DC, AC, and transient analyses, so you can test the response of your circuit to different inputs.
- Parametric, Monte Carlo, and sensitivity/worst-case analyses, so you can see how your circuit's behavior varies with changing component values.

The range of models built into PSpice include not only those for resistors, inductors, capacitors, and bipolar transistors, but also these:

- transmission line models, including delay, reflection, loss, dispersion, and crosstalk
- nonlinear magnetic core models, including saturation and hysteresis
- six MOSFET models, including BSIM3 version 3
- five GaAsFET models, including Parker-Skellern and TriQuint's TOM2 model
- IGBTs

Use parts from MicroSim's extensive set of libraries The model libraries feature over 10,200 analog models of parts made in North America, Japan, and Europe.

Vary device characteristics without creating new parts PSpice has numerous built-in models with parameters that you can tweak for a given device. These include independent temperature effects.

Model behavior PSpice supports analog behavioral modeling so you can describe functional blocks of circuitry using mathematical expressions and functions.

Analyses You Can Run with PSpice

See [Chapter 2, Simulation Examples](#), for introductory examples showing how to run each type of analysis.

See [Part Three, Setting Up and Running Analyses](#), for a more detailed discussion of each type of analysis and how to set it up.

Basic Analyses

DC sweep & other DC calculations

These DC analyses evaluate circuit performance in response to a direct current source. [Table 1-1](#) summarizes what PSpice calculates for each DC analysis type.

Table 1-1 *DC Analysis Types*

For this DC analysis...	PSpice computes this...
DC sweep	Steady-state voltage and currents when sweeping a source, a model parameter, or temperature over a range of values.
Bias point detail	Bias point data in addition to what is automatically computed in any simulation.
DC sensitivity	Sensitivity of a net or part voltage as a function of bias point.
Small-signal DC transfer	Small-signal DC gain, input resistance, and output resistance as a function of bias point.

AC sweep and noise

These AC analyses evaluate circuit performance in response to a small-signal alternating current source. [Table 1-2](#) summarizes what PSpice calculates for each AC analysis type.

Table 1-2 *AC Analysis Types*

For this AC analysis...	PSpice computes this...
AC sweep	Small-signal response of the circuit (linearized around the bias point) when sweeping one or more sources over a range of frequencies. Outputs include voltages and currents with magnitude and phase; you can use this information to obtain Bode plots.
Noise	For each frequency specified in the AC analysis: <ul style="list-style-type: none">• Propagated noise contributions at an output net from every noise generator in the circuit.• RMS sum of the noise contributions at the output.• Equivalent input noise.

Note *To run a noise analysis, you must also run an AC sweep analysis.*

Transient and Fourier

These time-based analyses evaluate circuit performance in response to time-varying sources. [Table 1-3](#) summarizes what PSpice calculates for each time-based analysis type.

Table 1-3 *Time-Based Analysis Types*

For this time-based analysis...	PSpice computes this...
Transient	Voltages and currents tracked over time.
Fourier	DC and Fourier components of the transient analysis results.

Note *To run a Fourier analysis, you must also run a transient analysis.*

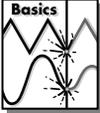
Advanced Multi-Run Analyses

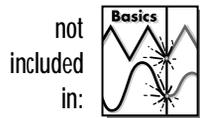
The multi-run analyses—parametric, temperature, Monte Carlo, and sensitivity/worst-case—result in a series of DC sweep, AC sweep, or transient analyses depending on which basic analyses you enabled.

Parametric and temperature

For parametric and temperature analyses, PSpice steps a circuit value in a sequence that you specify and runs a simulation for each value. [Table 1-4](#) shows the circuit values that you can step for each kind of analysis.

Table 1-4 *Parametric and Temperature Analysis Types*

	For this analysis...	You can step one of these...
not included in: 	Parametric	global parameter model parameter component value DC source operational temperature
	Temperature	operational temperature



Monte Carlo and sensitivity/worst-case

Monte Carlo and sensitivity/worst-case analyses are statistical. PSpice changes device model parameter values with respect to device and lot tolerances that you specify, and runs a simulation for each value.

Table 1-5 summarizes how PSpice runs each statistical analysis type.

Table 1-5 *Statistical Analysis Types*

For this statistical analysis...	PSpice does this...
Monte Carlo	For each simulation, <i>randomly</i> varies all device model parameters that you have defined a tolerance for.
Sensitivity/worst-case	Computes the probable worst-case response of the circuit in two steps: <ol style="list-style-type: none"> 1 Computes component sensitivity to changes in the device model parameters. This means PSpice <i>nonrandomly</i> varies device model parameters that you have defined a tolerance for, doing it one at a time for each device and running a simulation with each change. 2 Sets <i>all</i> model parameters for <i>all</i> devices to their worst-case values (assumed to be at one of the tolerance limits) and runs a final simulation.

Using PSpice with Other MicroSim Programs

Figure 1-1 illustrates the design flow for simulating a circuit and the programs that you use at each step.

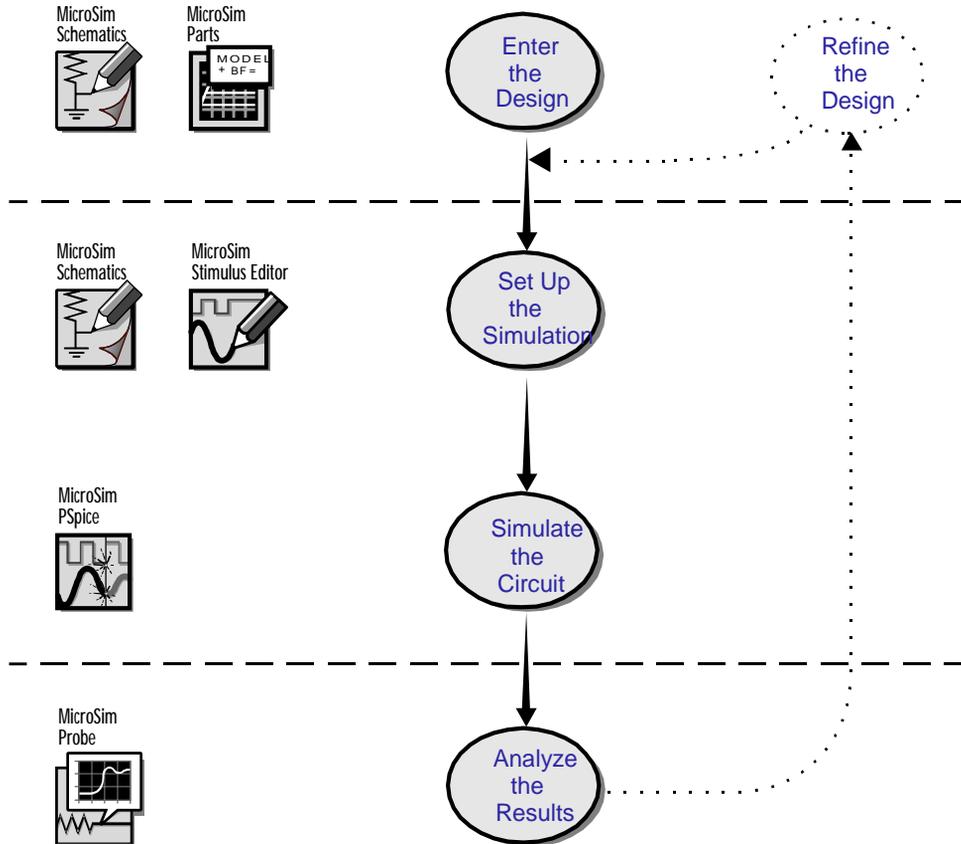


Figure 1-1 *Simulation Design Flow*

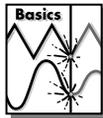
Using Schematics to Prepare for Simulation

Schematics is a design entry program you need to prepare your circuit for simulation. This means:

- placing and connecting part symbols,
- defining component values and other attributes,
- defining input waveforms,
- enabling one or more analyses, and
- marking the points in the circuit where you want to see results.

Schematics is also the control point for running other programs used in the simulation design flow.

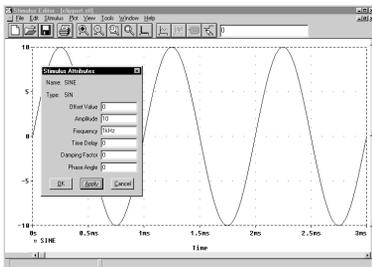
not
included
in:



What is the Stimulus Editor?

The Stimulus Editor is a graphical input waveform editor that lets you define the shape of time-based signals used to test your circuit's response during simulation. Using the Stimulus Editor, you can define analog stimuli with sine wave, pulse, piecewise linear, exponential pulse, and single-frequency FM shapes.

The Stimulus Editor lets you draw an analog piecewise linear stimulus by clicking at the points along the timeline that correspond to the input values you want at transitions.



Files Needed for Simulation

To simulate your design, PSpice needs to know about:

- the parts in your circuit and how they are connected,
- what analyses to run,
- the simulation models that correspond to the parts in your circuit, and
- the stimulus definitions to test with.

This information is provided in various data files. Some of these are generated by Schematics, others come from libraries (which can also be generated by other programs like the Stimulus Editor and Parts), and still others are user-defined.

Files That Schematics Generates

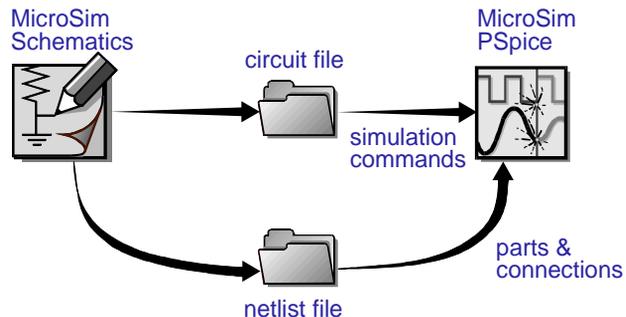


Figure 1-2 *Schematics-Generated Data Files That PSpice Reads*

When you begin the simulation process, Schematics first generates files describing the parts and connections in your circuit. These files are the netlist file and the circuit file that PSpice reads before doing anything else.

Netlist file

The netlist file contains a list of device names, values, and how they are connected with other devices. The name that Schematics generates for this file is *schematic_name.net*.

Refer to the online *MicroSim PSpice A/D Reference Manual* for the syntax of the statements in the netlist file and the circuit file.

Circuit file

The circuit file contains commands describing how to run the simulation. This file also refers to other files that contain netlist, model, stimulus, and any other user-defined information that apply to the simulation. The name that Schematics generates for this file is *schematic_name.cir*.

Other Files That You Can Configure for Simulation

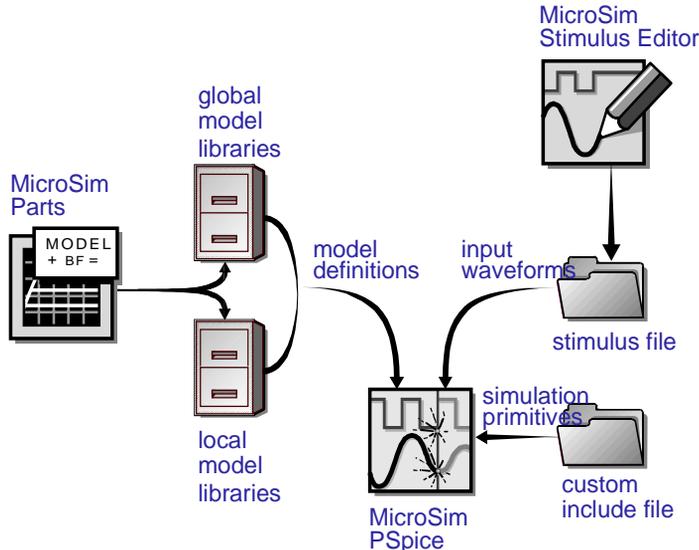
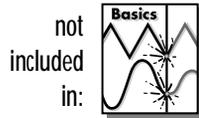


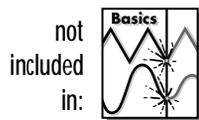
Figure 1-3 User-Configurable Data Files That PSpice Reads

The circuit file (.cir) that Schematics generates contains references to the other user-configurable files that PSpice needs to read.



A subcircuit, sometimes called a *macromodel*, is analogous to a procedure call in a software programming language.

See [What is the Parts Utility? on page 1-9](#) for a description.



Before starting simulation, PSpice needs to read other files that contain simulation information for your circuit. These are model libraries, and if required, stimulus files and include files.

You can create these files using MicroSim programs like the Stimulus Editor and the Parts utility. These programs automate file generation and provide graphical ways to verify the data. Or, you can use any text editor, like the MicroSim Text Editor, to enter the data manually.

Model library

A model library is a file that contains the electrical definition of one or more parts. PSpice uses this information to determine how a part will respond to different electrical inputs.

These definitions take the form of either a:

- *model parameter set*, which defines the behavior of a part by fine-tuning the underlying model built into PSpice, or
- *subcircuit netlist*, which describes the structure and function of the part by interconnecting other parts and primitives.

The most commonly used models are available in the MicroSim model libraries shipped with your programs. The model library names have a .lib extension.

If needed, however, you can create your own models and libraries, either:

- manually using the model editor in Schematics or some other text editor, or
- automatically using the Parts utility.

Stimulus file

A stimulus file contains time-based definitions for analog input waveforms. You can create a stimulus file either:

- manually using a text editor to create the definition (a typical file extension is `.stm`), or
- automatically using the Stimulus Editor (which generates a `.stl` file extension).

Include file

An include file is a user-defined file that contains:

- PSpice commands, or
- supplemental text comments that you want to appear in the PSpice output file (see page [1-15](#)).

You can create an include file using the MicroSim Text Editor. Typically, include file names have a `.inc` extension.

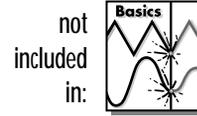
Configuring model library, stimulus, and include files

PSpice searches model libraries, stimulus files, and include files for any information it needs to complete the definition of a part or to run a simulation.

The files that PSpice searches depend on how you configure your model libraries and other files. Much of the configuration is set up for you automatically, however, you can do the following yourself:

- Add and delete files from the configuration.
- Change the scope of a file: that is, whether the file applies to one design only (local) or to any design (global).
- Change the search order.

Note *Not all stimulus definitions require a stimulus file. In some cases, like DC and AC sources, you must use a schematic symbol and set its attributes.*



See [What is the Stimulus Editor?](#) on page 1-8 for a description.

Example: An include file that contains definitions, using the PSpice `.FUNC` command, for functions that you want to use in numeric expressions elsewhere in the circuit.

More on libraries...

Configuration for model libraries is similar to that for other libraries that Schematics uses. These include symbol and package libraries. To find out more, refer to your *MicroSim Schematics User's Guide*.

Files That PSpice Generates

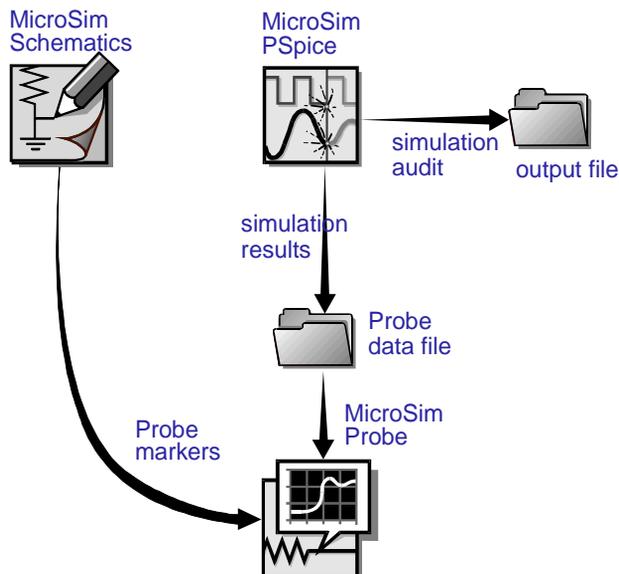


Figure 1-4 Data Files That PSpice Creates

After first reading the circuit file, netlist file, model libraries, and any other required inputs, PSpice starts the simulation. As simulation progresses, PSpice saves results to two files—the Probe data file and the PSpice output file.

For a description of how to use Probe to display simulation results, see [Part Four, Viewing Results](#).

For a description of the waveform analyzer program, see [What is Probe? on page 1-9](#).

There are two ways to add waveforms to the Probe display:

- From within Probe, by specifying trace expressions.
- From within Schematics, by cross-probing.

Probe data file

The Probe data file contains simulation results in a format that Probe can read. Probe reads this file automatically and displays waveforms reflecting circuit response at nets, pins, and parts that you marked in your schematic (cross-probing). You can set up your simulation so Probe displays the results as the simulation progresses or after the simulation completes.

Once Probe has read the Probe data file and displays the initial set of results, you are free to add more waveforms and to perform post-simulation analysis of the data.

PSpice output file

The PSpice output file is an ASCII text file that contains:

- the netlist representation of the circuit,
- the PSpice command syntax for simulation commands and options (like the enabled analyses),
- simulation results, and
- warning and error messages for problems encountered during read-in or simulation.

Its content is determined by:

- the types of analyses you run,
- the options you select for running PSpice, and
- the simulation control symbols (like VPRINT1 and VPLOT1) that you place and connect to nets in your schematic.

Example: Each instance of a VPRINT1 symbol placed in your schematic causes PSpice to generate a table of voltage values for the connecting net, and to write the table to the PSpice output file.

Simulation Examples

2

Chapter Overview

The examples in this chapter provide an introduction to the methods and tools for creating circuit designs, running simulations with PSpice, and analyzing simulation results using Probe. All analyses are performed on the same example circuit to clearly illustrate analysis setup, simulation, and result analysis procedures for each analysis type.

This chapter includes the following sections:

[Example Circuit Creation on page 2-2](#)

[Bias Point Analysis on page 2-6](#)

[DC Sweep Analysis on page 2-10](#)

[Transient Analysis on page 2-16](#)

[AC Sweep Analysis on page 2-20](#)

[Parametric Analysis on page 2-24](#)

[Probe Performance Analysis on page 2-30](#)

Example Circuit Creation

This section describes how to use MicroSim Schematics to create the simple diode clipper circuit shown in Figure 2-1.

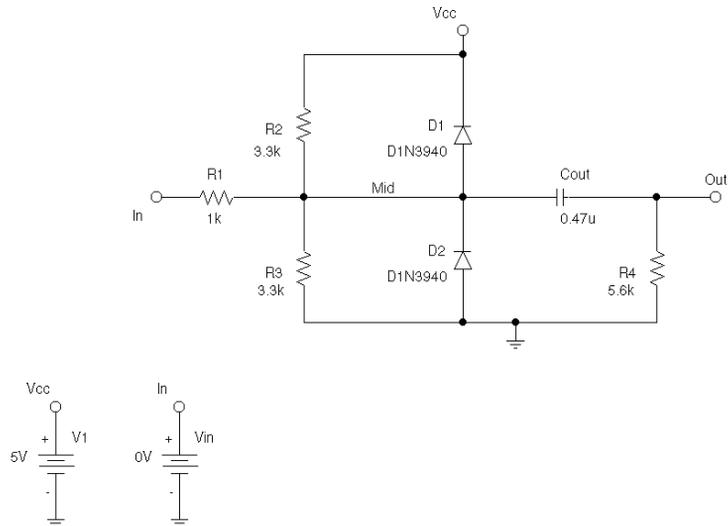


Figure 2-1 Diode Clipper Circuit

To open a new schematic window

- 1 Start Schematics. If Schematics is already running, be sure you are in the schematic editor. If you are in a blank schematic window (indicated by “Schematicn” in the title bar at the top of the window), you can begin creating the circuit.



If you need to open a new schematic window, from the File menu, select New.

To place the voltage sources

- 1 From the Draw menu, select Get New Part to display the Part Browser dialog box.
- 2 In the Part Name text box, type VDC.
- 3 Click Place & Close.



or press **Ctrl** + **G**

If you have enough room on your screen, click Place to leave the Part Browser dialog box open.

- 4 Move the pointer to the correct position on the schematic (see Figure 2-1) and click to place the first source.
- 5 Move the cursor and click again to place the second source.
- 6 Right-click to cancel placement mode.

To place the diodes

- 1 Go to the Part Browser dialog box.
- 2 In the Part name text box, type D1N39* to display a list of diodes.
- 3 Click D1N3940.
- 4 Click Place (to leave the dialog box open) or Place & Close (to close the dialog box).
- 5 Press **Ctrl**+**R** to rotate the diode outline to the correct orientation.
- 6 Click to place the first diode (D1), then click to place the second diode (D2).
- 7 Right-click to cancel placement mode.

To move the text associated with the diodes (or any other object)

- 1 Click the text once to select it.
- 2 Drag the text to a new location.

To place the other components

Follow similar steps as described for the diodes to place the components listed below. The symbol names you need to type in the Part name text box of the Part Browser dialog box are shown in parentheses:

- resistors (R)
- capacitor (C)
- ground symbols (EGND)
- bubble symbols (BUBBLE)

If needed, click  to redisplay the Part Browser dialog box.

When placing components:

- Leave space to connect the components with wires.
- You will change device names and values that don't match those shown in Figure 2-1 later in this section.

To refresh the schematic display, select Redraw from the View menu or press **Ctrl**+**L**.



You can right-click at any time to stop the wiring mode. The cursor changes to the default arrow.

If necessary, double-right click or press **Spacebar** to resume wiring mode. The cursor changes back to a pencil.

Clicking on any valid connection point terminates a wire. A valid connection point is shown as an *x* (see Figure 2-2).

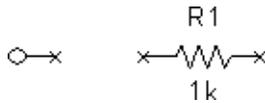


Figure 2-2 Connection Points

If you make a mistake when placing or connecting components:

- 1 From the Edit menu, select Undo, or click .

Bubbles serve as *wireless* connections where connectivity is implied by identical labels.

To connect the components

- 1 From the Draw menu, select Wire to enter wiring mode. The cursor changes to a pencil.
- 2 Click the connection point (the very end) of the pin on the bubble at the input of the circuit.
- 3 Click the nearest connection point of the input resistor R1.
- 4 Connect the other end of R1 to the output capacitor.
- 5 Connect the diodes to each other and to the wire between them:
 - a Click the connection point of the anode for the lower diode.
 - b Move the cursor straight up and click the wire between the diodes. The wire terminates and the junction of the wire segments is made visible.
 - c Click again on the junction to continue wiring.
 - d Click the end of the upper diode's cathode pin.
- 6 Continue connecting components until the circuit is wired as shown in Figure 2-1 on page 2-2.

To assign names (labels) to the nets and bubbles

- 1 Double-click any segment of the wire that connects R1, R2, R3, the diodes, and the capacitor.
- 2 In the Label text box, type `Mid`.
- 3 Click OK.
- 4 Double-click each bubble to label it as shown in Figure 2-1 on page 2-2.

To assign names to devices

- 1 Double-click the reference designator of the VDC symbol, V2.
- 2 In the Edit Reference Designator dialog box, type `Vin` in the Package Reference Designator text box.
- 3 Click OK.

- 4 Continue naming devices until all circuit devices are named as in Figure 2-1 on page 2-2.

To change the attribute values of devices

- 1 Double-click the attribute value (0V) of the VDC symbol, V1.
- 2 In the Set Attribute Value dialog box, type 5v.
- 3 Click OK.
- 4 Continue changing the attribute values of the circuit devices until all devices are named as in Figure 2-1 on page 2-2.

Your schematic should now have the same symbols, wiring, labels, and attributes as Figure 2-1 on page 2-2.

To save your schematic

- 1 From the File menu, select Save.
- 2 Type `clipper` in the File name text box.
- 3 Click OK to save the file as `clipper.sch`.



or press **Ctrl** + **S**

Finding Out More about Setting Up Your Schematic

About setting up a schematic for simulation

For a checklist of all of the things you need to do to set up your schematic for simulation, and how to avoid common problems, see [Chapter 3, Preparing a Schematic for Simulation](#).

About tracking versions of your design using Design Journal

As you develop and test your design, you can use the Design Journal feature in Schematics to create checkpoint schematics. This allows you to create an electronic record of design development and perform what-if analyses. To find out more, refer to the online Help in Schematics.

Bias Point Analysis

Running PSpice

When you perform a simulation, PSpice generates an output file (for this example, `clipper.out`). PSpice also generates bias information that Schematics can read and display.

While PSpice is running, the progress of the simulation appears and is updated in the PSpice simulation status window (see Figure 2-3).

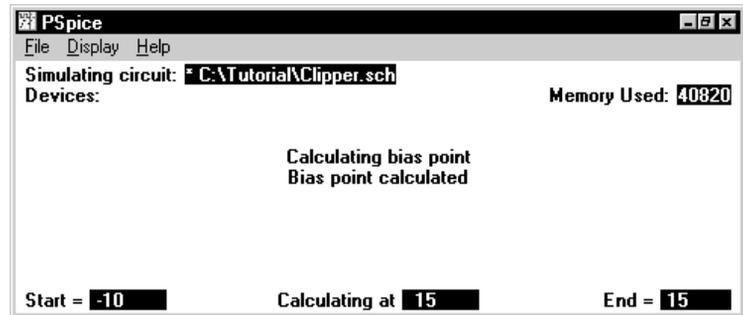


Figure 2-3 PSpice Simulation Status Window

To simulate the circuit using PSpice

- 1 In Schematics, make the `clipper.sch` window active.
- 2 From the Analysis menu, select Simulate.



or press **F11**

After the simulation, you may see a Schematics dialog box notifying you that it is back-annotating your schematic with simulation data. Do the following:

- 1 If you do not want to be notified of this after the next simulation, select Don't Show this Dialog Again.
- 2 Click OK to continue this example.

Using the Bias Information Display

You can display bias information on your schematic, including voltages for all nets and currents into all pins. You can also control which nets and pins have voltage and current measurements displayed at any given time.

To display bias voltage information at all nets

- 1 In Schematics, make the `clipper.sch` window active.
- 2 If the Simulation toolbar is not displayed, do the following:
 - a From the View menu, select Toolbars.
 - b Select (✓) the Simulation check box, then click Close.
- 3 If voltages are not displayed, then do the following: On the Simulation toolbar, click the Enable Bias Voltage Display button. DC bias point voltages appear at all nets (Figure 2-4).

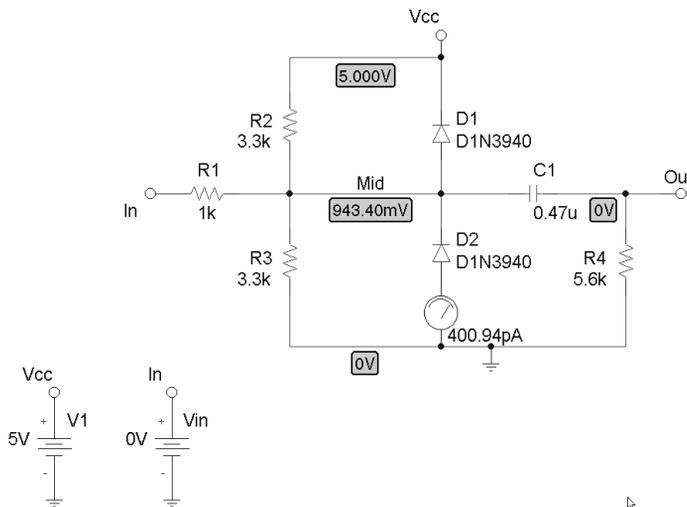


Figure 2-4 Clipper Circuit with Bias Point Voltages Displayed

Because the diodes are both reverse biased (off), and the input source V_{in} is 0V (a short circuit to ground), the bias point is dependent only on the values of V_{cc} , R_1 , R_2 , and R_3 .



The bias information display commands are also available from the Analysis menu by pointing to Display Results on Schematic.



You can move an individual voltage label as needed by selecting and dragging it. When you select a voltage, the wire with which the voltage is associated is highlighted for clarity. Voltage labels remain wherever you move them unless you delete or move the associated wire.

Individual currents have the same properties as voltages except that they are associated with device pins and the association is illustrated by arrows.

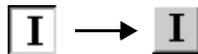
The voltage at net *Mid* is in agreement with manual calculation

$$V(\text{Mid}) = \frac{R_3}{R_2 + R_3} \times V_{cc}$$

where

$$R_{eq} = \frac{R_1 \times R_3}{R_1 + R_3}$$

Correct, expected bias point analysis results provide assurance of proper circuit connectivity.



Your settings for the display of voltages and currents are stored with the schematic. For this reason, if you re-enable the bias display (described later in this chapter), all net voltages and the currents into the pins at V1, R2, and D1 will display, reflecting the latest simulation results.

To display bias current through V1, R2, and D1

- 1 In Schematics, make the `clipper.sch` window active.
- 2 On the Simulation toolbar, click the Enable Bias Current Display button.
- 3 From the Edit menu, select the Select All command.
- 4 On the Simulation toolbar, click the Show/Hide Currents on Selected Part(s) button.
- 5 Make sure that no schematic components are selected (by clicking a blank space on the schematic), then shift-click the V1, R2, and D1 symbols.
- 6 On the Simulation toolbar, click the Show/Hide Currents on Selected Part(s) button. The currents into the pins of V1, R2, and D1 appear.

To turn the display of bias information off

- 1 On the Simulation toolbar, click the Enable Bias Voltage Display button.
- 2 On the Simulation toolbar, click the Enable Bias Current Display button.

Voltage and current levels no longer display on the schematic.

Using the Simulation Output File

The simulation output file acts as an audit trail of the simulation. This file optionally echoes the contents of the circuit file as well as the results of the bias point calculation. If there are any syntax errors in the netlist declarations or simulation commands, or anomalies while performing the calculation, PSpice writes error or warning messages to the output file.

To view the results of the bias point calculation directly on your schematic, see [Using the Bias Information Display on page 2-7](#).

To view the simulation output file

- 1 In Schematics, from the Analysis menu, select Examine Output to display the output file in the MicroSim Text Editor window.

Figure 2-5 shows the results of the bias point calculation as written in the simulation output file (`clipper.out`).

```

* C:\Tutorial\Clipper.sch
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C
*****
NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
( In) 0.0000 ( Mid) .9434 ( Out) 0.0000 ( Vcc) 5.0000

VOLTAGE SOURCE CURRENTS
NAME CURRENT
V_Vin 9.434E-04
V_V1 -1.229E-03
TOTAL POWER DISSIPATION 6.15E-03 WATTS

|
**** 01/07/97 09:20:52 **** Win95 PSpice 7.1 (October 1996) *** ID# 10807 ****
* C:\Tutorial\Clipper.sch
**** OPERATING POINT INFORMATION TEMPERATURE = 27.000 DEG C

```

Figure 2-5 Simulation Output File

- 2 When finished, close the MicroSim Text Editor window.

Note that the current through VIN is negative. By convention, PSpice measures the current through a two terminal device into the first terminal and out of the second terminal. For voltage sources, current is measured from the positive terminal to the negative terminal; this is opposite to the positive current flow convention and results in a negative value in the output file.

Finding Out More about Bias Point Calculations

To find out more about this...	See this...
Bias point calculations	Bias Point Detail on page 8-8
Viewing bias information on your schematic	Viewing Bias Point Voltages and Currents on page 14-2

DC Sweep Analysis

You can visually verify the DC response of the clipper by performing a DC sweep of the input voltage source and displaying the waveform results in Probe. This example sets up DC sweep analysis parameters to sweep Vin from -10 to 15 volts in 1 volt increments.

Setting Up and Running a DC Sweep Analysis

To set up and run a DC sweep analysis



- 1 From the Analysis menu, select Setup.

- 2 In the Analysis Setup dialog box, click the DC Sweep button.
- 3 Set up the DC Sweep dialog box as shown in Figure 2-6.

Note *The default settings for the DC Sweep dialog box are Voltage Source as the swept variable type and Linear as the sweep type. To choose a different swept variable type or sweep type, click the appropriate button.*

- 4 Click OK to close the DC Sweep dialog box.
- 5 Click Close to exit the Analysis Setup dialog box.
- 6 From the File menu, select Save.
- 7 From the Analysis menu, select Simulate to run the analysis as specified.

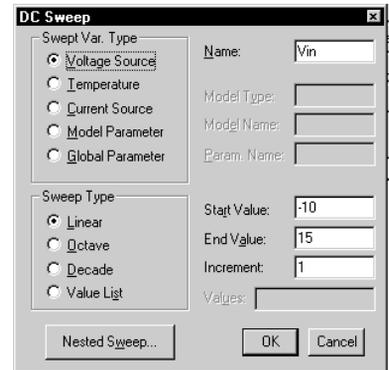


Figure 2-6 DC Sweep Dialog Box



or press **F11**

Displaying DC Analysis Results in Probe

If Probe is set up to automatically open upon successful completion of a simulation (the default setting), the Probe window appears when the simulation is finished. The Probe window includes one or more plot windows like the one shown in Figure 2-7.

To plot voltages at nets In and Mid

- 1 If the Probe window is not yet opened, from the Analysis menu, select Run Probe.
- 2 From the Trace menu, select Add.
- 3 Click V(In) and V(Mid) in the Add Traces dialog box.
- 4 Click OK.



or press **Insert**

To display a trace using a marker

- 1 In Schematics, from the Markers menu, select Mark Voltage/Level.

press **Ctrl + M**

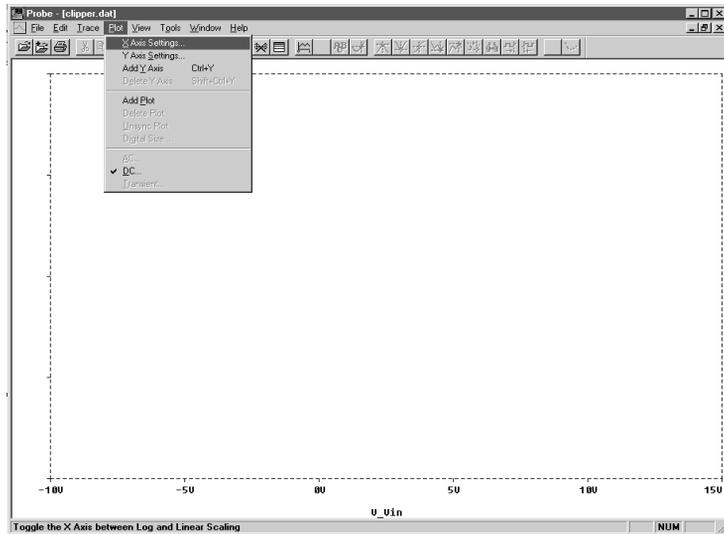


Figure 2-7 Probe Plot

- 2 Click to place a marker on net Out (Figure 2-8).

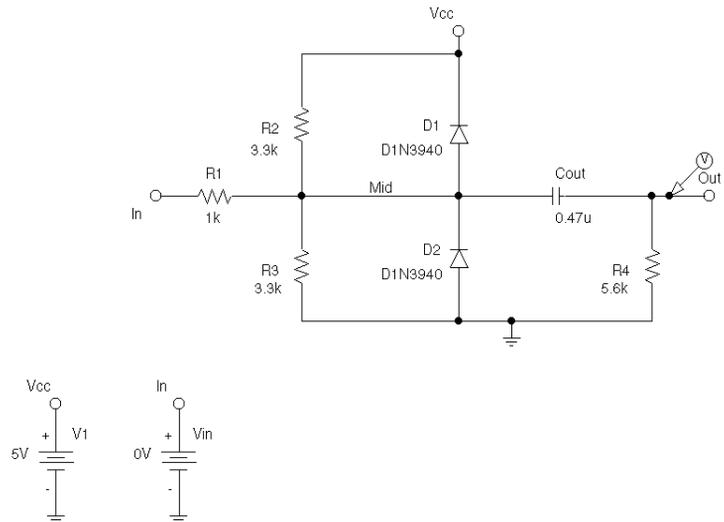


Figure 2-8 Clipper Circuit with Voltage Marker on Net Out

Schematics saves markers with the schematic files.

or press  **Ctrl** + **S**

- 3 Right-click to cancel marker mode.
- 4 Activate the Probe window. The V(Out) waveform trace appears as shown in Figure 2-9.
- 5 From the File menu, select Save.

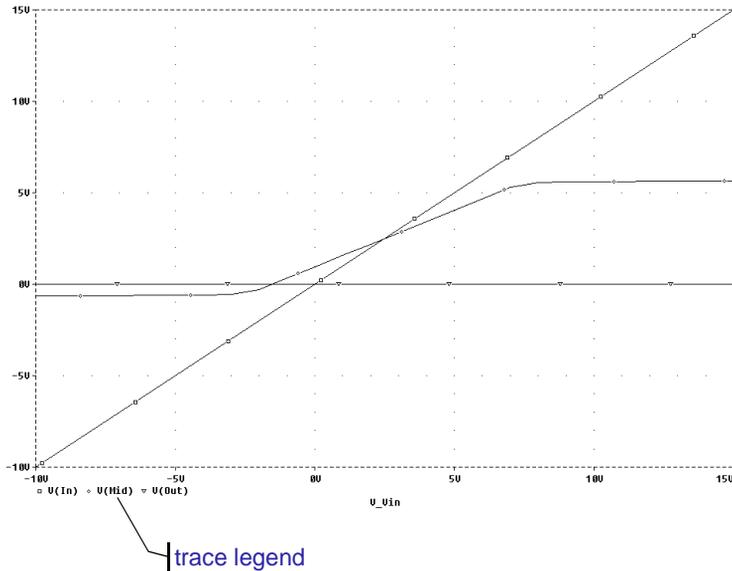


Figure 2-9 Voltage at In, Mid, and Out

To place cursors on V(In) and V(Mid)

- 1 In Probe, from the Tools menu, point to Cursor, then select Display.

Two cursors appear for the first trace defined in the legend below the x-axis—V(In) in this example. The Probe Cursor window also appears.

- 2 To display the cursor crosshairs:
 - a Position the mouse anywhere inside the plot window.
 - b Click to display the crosshairs for the first cursor.
 - c Right-click to display the crosshairs for the second cursor.

In the trace legend, the symbol for V(In) is outlined in the crosshair pattern for each cursor, resulting in a dashed line as shown in Figure 2-10.

This example uses the cursors feature to view the numeric values for two traces and the difference between them by placing a cursor on each trace.

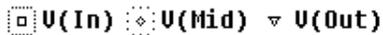
Table 2-1 Association of Probe Cursors with Mouse Buttons

Cursor 1	left mouse button
Cursor 2	right mouse button

V(In) ♦ V(Mid) ▽ V(Out)

Figure 2-10 Trace Legend

Your ability to get as close to 4.0 as possible depends on screen resolution and window size.



V(In) **V(Mid)** ▾ **V(Out)**

Figure 2-11 Trace Legend with V(Mid) Symbol Outlined

- 3** Place the first cursor on the V(In) waveform:
 - a** Click the portion of the V(In) trace in the proximity of 4 volts on the x-axis. The cursor crosshair appears, and the current X and Y values for the first cursor appear in the Probe Cursor window.
 - b** To fine-tune the cursor location to 4 volts on the x-axis, drag the crosshairs until the x-axis value of the A1 cursor in the Probe Cursor window is approximately 4.0. You can also press **→** and **←** for tighter control.
- 4** Place the second cursor on the V(Mid) waveform:
 - a** Right-click the trace legend symbol (diamond) for V(Mid) to associate the second cursor with the Mid waveform. The crosshair pattern for the second cursor outlines the V(Mid) trace symbol as shown in Figure 2-11.
 - b** Right-click the portion on the V(Mid) trace that is in the proximity of 4 volts on the x-axis. The X and Y values for the second cursor appear in the Probe Cursor window along with the difference (dif) between the two cursors' X and Y values.
 - c** To fine-tune the location of the second cursor to 4 volts on the x-axis, drag the crosshairs until the x-axis value of the A2 cursor in the Probe Cursor window is approximately 4.0. You can also press **Shift+→** and **Shift+←** for tighter control.

Figure 2-12 shows the Probe window when both cursors are placed.

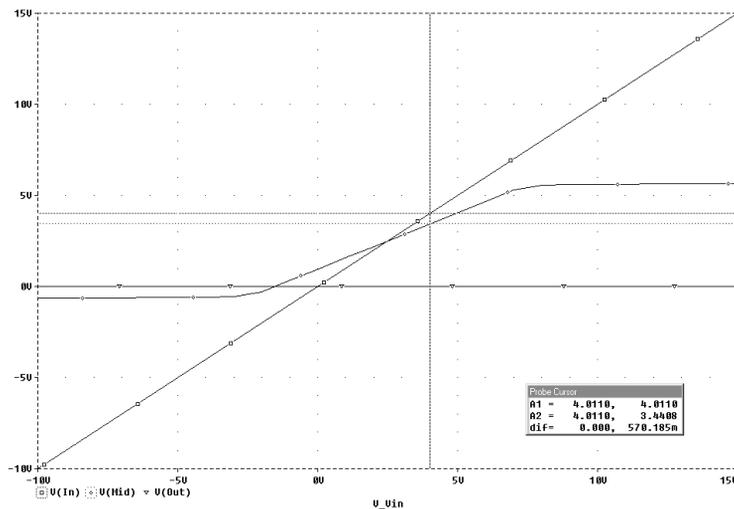


Figure 2-12 Voltage Difference at $V(In) = 4$ Volts

To delete all of the traces

- 1 From the Trace menu, select Delete All.

At this point, the schematic has been saved. If needed, you can quit Schematics and Probe and complete the remaining analysis exercises later using the saved schematic.

Finding Out More about DC Sweep Analysis

To find out more about this...

See this...

DC sweep analysis

[DC Sweep on page 8-2](#)



There are also ways to display the difference between two voltages as a trace:

- In Probe, add the trace expression $V(In)-V(Mid)$.
- In Schematics, from the Markers menu, select Mark Voltage Differential and place the two markers on different pins or wires.



You can also delete an individual trace by selecting its name in the trace legend and then pressing **[Del]**.

Example: To delete the $V(In)$ trace, click the text, $V(In)$, located under the plot's x-axis, and then press **[Del]**.

Transient Analysis

This example shows how to run a transient analysis on the clipper circuit. This requires adding a time-domain voltage stimulus as shown in Figure 2-13.

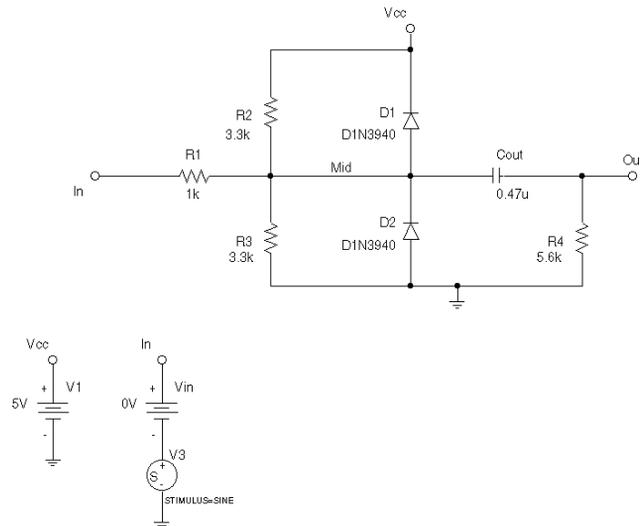


Figure 2-13 Diode Clipper Circuit with a Voltage Stimulus

To add a time-domain voltage stimulus

- 1 In Schematics, from the Markers menu, select Clear All.
- 2 Select the ground symbol beneath the VIN source.
- 3 From the Edit menu, select Cut.
- 4 Scroll down or select Out from the View menu.
- 5 Place a VSTIM symbol as shown in Figure 2-13.



If you do not have the Stimulus Editor:

- 1 Place a VSIN symbol instead of VSTIM, then double click it.
- 2 Set values for the VOFF, VAMPL, and FREQ attributes as defined in step 13. Click Save Attr after typing each attribute's value to accept the changes. When finished, click OK.

- 6 From the Edit menu, select Paste.
- 7 Place the ground symbol under the VSTIM symbol as shown in Figure 2-13.
- 8 From the View menu, select Fit.
- 9 From the File menu, select Save As, and then type `clippert.sch` as the name of the schematic file you want to save.
- 10 Double-click the VSTIM symbol.
- 11 In the Set Attribute Value dialog box, type `SINE`, then click OK. The New Stimulus dialog box and the Stimulus Editor appear.
- 12 In the Stimulus Editor, click `SIN`, then click OK.
- 13 In the SIN Attributes dialog box, set the first three parameters as follows:
 - Offset Voltage = 0
 - Amplitude = 10
 - Frequency = 1kHz
- 14 Click Apply to view the waveform. The Stimulus Editor window redisplay and looks like Figure 2-14.



or press **Ctrl** + **V**

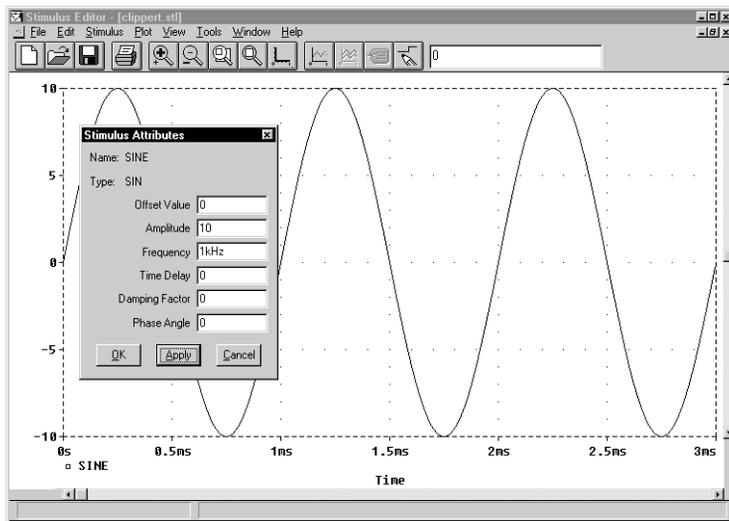


Figure 2-14 Stimulus Editor Window



or press **Shift** + **F12**

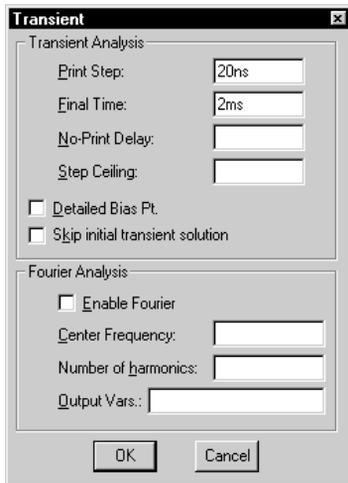


Figure 2-15 *Transient Analysis*



or press **F11**

15 Click OK.

16 From the File menu, select Save to save the stimulus information.

17 From the File menu, select Exit.

To set up and run the transient analysis

1 In Schematics, from the Analysis menu, select Setup.
2 In the Analysis Setup dialog box, click Transient to display the Transient Analysis dialog box.

3 Set up the Transient dialog box as shown in Figure 2-15.

4 Click OK.

5 Clear the DC Sweep check box to disable the DC sweep from the previous example.

DC Sweep is disabled here so you can see the results of a transient analysis run by itself. PSpice can run multiple analyses during simulation (for example, both DC sweep and transient analyses).

6 Click Close to exit the Analysis Setup dialog box.

7 From the File menu, select Save.

8 From the Analysis menu, select Simulate.

PSpice uses its own internal time steps for computation. The internal time step is adjusted according to the requirements of the transient analysis as it proceeds. PSpice saves data to the Probe data file for each internal time step.

Note *The internal time step is different from the Print Step value. Print Step controls how often optional text format data is written to the simulation output file (.OUT).*

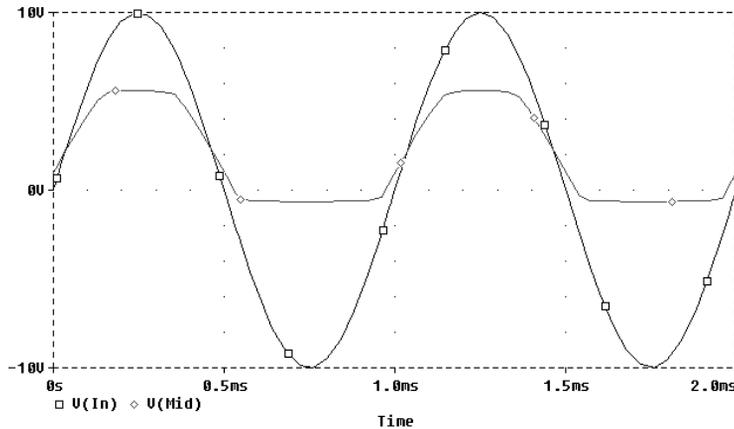
To display the input sine wave and clipped wave at V(Out)

1 In Probe, from the Trace menu, select Add.

2 Select V(In) and V(Out) by clicking them in the trace list.

3 Click OK to display the traces.

- 4 Place the symbols shown in the trace legend on the traces themselves as shown in Figure 2-16:
 - a From the Tools menu, select Options to display the Probe Options dialog box.
 - b In the Use Symbols frame, click Always.
 - c Click OK.



These waveforms illustrate the clipping of the input signal.

Figure 2-16 *Sinusoidal Input and Clipped Output Waveforms*

Finding Out More about Transient Analysis

To find out more about this...

See this...

Transient analysis for analog designs*

[Chapter 10, Transient Analysis](#)

*. Includes how to set up time-based stimuli using the Stimulus Editor.

AC Sweep Analysis

The AC sweep analysis in PSpice is a linear (or small signal) frequency domain analysis that can be used to observe the frequency response of any circuit at its bias point.

Setting Up and Running an AC Sweep Analysis

In this example, you will set up the clipper circuit for AC analysis by adding an AC voltage source for a stimulus signal (Figure 2-17) and by setting up AC sweep parameters.

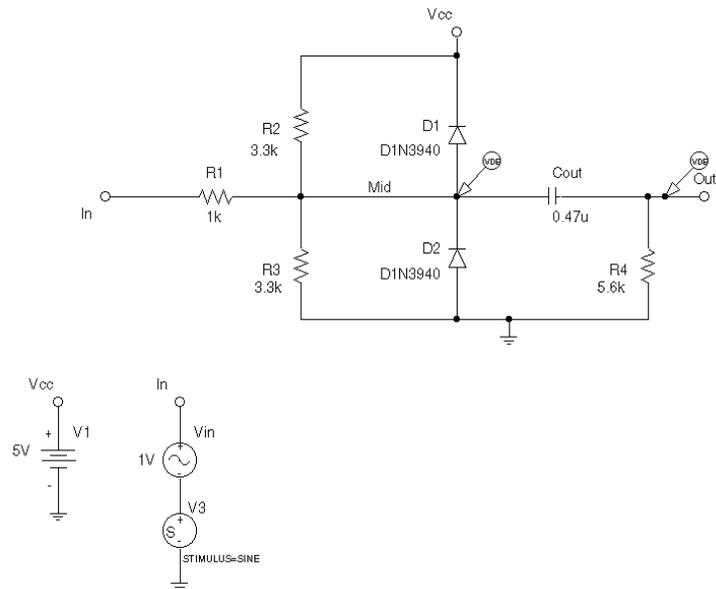


Figure 2-17 *Clipper Circuit with AC Stimulus*

Time-domain and AC stimuli are independent of one another. For example, the SINE stimulus is ignored (0V) during AC analysis.

To change Vin to include the AC stimulus signal

- 1 In Schematics, open `clippert.sch`.
- 2 Click the DC voltage source, `Vin`, to select it.
- 3 From the Edit menu, select Replace.

- 4 In the Replace Part dialog box, type `VAC`.
- 5 Select (✓) the Keep Attribute Values check box.
- 6 Click OK. The input voltage source changes to an AC voltage source.
- 7 Double-click the displayed (AC) value of the new Vin.
- 8 In the Set Attribute Value dialog box, set the value to 1V.

To set up the AC sweep and start simulation

- 1 From the Analysis menu, select Setup.
- 2 In the Analysis Setup dialog box, click AC Sweep.
- 3 Set up the AC Sweep and Noise Analysis dialog box as shown in Figure 2-18.

Note *PSpice is not case sensitive, so both M and m can be used as “milli,” and MEG, Meg, and meg can all be used for “mega.” However, Probe is case sensitive for M and m, and will read them as mega and milli, respectively.*

- 4 Click OK to close the AC Sweep dialog box.
- 5 Click Close to exit the Analysis Setup dialog box.
- 6 From the Markers menu, select Mark Advanced.
- 7 Double-click Vdb.
- 8 Place one Vdb marker on the output net, and place another on the Mid net.
- 9 From the File menu, select Save As, and then type `clippera.sch` as the name of the schematic file you want to save.
- 10 From the Analysis menu, select Simulate to start the simulation.

Because the transient analysis was still enabled, PSpice performs both the transient and AC analyses. The Probe window and the Analysis Type dialog box appear.

- 11 In the Analysis Type dialog box, click AC.

The new Vin still has a DC attribute that you can use to include a bias with the AC source. Double-click the AC source to see the DC attribute value.

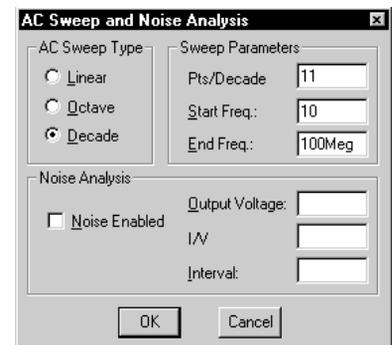


Figure 2-18 AC Sweep and Noise Analysis Dialog Box



or press **F11**

If Probe is not set to run automatically after simulation, from the Analysis menu, select Run Probe.

AC Sweep Analysis Results

Probe displays the dB magnitude ($20\log_{10}$) of the voltage at the marked nets, Out and Mid, as shown in Figure 2-19. VDB(Mid) has a lowpass response due to the diode capacitances to ground. The output capacitance and load resistor act as a highpass filter, so the overall response, illustrated by VDB(out), is a bandpass response. Because AC is a linear analysis and the input voltage was set to 1V, the output voltage is the same as the gain (or attenuation) of the circuit.

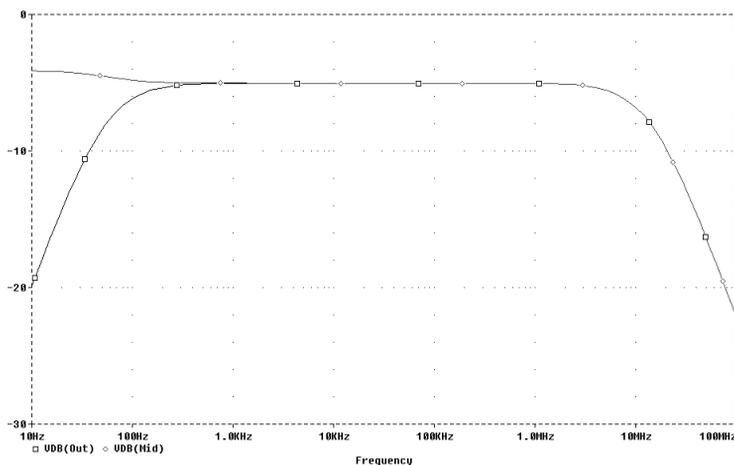


Figure 2-19 dB Magnitude Curves for “Gain” at Mid and Out

To display a Bode plot of the output voltage, including phase

- 1 In Schematics, from the Markers menu, select Mark Advanced.
- 2 Place a Vphase marker on the output next to the Vdb marker.
- 3 Delete the Vdb marker on Mid.
- 4 Activate the Probe window. The gain and phase plots both appear on the same graph with the same scale.
- 5 Click the trace name VP(Out) to select it.

Note Depending upon where the Vphase marker was placed, the trace name may be different, such as VP(Cout:2), VP(R4:1), or VP(R4:2).

- 6 From the Edit menu, select Cut.
- 7 From the Plot menu, select Add Y Axis.
- 8 From the Edit menu, select Paste. The Bode plot appears as shown in Figure 2-20.



or press **Ctrl** + **X**



or press **Ctrl** + **V**

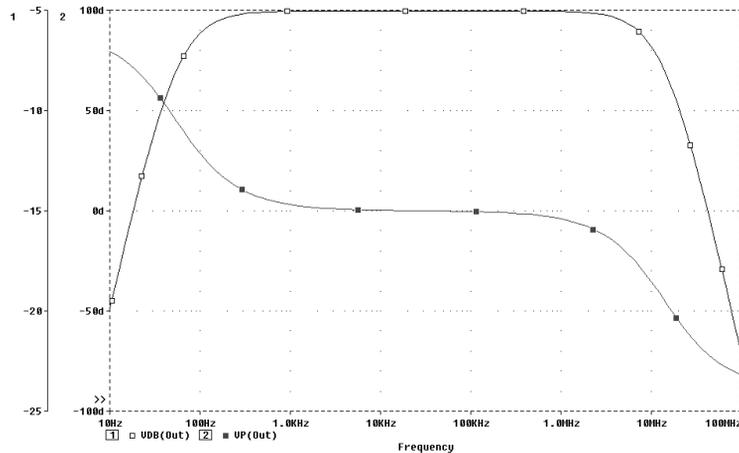


Figure 2-20 Bode Plot of Clipper's Frequency Response

Finding Out More about AC Sweep and Noise Analysis

To find out more about this...

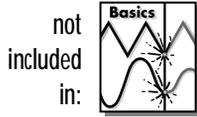
See this...

AC sweep analysis

[AC Sweep Analysis on page 9-2](#)

Noise analysis based on an AC sweep analysis

[Noise Analysis on page 9-9](#)



Parametric Analysis

This example shows the effect of varying input resistance on the bandwidth and gain of the clipper circuit by:

- Changing the value of R1 to the expression {Rval}.
- Adding a PARAM symbol to declare the parameter Rval.
- Specifying a parametric analysis to step the value of R1 using Rval.

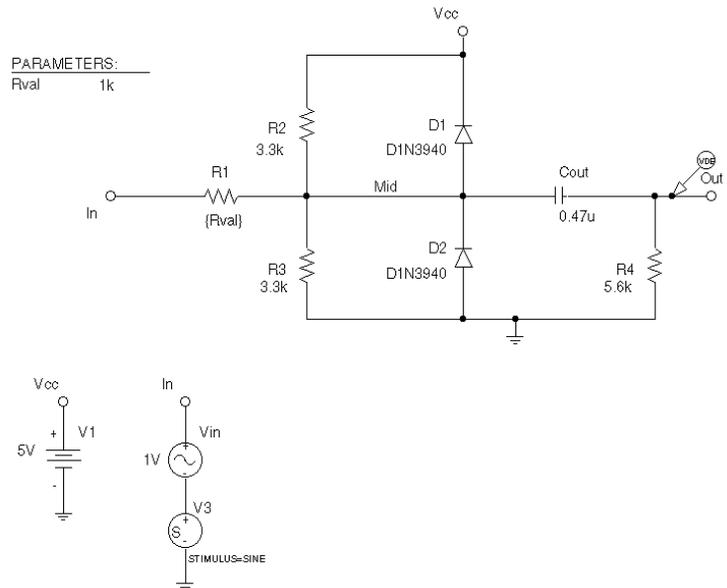


Figure 2-21 *Clipper Circuit with Global Parameter Rval*

The example results in multiple analysis runs, each with a different value of R1. Once the analysis is complete, you can analyze curve families for the analysis runs in Probe.

Setting Up and Running the Parametric Analysis

To change the value of R1 to the expression {Rval}

- 1 In Schematics, open `clippera.sch`.
- 2 Double-click the value label for R1.
- 3 In the Set Attribute Value dialog box, type `{Rval}`.
- 4 Click OK.

PSpice interprets text in curly braces as an expression that evaluates to a numerical value. This example uses the simplest form of an expression—a constant. The value of R1 will take on the value of the Rval parameter, whatever it may be.

To add a PARAM symbol to declare the parameter Rval

- 1 From the Draw menu, select Get New Part.
- 2 In the Part Name text box, type `PARAM`, then click Place & Close.
- 3 Place one `PARAM` symbol on any open space on the schematic.
- 4 Double-click the `PARAM` symbol to display the attributes list.
- 5 Double-click `NAME1` and type `Rval` (no curly braces) in the Value text box.
- 6 Click Save Attr to accept the change.
- 7 Double-click `VALUE1`, type `1k`, then click Save Attr.
- 8 Click OK. `Rval 1k` appears in the `PARAMETERS` list on the schematic.



or press **Ctrl**+**G**



This setup specifies that the parameter Rval is to be stepped from 100 to 10k logarithmically with a resolution of 10 points per decade.

The analysis is run for each value of Rval. Because the value of R1 is defined as {Rval}, the analysis is run for each value of R1 as it logarithmically increases from 100Ω to 10 kΩ in 20 steps, resulting in a total of 21 runs.



or press **F11**

To set up and run a parametric analysis to step the value of R1 using Rval

- 1 From the Analysis menu, select Setup.
- 2 In the Analysis Setup dialog box, click Parametric.
- 3 Set up the Parametric dialog box as shown below.

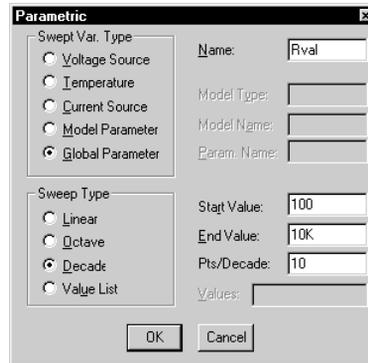


Figure 2-22 Parametric Dialog Box

- 4 Click OK.
- 5 Clear the Transient check box, and click Close to exit the Analysis Setup dialog box.
- 6 From the File menu, select Save As, and save the schematic as clipperp.sch.
- 7 Delete the VP marker. (For this example, we are only interested in the magnitude of the response.)
- 8 From the Analysis menu, select Simulate to run the analysis as specified.

Analyzing Waveform Families in Probe

There are 21 analysis runs, each with a different value of R1. When Probe starts, it displays the Available Sections dialog box that lists all 21 runs and the Rval parameter value for each. You have the option to select one or more runs.

To display all 21 traces in Probe

- 1 In the Available Sections dialog box, click OK to accept the default of all runs. All 21 traces (the entire family of curves) for VDB(Out) appear in Probe as shown in Figure 2-23.

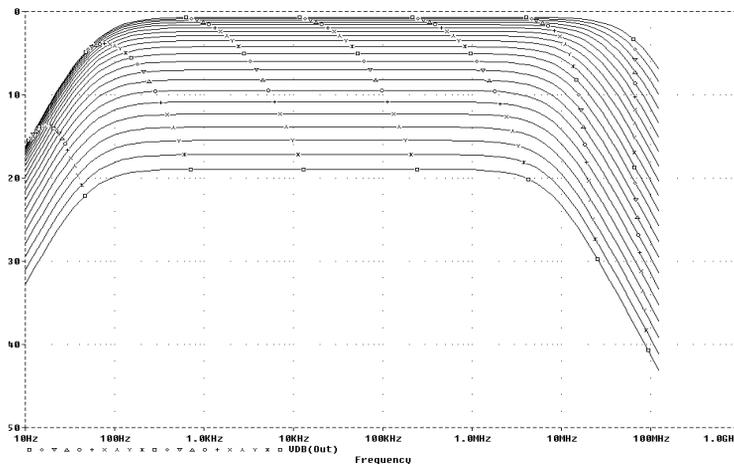


Figure 2-23 *Small Signal Response as R1 is Varied from 100Ω to 10 kΩ*

- 2 Click the trace name to select it and then press **Del** to remove the traces shown.

To compare the last run to the first run

- 1 From the Trace menu, select Add.
- 2 In the Trace Expression text box, type the following:
`Vdb(Out)@1 Vdb(Out)@21`
- 3 Click OK.

If Probe is not set to run automatically after simulation, from the Analysis menu, select Run Probe.

To select individual runs, click each one separately.

To see more information about the section that produced a specific trace, double-click the corresponding symbol in the legend below the x-axis.

You can also remove the traces by deleting the VDB marker in Schematics.



or press **Insert**

You can avoid some of the typing for the Trace Expression text box by selecting V(OUT) twice in the trace list and inserting text where appropriate in the resulting Trace Expression.

Note *The difference in gain is apparent. You can also plot the difference of the waveforms for runs 21 and 1 and then use the search commands feature to find certain characteristics of the difference.*

or press  **Insert**

4 Plot the new trace by specifying a waveform expression:

a From the Trace menu, select Add.

b In the Trace Expression text box, type the following waveform expression:

```
Vdb(Out )@1-Vdb(OUT )@21
```

c Click OK.

5 Use the search commands feature to find the value of the difference trace at its maximum and at a specific frequency:

a From the Tools menu, point to Cursor, then select Display.

b Right-click then left-click the trace symbol (triangle) for Vdb(Out)@1 - Vdb(Out)@21. Make sure that you left-click last to make cursor 1 the active cursor.

c From the Tools menu, point to Cursor, then select Max.

d From the Tools menu, point to Cursor, then select Search Commands.

e In the Search Command text box, type the following:

```
search forward x value (100)
```

f Choose 2 as the Cursor to Move option.

g Click OK.

Figure 2-24 shows the Probe window when the cursors are placed.

Note that the Y value for cursor 2 in the cursor box is about 17.87. This indicates that when R1 is set to 10 k Ω , the small signal attenuation of the circuit at 100 Hz is 17.87 dB greater than when R1 is 100 Ω .

The search command instructs Probe to search for the point on the trace where the x-axis value is 100.



6 From the Tools menu, point to Cursor, then select Display to deactivate the cursors.

7 Delete the trace.

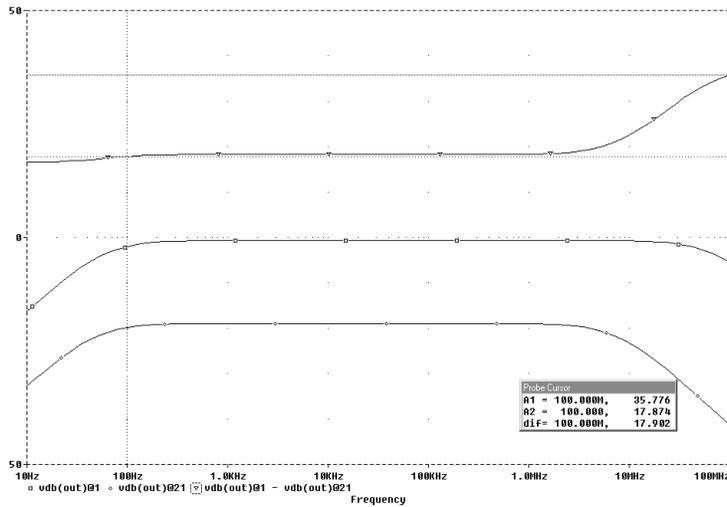


Figure 2-24 Comparison of Small Signal Frequency Response at 100 and 10 k Ω Input Resistance

Finding Out More about Parametric Analysis

To find out more about this...

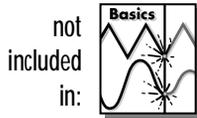
See this...

Parametric analysis

[Parametric Analysis on page 11-2](#)

Using global parameters

[Using Global Parameters and Expressions for Values on page 3-12](#)



Probe Performance Analysis

Performance analysis is an advanced feature in Probe that you can use to compare the characteristics of a family of waveforms. Performance analysis uses the principle of search commands introduced earlier in this chapter to define functions that detect points on each curve in the family.

Once you have defined these functions, you can apply them to a family of waveforms and produce traces that are a function of the variable that changed within the family.

This example shows how to use the performance analysis feature of Probe to view the dependence of circuit characteristics on a swept parameter. In this case, the small signal bandwidth and gain of the clipper circuit are plotted against the swept input resistance value.

To plot bandwidth vs. Rval using the performance analysis wizard

- 1 In Schematics, open `clipperp.sch` and run Probe.
- 2 In Probe, from the Trace menu, select Performance Analysis.

The Performance Analysis dialog box appears with information about the currently loaded data and performance analysis in general.

- 3 Click Wizard.
- 4 Click Next>.
- 5 In the Choose a Goal Function list, click Bandwidth, then click Next>.
- 6 Click in the Name of Trace text box and type `V(Out)`.
- 7 Click in the db level down for bandwidth calc text box and type 3.
- 8 Click Next>. The wizard displays the gain trace for the first run ($R=100$) and shows how the bandwidth is measured. This is done to test the goal function.

At each step, the wizard provides information and guidelines.

Click , then double-click `V(Out)`.

- 9 Click Next> or Finish. Probe displays a plot of the 3 dB bandwidth vs. Rval.
- 10 Change the x-axis to log scale.
 - a From the Plot menu, select X Axis Settings.
 - b In the Scale frame of the X Axis dialog box, choose Log.
 - c Click OK.

Double-click the x-axis.

To plot gain vs. Rval manually

- 1 From the Plot menu, select Add Y Axis.
- 2 From the Trace menu, select Add.
- 3 In the Functions and Macros frame, change to the Goal Functions list, and then click the Max(1) goal function.
- 4 In the Simulation Output Variables list, click V(out).
- 5 In the Trace Expression text box, edit the text to be $\text{Max}(V_{db}(\text{out}))$, then click OK. Probe displays gain on the second y-axis vs. Rval.



or press **Insert**

The Trace list includes goal functions only in performance analysis mode when the x-axis variable is the swept parameter.

Figure 2-25 shows the final performance analysis plot of 3 dB bandwidth and gain in dB vs. the swept input resistance value.

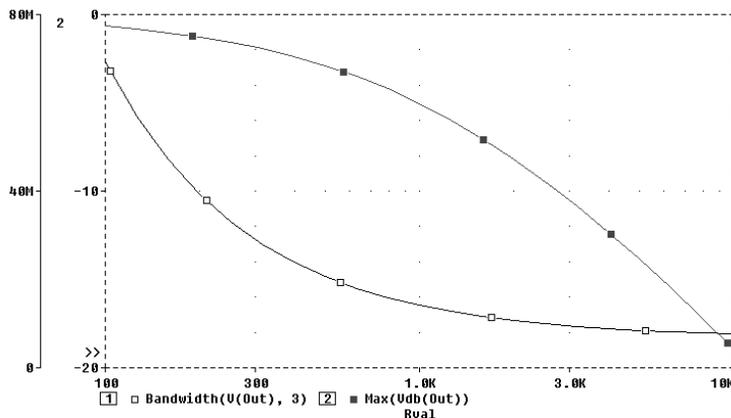


Figure 2-25 Performance Analysis Plots of Bandwidth and Gain vs. Rval

Finding Out More about Performance Analysis

To find out more about this...	See this...
How to use performance analysis	Example: RLC Filter on page 11-3 Tutorial: Monte Carlo Analysis of a Pressure Sensor on page 12-10
How to use search commands and create goal functions	Probe online help

Part Two

Design Entry

Part Two provides information about how to enter circuit designs that you want to simulate in MicroSim Schematics.

[Chapter 3, Preparing a Schematic for Simulation](#), outlines the things you need to do to successfully simulate your schematic including troubleshooting tips for the most frequently asked questions.

[Chapter 4, Creating and Editing Models](#), describes how to use the tools to create and edit model definitions, and how to configure the models for use.

[Chapter 5, Creating Symbols for Models](#), explains how to create symbols for existing or new model definitions so you can use the models when simulating from your schematic.

[Chapter 6, Analog Behavioral Modeling](#), describes how to model analog behavior mathematically or using table lookups.

Preparing a Schematic for Simulation

3

Chapter Overview

This chapter provides introductory information to help you enter circuit designs that simulate properly. If you want an overview, use the checklist on page [3-2](#) to guide you to specific topics.

Topics include:

[Checklist for Simulation Setup on page 3-2](#)

[Using Parts That You Can Simulate on page 3-6](#)

[Using Global Parameters and Expressions for Values on page 3-12](#)

[Defining Power Supplies on page 3-18](#)

[Defining Stimuli on page 3-19](#)

[Things to Watch For on page 3-22](#)

Refer to your *MicroSim Schematics User's Guide* for information that is general to schematic entry.

Checklist for Simulation Setup

This section is provided so you can quickly step through what you need to do to set up your circuit for simulation.

- 1 Find the topic that is of interest in the first column of any of these tables.
- 2 Go to the referenced section. For those sections that provide overviews, you will find references to more detailed discussions.

Typical Simulation Setup Steps

For more information on this step...	See this...	To find out this...
✓ Set component values and other attributes.	Using Parts That You Can Simulate on page 3-6 Using Global Parameters and Expressions for Values on page 3-12	An overview of vendor, passive, breakout, and behavioral parts. How to define values using variable parameters, function calls, and mathematical expressions.
✓ Define power supplies.	Defining Power Supplies on page 3-18	An overview of DC power.
✓ Define input waveforms.	Defining Stimuli on page 3-19	An overview of DC, AC, and time-based stimulus symbols.
✓ Set up one or more analyses.	Chapter 7, Setting Up Analyses and Starting Simulation Chapter 8 through Chapter 15 (see the table of contents)	Procedures, general to all analysis types, to set up and start the simulation. Detailed information about DC, AC, transient, parametric, temperature, Monte Carlo, and sensitivity/worst-case.

For more information on this step...	See this...	To find out this...
✓ Place markers.	Using Schematic Markers to Add Traces on page 13-12 Limiting Probe Data File Size on page 13-15	How to display results in Probe by picking schematic nets. How to limit the Probe data file size using markers.

Advanced Design Entry and Simulation Setup Steps

For more information on this step...	See this...	To find out how to...
✓ Create new models.	Chapter 4, Creating and Editing Models	Define models using the Parts utility, model editor, or Create Subcircuit command.
	Chapter 6, Analog Behavioral Modeling	Define the behavior of a block of analog circuitry as a mathematical function or lookup table.
✓ Create new symbols.	Chapter 5, Creating Symbols for Models	Create symbols either automatically for models using the symbol wizard or the Parts utility, or by manually defining AKO symbols; define simulation-specific attributes.
	In your <i>MicroSim Schematics User's Guide: Using the Symbol Editor</i> and <i>Creating and Editing Symbols</i> chapters	Create and edit symbol graphics, pins, and attributes in general.

When Netlisting Fails or the Simulation Does Not Start

If you have problems starting the simulation, there may be problems with the schematic or with system resources. If there are problems with the schematic, Schematics or PSpice issues errors and warnings to the Message Viewer. You can use the Message Viewer to get more information quickly about the specific problem.

To get online information about an error or warning shown in the Message Viewer

- 1 Select the error or warning message.
- 2 Press **F1**.

The following tables list the most commonly encountered problems and where to find out more about what to do.

Things to check in your schematic

Make sure that...	To find out more, see this...
✓ The model libraries, stimulus files, and include files are configured.	Configuring Model Libraries on page 4-41
✓ You are using symbols with models.	Unmodeled Parts on page 3-22 and Defining Symbol Attributes Needed for Simulation on page 5-18
✓ You are not using unmodeled pins.	Unmodeled Pins on page 3-25
✓ You have defined the grounds.	Missing Ground on page 3-26
✓ Every analog net has a DC path to ground.	Missing DC Path to Ground on page 3-27
✓ The symbol template is correct.	Defining Symbol Attributes Needed for Simulation on page 5-18
✓ Hierarchical parts, if used, are properly defined.	In your <i>MicroSim Schematics User's Guide</i> , the <i>Creating and Editing Hierarchical Designs</i> chapter
✓ Ports that connect to the same net have the same name.	In your <i>MicroSim Schematics User's Guide</i> , the <i>Using Ports</i> section in the <i>Creating and Editing Designs</i> chapter

Things to check in your system configuration

Make sure that...	To find out more, see this...
✓ Path to the PSpice and Probe programs is correct.	In your <i>MicroSim Schematics User's Guide</i> : the <i>Changing Application Settings</i> section in the <i>Using the Schematic Editor</i> chapter
✓ Directory containing your schematic file has write permission.	Your operating system manual
✓ Your system has sufficient free memory and disk space.	Your operating system manual

Using Parts That You Can Simulate



The MicroSim libraries

include special symbols that you can use for simulation only. These include:

- **stimulus symbols** to generate input signals to the circuit (see [Defining Stimuli on page 3-19](#))
- **ground symbols** required by all circuits, which need reference to ground
- **simulation control symbols** to do things like set bias values (see [Appendix A, Setting Initial State](#))
- **output control symbols** to do things like generate tables and line-printer plots to the PSpice output file (see [Chapter 15, Other Output Options](#))

The MicroSim libraries supply numerous parts designed for simulation. These include:

- vendor-supplied parts
- passive parts
- breakout parts
- behavioral parts

At minimum, a part that you can simulate has these properties:

- A simulation model to describe the part's electrical behavior; the model can be:
 - explicitly defined in a model library,
 - built into PSpice, or
 - built into the symbol (for some kinds of analog behavioral parts).
- A symbol with modeled pins to form electrical connections on your schematic.
- A translation from schematic symbol to netlist statement so that PSpice can read it in.

Note *Not all parts in the libraries are set up for simulation. For example, connectors are parts destined for board layout only and do not have these simulation properties.*

Vendor-Supplied Parts

The MicroSim libraries provide an extensive selection of manufacturers' parts. Typically, the library name reflects the kind of parts contained in the library and the vendor that provided the models.

Example: `motor_rf.slb` and `motor_rf.lib` contain symbols and models, respectively, for Motorola-made RF bipolar transistors.

For a listing of vendor-supplied parts contained in the MicroSim libraries, refer to the online *Library List*.

To find out more about each model library, read the comments in the `.lib` file header.

Part naming conventions

The part names in the MicroSim libraries usually reflect the manufacturers' part names. If multiple vendors supply the same part, each part name includes a suffix that indicates the vendor that supplied the model.

Example: The MicroSim libraries include several models for the OP-27 opamp as shown by these entries in the online *Library List*.

Device Type	Generic Name	Mfg. Name	Symbol Name	Library	Tech Type	Model	Package	New for 8.0
Operational Amplifier	OP-249	Analog Devices Inc.	OP-249G/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-260	Analog Devices Inc.	OP-260/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27A/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27B/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27C/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27E/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27F/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Analog Devices Inc.	OP-27G/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27	Linear Technology Corp.	OP-27/LT	LIN_TECH.SLB	*	*		
Operational Amplifier	OP-27		OP-27	OPAMP.SLB	*	*		
Operational Amplifier	OP-275	Analog Devices Inc.	OP-275/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-275	Analog Devices Inc.	OP-275G/AD	ANLG_DEV.SLB	*	*		
Operational Amplifier	OP-27A	Linear Technology Corp.	OP-27A/LT	LIN_TECH.SLB	*	*		
Operational Amplifier	OP-27C	Linear Technology Corp.	OP-27C/LT	LIN_TECH.SLB	*	*		

Notice the following:

- There is a generic OP-27 symbol provided by MicroSim, the OP-27/AD from Analog Devices, Inc., and the OP-27/LT from Linear Technology Corporation.
- The Model column for all of these parts contains an asterisk. This indicates that this part is modeled and that you can simulate it.



Finding the part that you want

If you are having trouble finding a part, you can search the libraries for parts with similar names by using either:

- the parts browser in Schematics and restricting the parts list to those names that match a specified wildcard text string, or
- the online *Library List* and searching for the generic part name using capabilities of the Adobe Acrobat Reader.

To find parts using the parts browser

- 1 In Schematics, from the Draw menu, select Get New Part.
- 2 In the Part Name text box, type a text string with wildcards that approximates the part name that you want to find. Use this syntax:

`<wildcard><part_name_fragment><wildcard>`

where `<wildcard>` is one of the following:

- * to match zero or more characters
- ? to match exactly one character

The parts browser displays only the matching part names.



Note This method finds any part contained in the current symbol libraries configuration, including symbols for user-defined parts.

If you want to find out more about a part supplied in the MicroSim libraries, such as manufacturer or whether you can simulate it, then search the online *Library List* (see page [3-9](#)).

To find parts using the online Library List

- 1 From the Help menu in Schematics, PSpice, or the Parts utility, select Library List.
- 2 From the Library List Help topic, click the button for the analog device types that you want to search.
- 3 From the Tools menu, select Find.
- 4 In the Find What text box, type the generic part name.
- 5 Enter any other search criteria, and then click Find.

The Acrobat Reader displays the first page where it finds a match. Each page maps the generic part name to the symbols (and corresponding vendor and symbol library name) in the MicroSim libraries.

- 6 If you want to repeat the search, from the Tools menu, select Find Again.

Note *If you are unsure of the device type, you can scan all of the device type lists using the Acrobat search capability. The first time you do this, you need to set up the across-list index. To find out more, refer to the online Library List help topics and the online Adobe Acrobat manuals.*

Note *This method finds only parts that MicroSim supplies.*

If you want to include user-defined parts in the search, use the parts browser in Schematics (see page [3-8](#)).

 or press **Ctrl** + **F**

Instead of the generic part name, you can enter other kinds of search information such as device type or manufacturer.

press **Ctrl** + **G**

Passive Parts

The MicroSim libraries supply several basic parts based on the passive device models built-in to PSpice. These are summarized in the following table.

To find out more about how to use these symbols and define their attributes, look up the corresponding PSpice device letter in the *Analog Devices* chapter in the online *MicroSim PSpice A/D Reference Manual*, and then look in the *Schematic Symbols* section.

These symbols are available...	For this part type...	Which is this PSpice device letter...
C C_VAR	capacitor	C
L	inductor	L
R R_VAR	resistor	R
XFRM_LINEAR K_LINEAR	transformer	K and L
T	ideal transmission line	T
TLOSSY*	Lossy transmission line	T
TnCOUPLED** TnCOUPLEDX** KCOUPLEn**	coupled transmission line	T and K

*. TLOSSY is not available in Basics packages.

** For these device types, the MicroSim libraries supply several parts. Refer to the online *MicroSim PSpice A/D Reference Manual* for the available symbols.

Breakout Parts

The MicroSim libraries supply passive and semiconductor parts with default model definitions that define a basic set of model parameters. This way, you can easily:

- assign device and lot tolerances to model parameters for Monte Carlo and sensitivity/worst-case analyses,
- define temperature coefficients, and
- define device-specific operating temperatures.

These are called breakout parts and are summarized in the following table.

Use this breakout part...	For this part type...	Which is this PSpice device letter...
BBREAK	GaAsFET	B
CBREAK	capacitor	C
DBREAK _x *	diode	D
JBREAK _x *	JFET	J
KBREAK	inductor coupling	K
LBREAK	inductor	L
MBREAK _x *	MOSFET	M
QBREAK _x *	bipolar transistor	Q
RBREAK	resistor	R
SBREAK	voltage-controlled switch	S
TBREAK	transmission line	T
WBREAK	current-controlled switch	W
XFRM_NONLINEAR	transformer	K and L
ZBREAKN	IGBT	Z

*. For this device type, the MicroSim libraries supply several breakout parts. Refer to the online *MicroSim PSpice A/D Reference Manual* for the available symbols.

To find out more about models, see [What Are Models? on page 4-3](#).

To find out more about Monte Carlo and sensitivity/worst-case analyses, see [Chapter 12, Monte Carlo and Sensitivity/Worst-Case Analyses](#).

To find out more about setting temperature parameters, see the *Analog Devices* chapter in the online *MicroSim PSpice A/D Reference Manual* and find the device type that you are interested in.

To find out more about how to use these symbols and define their attributes, look up the corresponding PSpice device letter in the *Analog Devices* chapter in the online *MicroSim PSpice A/D Reference Manual*, and then look in the *Schematics Symbols* section.

Behavioral Parts

Behavioral parts allow you to define how a block of circuitry should work without having to define each discrete component.

For more information, see [Chapter 6, Analog Behavioral Modeling](#).

These parts use analog behavioral modeling (ABM) to define each part's behavior as a mathematical expression or lookup table. The MicroSim libraries provide ABM parts that operate as math functions, limiters, Chebyshev filters, integrators, differentiators, and others that you can customize for specific expressions and lookup tables. You can also create your own ABM parts.

Using Global Parameters and Expressions for Values

In addition to literal values, you can use global parameters and expressions to represent numeric values in your circuit design.

Global Parameters

When multiple parts are set to the same value, global parameters provide a convenient way to change all of their values for "what-if" analyses.

Example: If two independent sources have a value defined by the parameter VSUPPLY, then you can change both sources to 10 volts by assigning the value *once* to VSUPPLY.

A global parameter is like a programming variable that represents a numeric value by name.

Once you have defined a parameter (declared its name and given it a value), you can use it to represent circuit values anywhere in the schematic; this applies to any hierarchical level.

Some ways that you can use parameters are as follows:

- Apply the same value to multiple part instances.
- Set up an analysis that sweeps a variable through a range of values (for example, DC sweep or parametric analysis).

Declaring and using a global parameter

To use a global parameter in your schematic, you need to:

- define the parameter using a PARAM symbol, and
- use the parameter in place of a literal value somewhere in your design.

To declare a global parameter

- 1 Place a PARAM symbol in your schematic.
- 2 Double-click the PARAM symbol instance.
- 3 In the Attributes dialog box, declare up to three global parameters. For each global parameter:
 - a Click the NAME n attribute, type the parameter name in the Value text box, and then click Save Attr.
 - b Click the corresponding VALUE n attribute, type a default value for the parameter in the Value text box, and then click Save Attr.

Note *The system variables in [Table 3-3 on page 3-18](#) have reserved parameter names. Do not use these parameter names when defining your own parameters.*

To use the global parameter in your circuit

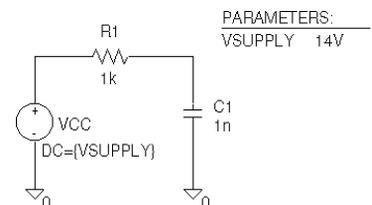
- 1 Find the numeric value that you want to replace: a component value, model parameter value, or other attribute value.
- 2 Replace the value with the name of the global parameter using the following syntax:

{ *global_parameter_name* }

The curly braces tell PSpice to *evaluate* the parameter and use its value.

Example: To declare the global parameter VSUPPLY that will set the value of an independent voltage source to 14 volts, place the PARAM symbol, and then set its NAME1 attribute to VSUPPLY and the VALUE1 attribute to 14v.

Example: To set the independent voltage source, VCC, to the value of the VSUPPLY parameter, set its DC attribute to {VSUPPLY}.



Expressions

An expression is a mathematical relationship that you can use to define a numeric or boolean (TRUE/FALSE) value.

PSpice evaluates the expression to a single value every time:

- it reads in a new circuit, and
- a parameter value used within an expression changes during an analysis.

Example: A parameter that changes with each step of a DC sweep or parametric analysis.

Specifying expressions

To use an expression in your circuit

Example: Suppose you have declared a parameter named FACTOR (with a value of 1.2) and want to scale a -10 V independent voltage source, VEE, by the value of FACTOR. To do this, set the DC attribute of VEE to:

```
{-10*FACTOR}
```

PSpice evaluates this expression to:

```
(-10 * 1.2) or -12 volts
```

For more information on user-defined functions, see the .FUNC command in the *Commands* chapter in the online *MicroSim PSpice A/D Reference Manual*.

For more information on user-defined parameters, see [Using Global Parameters and Expressions for Values on page 3-12](#).

- 1 Find the numeric or boolean value you want to replace: a component value, model parameter value, other attribute value, or logic in an IF function test (see page [3-17](#) for a description of the IF function).
- 2 Replace the value with an expression using the following syntax:

```
{ expression }
```

where *expression* can contain any of the following:

- standard operators listed in [Table 3-1](#)
- built-in functions listed in [Table 3-2](#)
- user-defined functions
- system variables listed in [Table 3-3](#)
- user-defined global parameters
- literal operands

The curly braces tell PSpice to *evaluate* the expression and use its value.

Note *Though PSpice accepts expressions of any length, Schematics does not. Value assignments to symbol attributes are limited to 1,024 characters. If your expression exceeds this limit, create a user-defined function (saved in an include file) and use the function in the expression. Remember to configure the include file.*

Table 3-1 *Operators in Expressions*

This operator class...	Includes this operator..	Which means...
	.	
arithmetic	+	addition or string concatenation
	-	subtraction
	*	multiplication
	/	division
	**	exponentiation
logical*	~	unary NOT
		boolean OR
	^	boolean XOR
	&	boolean AND
relational*	==	equality test
	!=	non-equality test
	>	greater than test
	>=	greater than or equal to test
	<	less than test
	<=	less than or equal to test

*. Logical and relational operators are used within the IF() function.

Table 3-2 *Functions in Arithmetic Expressions*

This function...	Means this...	
ABS(x)	$ x $	
SQRT(x)	$x^{1/2}$	
EXP(x)	e^x	
LOG(x)	$\ln(x)$	which is log base e
LOG10(x)	$\log(x)$	which is log base 10
PWR(x,y)	$ x ^y$	
PWRS(x,y)	$+ x ^y$ (if $x > 0$) $- x ^y$ (if $x < 0$)	
SIN(x)	$\sin(x)$	where x is in radians
ASIN(x)	$\sin^{-1}(x)$	where the result is in radians
SINH(x)	$\sinh(x)$	where x is in radians
COS(x)	$\cos(x)$	where x is in radians
ACOS(x)	$\cos^{-1}(x)$	where the result is in radians
COSH(x)	$\cosh(x)$	where x is in radians
TAN(x)	$\tan(x)$	where x is in radians
ATAN(x) ARCTAN(x)	$\tan^{-1}(x)$	where the result is in radians
ATAN2(y,x)	$\tan^{-1}(y/x)$	where the result is in radians
TANH(x)	$\tanh(x)$	where x is in radians
M(x)	magnitude of x^*	which is the same as ABS(x)
P(x)	phase of x^*	in degrees; returns 0.0 for real numbers
R(x)	real part of x^*	
IMG(x)	imaginary part of x^*	which is applicable to AC analysis only
DDT(x)	time derivative of x	which is applicable to transient analysis only

Note *In Probe, this function is D(x).*

Table 3-2 *Functions in Arithmetic Expressions (continued)*

This function...	Means this...	
SDT(x)	time integral of x	which is applicable to transient analysis only
TABLE(x,x ₁ ,y ₁ ,...)	y value as a function of x	where x _n ,y _n point pairs are plotted and connected by straight lines
MIN(x,y)	minimum of x and y	
MAX(x,y)	maximum of x and y	
LIMIT(x,min,max)	min if x < min max if x > max else x	
SGN(x)	+1 if x > 0 0 if x = 0 -1 if x < 0	
STP(x)	1 if x > 0 0 otherwise	which is used to suppress a value until a given amount of time has passed
IF(t,x,y)	x if t is true y otherwise	where t is a relational expression using the relational operators shown in Table 3-1

Note *In Probe, this function is S(x).*

Example: {v(1)*STP(TIME-10ns)} gives a value of 0.0 until 10 nsec has elapsed, then gives v(1).

*. M(x), P(x), R(x), and IMG(x) apply to Laplace expressions only.

Note *If a passive or semiconductor device has an independent temperature assignment, then TEMP does not represent that device's temperature.*

To find out more about customizing temperatures for passive or semiconductor devices, refer to the .MODEL command in the *Commands* chapter in the online *MicroSim PSpice A/D Reference Manual*.

To find out how to use these symbols and specify their attributes, see the following:

- [Setting Up a DC Stimulus on page 8-4](#)
- [Using VSRC or ISRC symbols on page 3-21](#)

Table 3-3 *System Variables*

This variable...	Evaluates to this...
TEMP	<p>Temperature values resulting from a temperature, parametric temperature, or DC temperature sweep analysis.</p> <p>The default temperature, TNOM, is set in the Options dialog box (from the Analysis menu, select Setup and click Options). TNOM defaults to 27°C.</p> <p>Note <i>TEMP can only be used in analog behavioral modeling expressions.</i></p>
TIME	<p>Time values resulting from a transient analysis. If no transient analysis is run, this variable is undefined.</p> <p>Note <i>TIME can only be used in analog behavioral modeling expressions.</i></p>

Defining Power Supplies

If your circuit requires DC power, then you need to include a DC source in your design. To specify a DC source, use one of the following symbols.

For this source type...	Use this symbol...
voltage	VDC or VSRC
current	IDC or ISRC

Defining Stimuli

To simulate your circuit, you need to connect one or more source symbols that describe the input signal that the circuit must respond to.

The MicroSim libraries supply several source symbols that are described in the tables that follow. These symbols depend on:

- the kind of analysis you are running, and
- how you want to define the stimulus: using the Stimulus Editor, using a file specification, or by defining symbol attribute values.

Stimuli include both voltage and current sources. The following table shows the symbol names for voltage sources.

If you want this kind of input...	Use this symbol for voltage...
For DC analyses	
DC bias	VDC or VSRC
For AC analyses	
AC magnitude and phase	VAC or VSRC
For transient analyses	
exponential	VEXP or VSTIM*
periodic pulse	VPULSE or VSTIM*
piecewise-linear	VPWL or VSTIM*
piecewise-linear that repeats forever	VPWL_RE_FOREVER or VPWL_F_RE_FOREVER**
piecewise-linear that repeats n times	VPWL_N_TIMES or VPWL_F_N_TIMES**
frequency-modulated sine wave	VSFFM or VSTIM*
sine wave	VSIN or VSTIM*

*. VSTIM and ISTIM symbols require the Stimulus Editor to define the input signal; these symbols are not available in Basics.

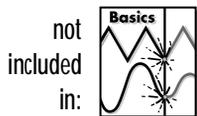
** . VPWL_F_RE_FOREVER and VPWL_F_N_TIMES are file-based symbols; the stimulus specification resides in a file and adheres to PSpice netlist syntax.

See [Setting Up a DC Stimulus on page 8-4](#) for more details.

See [Setting Up an AC Stimulus on page 9-3](#) for more details.

See [Defining a Time-Based Stimulus on page 10-3](#) for more details.

Example: The current source equivalent to VDC is IDC, to VAC is IAC, to VEXP is IEXP, and so on.



To determine the symbol name for an equivalent current source

- 1 In the table of voltage source symbols, replace the first V in the symbol name with I.

Using VSTIM and ISTIM

You can use VSTIM and ISTIM symbols to define any kind of time-based input signal. To specify the input signal itself, you need to use the Stimulus Editor.

To start the Stimulus Editor for a VSTIM or ISTIM symbol

- 1 Double-click the symbol instance on your schematic.

You are now ready to specify the input signal's behavior. To find out how, see [The Stimulus Editor Utility on page 10-5](#).

If you want to specify multiple stimulus types

If you want to run more than one analysis type, including a transient analysis, then you need to use either of the following:

- time-based stimulus symbols with AC and DC attributes
- VSRC or ISRC symbols

Using time-based stimulus symbols with AC and DC attributes

The time-based stimulus symbols that you can use to define a transient, DC, and/or AC input signal are listed below.

VEXP	IEXP
VPULSE	IPULSE
VPWL	IPWL
VPWL_F_RE_FOREVER	IPWL_F_RE_FOREVER
VPWL_F_N_TIMES	IPWL_F_N_TIMES
VPWL_RE_FOREVER	IPWL_RE_FOREVER
VPWL_RE_N_TIMES	IPWL_RE_N_TIMES
VSFFM	ISFFM
VSIN	ISIN

In addition to the transient attributes, each of these symbols also has a DC and AC attribute. When you use one of these symbols, you must define all of the transient- attributes. However, it's common to leave DC and/or AC undefined (blank). When you give them a value, the syntax you need to use is as follows.

This attribute...	Has this syntax...
DC	<i>DC_value[units]</i>
AC	<i>magnitude_value[units] [phase_value]</i>

Using VSRC or ISRC symbols

The VSRC and ISRC symbols have one attribute for each analysis type: DC, AC, and TRAN. You can set any or all of them using PSpice netlist syntax. When you give them a value, the syntax you need to use is as follows.

This attribute...	Has this syntax...
DC	<i>DC_value[units]</i>
AC	<i>magnitude_value[units] [phase_value]</i>
TRAN	<i>time-based_type (parameters)</i> where <i>time-based_type</i> is EXP, PULSE, PWL, SFFM, or SIN, and the <i>parameters</i> depend on the <i>time-based_type</i> .

Note *MicroSim recommends that if you are running only a transient analysis, use a VSTIM or ISTIM symbol if you have the standard package, or one of the other time-based source symbols that has attributes specific for a waveform shape.*

For the meaning of transient source attributes, refer to the I/V (independent current and voltage source) device type syntax in the *Analog Devices* chapter in the online *MicroSim PSpice A/D Reference Manual*.

For the syntax and meaning of transient source specifications, refer to the I/V (independent current and voltage source) device type in the *Analog Devices* chapter in the online *MicroSim PSpice A/D Reference Manual*.

Things to Watch For

For a roadmap to other commonly encountered problems and solutions, see [When Netlisting Fails or the Simulation Does Not Start on page 3-4](#).

This section includes troubleshooting tips for some of the most common reasons why your circuit might fail to netlist or simulate.

Unmodeled Parts

If you see messages like this in the Message Viewer,

```
Warning: Part part_name has no simulation model.
```

then you may have done one of the following things:

- Placed a part from the MicroSim libraries that is not available for simulation (used only for board layout).
- Placed a custom part that has been incompletely defined for simulation.



The libraries listed in the tables that follow all contain parts that you can simulate. Some files also contain parts that you can only use for board layout. That's why you need to check the TEMPLATE attribute if you are unsure or still getting warnings when you try to simulate your circuit.

Do this if the part in question is from the MicroSim libraries

- Replace the part with an equivalent part from one of the libraries listed in the tables that follow.
- Make sure that you can simulate the part by checking the following:
 - That it has a TEMPLATE attribute and that its value is non-blank.
 - That it has a MODEL attribute and that its value is non-blank.

Analog Libraries with Modeled Parts

I_SHOT	EPWRBJT	NAT_SEMI
ABM	FILTSUB	OPAMP
ADV_LIN	FWBELL	OPTO
AMP	HARRIS	PHIL_BJT
ANALOG	IGBT*	PHIL_FET
ANA_SWIT	JBIPOLAR	PHIL_RF
ANLG_DEV	JDIODE	POLYFET
ANL_MISC	JFET	PWRBJT
APEX	JJFET	PWRMOS
BIPOLAR	JOPAMP	SIEMENS
BREAKOUT	JPWRBJT	SWIT_RAV
BUFFER	JPWRMOS	SWIT_REG
BURR_BRN	LIN_TECH	TEX_INST
CD4000	MAGNETIC*	THYRISTR*
COMLINR	MAXIM	TLINE*
DIODE	MOTORAMP	XTAL
EBIPOLAR	MOTORMOS	ZETEX
EDIODE	MOTORSEN	
ELANTEC	MOTOR_RF	

* Not included in Basics.

To find out more about a particular library, refer to the online *Library List* or read the header of the model library file itself.



Check for this if the part in question is custom-built

Are there blank (or inappropriate) values for the symbol's MODEL and TEMPLATE attributes?

If so, load this symbol into the symbol editor and set these attributes appropriately. One way to approach this is to edit the symbol that appears on your schematic.

To find out more about setting the simulation attributes for symbols, see [Defining Symbol Attributes Needed for Simulation on page 5-18](#).

To find out more about using the symbol editor, refer to your *MicroSim Schematics User's Guide*.

To edit the attributes for the symbol in question

- 1 In the schematic editor, select the symbol.
- 2 From the Edit menu, select Symbol.
The symbol editor window appears with the symbol already loaded.
- 3 From the Part menu, select Attributes and proceed to change the attributes values.

Unconfigured Model, Stimulus, or Include Files

If you see messages like these in the Message Viewer,

```
(schematic_name) Floating pin: refdes pin  
    pin_name  
Floating pin: pin_id  
File not found  
Can't open stimulus file
```

or messages like these in the PSpice output file,

```
Model model_name used by device_name is  
    undefined.  
Subcircuit subckt_name used by device_name  
    is undefined.  
Can't find .STIMULUS "refdes" definition
```

then you may be missing a model library, stimulus file, or include file from the configuration list, or the configured file is not on the library path.

Check for this

- Does the relevant model library, stimulus file, or include file appear in the configuration list?
- If the file is configured, does the default library search path include the directory path where the file resides, or explicitly define the directory path in the configuration list?

If the file is not configured, add it to the list and make sure that it appears before any other library or file that has an identically-named definition.

To view the configuration list

- 1 In the schematic editor, from the Analysis menu, select Library and Include Files.

If the directory path is not specified, update the default library search path or change the file entry in the configuration list to include the full path specification.

To view the default library search path

- 1 In the schematic editor, from the Options menu, select Editor Configuration.



To find out more about how to configure these files and about search order, see [Configuring Model Libraries on page 4-41](#).

To find out more about the default configuration, see [How Are Models Organized? on page 4-4](#).

To find out more about the library search path, see [Changing the Library Search Path on page 4-46](#).

Unmodeled Pins

If you see messages like these in the Message Viewer,

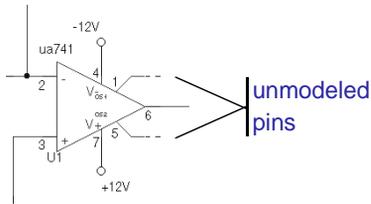
```
Warning: Part part_name pin pin_name is
unmodeled.
```

```
Warning: Less than 2 connections at node
node_name.
```

or messages like this in the PSpice output file,

```
Floating/unmodeled pin fixups
```

then you may have drawn a wire to an unmodeled pin.



The MicroSim libraries include parts that are suitable for both simulation and board layout. These parts may have a mix of modeled pins (solid line) and unmodeled pins (broken line). The unmodeled pins map into packages but have no electrical significance; PSpice ignores unmodeled pins during simulation.



Check for this

Are there connections to unmodeled pins?

If so, do one of the following:

- Remove wires connected to unmodeled pins.
- If you expect the connection to affect simulation results, find an equivalent part that models the pins in question and draw the connections.

To find out more about searching for parts, see [Finding the part that you want on page 3-8](#).

Missing Ground

If for *every net* in your circuit you see this message in the PSpice output file,

```
ERROR -- Node node_name is floating.
```

then your circuit may not be tied to ground.



Check for this

Are there AGND or EGND symbols connected appropriately on your schematic?

If not, place and connect one (or more, as needed) on your schematic.



AGND



EGN

Missing DC Path to Ground

If for *selected nets* in your circuit you see this message in the PSpice output file,

```
ERROR -- Node node_name is floating.
```

then you may be missing a DC path to ground.

Check for this

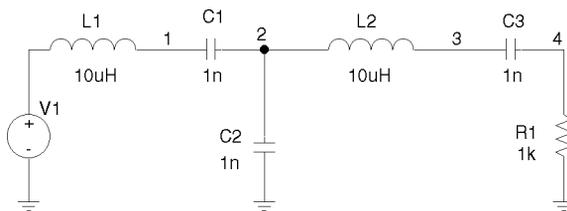
Are there any nets that are isolated from ground by either open circuits or capacitors?



If so, then add a very large (for example, 1 Gohm) resistor either:

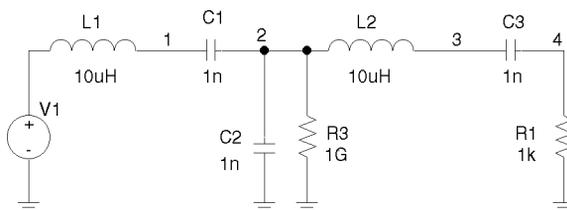
- in parallel with the capacitor or open circuit, or
- from the isolated net to ground.

Example: The circuit shown below connects capacitors (DC open circuits) such that both ends of inductor L2 are isolated from ground.



Note When calculating the *bias point solution*, PSpice treats capacitors as open circuits and inductors as short circuits.

When simulated, PSpice flags nets 2 and 3 as floating. The following topology solves this problem.



Creating and Editing Models

4

Chapter Overview

This chapter provides information about creating and editing models for parts that you want to simulate.

Topics are grouped into four areas introduced later in this overview. If you want to find out quickly which tools to use to complete a given task and how to start, then:

- 1 Go to the roadmap in [Ways to Create and Edit Models on page 4-8](#).
- 2 Find the task you want to complete.
- 3 Go to the sections referenced for that task for more information about how to proceed.

Background information These topics present model library concepts and an overview of the tools that you can use to create and edit models. Topics include:

- [What Are Models? on page 4-3](#)
- [How Are Models Organized? on page 4-4](#)
- [Tools to Create and Edit Models on page 4-7](#)

Task roadmap This topic helps you find the sections in this chapter that are relevant to the model editing task that you want to complete. This topic is:

- [Ways to Create and Edit Models on page 4-8](#)

How to use the tools These topics explain how to use different tools to create and edit models on their own and when editing schematics or symbols. Topics include:

- [Using the Parts Utility to Edit Models on page 4-10](#)
- [Using the Model Editor on page 4-29](#)
- [Using the Create Subcircuit Command on page 4-37](#)

Other useful information These topics explain how to configure and reuse models after you have created or edited them. Topics include:

- [Changing the Model Reference to an Existing Model Definition on page 4-38](#)
- [Reusing Instance Models on page 4-39](#)
- [Configuring Model Libraries on page 4-41](#)

What Are Models?

A model defines the electrical behavior of a part. On your schematic, this correspondence is defined by a symbol's MODEL attribute, which is assigned the model name.

A model is defined as either a:

- model parameter set, or
- subcircuit netlist,

depending on the device type that it describes. Both ways of defining a model are text-based with specific rules of syntax.

For a description of the MODEL attribute, see [MODEL on page 5-18](#).

Models defined as model parameter sets

PSpice has built-in algorithms or models that describe the behavior of many device types. The behavior of these *built-in models* is described by a *set of model parameters*.

The behavior for a part that is based on a built-in model is defined by setting all or any of the corresponding model parameters to new values using the PSpice .MODEL syntax.

Example:

```
.MODEL MLOAD NMOS
+ (LEVEL=1 VTO=0.7 CJ=0.02pF)
```

Models defined as subcircuit netlists

For some parts, there are no PSpice built-in models that can describe their behavior fully. These kinds of parts are defined using the PSpice .SUBCKT/.ENDS or *subcircuit syntax* instead.

Subcircuit syntax includes:

- *Netlists* to describe the structure and function of the part.
- *Variable input parameters* to fine tune the model.

To find out more about PSpice command and netlist syntax, refer to the online *MicroSim PSpice A/D Reference Manual*.

Example:

```
* FIRST ORDER RC STAGE
.SUBCKT LIN/STG IN OUT AGND
+ PARAMS: C1VAL=1 C2VAL=1 R1VAL=1 R2VAL=1
+          GAIN=10000
C1 IN  N1  {C1VAL}
C2 N1  OUT {C2VAL}
R1 IN  N1  {R1VAL}
R2 N1  OUT {R2VAL}
EAMP1 OUT AGND VALUE={V(AGND,N1)*GAIN}
.ENDS
```

How Are Models Organized?

The key concepts behind model organization are as follows:

- Model definitions are saved in files called libraries.
- Model libraries must be configured so PSpice searches them for definitions.
- Depending on the configuration, model libraries are available either to a specific design or to all designs.

Model Libraries

You can use the MicroSim Text Editor, or any other standard text editor, to view model definitions in the libraries.

Example: `motor_rf.lib` contains models for Motorola-made RF bipolar transistors.

Device model and subcircuit definitions are organized into model libraries. Model libraries are text files that contain one or more model definitions. Typically, model library names have a `.lib` extension.

Most model libraries contain parts of similar type. For vendor-supplied parts, libraries are also partitioned by manufacturer. To find out more about the parts contained in a library, read the comments in the file header.

Model Library Configuration

PSpice searches model libraries for the model names specified by the MODEL attribute value on symbols in your schematic. These are the model definitions that PSpice uses to simulate your circuit.

For PSpice to know where to look for these model definitions, you must configure the libraries. This means:

- Specifying the directory path or paths to the model libraries.
- Naming each model library that PSpice should search and listing them in the needed search order.
- Assigning global or local scope to the model library.

To optimize the search, PSpice uses indexes. To find out more about this and how to add, delete, and rearrange configured libraries, see [Configuring Model Libraries on page 4-41](#).

Global vs. Local Models and Libraries

Model libraries and the models they contain have either local or global application to your designs.

Local models Local models apply to one design. The *schematic editor* automatically creates a local model whenever you modify the model definition for a part instance on your schematic. You can also create models externally and then manually configure the new libraries for a specific design.

Global models Global models are available to all designs you might create. The *symbol editor* automatically creates a global model whenever you create a symbol with a new model definition. The Parts utility also creates global models. You can also create models externally and then manually configure the new libraries for use in all designs.

To find out how to change the local and global configuration of model libraries, see [Changing Local and Global Scope on page 4-45](#).

Example usage: To set up device and lot tolerances on the model parameters for a particular part instance when running a Monte Carlo or sensitivity/worst-case analysis.

PSpice searches local libraries before global libraries. To find out more, see [Changing Model Library Search Order on page 4-45](#).

Nested Model Libraries

Besides device model and subcircuit definitions, model libraries can also contain references to other model libraries using the PSpice `.LIB` syntax. When searching model libraries for matches, PSpice also scans these referenced libraries.

Example: Suppose you have two custom model libraries, `mydiodes.lib` and `myopamps.lib`, that you want PSpice to search any time you simulate a design. Then you can create a third model library, `mymodels.lib`, that contains these two statements:

```
.LIB mydiodes.lib  
.LIB myopamps.lib
```

and configure `mymodels.lib` for global use. Because `mydiodes.lib` and `myopamps.lib` are referenced from `mymodels.lib`, they are automatically configured for global use as well.

For a listing of parts provided by MicroSim, refer to the online *Library List*.

MicroSim-Provided Models

The model libraries that you initially install with your MicroSim programs are listed in `nom.lib`. This file demonstrates how you can nest references to other libraries and models.

If you select Library and Include Files from the Analysis menu in Schematics immediately after installation, you will see the `nom.lib*` entry in the Library Files list. The asterisk means that this model library, and any of the model libraries it references, contain global model definitions.

Tools to Create and Edit Models

There are three tools that you can use to create and edit model definitions. Use the:

- **Parts utility** when you want to:
 - derive models from data sheet curves provided by manufacturers, or
 - modify the behavior of a Parts-supported model.
- **Model editor** when:
 - model parameters are already defined (such as for models from a vendor), or
 - the model is not supported by the Parts utility, or
 - you want to edit the PSpice command syntax (text) for .MODEL and .SUBCKT definitions.
- **Create Subcircuit command** when you have a hierarchical level in your schematic that you want to set up as an equivalent symbol with behavior described as a subcircuit netlist (.SUBCKT syntax).

Note *If you created a subcircuit definition using the Create Subcircuit command and want to alter it, use the model editor to edit the definition, or modify the original hierarchical schematic and run Create Subcircuit again to replace the definition.*



Note *The Parts utility is not included in PSpice Basics.*

For a description of models supported by the Parts utility, see [Parts-Supported Device Types on page 4-12](#).



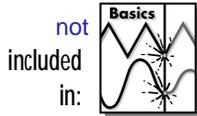
The Create Subcircuit command will not help you create a hierarchical design. You need to do this yourself before using the Create Subcircuit command. For information on hierarchical schematics and how to create them, refer to your *MicroSim Schematics User's Guide*.

Ways to Create and Edit Models

This section is a roadmap to other information in this chapter. Find the task that you want to complete, then go to the referenced sections for more information.

If you want to...	Then do this...	To find out more, see this...
<p>➔ Create or edit the model for an existing symbol and have it affect all schematics that use that symbol.</p>	<p>Create or load the symbol first in the symbol editor, then edit the model using either the:</p> <ul style="list-style-type: none"> • Parts utility *, or • model editor. 	<p>Running the Parts Utility from the Symbol Editor on page 4-18. Running the Model Editor from the Symbol Editor on page 4-31.</p>
<p>➔ Create a model from scratch and automatically create a symbol for it to use in any schematic.</p>	<p>Start the Parts* utility and enable/disable automatic symbol creation as needed; then create or view the model.</p>	<p>Running the Parts Utility Alone on page 4-16.</p>
<p>➔ Create a model from scratch without a symbol and have the model definition available to any design.</p>		
<p>➔ View model characteristics for a part.</p>		
<p>➔ Define tolerances on model parameters for statistical analyses.</p>	<p>Select the part instance on your schematic then edit the model using the model editor.</p>	<p>Running the Model Editor from the Schematic Editor on page 4-33.</p>
<p>➔ Test behavior variations on a part.</p>	<p>Select the part instance on your schematic then edit the model using either the:</p>	<p>Running the Parts Utility from the Schematic Editor on page 4-20</p>
<p>➔ Refine a model before making it available to all schematics.</p>	<ul style="list-style-type: none"> • Parts utility*, or • model editor. 	<p>Running the Model Editor from the Schematic Editor on page 4-33.</p>
<p>➔ Derive subcircuit definitions from a hierarchical schematic.</p>	<p>Use the Create Subcircuit command in the schematic editor.</p>	<p>Using the Create Subcircuit Command on page 4-37.</p>

For a list of device types that the Parts utility supports, see [Parts-Supported Device Types on page 4-12](#). If the Parts utility does not support the device type for the model definition that you want to create, then you can use a standard text editor to create a model definition using the PSpice `.MODEL` and `.SUBCKT` command syntax.



Using the Parts Utility to Edit Models

The Parts utility converts information that you enter from the part manufacturer's data sheet into either:

- model parameter sets using PSpice .MODEL syntax, or
 - subcircuit netlists using PSpice .SUBCKT syntax,
- and saves these definitions to model libraries that PSpice can search when looking for simulation models.

The Parts utility does not support the following subcircuit constructs:

- optional nodes construct, OPTIONAL:
- variable parameters construct, PARAMS:
- local .PARAM command
- local .FUNC command

To refine the subcircuit definition for these constructs, use the model editor described in [Using the Model Editor on page 4-29](#).

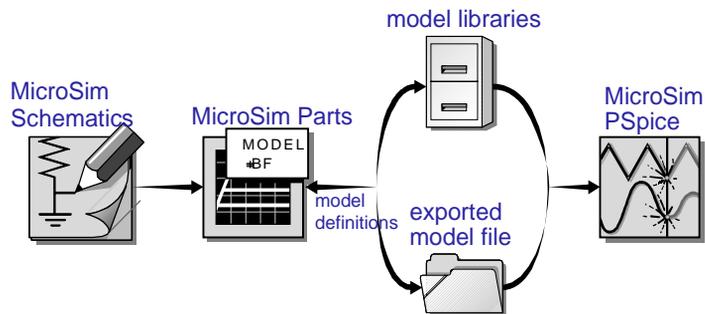


Figure 4-1 Relationship of Parts Utility to Schematics and PSpice

Note By default, the Parts utility creates or updates model libraries. To create an exported model file, use the Export command from the Part menu and configure it as an include file. For more information, see [How PSpice Uses Model Libraries](#) and the companion sidebar on page [4-43](#).

Ways to Use the Parts Utility

You can use the Parts utility five ways:

- **To define a new model, and then automatically create a symbol.** Any new models and symbols are automatically available to any schematic.
- **To define a new model only (no symbol).** You can optionally turn off the symbol creation feature for new models. The model definition is available to any design, for example, by changing a model reference on a part instance.
- **To edit the model definition linked to symbol definition in the symbol library.** This means you need to start Parts from the symbol editor after having loaded or created a symbol. Schematics automatically links the new model definition (that the Parts utility creates) to the symbol definition.
- **To edit a model definition for a part instance on your schematic.** This means you need to start the Parts utility from the schematic editor after having selected a part instance on your schematic. The schematic editor automatically links the new model definition (that the Parts utility creates) to the selected part instance.
- **To examine or verify the electrical characteristics of a model without running PSpice.** This means you can run the Parts utility alone to:
 - check characteristics of a model quickly, given a set of model parameter values, or
 - compare characteristic curves to data sheet information or measured data.

To find out more, see [Running the Parts Utility Alone on page 4-16](#).

To find out more, see [Running the Parts Utility Alone on page 4-16](#).

To find out more, see [Running the Parts Utility from the Symbol Editor on page 4-18](#).

To find out more, see [Running the Parts Utility from the Schematic Editor on page 4-20](#).

To find out more, see [Running the Parts Utility Alone on page 4-16](#).

Part types that the Parts utility models using the .MODEL statement are based on the models built into PSpice.

Note *The model parameter defaults used by the Parts utility are different from those used by the models built into PSpice.*

Parts-Supported Device Types

Table 4-1 summarizes the device types supported in the Parts utility.

Table 4-1 *Models Supported in the Parts Utility*

This part type...	Uses this definition form...	And this name prefix * ...
diode	.MODEL	D
bipolar transistor	.MODEL	Q
IGBT	.MODEL	Z
JFET	.MODEL	J
power MOSFET	.MODEL	M
operational amplifier**	.SUBCKT	X
voltage comparator**	.SUBCKT	X
nonlinear magnetic core	.MODEL	K
voltage regulator**	.SUBCKT	X
voltage reference**	.SUBCKT	X

*. This is the standard PSpice device letter notation. Refer to the online *MicroSim PSpice A/D Reference Manual*.

** . The Parts utility only supports .SUBCKT models that were generated by the Parts utility. This means that you cannot load a .SUBCKT model created manually or by another tool into the Parts utility for editing. When you try to load a .SUBCKT model that the Parts utility did not create, Parts displays the “model not supported” message.

Ways To Characterize Models

Figure 4-2 shows two ways to characterize models using the Parts utility.

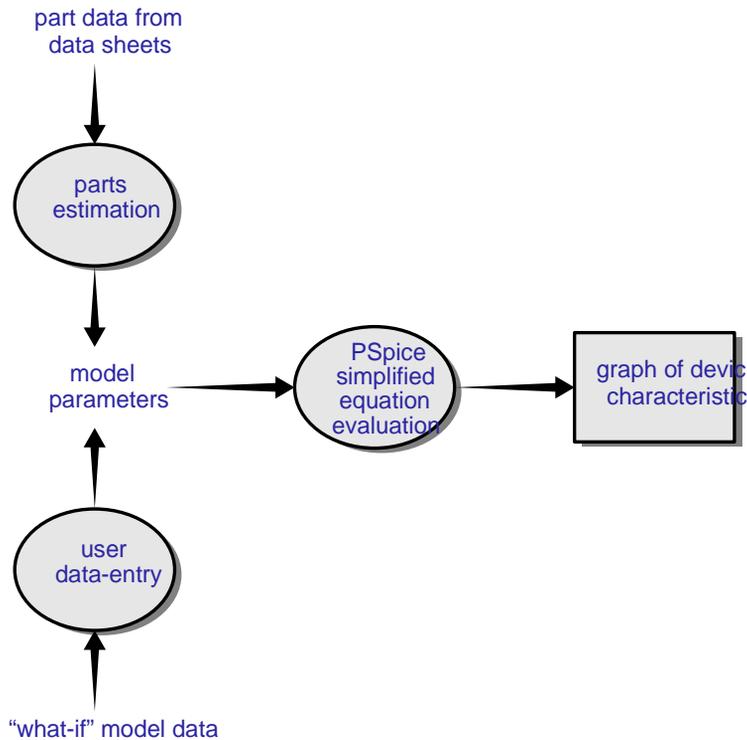


Figure 4-2 Process and Data Flow for the Parts Utility

Creating models from data sheet information

The most common way to characterize models is to enter data sheet information for each device characteristic. After you are satisfied with the behavior of each characteristic, you can have the Parts utility estimate (or extract) the corresponding model parameters and generate a graph showing the behavior of the characteristic. This is called the fitting process. You can repeat this process and when you are satisfied with the results, save them; the Parts utility creates model libraries containing appropriate device model and subcircuit definitions.



Testing and verifying models created with Parts

Each curve in the Parts utility is defined only by the parameters being adjusted. For the diode, the forward current curve *only* shows the part of the current equation which is associated with the forward characteristic parameters (such as I_S , N , R_s).

However, PSpice uses the *full* equation for the diode model, which includes a term involving the reverse characteristic parameters (such as I_{SR} , N_R). These parameters could have a significant effect at low current.

This means that the curve displayed in the Parts utility does not exactly match what is displayed in Probe after a simulation. Be sure to test and verify models using PSpice. If needed, fine-tune the models.

Note When specifying operating characteristics for a part, you can use typical values found on data sheets effectively for most simulations. To verify your design, you may also want to use best- and worst-case values to create separate models, and then swap them into the circuit.

Analyzing the effect of model parameters on device characteristics

You can also edit model parameters directly and investigate how changing their values affects a device characteristic. As you change model parameters, the Parts utility recalculates the behavior of the part characteristics and displays a new curve for each of the affected ones.

How to Fit Models

For more information about the properties of devices supported by the Parts utility, refer to the online *MicroSim PSpice A/D Reference Manual*.

For a given model, the Parts utility displays a window with a list of the device characteristics and a list of all model parameters. You can also view performance curves (see Figure 4-3).

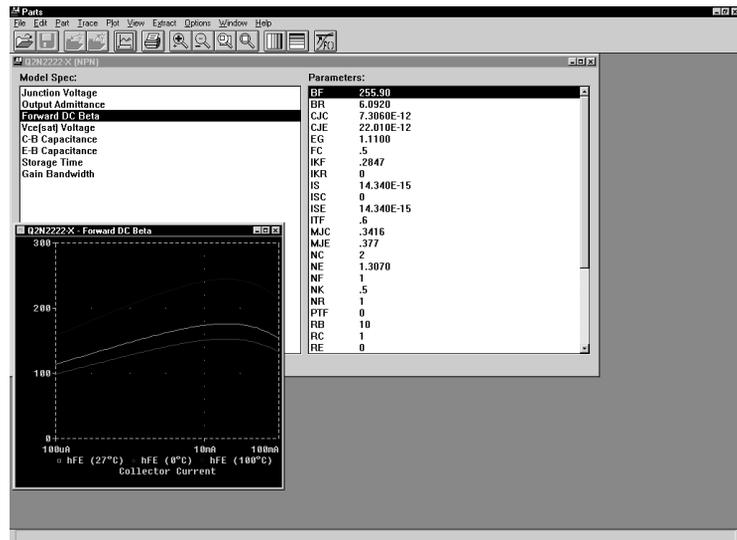


Figure 4-3 Parts Utility Window with Data for a Bipolar Transistor

To fit the model

- 1 For each device characteristic that you want to set up:
 - a In the Model Spec list, select the device characteristic.
 - b From the Edit menu, select Spec.
 - c In the Edit Model Spec dialog box, type in the device information from the data sheet.
 - d Click Add.
 - e Click OK.
- 2 From the Extract menu, select Parameters to extract all relevant model parameters for the current specification.

An asterisk (*) appears next to each extracted model parameter.
- 3 Repeat steps **1-2** until the model meets target behaviors.

Or, instead of steps **a** and **b**, double-click the device characteristic in the Model Spec list.

To view performance curves

- 1 From the Model Spec list, select a device characteristic.
- 2 From the Plot menu, select Display.

Note *If you view performance curves before fitting, then your data points and the curve for the current model specification may not match.*

After you have selected the part that you want to model, you can proceed with entering data sheet information and model fitting as described in [How to Fit Models on page 4-14](#).

If you have already started the Parts utility from Schematics and want to continue working on new models, then:

- 1 Save the opened model library.
- 2 Open or create a different model library.
- 3 Get a model, or create a new one.

Instead of using the MicroSim default symbol set for new models, you can have the Parts utility use your own set of standard symbols. To find out more, see [Basing New Symbols On a Custom Set of Symbols on page 5-13](#).

Running the Parts Utility Alone

If you want to:

- model a new part and use the part in any schematic (and automatically create a symbol),
- create a model and have the model definition available to any design (do not create a symbol), or
- examine or verify the characteristics of a given model without using PSpice,

then run the Parts utility alone. This means that the model you are creating or examining is not currently tied to a part instance on your schematic or to a symbol editing session.

Note *You can only edit models for device types that the Parts utility supports. See [Parts-Supported Device Types on page 4-12](#) for details.*

Starting the Parts utility

To start the Parts utility alone

- 1 From the MicroSim program folder, select Parts.
- 2 From the File menu, select Open/Create, and enter an existing or new model library name.
- 3 From the Part menu, select New, Copy, or Import to load a device model.

Enabling and disabling automatic symbol creation

Symbol creation in the Parts utility is optional. By default, automatic symbol creation is enabled. However, if you previously disabled symbol creation, you will need to enable it before creating a new model and symbol.

To automatically create symbols for new models

- 1 From the Options menu, select Symbol Creation Setup.

- 2 If not already checked, select Always Create Symbol to enable automatic symbol creation.
- 3 In the Save Symbol To frame, define the name of the symbol library for the new symbol. Choose either:
 - Symbol Library Path Same As Model Library to create or open the `.slb` file that has the same name prefix as the currently open model library (`.lib`).
 - User-Defined Symbol Library, and then enter a file name into the Symbol Library Name text box.

Note *If you select a user-defined symbol library, the Parts utility saves all new symbols to the specified file until you change it.*

Example: If the model library is `myparts.lib`, then the Parts utility creates the symbol library `myparts.slb`.

Saving global models (and symbols)

When you save your edits, the Parts utility does the following for you:

- Saves the model definition to the model library that you originally opened.
- If you had the automatic symbol creation option enabled, saves the symbol definition to `model_library_name.slb`.
- If the libraries are new, configures them for *global* use.

To save the new model (and symbol)

- 1 From the File menu, select Save Library to update `model_library_name.lib` (and, if you enabled symbol creation, `model_library_name.slb`), and save them to disk.

If you want to save the open model library to a new library, then

- 1 From the File menu, select SaveAs.
- 2 Enter the name of the new model library.

The Parts utility still automatically configures the model library as global. If the Parts utility created symbols, Parts saves the symbol library to `new_model_library_name.slb`, which is also global.

If you want to save only the model definition that you are currently editing to a different library, then

- 1 From the Part menu, select Export.
- 2 Enter the name of the new file.
- 3 If you want PSpice to search this file automatically,

After you have started the Parts utility, you can proceed with entering data sheet information and model fitting as described in [How to Fit Models on page 4-14](#).

Running the Parts Utility from the Symbol Editor

If you want to:

- base a new part on an existing symbol, or
- edit the model for an existing symbol and have it affect all schematics that use the symbol,

then run the Parts utility from the symbol editor in Schematics.

Note *You can only edit models for device types that the Parts utility supports. See [Parts-Supported Device Types on page 4-12](#) for details.*

Starting the Parts utility

To start the Parts utility from the Schematics symbol editor

Start the symbol editor.

- 1 From the File menu in the schematic editor, select Edit Library.

Start the Parts utility.

- 2 In the symbol editor, get, copy, or import a symbol definition.
- 3 From the Edit menu, click Attribute.
 - a Make sure that the MODEL attribute is assigned a new or existing model name. If the model name exists, make sure that the Parts utility supports the device type.
 - b If you are creating a new model, change the PART attribute value as needed to match the new model name.
 - c Click OK.
- 4 From the Edit menu, click Model.
- 5 Click Edit Model (Parts).

For general information about creating symbols, see [Chapter 5, Creating Symbols for Models](#).

The symbol editor searches the model libraries for the model.

- If found, the symbol editor opens the model library containing the original model and initializes Parts with the model information.
- If not found, the symbol editor assumes that it is a new model; at startup, the Parts utility displays the Create New Part dialog box.

To find out how Schematics searches the library, see [Changing Model Library Search Order on page 4-45](#).

Saving global models

When you save your edits, the following is done for you to make sure that the symbol saved to the symbol library and new model definition are linked and available to all schematics:

- The Parts utility saves the model definition to `symbol_library_name.lib`.
- If the library is new, the Parts utility configures it for global use.

To save models created from the symbol editor

- 1 From the File menu, select Save to update `symbol_library_name.lib` and write it to disk.

If you want to save the open model library to a new library, then

- 1 From the File menu, select SaveAs.
- 2 Enter the name of the new model library.

The Parts utility automatically configures the model library as global.

Once you have started the Parts utility, you can proceed with entering data sheet information and model fitting as described in [How to Fit Models on page 4-14](#).

For more information on instance models, see [Reusing Instance Models on page 4-39](#).

Running the Parts Utility from the Schematic Editor

If you want to:

- test behavior variations on a part, or
- refine a model before making it available to all schematics,

then run the Parts utility from the schematic editor in Schematics.

This means editing models for part instances on your schematic. When you select a part instance and edit its model, the schematic editor automatically creates an *instance model* that you can then change.

Note *You can only edit models for device types that the Parts utility supports. See [Parts-Supported Device Types on page 4-12](#) for details.*

What is an instance model?

An instance model is a *copy* of the symbol's original model. The copied model is local to the design. You can customize the instance model without impacting any other schematic that uses the original symbol from the library.

When the schematic editor creates the copy, it assigns a unique name that is by default:

original_model_name-Xn

where *n* is <blank 1 | 2 | ... > depending on the number of different instance models derived from the original model for the current schematic.

Starting the Parts utility

To start editing an instance model

- 1 In the schematic editor, select one symbol on your schematic.
- 2 From the Edit menu, select Model.
- 3 Click Edit Instance Model (Parts).

The schematic editor searches the model libraries for the instance model.

- If found, the schematic editor initializes the Parts utility with the name of the model library that contains the instance model; the Parts utility opens the model library and reads in the instance model.
- If not found, the schematic editor assumes that this is a new instance model and does the following: makes a copy of the original model definition, names it *original_model_name-Xn*, and initializes the Parts utility with the new model.

Saving local models

When you save your edits, the Parts utility writes the model definition to *schematic_name.lib*, which is already configured for local use (see [What happens if you don't save the instance model on page 4-22](#)).

To save instance models

- 1 From the File menu, select Save to update *schematic_name.lib* and write it to disk.

To find out how Schematics searches the library, see [Changing Model Library Search Order on page 4-45](#).



Actions that automatically configure the instance model file for global use instead

Instance model libraries are normally configured for local use. However, if you perform any of the following actions, the Parts utility configures this library for global use instead:

- Save the model to a different library using SaveAs from the File menu.
- After saving the instance model, you decide to create additional models and save them to the same instance model library *schematic_name.lib*.

What happens if you don't save the instance model

Before the schematic editor starts the Parts utility, it does these things:

- Makes a copy of the original model and saves it as an instance model in `schematic_name.lib`.
- Configures `schematic_name.lib` for local use, if not already done.
- Assigns the new instance model name to the MODEL attribute for the selected part instance.

This means that if you:

- cancel the Parts utility session, or
- return to Schematics to simulate the design

without first saving the model you are editing, the part instance on your schematic still refers to the instance model.

In this case, the instance model is identical to the original model. If you decide to edit this model later, be sure to do one of the following:

- If you want the changes to remain local, edit the instance model in the local library, using the model editor.
- If you want the change to be global, change the model reference for the part instance on your schematic back to the original model name in the global library, and then edit the original model in the symbol editor.

To find out how to change model references, see [Changing the Model Reference to an Existing Model Definition on page 4-38](#).

The Parts Utility Tutorial

In this tutorial, you will model a simple diode device as follows:

- Create the schematic for a simple half-wave rectifier.
- Run the Parts utility from the schematic editor to create an instance model for the diode in your schematic.

Creating the half-wave rectifier schematic

To draw the schematic

- 1 Start Schematics.
- 2 From the Draw menu in the schematic editor, select Get New Part.
- 3 Place one each of the following symbols (reference designator shown in parentheses) as shown in Figure 4-4:
 - Dbreak (D1 diode)
 - C (C1 capacitor)
 - R (R1 resistor)
 - VSIN (V1 sine wave source)
 - AGND (0 analog ground).
- 4 From the Draw menu, select Wire, and draw the connections between symbols as shown in Figure 4-4.
- 5 From the File menu, select Save As.
- 6 Type `rectfr` and click OK to save the schematic to `rectfr.sch`.

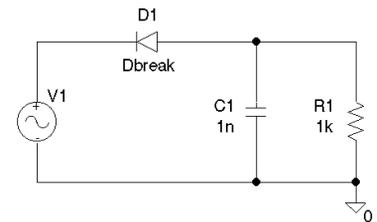


Figure 4-4 Schematic for a Half-Wave Rectifier



or press

Ctrl + **W**

Note If you were to simulate this design using a transient analysis, you would also need to set up a transient specification for V1; most likely, this would mean defining the *VOFF* (offset voltage), *VAMPL* (amplitude), and *FREQ* (frequency) attributes for V1. For this tutorial, however, you will not run a simulation, so you can skip this step.

Starting the Parts utility for the D1 diode

To start the Parts utility

- 1 Click the D1 symbol to select it.
- 2 From the Edit menu, select Model.
- 3 Click Edit Instance Model (Parts).

The schematic editor searches the configured set of model libraries for an instance model corresponding to this symbol.

- 4 Click OK.

Three things happen:

- Schematics automatically creates `rectfr.lib` and configures it into the set of local model libraries.
- The Parts window displays.
- The D1 instance in the schematic references a unique instance model name, `Dbreak-X`.

Entering data sheet information

As shown in Figure 4-5, the Parts window initially displays:

- diode model characteristics listed in the Model Spec box, and
- `Dbreak-X` model parameter values listed in the Parameters box.

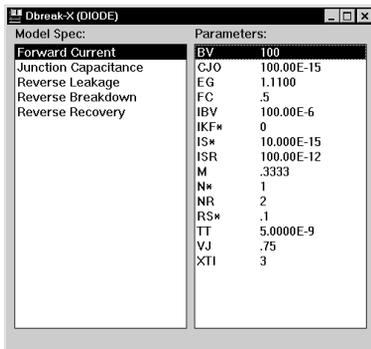


Figure 4-5 Diode Model Characteristics and Parameter Values for the `Dbreak-X`

You can modify each model characteristic listed in the Model Spec list with new values from the data sheets. The Part utility takes the new information and fits new model parameter values.

When updating the entered data, the Parts utility expects either:

- device curve data (point pairs), or
- single-valued data,

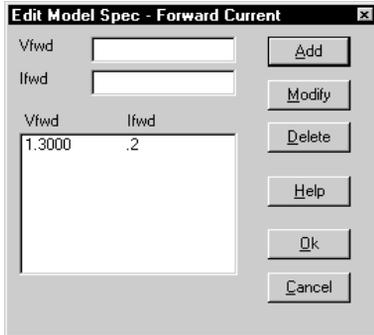
depending on the device characteristic.

For the diode, Forward Current, Junction Capacitance, and Reverse Leakage require device curve data. Reverse Breakdown and Reverse Recovery require single-valued data.

Table 4-2 lists the data sheet information for the Dbreak-X model.

Table 4-2 *Sample Diode Data Sheet Values*

For this model characteristic...	Enter this...
forward current	(1.3, 0.2)
junction capacitance	(1m, 120p) (1, 73p) (3.75, 45p)
reverse leakage	(6, 20n)
reverse breakdown	(Vz=7.5, Iz=20m, Zz=5)
reverse recovery	no changes



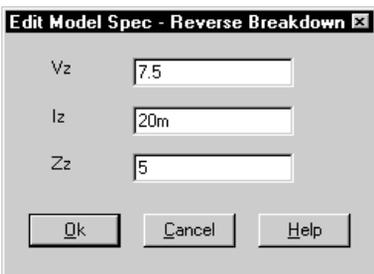
To change the Forward Current characteristic

- 1 In the Model Spec list, double-click Forward Current.
The Edit Model Spec-Forward Current dialog box appears. This dialog box requires curve data.
- 2 In the Vfwd text box, type 1 . 3.
- 3 Press **[Tab]** to move to the Ifwd text box, and then type 0 . 2.
- 4 Click Add.
The new values appear in the Vfwd-Ifwd table.
- 5 Click OK.

To change the values for Junction Capacitance and Reverse Leakage

- 1 Follow the same steps as for Forward Current, entering the data sheet information listed in [Table 4-2](#) that corresponds to the current device characteristic.

To change the Reverse Breakdown characteristic



- 1 In the Model Spec list, double-click Reverse Breakdown.
The Edit Model Spec-Reverse Breakdown dialog box appears. This dialog box accepts single-valued data.
- 2 In the Vz text box, type 7 . 5.
- 3 Press **[Tab]** to move to the Iz text box, and then type 20m.
Note that the Parts utility accepts the same scale factors normally accepted by PSpice.
- 4 Press **[Tab]** to move to the Zz text box, and then type 5.
- 5 Click OK.

Extracting model parameters

To generate new model parameter values

- 1 From the Extract menu, select Parameter.

The new values appear in the Parameters box with an asterisk appearing to the right of the ones that have changed.

To display the curves for the five diode characteristics

- 1 Click Forward Current and drag the mouse down to the end of the list to select all of the entries in the Model Spec box.
- 2 From the Plot menu, select Display.
- 3 From the Window menu, select Tile.

A few of the plots are shown in Figure 4-6.

You can also do the following with an active plot window:

- Drag the plot to a new position.
- Pan and zoom within the plot using options in the View menu.
- Rescale axes using options in the Plot menu.

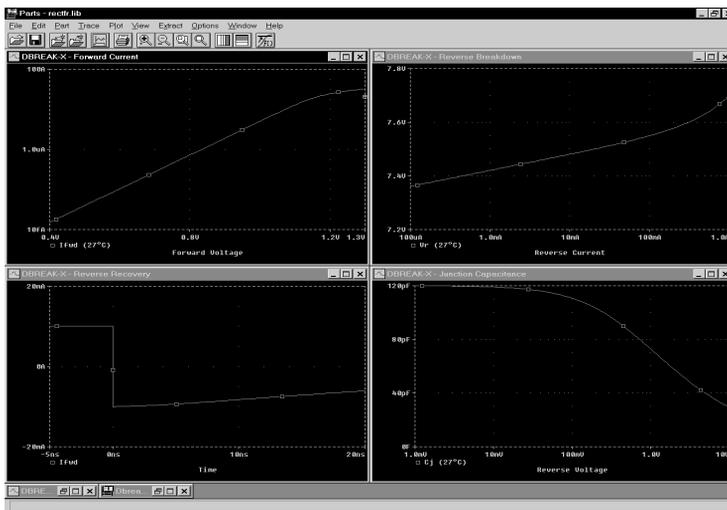


Figure 4-6 Assorted Device Characteristic Curves for a Diode

Adding curves for more than one temperature

By default, the Parts utility computes device curves at 27°C. For any characteristic, you can add curves to the plot at other temperatures.

To add curves for Forward Current at a different temperature

press **[Ins]**

- 1 Click the Forward Current plot window to bring it to the foreground.
- 2 From the Trace menu, select Add.
- 3 Type 100 (in °C).
- 4 Click OK.

The Forward Current plot should appear as shown in Figure 4-7.

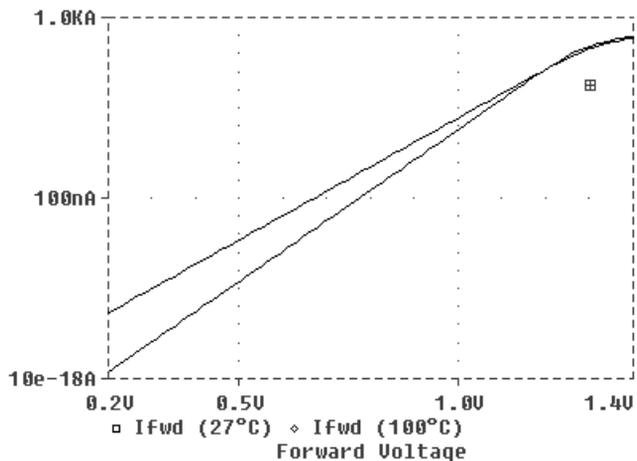


Figure 4-7 Forward Current Device Curve at Two

Completing the model definition

You can refine the model definition by:

- modifying the entered data as described before, or
- editing model parameters directly.

You can update individual model parameters by double-clicking the entry in the Parameters list of the main parts window, and then updating the parameter values in the dialog box. When you exit from the dialog box, the Parts utility automatically updates the device curves.

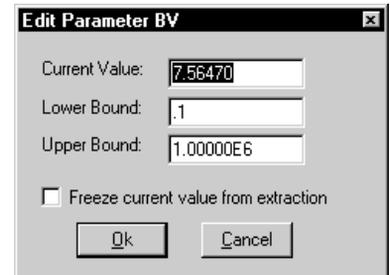
For now, you will leave the model parameters at their current settings.

To save the model definition with the current parameter values and to make the model available to your schematic

- 1 From the File menu, select Save to update `rectfr.lib` and write the library to disk.

Your schematic is now ready to simulate with the model definition you just created.

Example: If you double-click BV in the Parameters list, the Edit Parameter BV dialog box displays.



This dialog box lets you define a valid range for the model parameter and specify whether it should be excluded from the extraction process.

Using the Model Editor

The model editor displays the PSpice syntax for model definitions:

- `.MODEL` syntax for models defined as parameter sets
- `.SUBCKT` syntax for models defined as netlist subcircuits

You can use the model editor to:

- change definitions, and
- create new definitions

by typing in PSpice commands and netlist entries. When you are finished, Schematics automatically configures the model definitions into the model libraries.

Changing Model Properties

To find out more about PSpice command and netlist syntax, refer to the online *MicroSim PSpice A/D Reference Manual*.

The model editor window contains an edit area that displays the PSpice commands and netlist entries for the model definition. You can freely edit the definition just as you would in any standard text editor.

Editing .MODEL definitions

For definitions implemented as model parameter sets using PSpice .MODEL syntax, the model editor lists one parameter per line. This makes it easier to add DEV/LOT tolerances to model parameters for Monte Carlo or sensitivity/worst-case analysis.

AKO (A Kind Of) models are based on another model, and inherit the parameter values of the base model. Any parameters defined for the AKO model supersede those inherited from the base model. See Figure 4-8.

If the model is an AKO (see sidebar) and you want to view all of the parameters that are available, you can have the model editor *flatten* the AKO hierarchy into one model definition.

To convert an AKO model into a non-AKO model (showing all parameters)

- 1 Click Expand AKO(s).

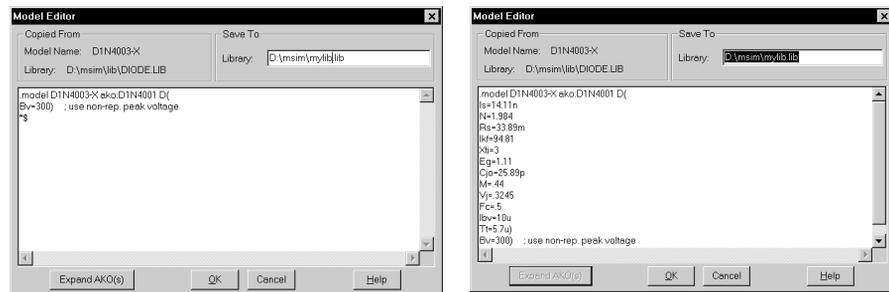


Figure 4-8 AKO Model Definition Before and After Flattening

Editing .SUBCKT definitions

For definitions implemented as subcircuit netlists using PSpice .SUBCKT syntax, the model editor displays the subcircuit syntax exactly as it appears in the model library. The model editor also includes all of the comments immediately before or after the subcircuit definition.

Changing the model name

You can change the model name directly in the PSpice .MODEL or .SUBCKT syntax, but should double-check that the new name does not conflict with models already contained in the libraries.

Note *If you do create a model with the same name as another model and want PSpice to always use your model, make sure the configured model libraries are ordered so your definition precedes any other definitions.*

To find out more about instance model naming conventions, see [What is an instance model? on page 4-33](#).

To find out more about search order in the model library, see [Changing Model Library Search Order on page 4-45](#).

Running the Model Editor from the Symbol Editor

If you want to:

- base a new part on an existing symbol, or
- edit the model for an existing symbol and have it affect all schematics that use the symbol,

then run the model editor from the symbol editor in Schematics.

Starting the model editor

To start the model editor from the Schematics symbol editor

Start the symbol editor.

- 1 From the File menu in the schematic editor, select Edit Library.

For general information about creating symbols, see [Chapter 5, Creating Symbols for Models](#).

Once you have started the model editor, you can proceed to change the text as described in [Changing Model Properties on page 4-30](#).

To find out how Schematics searches the library, see [Changing Model Library Search Order on page 4-45](#).

If you want to save the model to a different library, then

- 1 In the Library text box in the Save To frame, type a different name.

The model editor still automatically configures the model library as global.

Start the model editor.

- 2 Create or load a symbol definition.
- 3 From the Edit menu, click Attribute.
 - a Make sure that your symbol has a MODEL attribute and an assigned value.
 - b Change the PART attribute value as needed to match the model name.
 - c Click OK.
- 4 From the Edit menu, select Model.
- 5 Click Edit Model (Text).

The model editor searches the configured model libraries for the model.

- If found, the model editor opens the library containing the original model and displays the definition in the edit area.
- If not found, the model editor assumes that this is a new model and displays an empty edit area.

Saving global models

When you save your edits, the following is done for you to make sure the symbol saved to the symbol library and new model definition are linked and available to any schematic:

- The model editor saves the model definition to `symbol_library_name.lib`.
- If the library is new, the model editor configures it for global use.
- The symbol editor assigns the new model name to the MODEL attribute for the symbol in the library.

To save models created in the symbol editor

- 1 In the model editor window, click OK when you have finished editing the model.

Running the Model Editor from the Schematic Editor

If you want to:

- define tolerances on model parameters for statistical analyses,
- test behavior variations on a part, or
- refine a model before making it available to all schematics,

then run the model editor from the schematic editor in Schematics.

This means editing models for part instances on your schematic. When you select a part instance and edit its model, the schematic editor automatically creates an *instance model* that you can then change.

What is an instance model?

An instance model is a *copy* of the symbol's original model. The copied model is local to the design. You can customize the instance model without impacting any other schematic that uses the original symbol from the library.

When the schematic editor creates the copy, it assigns a unique name that is by default:

original_model_name-Xn

where *n* is *<blank 1 | 2 | ... >* depending on the number of different instance models derived from the original model for the current schematic.

You can also use the model editor to view the syntax for a model definition. When you are finished, be sure to click **Cancel** so the schematic editor does not create an instance model.

For more information on instance models, see [Reusing Instance Models on page 4-39](#).

Once you have started the model editor, you can proceed to change the text as described in [Changing Model Properties on page 4-30](#).

To find out how Schematics searches the library, see [Changing Model Library Search Order on page 4-45](#).



Actions that automatically configure the instance model library for global use instead

Instance model libraries are normally configured for local use. However, if you perform the following action, the model editor configures the library for global use instead:

- Save the model to a different library by typing a new file name in the Library text box in the Save To frame.

Starting the model editor

To start editing an instance model

- 1 In the schematic editor, select the symbol on your schematic.
- 2 From the Edit menu, select Model.
- 3 Click Edit Instance Model (Text).

The model editor searches the configured libraries for the instance model:

- If found, the model editor opens the library containing the instance model and displays the instance model definition in the edit area.
- If not found, the model editor assumes that this is a new instance model and does the following: makes a copy of the original model definition, names it *original_model_name-Xn*, and displays the new model definition.

Saving local models

When you save your edits, the following is done for you to make sure the instance model is linked to the selected part instances on your schematic:

- The model editor saves the model definition to *schematic_name.lib*.
- If the library is new, the model editor configures *schematic_name.lib* for local use.
- The schematic editor assigns the new model name to the MODEL attribute for each of the selected part instances.

To save instance models created in the model editor

- 1 In the model editor window, click OK when you have finished editing the model.

Example: Editing a Q2N2222 Instance Model

Suppose you have a schematic named `my.sch` that contains several instances of a Q2N2222 bipolar transistor. Suppose also, that you are interested in the effect of base resistance variation on one specific device—Q6. To do this you need to do the following:

- Define a tolerance (in this example, 5%) on the `Rb` model parameter.
- Set up and run a Monte Carlo analysis.

The following example demonstrates how to set up the instance model for Q6.

Starting the model editor

To start the model editor, you need to:

- 1 Select Q6 on your schematic.
- 2 From the Edit menu, select Model.
- 3 Click Edit Instance Model (Text).

The model editor automatically creates a copy of the Q2N2222 base model definition and changes the name to Q2N2222-X. The model editor also displays the PSpice syntax for the copied model in the edit area.

Editing the Q2N2222-X model instance

Text edits appropriate to this example are as follows:

- Add the `DEV 5%` clause to the `Rb` statement (required).
- Change the model name to Q2N2222-MC (optional, for descriptive purposes only).

Figure 4-9 shows how the model definition looks after having made these changes.

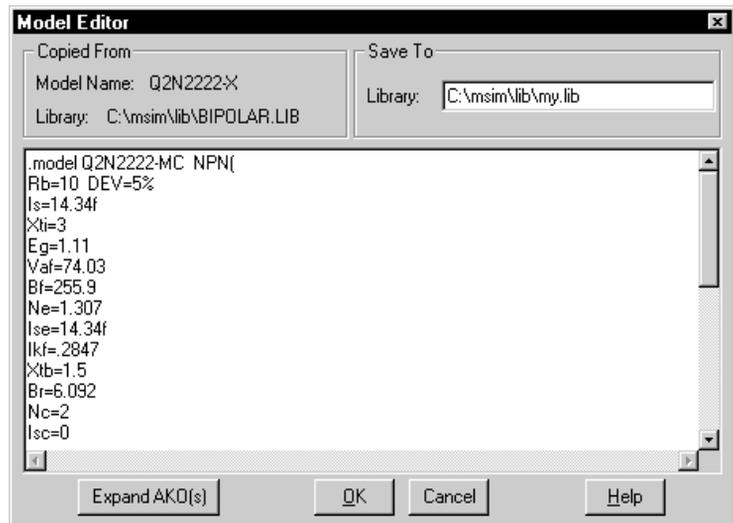


Figure 4-9 Model Editor Showing Q2N2222 with a DEV Tolerance Set on Rb

If you were to verify the model library configuration (from the Analysis menu in the schematic editor, select Library and Include Files), you would see entries for `nom.lib*` (global as denoted by the asterisk) and `my.lib` (local, no asterisk) in the model library list.

If you wanted, you could change the model reference for this part back to the original Q2N2222 by following the procedure [To change model references for part instances on your schematic on page 4-38](#).

Saving the edits and updating the schematic

When you click OK, two things happen:

- The model editor writes the model definition to the library showing in the Library text box, and closes the editing window.
- The schematic editor updates the MODEL attribute value to Q2N2222-MC for the Q6 part instance.

In this example, the library defaults to `my.lib` (see Figure 4-9). If `my.lib` does not already exist, the model editor creates and saves it in the current working directory. The schematic editor then automatically configures it as a local model library for use with the current schematic only.

Now you are ready to set up and run the Monte Carlo analysis.

Using the Create Subcircuit Command

The Create Subcircuit command in the schematic editor creates a subcircuit netlist definition for the displayed level of hierarchy and all lower levels in your schematic.

The schematic editor does the following things for you:

- Maps any named interface ports at the active level of hierarchy to terminal nodes in the PSpice `.SUBCKT` statement.
- Writes the subcircuit definition to a file named `schematic_name.sub`.

Before you can use the subcircuit definition in your schematic, you need to:

- Create a symbol for the subcircuit.
- Configure the `schematic_name.sub` file so PSpice knows where to find it.

To create a subcircuit definition for a portion of your schematic

- 1 Move to the level of hierarchy for which you want to create a subcircuit (`.SUBCKT`) definition.
- 2 From the Draw menu, select Get New Part.
- 3 Place interface ports for your subcircuit:
 - IF_IN for input ports
 - IF_OUT for output ports
- 4 From the File menu, select Save.
- 5 From the Tools menu, select Create Subcircuit to generate the subcircuit definition and save it to `schematic_name.sub`.
- 6 In the schematic editor, from the Analysis menu, select Library and Include Files, and then configure



The Create Subcircuit command will not help you create a hierarchical design. You need to do this yourself before using the Create Subcircuit command. For information on hierarchical schematics and how to create them, refer to your *MicroSim Schematics User's Guide*.

Refinements can include extending the subcircuit definition using the optional nodes construct, OPTIONAL:, the variable parameters construct, PARAMS:, and the .FUNC and local .PARAM commands.

schematic_name.sub as either a model library or include file (see [Configuring Model Libraries on page 4-41](#)).

- 7 If necessary, refine the subcircuit definition for the new symbol or for a part instance on your schematic using the model editor (see [Using the Model Editor on page 4-29](#)).
- 8 From the File menu, select Edit Library to start the symbol editor.
- 9 Define a new symbol for the subcircuit definition.

One way to do this is to use the symbol wizard. See [Chapter 5, Creating Symbols for Models](#) for a complete discussion.

Changing the Model Reference to an Existing Model Definition

Symbols are linked to models by the model name assigned to the symbols' MODEL attribute. You can change this assignment by replacing the MODEL attribute value with the name of a different model that already exists in the library.

You can do this for:

- A part instance in your schematic.
- A symbol in the symbol library.

To change model references for part instances on your schematic

- 1 Find the name of the model that you want to use.
- 2 In the schematic editor, select one or more symbols on your schematic.
- 3 From the Edit menu, select Model.
- 4 Click Change Model Reference.

- 5 In the Model Name text box, type the name of the existing model that you want to use.
- 6 Click OK.

To change the model reference for a symbol in the symbol library

- 1 Find the name of the model that you want to use.
- 2 In the schematic editor, from the File menu, select Edit Library to start the symbol editor.
- 3 From the File menu, select Open, and then select the symbol library that contains the symbol that you want to change.
- 4 From the Part menu, select Get, and then select the symbol that you want to change.
- 5 From the Part menu, select Attributes.
- 6 Click MODEL, and then change its value to the name of the existing model that you want to use.
- 7 Click PART, and then change its value to reflect the new symbol name.

In general, the symbol name should match the MODEL name.

Or you can replace steps [2-4](#) as follows:

- 1 In the schematic editor, select the symbol that you want to change.
- 2 From the Edit menu, select Symbol to start the symbol editor.

Now you can continue with step [5](#) in the main procedure.

Reusing Instance Models

If you created instance models in your schematic and want to reuse them, there are two things you can do:

- Assign the instance model to other part instances in the same schematic.
- Change the instance model to a global model and create a symbol that corresponds to it.

For information on how to create instance models, see:

- [Running the Parts Utility from the Schematic Editor on page 4-20](#)
- [Running the Model Editor from the Schematic Editor on page 4-33](#)

Reusing Instance Models in the Same Schematic

There are two ways to use the instance model elsewhere in the same schematic.

To use the instance model elsewhere on your schematic

- 1 Do one of the following:
 - Change the model reference for other part instances to the name of the new model instance.
 - From the Edit menu, use Copy and Paste to place more part instances.

See [Changing the Model Reference to an Existing Model Definition on page 4-38](#).

Making Instance Models Available To All Schematics

If you are refining model behavior locally in your schematic, and are ready to make it available to any schematic, then you need to link the model definition to a symbol and configure it for global use.

To make your instance model available to any schematic

- 1 Create a symbol and assign the instance model name to the MODEL attribute.
- 2 If needed, move the instance model definition to an appropriate model library, and make sure the library is configured for global use.

Note *If you use the symbol wizard to create the symbol automatically from the model definition, then this step is handled for you.*

See [Chapter 5, Creating Symbols for Models](#) for more information.

See [Configuring Model Libraries on page 4-41](#) for more information.

Configuring Model Libraries

Though model libraries are usually configured for you, there are things that you sometimes must do manually. These are:

- add new model libraries that were created outside of Schematics or the Parts utility
- change the global or local scope of a model library
- change the library search order
- change or add directory search paths

The Library and Include Files dialog box

The Library and Include Files dialog box is where you can add, change, and delete model libraries from the configuration, or resequence the search order.

Note *Deletion in this context means you are removing the model library from the configured list. The library still exists on disk and you can add it back to the configuration later.*

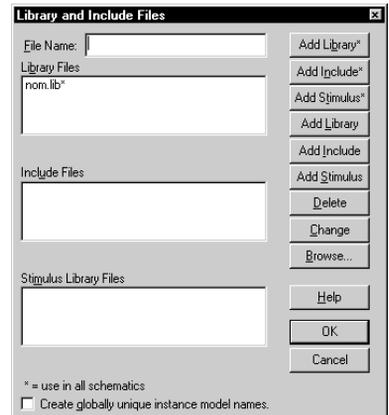
To display the Library and Include Files dialog box

- 1 From the Analysis menu in the schematic editor, select Library and Include Files.

The Library Files box lists the model libraries that PSpice searches for definitions matching the parts in your design. Files showing an asterisk (*) after their name have global scope; files with names left unmarked have local scope.

The buttons for adding model libraries to the configuration follow the same local/global syntax convention. Use:

- Add Library for local models.



A second list box contains include files. You can manually add local and global include files to your configuration using the Add Include and Add Include* buttons, respectively.

A third list box contains stimulus files. See [Configuring Stimulus Files on page 10-6](#) for more information.

- Add Library* for global models.

How PSpice Uses Model Libraries

PSpice searches libraries for any information it needs to complete the definition of a part or to run a simulation. If an up-to-date index does not already exist, PSpice automatically generates an index file and uses the index to access only the model definitions relevant to the simulation. This means:

- Memory is not used up with definitions that your design does not use.
- There is no memory penalty for having large model libraries.
- Read-in time is kept to a minimum.

Search order

When searching for model definitions, PSpice scans the model libraries using these criteria:

- local model libraries before global model libraries
- model library sequence as listed in the Library Files list box in the Analysis and Include Files dialog box
- local directory (where the current schematic resides) first, then the list of directories specified in the library search path in the order given (see [Changing the Library Search Path on page 4-46](#))

Handling duplicate model names

If your model libraries contain duplicate model names, PSpice always uses the first model it finds. This means you might need to resequence the search order to make sure PSpice uses the model that you want. See [Changing Model Library Search Order on page 4-45](#).

Note *Keep in mind that PSpice searches local libraries before global libraries, if the new model you want to use is local and the duplicate definition is global, you do not need to make any changes.*



When you use include files instead

PSpice treats model library and include files differently as follows:

- For model library files, PSpice reads in only the definitions it needs to run the current simulation.
- For include files, PSpice reads in the file in its entirety.

This means if you configure a model library (.lib extension) as an include file using the Add Include or Add Include* button, PSpice reads in every model definition contained in that file.

If the model library is large, you may overload the memory capacity of your system. However, when developing models, you can do the following:

- 1 Initially configure the model library as an include file; this avoids rebuilding the index files every time the model library changes.
- 2 When your models are stable, reconfigure the include file containing the model definitions as a library file.

To reconfigure an include file as a library file:

- 1 From the Analysis menu, select Library and Include Files.
- 2 Select the include file that you want to change.
- 3 Click Add Library* or Add Library.
- 4 Click Delete to remove the include file entry.

Adding Model Libraries to the Configuration

Schematics always adds new libraries above the selected library name in the Library Files list box.

To add model libraries to the configuration

- 1 From the Analysis menu, select Library and Include Files.
- 2 Click the library name positioned one entry below where you want to add the new library.
- 3 In the File Name text box, either:
 - type the name of the model library, or
 - use Browse to locate and select the library.
- 4 Do one of the following:
 - If the model definitions are for local use in the current schematic, click Add Library.
 - If the model definitions are for global use in any schematic, click Add Library* instead.
- 5 Click OK.

Note *If the model libraries reside in a directory that is not on the library search path, and you use the Browse button in step 3 to select the libraries you want to add, then the schematic editor automatically updates the library search path. Otherwise, you need to add the directory path yourself. See [Changing the Library Search Path on page 4-46](#).*

Changing Local and Global Scope

There are times when you might need to change the scope of a model library from local to global, or vice versa.

To change the scope of a local model to global

- 1 From the Analysis menu, select Library and Include Files.
- 2 Select the model library that you want to change.
- 3 Click Add Library* to add a global entry.
- 4 Click Delete to remove the local entry.

If you have a global model that you want to make local, use the Add Library button instead in step [3](#).

Example: If you have an instance model that you now want to make available to any design, then you need to change the local model library that contains it to have global scope.

For more information, see [Global vs. Local Models and Libraries on page 4-5](#).

Changing Model Library Search Order

Two reasons why you might need to modify the search order are to:

- reduce the search time
- avoid using the wrong model when there are model names duplicated across libraries; PSpice always uses the first instance

See [Handling duplicate model names on page 4-43](#) for more information.

To change the order of libraries

- 1 In the Library and Include Files dialog box:
 - a In the Library Files list, delete the library that you want to move.
 - b Add back the same library in the location where you want it. (Remember that Schematics adds a new library above the selected entry in the list.)

See [Adding Model Libraries to the Configuration on page 4-44](#) for more information.

- If you have listed multiple `.lib` commands within a single library (like `nom.lib`), then edit the library using a text editor to change the order.

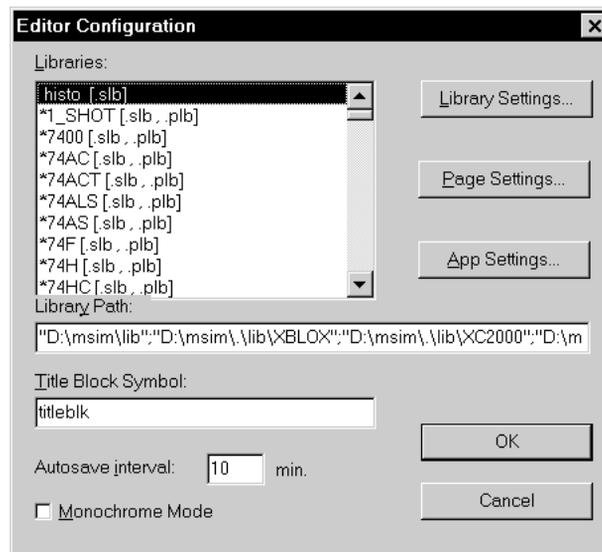
Example: The model libraries `diodes.lib` and `ediodes.lib` (European manufactured diodes) shipped with your MicroSim programs have identically named device definitions. If your schematic uses a device out of one of these libraries, you need to position the model library containing the definition of choice earlier in the list. If your system is configured as originally shipped, this means you need to add the specific library to the list *before* `nom.lib`.



Do not edit `nom.lib`. If you do, PSpice will recreate the indexes for every model library referenced in `nom.lib`. This can take some time.

Changing the Library Search Path

For model libraries that are configured without explicit path names, PSpice first searches the directory where the working schematic resides, then steps down the list of directories specified in the Library Path text box in the Editor Configuration dialog box.



To change the library search path

- 1 In the schematic editor, from the Options menu, select Editor Configuration.
- 2 In the Library Path text box, position the pointer after the directory path that PSpice should search before the new path.
- 3 Type in the new path name following these rules:
 - Use a semi-colon character (;) to separate two path names.
 - Do not follow the last path name with a semi-colon.

Example: To search first C:\MSIM\LIB then C:\MYLIBS for model libraries, type

```
"C:\MSIM\LIB" ; "C:\MYLIBS"
```

in the Library Path text box.

Creating Symbols for Models

5

Chapter Overview

This chapter provides information about creating symbols for model definitions so you can simulate the part from your schematic.

Topics are grouped into four areas introduced later in this overview. If you want to find out quickly which tools to use to complete a given task and how to start, then:

- 1 Go to the roadmap in [Ways to Create Symbols for Models on page 5-4](#).
- 2 Find the task you want to complete.
- 3 Go to the sections referenced for that task for more information about how to proceed.

For general information about creating symbols, including from scratch or from existing symbols, refer to your *MicroSim Schematics User's Guide*.

Background information These topics provide background on the things you need to know and do to prepare for creating symbols.

- [What's Different About Symbols Used for Simulation? on page 5-3](#)
- [Preparing Your Models for Symbol Creation on page 5-5](#)

Task roadmap This topic helps you find the sections in this chapter that are relevant to the symbol creation task that you want to complete. This topic is:

- [Ways to Create Symbols for Models on page 5-4](#)

How to use the tools These topics explain how to use different tools to create symbols for model definitions. Topics include:

- [Using the Symbol Wizard on page 5-6](#)
- [Using the Parts Utility to Create Symbols on page 5-11](#)
- [Creating AKO Symbols on page 5-8](#)
- [Basing New Symbols On a Custom Set of Symbols on page 5-13](#)

Other useful information These topics explain how to refine symbol graphics and attributes. Topics include:

- [Editing Symbol Graphics on page 5-15](#)
- [Defining Symbol Attributes Needed for Simulation on page 5-18](#)

What's Different About Symbols Used for Simulation?

A symbol used for simulation has these special properties:

- a link to a simulation model
- a netlist translation
- modeled pins
- other simulation properties specific to the part

For information on adding simulation models to a model library, see [Chapter 4, Creating and Editing Models](#).

Note *To use the symbol for board layout, you must link a package definition. For information on creating and linking package definitions, refer to your MicroSim Schematics User's Guide.*

Ways to Create Symbols for Models

If you want to...	Then do this...	To find out more, see this...
<ul style="list-style-type: none">➔ Automatically create symbols for a set of vendor or user-defined models saved in a model library.➔ Change the graphic standard for an existing model library.	Run the symbol wizard to create symbols from a model library.	Using the Symbol Wizard on page 5-6 Basing New Symbols On a Custom Set of Symbols on page 5-13
<ul style="list-style-type: none">➔ Produce a compact symbol library for a set of vendor or user-defined models.	Create AKO symbols using the symbol editor.	Creating AKO Symbols on page 5-8
<ul style="list-style-type: none">➔ Automatically create one symbol each time you extract a new model.	Run the Parts utility* and enable automatic creation of symbols.	Using the Parts Utility to Create Symbols on page 5-11 Using the Parts Utility to Edit Models on page 4-10 Basing New Symbols On a Custom Set of Symbols on page 5-13

*. For a list of device types that the Parts utility supports, see [Parts-Supported Device Types on page 4-12](#).

Preparing Your Models for Symbol Creation

If you already have model definitions and want to create symbols for them, you should organize the definitions into libraries containing similar device types.

To set up a model library for symbol creation

- 1 If all of your models are in one file and you wish to keep them that way, rename the file to:
 - reflect the kinds of models contained in the file, and
 - have the `.lib` extension.
- 2 If each model is in its own file, and you want to concatenate them into one file, use the DOS `copy` command.
Example: You can append a set of files with `.mod` extensions into a single `.lib` file using the DOS command:

```
copy *.mod mylib.lib
```
- 3 Make sure the model names in your new library do not conflict with model names in any other model library.

Model libraries typically have a `.lib` extension. However, you can use a different file extension as long as the file format conforms to the standard model library file format.

For information on managing model libraries, including the search order PSpice uses, see [Configuring Model Libraries on page 4-41](#).

Using the Symbol Wizard

If:

- you want to automatically create symbols for a set of similar model definitions that are saved in a model library, and
- you do not need to minimize the size of the new symbol library to save disk space,

then use the symbol wizard.

How to Start the Symbol Wizard

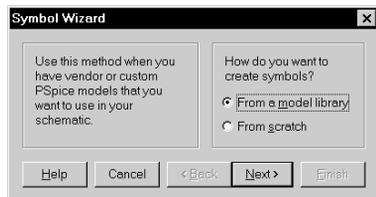
To start the symbol wizard for a set of model definitions in a library

Start the symbol editor

- 1 In the schematic editor, from the File menu, select Edit Library.

Start the symbol wizard

- 2 From the Part menu, select Symbol Wizard.
- 3 In the first wizard screen, choose From a Model Library.
- 4 Click Next.
- 5 Continue to follow the instructions.



How the Symbol Wizard Works

The symbol wizard operates in four phases: setup, automatic symbol creation, refinement, and global configuration.

Phase 1: Setup To begin, the symbol wizard asks you for:

- the name of the model library that contains the model definitions, and
- the name of the symbol library to save the new symbol definitions to.

Instead of using the MicroSim default symbol set, you can use your own set of standard symbols. To find out more, see [Basing New Symbols On a Custom Set of Symbols on page 5-13](#).

Phase 2: Automatic symbol creation The symbol wizard automatically creates symbols for:

- all of the models defined as model parameter sets (.MODEL syntax), and
- as many of the models defined as subcircuits (.SUBCKT syntax) as it recognizes.

When phase 2 completes, the wizard displays two lists showing:

- subcircuits that it could not create symbols for, and
- subcircuits with symbols.

Phase 3: Refinement The wizard arranges the subcircuits into groups that have the same terminal node names (pins) in the same order.

If you want to replace a symbol that the wizard created or want to create a symbol for a group (or subset) of subcircuits that don't have a symbol, then you can do so by choosing to base the new symbol on either:

- an existing symbol, or
- a generic rectangle,

and then continue to follow the instructions.

You can repeat this process until you are satisfied with the symbol assignments.

If needed, you can refine the graphics after finishing the wizard. See [Editing Symbol Graphics on page 5-15](#).

Phase 4, Global library configuration When you click Finish, the wizard saves the symbols to the symbol library you named in the setup phase and does the following:

- Configures the symbol library for global use.
- If the model library is not yet configured, configures it for global use.

Creating AKO Symbols

If you want to create a *compact* symbol library for a set of similar models, then create AKO symbols. AKO symbols are a convenient way to define new symbols that are only slightly different from another symbol.

Example: The symbol library, `bipolar.slb`, contains two base symbols named `qnpn` and `qnpn`. Every other symbol in this library is derived from one of these base symbols. For example, `DH3467CD` is an AKO of the `qnpn` base symbol, with changes to the PART and MODEL attributes to reflect individual part behaviors. This means that `DH3467CD` inherits all of the graphics and attribute values of `qnpn`, except that the specific PART and MODEL attribute values for `DH3467CD` supersede the corresponding attribute values for `qnpn`.

What Are Base vs. AKO Symbols?

Base symbols A base symbol defines the graphical properties of the symbol and the minimum set of attributes needed to make the symbol functional in any schematic.

AKO (A Kind Of) symbols An AKO symbol inherits all of the graphics and attributes of the base symbol that it references, and it can alter or add to the base symbol's attributes.

Base and AKO Symbols in Symbol Libraries

A symbol library contains definitions for both base symbols and AKO symbols. In any symbol library provided by MicroSim, the base symbols appear at the end of the file.

Note *An AKO symbol can only reference base symbols contained in its own library.*

How to Create AKO Symbols

AKO symbol creation is a two step process as follows:

- 1 Create the base symbol.
- 2 Add one or more AKO symbols.

The following procedure explains how to create a new library with the base and AKO symbols.

To create a new library with a base symbol and AKO symbols

Start the symbol editor

- 1 From the File menu in the schematic editor, select Edit Library.

Add the base symbol

- 2 Create the base symbol using either of these methods:
 - Copy (and modify) an existing base part.
 - Create the base part from scratch.

Refer to your *MicroSim Schematics User's Guide* for instructions.

Note *When you are creating new symbols, MicroSim recommends that you save the symbols to a library other than those provided by MicroSim. This way, you'll avoid losing custom symbols when you next install a MicroSim program update.*

Here are a few things to keep in mind:

- If you need to create new symbols with different graphics, you can add more than one base symbol to the same custom symbol library.
- If you need to add symbols over the course of several editing sessions, you can open your custom symbol library using Open from the File menu and proceed to add new base and AKO symbols as described in this procedure.
- AKO symbols must reside in the same symbol library as the base symbols they reference.

- 3** Save the base symbol to a new library:
 - a** From the File menu, select Save As.
 - b** Type the name of the new library *without* the .slb extension. Make sure the new library name:
 - matches the name of the model library with the corresponding definitions, and
 - does not duplicate an existing library name.
 - c** Click OK.
- 4** Click YES when you are prompted to add this library to the list of Schematics' configured libraries.

Schematics automatically configures the symbol library for global use, which makes the symbols available to any schematic.

Add one or more AKO symbols

- 5** From the Part menu, select New.
- 6** In the Description text box, type a description for the part.
- 7** In the Part Name text box, type the name of the part.

This should match the name of the corresponding model definition.
- 8** In the AKO Name text box, type the name of the *base* part.
- 9** Click OK.

The status bar at the top of the screen lists the AKO symbol name.
- 10** From the Part menu, select Attributes.
- 11** Change the MODEL attribute and any other attributes as needed.
- 12** Click OK.
- 13** From the File menu, select Save.
- 14** For each AKO symbol that you want to create, repeat steps **5** through **13**.

In general, you will not need to edit the TEMPLATE attribute.

Completing the Configuration of Your Part

The only thing left to do is to make sure PSpice knows where to find the model library that contains the model definitions corresponding to the symbols you just created.

To configure the model library

- 1 Click in the schematic editor window.
- 1 From the Analysis menu, select Library and Include Files.
- 2 In the File Name text box, type the name of the library *including* the file extension.
- 3 Click Add Library* (with an asterisk) to configure the model library for global use.
- 4 Click OK.

Using the Parts Utility to Create Symbols

If you want to run the Parts utility and enable automatic creation of symbols for any model that you create or change, then run the Parts utility alone. This means any models you create are not currently tied to a part instance on your schematic or to a symbol editing session.

Note *If you open an existing model library, the Parts utility creates symbols for only the models that you change or add to it. If you want to create symbols for all model definitions in a library, or for model definitions that the Parts utility does not support, then use the symbol wizard (see [Using the Symbol Wizard on page 5-6](#)).*



To find out how to use Parts to create models, see [Using the Parts Utility to Edit Models on page 4-10](#).

To find out which device types the Parts utility supports, see [Parts-Supported Device Types on page 4-12](#).

Starting the Parts Utility

If you have already started the Parts utility from Schematics, and want to continue working on new models and symbols, then:

- 1 Close the opened model library.
- 2 Open a new model library.
- 3 Load a device model or create a new one.

To start the Parts utility alone

- 1 From the MicroSim program folder, select Parts.
- 2 From the File menu, select Open/Create, and enter an existing or new model library name.
- 3 From the Part menu, select New, Copy, or Import to load a device model.

Setting Up Automatic Symbol Creation

Symbol creation from the Parts utility is optional. By default, automatic symbol creation is enabled. However, if you previously disabled symbol creation, you will need to enable it before creating a new model and symbol.

Instead of using the MicroSim default symbol set, you can use your own set of standard symbols. To find out more, see [Basing New Symbols On a Custom Set of Symbols on page 5-13](#).

Example: If the model library is named `myparts.lib`, then the Parts utility creates the symbol library named `myparts.slb`.

To automatically create symbols for new models

- 1 From the Options menu, select Symbol Creation Setup.
- 2 If not already checked, select Always Create Symbol to enable automatic symbol creation.
- 3 In the Save Symbols To frame, define the name of the symbol library for the new symbol. Choose one of the following:
 - Symbol Library Path Same As Model Library to create or open the `.slb` file that has the same name prefix as the currently open model library (`.lib`).
 - User-Defined Symbol Library, and then enter a library name into the Symbol Library Name text box.

Basing New Symbols On a Custom Set of Symbols

If you are using the symbol wizard or the Parts utility to automatically generate symbols for model definitions, and you want to base the new symbols on a custom graphic standard (rather than the MicroSim default symbols), then you can change which underlying symbols either utility uses by setting up your own set of symbols.

Note *If you use a custom symbol set, the symbol wizard and the Parts utility always check the custom symbol library first for a symbol that matches the model definition.*

To create a custom set of symbols for automatic symbol generation

- 1 Create a symbol library with the custom symbols.

Be sure to name these symbols by their device type as shown in Table 5-1; this is how the symbol wizard and the Parts utility determine which symbol to use for a model definition.

For information on creating symbols from scratch or from an existing symbol, refer to your *MicroSim Schematics User's Guide*.

Table 5-1 *Symbol Names for Custom Symbol Generation*

For this device type...	Use this symbol name...	For this device type...	Use this symbol name...
Bipolar transistor: LPNP	LPNP	MOSFET: N-channel	NMOS
Bipolar transistor: NPN	NPN	MOSFET: P-channel	PMOS
Bipolar transistor: PNP	PNP	OPAMP: 5-pin	OPAMP5
Capacitor*	CAP	OPAMP: 7-pin	OPAMP7
Diode	DIODE	Resistor*	RES
GaAsFET*	GASFET	Switch: voltage-controlled*	VSWITCH
IGBT: N-channel	NIGBT	Transmission line*	TRN
Inductor*	IND	Voltage comparator	VCOMP
JFET: N-channel	NJF	Voltage comparator: 6 pin	VCOMP6
JFET: P-channel	PJF	Voltage reference	VREF
Magnetic core	CORE	Voltage regulator	VREG

*. Does not apply to the Parts utility.

- 2 For each custom symbol, set its MODEL attribute to `M where ` is a back-single quote or grave symbol.

This tells the Parts utility or symbol wizard to substitute the correct model name.

To base new symbols on custom symbols using the Parts utility

- 1 From the Options menu in the Parts utility, select Symbol Creation Setup, and enable automatic symbol creation as described in the procedure, [To automatically create symbols for new models on page 5-12](#).
- 2 Click Advanced Options.
- 3 In the Base Symbols On frame, choose Symbols in Existing Symbol Library, and then enter the name of the symbol library that contains your custom symbols.
- 4 Click OK.

To base new symbols on custom symbols using the symbol wizard

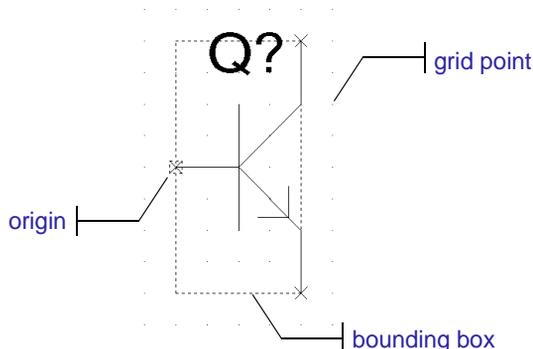
- 1 In the symbol wizard screen that asks, “Which symbol library do you want to save your symbols to?”, click Advanced Options.
- 2 Choose “Symbols in an existing symbol library...”, and then enter the name of the symbol library that contains your custom symbols.

Editing Symbol Graphics

If you created symbols using the symbol wizard or the Parts utility, and you want to make further changes, the following sections explain a few key things to remember when you edit the symbols.

How Schematics Places Symbols

When placing symbols in your schematic, the schematic editor uses the symbol's origin and bounding box as points of reference for different editing activities.



You will need to adjust these when you change your symbol in the symbol editor. The symbol editor helps you position the origin, bounding box, and also pins by using an adjustable snap grid.

Here are the things to check when changing symbol graphics:

- ✓ Is the origin at the connecting point of the upper leftmost pin of the symbol?
- ✓ Are all visible pins contained within the bounding box?
- ✓ Is the bounding box no larger than necessary?
- ✓ Are all pins on grid?

Defining Important Symbol Elements

Origin

The origin, denoted by a small box with a dashed outline, is the center point that the schematic editor uses when rotating a part instance. By convention, the origin of each symbol in the symbol library is placed at the point of connection to the upper far left pin on the device.

The point of connection of a wire or pin is known as the *hot-spot*.

To define the symbol origin

- 1 From the Graphics menu in the symbol editor, select Origin.
- 2 Double-click the connecting point of the pin that you want to use as the origin (usually the upper far-left pin).

Bounding box

The bounding box, denoted by a large rectangle with a dashed outline, defines the selection area for a part instance on the schematic. By convention, the bounding box encompasses the symbol graphics and pins.

You can position attributes that are visible on the schematic, outside of the bounding box.

To make proper connections, the bounding box *must* contain all visible pins. If you try to save a symbol that has pins outside the bounding box, the symbol editor issues a warning message.

Note *To make selection of closely-spaced part instances on a schematic as easy as possible, avoid defining symbol bounding boxes that are larger than necessary.*

To define the symbol bounding box

- 1 From the Graphics menu in the symbol editor, select Bbox.
- 2 Click to establish one corner, and then move the mouse to adjust the size of the bounding box.
- 3 Click again to quit.

Grid spacing for graphics

The grid, denoted by evenly spaced grid points, regulates the sizing and positioning of graphic objects and the positioning of pins. The default grid spacing is set at 0.1", and the minimum grid spacing is 0.01".

You can change the grid spacing when you need to draw graphics in a tighter space.

To change the grid spacing

- 1 From the Options menu in the symbol editor, select Display Options.
- 2 In the Grid Spacing text box, type in a new value in inches.
- 3 Click OK.

Note *Before placing pins, be sure to set the grid spacing back to the default.*

Grid spacing for pins

Pins *must* be placed on the grid at integer multiples of the grid spacing. Because the default grid spacing in the schematic editor is set at 0.1", MicroSim recommends setting pin spacing in the symbol editor at 0.1" intervals from the origin of the symbol and at least 0.1" from any adjacent pins.

The symbol editor considers pins that are not placed at integer multiples of the grid spacing from the origin as *off-grid*, and displays a warning when you try to save the symbol.

Here are two rules of thumb:

- Make sure Stay on Grid is enabled when editing symbol pins and editing schematics so you can easily make connections.
- Make sure the grid spacing used to edit the symbol pins *matches* the grid spacing in the schematic editor.



Pin changes that alter the symbol template

If you either:

- change pin names, or
- delete pins

then you must adjust the value of the symbol's TEMPLATE attribute to reflect these changes. To find out how, see [Pin callout in subcircuit templates on page 5-24](#).

Here are the things to check when editing symbol attributes:

- ✓ Does the value of the MODEL attribute match the PSpice .MODEL or .SUBCKT name?
- ✓ Does the TEMPLATE specify the correct number of pins/nodes?
- ✓ Are the pins/nodes in the TEMPLATE specified in the proper order?
- ✓ Do the pin/node names in the TEMPLATE match the pin names on the symbol?

When in the schematic editor, you cannot edit the MODEL attribute directly through the Attributes dialog box. Instead, you must change the MODEL assignment using the Model command from the Edit menu and then either:

- change the model reference to an existing model definition, or
- create a new instance model.

The schematic editor automatically assigns the resulting model name to the MODEL attribute.

You can also edit the underlying model for a symbol from within the symbol editor, using the Model command from the Edit menu.

For more information on model editing in general, see [Chapter 4, Creating and Editing Models](#). For specific information on changing model references, see [Changing the Model Reference to an Existing Model Definition on page 4-38](#).

Defining Symbol Attributes Needed for Simulation

If you created your symbols using any of the methods discussed in this chapter, then your symbol will have these attributes already defined for it:

- MODEL and TEMPLATE for simulation, and
- PART and REFDES for identification.

You can also add other simulation-specific attributes like SIMULATION ONLY.

To find out more about this attribute...	See this...
MODEL	page 5-18
SIMULATION ONLY	page 5-19
TEMPLATE	page 5-19

MODEL

The MODEL attribute defines the name of the model that PSpice must use for simulation. When defining this attribute, this rule applies:

- The MODEL name should match the name of the .MODEL or .SUBCKT definition of the simulation model as it appears in the model library (.lib).

Example: If your design includes a 2N2222 bipolar transistor with a .MODEL name of Q2N2222, then the MODEL attribute for that part's symbol should be Q2N2222.

Note *Make sure that the model library containing the definition for the assigned model is configured into the model library. See [Configuring Model Libraries on page 4-41](#) for more information.*

SIMULATION ONLY

The SIMULATION ONLY attribute indicates that the part or special symbol applies only to simulation with PSpice. Parts like voltage sources, current sources, breakout parts (like RBREAK found in `breakout.slb`), and simulation control symbols (like VIEWPOINT found in `special.slb`) have the SIMULATION ONLY attribute. This attribute does not require a value.

TEMPLATE

The TEMPLATE attribute defines the PSpice syntax for the symbol's netlist entry. When netlisting, the schematic editor substitutes actual values from the circuit into the appropriate places in the TEMPLATE syntax, then writes the translated statement to the netlist file.

Any symbol that you want to simulate must have a defined TEMPLATE attribute. These rules apply:

- The pin names specified in the TEMPLATE attribute must match the pin names on the symbol.
- The number and order of the pins listed in the TEMPLATE attribute must match those for the associated .MODEL or .SUBCKT definition referenced for simulation.
- The first character in a TEMPLATE must be a PSpice device letter appropriate for the symbol (such as Q for a bipolar transistor).

When in the schematic editor, you cannot edit the TEMPLATE attribute. You must run the symbol editor to change this attribute.



Creating symbols not destined for simulation

Some symbol libraries contain parts designed only for board layout; PSpice cannot simulate these parts. This means they do not have TEMPLATE attributes or that the TEMPLATE attribute value is blank.

If you create a symbol that you don't want used for simulation, be sure to delete the TEMPLATE attribute that the symbol editor provides automatically.

Example: Connectors are used for board layout, but don't take part in simulation except to provide one or more pins where you can place Probe markers. Connectors have a PKGREF attribute but no TEMPLATE attribute.

TEMPLATE syntax

The TEMPLATE contains:

- *regular characters* that the schematic editor interprets verbatim, and
- *attribute names* and *control characters* that the schematic editor translates.

Regular characters in templates

Regular characters include the following:

- alphanumerics
- any keyboard symbol *except* the special syntactical symbols used with attributes (@ & ? ~ #).
- white space

An *identifier* is a collection of regular characters of the form:

alphabetic character [*any other regular character*]*.

Attribute names in templates

Attribute names are preceded by a special character as follows:

[@ | ? | ~ | # | &]<identifier>

The schematic editor processes the attribute according to the special character as shown in the following table.

This syntax...*	Is replaced with this...
@<id>	Value of <id>. Error if no <id> attribute or if no value assigned.
&<id>	Value of <id> if <id> is defined.
?<id>s...s	Text between s...s separators if <id> is defined.
?<id>s...ss...s	Text between the first s...s separators if <id> is defined, else the second s...s clause.

This syntax... *	Is replaced with this...
~<id>s...s	Text between s...s separators if <id> is undefined.
~<id> s...ss...s	Text between the first s...s separators if <id> is undefined, else the second s...s clause.
#<id>s...s	Text between s...s separators if <id> is defined, but delete rest of template if <id> is undefined.

*. s is a separator character

Separator characters include commas (,), periods (.), semi-colons (;), forward slashes (/), and vertical bars (|). You must always use the same character to specify an opening-closing pair of separators.

Note *You can use different separator characters to nest conditional attribute clauses.*

The ^ character in templates

The schematic editor replaces the ^ character with the complete hierarchical path to the device being netlisted.

The \n character sequence in templates

The symbol editor replaces the character sequence \n with a new line. Using \n, you can specify a multi-line netlist entry from a one-line template.

Example: The template fragment `?G|G=@G||G=1000|` uses the vertical bar as the separator between the if-then-else parts of this conditional clause. If G has a value, then this fragment translates to `G=<G attribute value>`. Otherwise, this fragment translates to `G=1000`.



Recommended scheme for netlist templates

Templates for devices in the symbol library start with a PSpice device letter, followed by the hierarchical path, and then the reference designator (REFDES) attribute.

MicroSim recommends that you adopt this scheme when defining your own netlist templates.

Example: `R^@REFDES ...` for a resistor

The % character and pin names in templates

Pin names are denoted as follows:

`%<pin name>`

where *pin name* is one or more regular characters.

The schematic editor replaces the `%<pin name>` clause in the template with the name of the net connected to that pin.

The end of the pin name is marked with a separator (see [Attribute names in templates on page 5-20](#)). To avoid name conflicts in Probe, the schematic editor translates the following characters contained in pin names.

This pin name character...	Is replaced with this...
<	l (L)
>	g
=	e
\XXX\	XXXbar

Note *To include a literal % character into the netlist output, enter %% in the template.*

TEMPLATE examples

Simple resistor (R) template

The R symbol has:

- two pins: 1 and 2
- two required attributes: REFDES and VALUE

Template

```
R^@REFDES %1 %2 @VALUE
```

Sample translation

```
R_R23 abc def 1k
```

where REFDES equals R23, VALUE equals 1k, and R is connected to nets abc and def.

Voltage source with optional AC and DC specifications (VAC) template

The VAC symbol has:

- two attributes: AC and DC
- two pins: + and -

Template

```
V^@REFDES %+ %- ?DC |DC=@DC | ?AC |AC=@AC |
```

Sample translation

```
V_V6 vp vm DC=5v
```

where REFDES equals v6, VSRC is connected to nodes vp and vm, DC is set to 5v, and AC is undefined.

Sample translation

```
V_V6 vp vm DC=5v AC=1v
```

where, in addition to the settings for the previous translation, AC is set to 1v.

Parameterized subcircuit call (X) template

Suppose you have a subcircuit Z that has:

- two pins: a and b
- a subcircuit parameter: G, where G defaults to 1000 when no value is supplied

To allow the parameter to be changed on the schematic, treat G as an attribute in the template.

Note For clarity, the *TEMPLATE* attribute value is shown here in multiple lines; in a symbol definition, it is specified in one line (no line breaks).

Template

```
X^@REFDES %a %b Z PARAMS: ?G|G=@G|
~G|G=1000|
```

Equivalent template (using the if...else form)

```
X^@REFDES %a %b Z PARAMS: ?G|G=@G| |G=1000|
```

Sample translation

```
X_U33 101 102 Z PARAMS: G=1024
```

where REFDES equals U33, G is set to 1024, and the subcircuit connects to nets 101 and 102.

Sample translation:

```
X_U33 101 102 Z PARAMS: G=1000
```

where the settings of the previous translation apply except that G is undefined.

Pin callout in subcircuit templates

To find out how to define subcircuits, refer to the .SUBCKT command in the online *MicroSim PSpice A/D Reference Manual*.

The number and sequence of pins named in a template for a subcircuit must agree with the definition of the subcircuit itself—that is, the node names listed in the .SUBCKT statement, which heads the definition of a subcircuit. These are the pinouts of the subcircuit.

Example: Consider the following first line of a (hypothetical) subcircuit definition:

```
.SUBCKT SAMPLE 10 3 27 2
```

The four numbers following the name SAMPLE—10, 3, 27, and 2—are the node names for this subcircuit's pinouts.

Now suppose that the symbol definition shows four pins:

```
IN+          OUT+          IN-          OUT-
```

The number of pins on the symbol equals the number of nodes in the subcircuit definition.

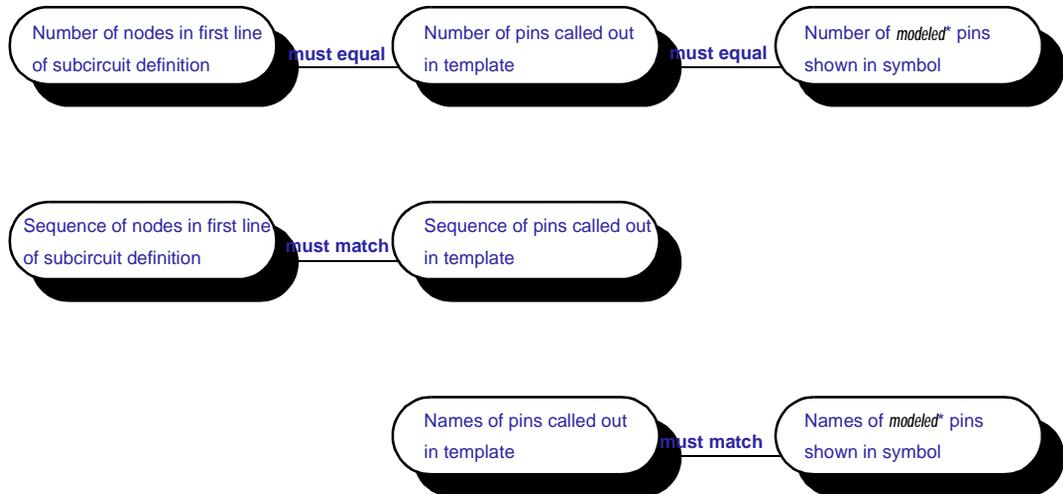
If the correspondence between pin names and nodes is as follows:

This node name...	Corresponds to this pin name...
10	IN+
3	IN-
27	OUT+
2	OUT-

then the template would look like this:

```
X^@REFDES %IN+ %IN- %OUT+ %OUT- @MODEL
```

The *rules of agreement* are outlined in Figure 5-1.



* Unmodeled pins may appear on a symbol (like the two voltage offset pins on a 741 opamp symbol). These pins are not netlisted, and do not appear on the template.

Figure 5-1 Rules for Pin Callout in Subcircuit Templates

Analog Behavioral Modeling

6

Chapter Overview

This chapter describes how to use Analog Behavioral Modeling (ABM) feature provided in PSpice A/D. This chapter includes the following sections:

[Overview of Analog Behavioral Modeling on page 6-2](#)

[The abm.slb Symbol Library File on page 6-3](#)

[Placing and Specifying ABM Parts on page 6-4](#)

[ABM Part Templates on page 6-6](#)

[Control System Parts on page 6-7](#)

[PSpice A/D-Equivalent Parts on page 6-28](#)

[Cautions and Recommendations for Simulation and Analysis on page 6-40](#)

[Basic Controlled Sources on page 6-46](#)

Overview of Analog Behavioral Modeling

The Analog Behavioral Modeling (ABM) feature provided in PSpice A/D allows for flexible descriptions of electronic components in terms of a transfer function or lookup table. In other words, a mathematical relationship is used to model a circuit segment so the segment need not be designed component by component.

The symbol library contains several ABM parts that can be classified as either control system parts or as PSpice A/D-equivalent parts. See [Basic Controlled Sources on page 6-46](#) for an introduction to these parts, how to use them, and the distinction between those with general-purpose application and those with special purpose application.

Control system parts are defined with the reference voltage preset to ground so that each controlling input and output are represented by a single pin in the symbol. These are described in [Control System Parts on page 6-7](#).

PSpice A/D-equivalent parts reflect the structure of the PSpice A/D “E” and “G” device types which respond to a differential input and have double-ended output. These are described in [PSpice A/D-Equivalent Parts on page 6-28](#).

The Device Equations option (described in the online *MicroSim PSpice A/D Reference Manual*) can also be used for modeling of this type, but we recommend using the ABM feature wherever possible. With Device Equations, the PSpice A/D source code is actually modified. While this is more flexible and the result executes faster, it is much more difficult to use and prone to error. In addition, any changes made to source code must be reapplied whenever a PSpice A/D update is installed. Parts built using ABM can be used for most cases of interest, are much easier to use, and are unaffected by PSpice A/D updates.

The abm.slb Symbol Library File

The symbol file `abm.slb` contains the ABM components. This file can logically be thought of as consisting of two sections.

The first section contains symbols that can be quickly connected to form “control system” types of circuits. These components have names like SUM, GAIN, LAPLACE, and HIPASS.

The second section contains symbols that are useful for more traditional “controlled source” forms of schematic parts. These PSpice A/D-equivalent symbols have names like EVALUE and GFREQ and are based on extensions to traditional PSpice A/D “E” and “G” device types.

ABM components are implemented using PSpice A/D primitives; there is no corresponding `abm.lib` file. A small number of components generate multi-line netlist entries, but the majority are implemented as single PSpice A/D “E” or “G” device declarations. See [ABM Part Templates on page 6-6](#) for a discussion of TEMPLATE attributes and their role in generating netlist declarations. See [Implementation of PSpice A/D-Equivalent Parts on page 6-29](#) for more on PSpice A/D “E” and “G” syntax.

Placing and Specifying ABM Parts

ABM parts are placed and connected in the same way as other part symbols. Once an ABM symbol is placed, the instance attributes can be edited, effectively customizing the operational behavior of the part. This is equivalent to defining an ABM expression describing how inputs are transformed into outputs. The following sections discuss some of the rules for specifying ABM expressions.

Net Names and Device Names in ABM Expressions

In ABM expressions, it is natural to refer to signals by name. This is also considerably more convenient than having to connect a wire from a pin on an ABM component to a point carrying the voltage of interest.

The name of an interface port does not extend to any connected nets. To refer to a signal originating at an interface port, connect the port to an offpage connector of the desired name.

If you used an expression such as $V(2)$, then the referenced net (2 in this case) is interpreted as the name of a local or global net. A local net is a labeled wire or bus segment in a hierarchical schematic, or a labeled offpage connector. A global net is a labeled wire or bus segment at the top level, or a global connector.

MicroSim Schematics recognizes these constructs in ABM expressions:

```
V(<net name>)  
V(<net name>,<net name>)  
I(<vdevice>)
```

When one of these is recognized, Schematics searches for *<net name>* or *<vdevice>* in the net name space or the device name space, respectively. Names are searched for first at the hierarchical level of the part being netlisted. If not found there, then the set of global names is searched. If the fragment is not found, then a warning is issued but Schematics still outputs the

resulting netlist. When a match is found, the original fragment is replaced by the fully qualified name of the net or device.

For example, suppose we have a hierarchical part U1. Inside the schematic representing U1 we have an ABM expression including the term V(Reference). If “Reference” is the name of a local net, then the fragment written to the netlist will be translated to V(U1_Reference). If “Reference” is the name of a global net, the corresponding netlist fragment will be V(Reference).

Names of voltage sources are treated similarly. For example, an expression including the term I(Vsense) will be output as I(V_U1_Vsense) if the voltage source exists locally, and as I(V_Vsense) if the voltage source exists at the top level.

Forcing the Use of a Global Definition

If a net name exists both at the local hierarchical level and at the top level, the search mechanism used by Schematics will find the local definition. You can override this, and force Schematics to use the global definition, by prefixing the name with a single quote (') character.

For example, suppose there is a net called Reference both inside hierarchical part U1 and at the top level. Then, the ABM fragment V(Reference) will result in V(U1_Reference) in the netlist, while the fragment V('Reference) will produce V(Reference).

ABM Part Templates

For most ABM symbols, a single PSpice A/D “E” or “G” device declaration is output to the netlist per symbol instance. The TEMPLATE attribute in these cases is straightforward. For example the LOG symbol defines an expression variant of the E device with its output being the natural logarithm of the voltage between the input pin and ground:

```
E^@REFDES %out 0 VALUE { LOG(V(%in)) }
```

The fragment E^@REFDES is standard. The “E” specifies a PSpice A/D controlled voltage source (E device); %in and %out are the input and output pins, respectively; VALUE is the keyword specifying the type of ABM device; and the expression inside the curly braces defines the logarithm of the input voltage.

Several ABM symbols produce more than one primitive PSpice A/D device per symbol instance. In this case, the TEMPLATE attribute may be quite complicated. An example is the DIFFER (differentiator) symbol. This is implemented as a capacitor in series with a current sensor together with an E device which outputs a voltage proportional to the current through the capacitor.

The template has several unusual features: it gives rise to three primitives in the PSpice A/D netlist, and it creates a local node for the connection of the capacitor and its current-sensing V device.

For clarity, the template is shown on three lines although the actual template is a single line.

```
C^@REFDES %in $$U^@REFDES 1\n
V^@REFDES $$U^@REFDES 0 0v\n
E^@REFDES %out 0 VALUE { @GAIN *
I(V^@REFDES) }
```

The fragments C^@REFDES, V^@REFDES, and E^@REFDES create a uniquely named capacitor, current sensing V device, and E device, respectively. The fragment \$\$U^@REFDES creates a name suitable for use as a local node. The E device generates an output proportional to the current through the local V device.

Control System Parts

Control system parts have single-pin inputs and outputs. The reference for input and output voltages is analog ground (0). An enhancement to PSpice A/D means these components can be connected together with no need for dummy load or input resistors.

Table 6-1 lists the control system parts, grouped by function. Also listed are characteristic attributes that may be set. In the sections that follow, each part and its attributes are described in more detail.

Table 6-1 *Control System Parts*

Category	Symbol	Description	Attributes
Basic Components	CONST	constant	VALUE
	SUM	adder	
	MULT	multiplier	
	GAIN	gain block	GAIN
	DIFF	subtractor	
Limiters	LIMIT	hard limiter	LO, HI
	GLIMIT	limiter with gain	LO, HI, GAIN
	SOFTLIM	soft (tanh) limiter	LO, HI, GAIN
Chebyshev Filters	LOPASS	lowpass filter	FP, FS, RIPPLE, STOP
	HIPASS	highpass filter	FP, FS, RIPPLE, STOP
	BANDPASS	bandpass filter	F0, F1, F2, F3, RIPPLE, STOP
	BANDREJ	band reject (notch) filter	F0, F1, F2, F3, RIPPLE, STOP
Integrator and Differentiator	INTEG	integrator	GAIN, IC
	DIFFER	differentiator	GAIN
Table Look-Ups	TABLE	lookup table	ROW1...ROW5
	FTABLE	frequency lookup table	ROW1...ROW5

Table 6-1 Control System Parts (continued)

Category	Symbol	Description	Attributes
Laplace Transform	LAPLACE	Laplace expression	NUM, DENOM
Math Functions (where 'x' is the input)	ABS	$ x $	
	SQRT	$x^{1/2}$	
	PWR	$ x ^{\text{EXP}}$	EXP
	PWRS	x^{EXP}	EXP
	LOG	$\ln(x)$	
	LOG10	$\log(x)$	
	EXP	e^x	
	SIN	$\sin(x)$	
	COS	$\cos(x)$	
	TAN	$\tan(x)$	
	ATAN	$\tan^{-1}(x)$	
	ARCTAN	$\tan^{-1}(x)$	
Expression Functions	ABM	no inputs, V out	EXP1...EXP4
	ABM1	1 input, V out	EXP1...EXP4
	ABM2	2 inputs, V out	EXP1...EXP4
	ABM3	3 inputs, V out	EXP1...EXP4
	ABM/I	no input, I out	EXP1...EXP4
	ABM1/I	1 input, I out	EXP1...EXP4
	ABM2/I	2 inputs, I out	EXP1...EXP4
	ABM3/I	3 inputs, I out	EXP1...EXP4

Basic Components

The basic components provide fundamental functions and in many cases, do not require specifying attribute values. These parts are described below.

CONST

VALUE constant value

The CONST part outputs the voltage specified by the VALUE attribute. This part provides no inputs and one output.

SUM

The SUM part evaluates the voltages of the two input sources, adds the two inputs together, then outputs the sum. This part provides two inputs and one output.

MULT

The MULT part evaluates the voltages of the two input sources, multiplies the two together, then outputs the product. This part provides two inputs and one output.

GAIN

GAIN constant gain value

The GAIN part multiplies the input by the constant specified by the GAIN attribute, then outputs the result. This part provides one input and one output.

DIFF

The DIFF part evaluates the voltage difference between two inputs, then outputs the result. This part provides two inputs and one output.

Limiters

The Limiters can be used to restrict an output to values between a set of specified ranges. These parts are described below.

LIMIT

HI	upper limit value
LO	lower limit value

The LIMIT part constrains the output voltage to a value between an upper limit (set with the HI attribute) and a lower limit (set with the LO attribute). This part takes one input and provides one output.

GLIMIT

HI	upper limit value
LO	lower limit value
GAIN	constant gain value

The GLIMIT part functions as a one-line opamp. The gain is applied to the input voltage, then the output is constrained to the limits set by the LO and HI attributes. This part takes one input and provides one output.

SOFTLIMIT

HI	upper limit value
LO	lower limit value
GAIN	constant gain value
A, B, V, TANH	internal variables used to define the limiting function

The SOFTLIMIT part provides a limiting function much like the LIMIT device, except that it uses a continuous curve limiting function, rather than a discontinuous limiting function. This part takes one input and provides one output.

Chebyshev Filters

The Chebyshev filters allow filtering of the signal based on a set of frequency characteristics. The output of a Chebyshev filter depends upon the analysis being done.

For DC and bias point, the output is simply the DC response of the filter. For AC analysis, the output for each frequency is the filter response at that frequency. For transient analysis, the output is then the convolution of the past values of the input with the impulse response of the filter. These rules follow the standard method of using Fourier transforms.

Note *PSpice A/D computes the impulse response of each Chebyshev filter used in a transient analysis during circuit read-in. This may require considerable computing time. A message is displayed on your screen indicating that the computation is in progress.*

Note *To obtain a listing of the filter Laplace coefficients for each stage, select Setup from the Analysis menu, click on Options, and enable LIST in the Options dialog box.*

Each of the Chebyshev filter parts is described in the following pages.

LOPASS

FS	stop band frequency
FP	pass band frequency
RIPPLE	pass band ripple in dB
STOP	stop band attenuation in dB

The LOPASS part is characterized by two cutoff frequencies that delineate the boundaries of the filter pass band and stop band. The attenuation values, RIPPLE and STOP, define the maximum allowable attenuation in the pass band, and the minimum required attenuation in the stop band, respectively. The LOPASS part provides one input and one output.

Figure 6-1 shows an example of a LOPASS filter device. The filter provides a pass band cutoff of 800 Hz and a stop band cutoff of 1.2 kHz. The pass band ripple is 0.1 dB and the

MicroSim recommends looking at one or more of the references cited in [Frequency-Domain Device Models on page 6-35](#), as well as some of the following references on analog filter design:

- 1 Ghavsi, M.S. & Laker, K.R., *Modern Filter Design*, Prentice-Hall, 1981.
- 2 Gregorian, R. & Temes, G., *Analog MOS Integrated Circuits*, Wiley-Interscience, 1986.
- 3 Johnson, David E., *Introduction to Filter Theory*, Prentice-Hall, 1976.
- 4 Lindquist, Claude S., *Active Network Design with Signal Filtering Applications*, Steward & Sons, 1977.
- 5 Stephenson, F.W. (ed), *RC Active Filter Design Handbook*, Wiley, 1985.
- 6 Van Valkenburg, M.E., *Analog Filter Design*, Holt, Rinehart & Winston, 1982.

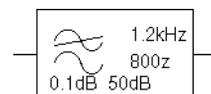


Figure 6-1 LOPASS Filter Example

minimum stop band attenuation is 50 dB. Assuming that the input to the filter is the voltage at net 10 and output is a voltage between nets 5 and 0, this will produce a PSpice A/D netlist declaration like this:

```
ELOWPASS 5 0 CHEBYSHEV {V(10)} = LP 800 1.2K .1dB 50dB
```

HIPASS

FS	stop band frequency
FP	pass band frequency
RIPPLE	pass band ripple in dB
STOP	stop band attenuation in dB

The HIPASS part is characterized by two cutoff frequencies that delineate the boundaries of the filter pass band and stop band. The attenuation values, RIPPLE and STOP, define the maximum allowable attenuation in the pass band, and the minimum required attenuation in the stop band, respectively. The HIPASS part provides one input and one output.

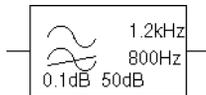


Figure 6-2 HIPASS Filter Part

Figure 6-2 shows an example of a HIPASS filter device. This is a high pass filter with the pass band above 1.2 kHz and the stop band below 800 Hz. Again, the pass band ripple is 0.1 dB and the minimum stop band attenuation is 50 dB. This will produce a PSpice A/D netlist declaration like this:

```
EHIGHPASS 5 0 CHEBYSHEV {V(10)} = HP 1.2K 800 .1dB 50dB
```

BANDPASS

RIPPLE	pass band ripple in dB
STOP	stop band attenuation in dB
F0, F1,	cutoff frequencies
F2, F3	

The BANDPASS part is characterized by four cutoff frequencies. The attenuation values, RIPPLE and STOP, define the maximum allowable attenuation in the pass band, and the minimum required attenuation in the stop band, respectively. The BANDPASS part provides one input and one output.

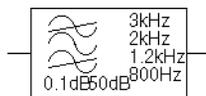


Figure 6-3 BANDPASS Filter Part Example

Figure 6-3 shows an example of a BANDPASS filter device. This is a band pass filter with the pass band between 1.2 kHz and 2 kHz, and stop bands below 800 Hz and above 3 kHz. The pass

band ripple is 0.1 dB and the minimum stop band attenuation is 50 dB. This will produce a PSpice A/D netlist declaration like this:

```
EBANDPASS 5 0 CHEBYSHEV
+ {V(10)} = BP 800 1.2K 2K 3K .1dB 50dB
```

BANDREJ

RIPPLE is the pass band ripple in dB
 STOP is the stop band attenuation in dB
 F0, F1, are the cutoff frequencies
 F2, F3

The BANDREJ part is characterized by four cutoff frequencies. The attenuation values, RIPPLE and STOP, define the maximum allowable attenuation in the pass band, and the minimum required attenuation in the stop band, respectively. The BANDREJ part provides one input and one output.

Figure 6-4 shows an example of a BANDREJ filter device. This is a band reject (or “notch”) filter with the stop band between 1.2 kHz and 2 kHz, and pass bands below 800 Hz and above 3 kHz. The pass band ripple is 0.1 dB and the minimum stop band attenuation is 50 dB. This will produce a PSpice A/D netlist declaration like this:

```
ENOTCH 5 0 CHEBYSHEV {V(10)} = BR 1.2K 800 3K 2K .1dB 50dB
```

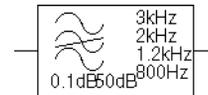


Figure 6-4 BANDREJ Filter

Integrator and Differentiator

The integrator and differentiator parts are described below.

INTEG

IC	initial condition of the integrator output
GAIN	gain value

The INTEG part implements a simple integrator. A current source/capacitor implementation is used to provide support for setting the initial condition.

DIFFER

GAIN	gain value
------	------------

The DIFFER part implements a simple differentiator. A voltage source/capacitor implementation is used. The DIFFER part provides one input and one output.

Table Look-Up Parts

TABLE and FTABLE parts provide a lookup table that is used to correlate an input and an output based on a set of data points. These parts are described below and on the following pages.

TABLE

If more than five values are required, the symbol can be customized through the symbol editor. Insert additional row variables into the template using the same form as the first five, and add ROW n attributes as needed to the list of attributes.

ROW n	is an (input, output) pair; by default, up to five triplets are allowed where $n=1, 2, 3, 4,$ or 5
---------	--

The TABLE part allows the response to be defined by a table of one to five values. Each row contains an input and a corresponding output value. Linear interpolation is performed between entries.

For values outside the table's range, the device's output is a constant with a value equal to the entry with the smallest (or largest) input. This characteristic can be used to impose an upper and lower limit on the output. The TABLE part provides one input and one output.

FTABLE

ROW n	either an (input frequency, magnitude, phase) triplet, or an (input frequency, real part, imaginary part) triplet describing a complex value; by default, up to five triplets are allowed where $n=1, 2, 3, 4, \text{ or } 5$	If more than five values are required, the symbol can be customized through the symbol editor. Insert additional row variables into the template using the same form as the first five, and add ROW n attributes as needed to the list of attributes.
DELAY	group delay increment; defaults to 0 if left blank	
R_I	table type; if left blank, the frequency table is interpreted in the (input frequency, magnitude, phase) format; if defined with any value (such as YES), the table is interpreted in the (input frequency, real part, imaginary part) format	
MAGUNITS	units for magnitude where the value can be DB (decibels) or MAG (raw magnitude); defaults to DB if left blank	
PHASEUNITS	units for phase where the value can be DEG (degrees) or RAD (radians); defaults to DEG if left blank	

The FTABLE part is described by a table of frequency responses in either the magnitude/phase domain (R_I=) or complex number domain (R_I=YES). The entire table is read in and converted to magnitude in dB and phase in degrees.

Interpolation is performed between entries. Magnitude is interpolated logarithmically; phase is interpolated linearly. For frequencies outside the table's range, 0 (zero) magnitude is used. This characteristic can be used to impose an upper and lower limit on the output.

The DELAY attribute increases the group delay of the frequency table by the specified amount. The delay term is particularly useful when a frequency table device generates a non-causality warning message during a transient analysis. The warning message issues a delay value that can be assigned to the symbol's DELAY attribute for subsequent runs, without otherwise altering the table.

The output of the part depends on the analysis being done. For DC and bias point, the output is the zero frequency magnitude times the input voltage. For AC analysis, the input voltage is linearized around the bias point (similar to EVALUE and

GVALUE parts, [Modeling Mathematical or Instantaneous Relationships on page 6-30](#)). The output for each frequency is then the input times the gain, times the value of the table at that frequency.

For transient analysis, the voltage is evaluated at each time point. The output is then the convolution of the past values with the impulse response of the frequency response. These rules follow the standard method of using Fourier transforms. We recommend looking at one or more of the references cited in [Frequency-Domain Device Models on page 6-35](#) for more information.

Note *The table's frequencies must be in order from lowest to highest. The TABLE part provides one input and one output.*

Example

A device, ELOFILT, is used as a frequency filter. The input to the frequency response is the voltage at net 10. The output is a voltage across nets 5 and 0. The table describes a low pass filter with a response of 1 (0 dB) for frequencies below 5 kilohertz and a response of 0.001 (-60 dB) for frequencies above 6 kilohertz. The phase lags linearly with frequency. This is the same as a constant time delay. The delay is necessary so that the impulse response is causal. That is, so that the impulse response does not have any significant components before time zero. The FTABLE part in Figure 6-5 could be used.

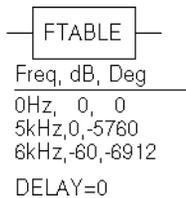


Figure 6-5 FTABLE Part

This part is characterized by the following attributes:

```

ROW1 = 0Hz      0      0
ROW2 = 5kHz    0      -5760
ROW3 = 6kHz    -60    -6912
DELAY =
R_I =
MAGUNITS =
PHASEUNITS =

```

Since R_I, MAGUNITS, and PHASEUNITS are undefined, each table entry is interpreted as containing frequency, magnitude value in dB, and phase values in degrees. Delay defaults to 0.

This produces a PSpice A/D netlist declaration like this:

```

ELOFILT 5 0 FREQ {V(10)} = (0,0,0) (5kHz,0,-5760)
+ (6kHz,-60,-6912)

```

Since constant group delay is calculated from the values for a given table entry as:

$$\text{group delay} = \text{phase} / 360 / \text{frequency}$$

An equivalent FTABLE instance could be defined using the DELAY attribute. For this example, the group delay is 3.2 msec ($6912 / 360 / 6k = 5760 / 360 / 6k = 3.2m$). Equivalent attribute assignments are:

```

ROW1 = 0Hz      0      0
ROW2 = 5kHz    0      0
ROW3 = 6kHz    -60    0
DELAY = 3.2ms
R_I =
MAGUNITS =
PHASEUNITS =

```

This produces a PSpice A/D netlist declaration like this:

```

ELOFILT 5 0 FREQ {V(10)} = (0,0,0) (5kHz,0,0) (6kHz,-60,0)
+ DELAY=3.2ms

```

Laplace Transform Part

The LAPLACE part specifies a Laplace transform which is used to determine an output for each input value.

LAPLACE

NUM	numerator of the Laplace expression
DENOM	denominator of the Laplace expression

The LAPLACE part uses a Laplace transform description. The input to the transform is a voltage. The numerator and denominator of the Laplace transform function are specified as attributes for the symbol.

Note *Voltages, currents, and TIME may not appear in a Laplace transform specification.*

The output of the part depends on the type of analysis being done. For DC and bias point, the output is the zero frequency gain times the value of the input. The zero frequency gain is the value of the Laplace transform with $s=0$. For AC analysis, the output is then the input times the gain times the value of the Laplace transform. The value of the Laplace transform at a frequency is calculated by substituting $j\cdot\omega$ for s , where ω is 2π -frequency. For transient analysis, the output is the convolution of the input waveform with the impulse response of the transform. These rules follow the standard method of using Laplace transforms.

Example 1

The input to the Laplace transform is the voltage at net 10. The output is a voltage and is applied between nets 5 and 0. For DC, the output is simply equal to the input, since the gain at $s = 0$ is 1. The transform, $1/(1+.001\cdot s)$, describes a simple, lossy integrator with a time constant of 1 millisecond. This can be implemented with an RC pair that has a time constant of 1 millisecond.

For AC analysis, the gain is found by substituting $j\cdot\omega$ for s . This gives a flat response out to a corner frequency of $1000/(2\pi) = 159$ hertz and a roll-off of 6 dB per octave after 159 Hz. There is also a phase shift centered around 159 Hz. In other words, the

gain has both a real and an imaginary component. For transient analysis, the output is the convolution of the input waveform with the impulse response of $1/(1+.001 \cdot s)$. The impulse response is a decaying exponential with a time constant of 1 millisecond. This means that the output is the “lossy integral” of the input, where the loss has a time constant of 1 millisecond. The LAPLACE part shown in Figure 6-6 could be used for this purpose.

The transfer function is the Laplace transform ($1/[1+.001*s]$). This LAPLACE part is characterized by the following attributes:

```
NUM = 1
DENOM = 1 + .001*s
```

The gain and phase characteristics are shown in Figure 6-7.

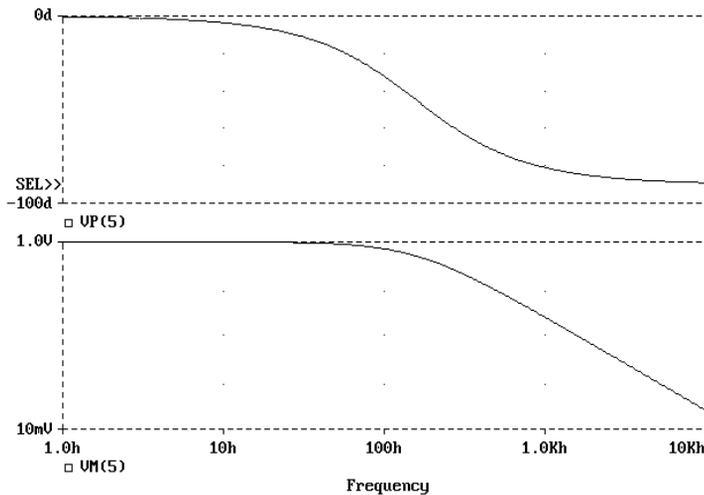


Figure 6-7 Lossy Integrator Example: Viewing Gain and Phase Characteristics with Probe

This produces a PSpice A/D netlist declaration like this:

```
ERC 5 0 LAPLACE {V(10)} = {1/(1+.001*s)}
```

Example 2

The input is V(10). The output is a current applied between nets 5 and 0. The Laplace transform describes a lossy transmission line. R, L, and C are the resistance, inductance, and capacitance of the line per unit length.

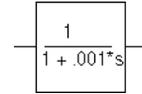


Figure 6-6 LAPLACE Part

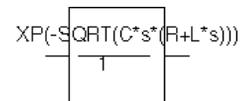


Figure 6-8 LAPLACE Part

If R is small, the characteristic impedance of such a line is $Z = ((R + j \cdot \omega \cdot L) / (j \cdot \omega \cdot C))^{1/2}$, the delay per unit length is $(L/C)^{1/2}$, and the loss in dB per unit length is $23 \cdot R/Z$. This could be represented by the device in Figure 6-8.

The parameters R , L , and C can be defined in a `.PARAM` statement contained in a model file. (Refer to the online *MicroSim PSpice A/D Reference Manual* for more information about using `.PARAM` statements.) More useful, however, is for R , L , and C to be arguments passed into a subcircuit. This part has the following characteristics:

```
NUM = EXP(-SQRT(C*s*(R+L*s)))
DENOM = 1
```

This produces a PSpice A/D netlist declaration like this:

```
GLOSSY 5 0 LAPLACE {V(10)} = {exp(-sqrt(C*s*(R + L*s)))}
```

The Laplace transform parts are, however, an inefficient way, in both computer time and memory, to implement a delay. For ideal delays we recommend using the transmission line part instead.

Math Functions

The ABM math function parts are shown in [Table 6-2](#). For each device, the corresponding template is shown, indicating the order in which the inputs are processed, if applicable.

Table 6-2 *ABM Math Function Parts*

For this device...	Output is the...
ABS	absolute value of the input
SQRT	square root of the input
PWR	result of raising the absolute value of the input to the power specified by EXP
PWRS	result of raising the (signed) input value to the power specified by EXP
LOG	LOG of the input
LOG10	LOG ₁₀ of the input
EXP	result of e raised to the power specified by the input value (e^x where x is the input)
SIN	<i>sin</i> of the input (where the input is in radians)
COS	<i>cos</i> of the input (where the input is in radians)
TAN	<i>tan</i> of the input (where the input is in radians)
ATAN, ARCTAN	\tan^{-1} of the input (where the output is in radians)

Math function parts are based on the PSpice A/D “E” device type. Each provides one or more inputs, and a mathematical function which is applied to the input. The result is output on the output net.

ABM Expression Parts

The expression parts are shown in [Table 6-3](#). These parts can be customized to perform a variety of functions depending on your

requirements. Each of these parts has a set of four expression *building block* attributes of the form:

$$\text{EXP}_n$$

where $n = 1, 2, 3,$ or 4 .

During netlist generation, the complete expression is formed by concatenating the building block expressions in numeric order, thus defining the transfer function. Hence, the first expression fragment should be assigned to the EXP1 attribute, the second fragment to EXP2, and so on.

Expression attributes can be defined using a combination of arithmetic operators and input designators. You may use any of the standard PSpice A/D arithmetic operators (see [Table 3-1 on page 3-15](#)) within an expression statement. You may also use the EXP n attributes as variables to represent nets or constants.

Table 6-3 *ABM Expression Parts*

Device	Inputs	Output
ABM	none	V
ABM1	1	V
ABM2	2	V
ABM3	3	V
ABM/I	none	I
ABM1/I	1	I
ABM2/I	2	I
ABM3/I	3	I

The following examples illustrate a variety of ABM expression part applications.

Example 1

Suppose you want to set an output voltage on net 4 to 5 volts times the square root of the voltage between nets 3 and 2. You could use an ABM2 part (which takes two inputs and provides a voltage output) to define a part like the one shown in Figure 6-9.

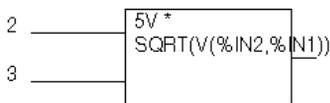


Figure 6-9 *ABM Expression Part Example 1*

In this example of an ABM device, the output voltage is set to 5 volts times the square root of the voltage between net 3 and net 2. The attribute settings for this part are as follows:

```
EXP1 = 5V *
EXP2 = SQRT(V(%IN2,%IN1))
```

This will produce a PSpice A/D netlist declaration like this:

```
ESQROOT 4 0 VALUE = {5V*SQRT(V(3,2))}
```

Example 2

GPSK is an oscillator for a PSK (Phase Shift Keyed) modulator. Current is pumped from net 11 through the source to net 6. Its value is a sine wave with an amplitude of 15 mA and a frequency of 10 kHz. The voltage at net 3 can shift the phase of GPSK by 1 radian/volt. Note the use of the TIME parameter in the EXP2 expression. This is the PSpice A/D internal sweep variable used in transient analyses. For any analysis other than transient, TIME = 0. This could be represented with an ABM1/I part (single input, current output) like the one shown in Figure 6-10.

This part is characterized by the following attributes:

```
EXP1 = 15ma * SIN(
EXP2 = 6.28*10kHz*TIME
EXP3 = + V(%IN))
```

This produces a PSpice A/D netlist declaration like this:

```
GPSK 11 6 VALUE = {15MA*SIN(6.28*10kHz*TIME+V(3))}
```

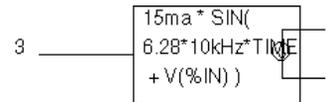


Figure 6-10 ABM Expression Part Example 2

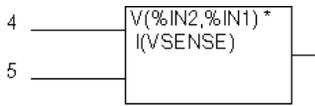


Figure 6-11 ABM Expression Part Example 3

Example 3

A device, EPWR, computes the instantaneous power by multiplying the voltage across nets 5 and 4 by the current through VSENSE. Sources are controlled by expressions which may contain voltages or currents or both. The ABM2 part (two inputs, current output) in Figure 6-11 could represent this.

This part is characterized by the following attributes:

```
EXP1 = V(%IN2,%IN1) *
EXP2 = I(VSENSE)
```

This produces a PSpice A/D netlist declaration like this:

```
EPWR 3 0 VALUE = {V(5,4)*I(VSENSE)}
```

Example 4

The output of a component, GRATIO, is a current whose value (in amps) is equal to the ratio of the voltages at nets 13 and 2. If $V(2) = 0$, the output depends upon $V(13)$ as follows:

```
if V(13) = 0, output = 0
if V(13) > 0, output = MAXREAL
if V(13) < 0, output = -MAXREAL
```

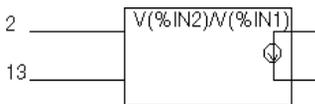


Figure 6-12 ABM Expression Part Example 4

where MAXREAL is a PSpice A/D internal constant representing a very large number (on the order of $1e30$). In general, the result of evaluating an expression is limited to MAXREAL. This is modeled with an ABM2/I (two input, current output) part like this one in Figure 6-12.

This part is characterized by the following attributes:

```
EXP1 = V(%IN2)/V(%IN1)
```

Note that output of GRATIO can be used as part of the controlling function. This produces a PSpice A/D netlist declaration like this:

```
GRATIO 2 3 VALUE = {V(13)/V(2)}
```

Note *Letting a current approach $\pm 1e30$ will almost certainly cause convergence problems. To avoid this, use the limit function on the ratio to keep the current within reasonable limits.*

An Instantaneous Device Example: Modeling a Triode

This section provides an example of using various ABM parts to model a triode vacuum tube. The schematic of the triode subcircuit is shown in Figure 6-13.

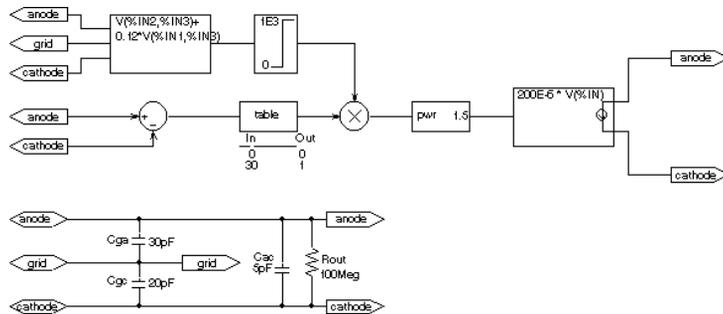


Figure 6-13 *Triode Circuit*

Assumptions: In its main operating region, the triode's current is proportional to the $3/2$ power of a linear combination of the grid and anode voltages:

$$i_{\text{anode}} = k_0 * (v_g + k_1 * v_a)^{1.5}$$

For a typical triode, $k_0 = 200\text{e-}6$ and $k_1 = 0.12$.

Looking at the upper left-hand portion of the schematic, notice the a general-purpose ABM part used to take the input voltages from anode, grid, and cathode. Assume the following associations:

- $V(\text{anode})$ is associated with $V(\%IN1)$
- $V(\text{grid})$ is associated with $V(\%IN2)$
- $V(\text{cathode})$ is associated with $V(\%IN3)$

The expression attribute EXP1 then represents $V(\text{grid}, \text{cathode})$ and the expression attribute EXP2 represents $0.12[V(\text{anode}, \text{cathode})]$. When the template substitution is performed, the resulting VALUE is equivalent to the following:

$$V = V(\text{grid}, \text{cathode}) + 0.12 * V(\text{anode}, \text{cathode})$$

The part would be defined with the following characteristics:

```
EXP1 = V(%IN2,%IN3)+  
EXP2 = 0.12*V(%IN1,%IN3)
```

This works for the main operating region but does not model the case in which the current stays 0 when combined grid and anode voltages go negative. We can accommodate that situation as follows by adding the LIMIT part with the following characteristics:

```
HI = 1E3  
LO = 0
```

This part instance, LIMIT1, converts all negative values of $v_g + 12*v_a$ to 0 and leaves all positive values (up to 1 kV) alone. For a more realistic model, we could have used TABLE to correctly model how the tube turns off at 0 or at small negative grid voltages.

We also need to make sure that the current becomes zero when the anode alone goes negative. To do this, we can use a DIFF device, (immediately below the ABM3 device) to monitor the difference between V(anode) and V(cathode), and output the difference to the TABLE part. The table translates all values at or below zero to zero, and all values greater than or equal to 30 to one. All values between 0 and 30 are linearly interpolated. The attributes for the TABLE part are as follows:

```
ROW1 = 00  
ROW2 = 301
```

The TABLE part is a simple one, and ensures that only a zero value is output to the multiplier for negative anode voltages.

The output from the TABLE part and the LIMIT part are combined at the MULT multiplier part. The output of the MULT part is the product of the two input voltages. This value is then raised to the 3/2 or 1.5 power using the PWR part. The exponential attribute of the PWR part is defined as follows:

```
EXP = 1.5
```

The last major component is an ABM expression component to take an input voltage and convert it into a current. The relevant ABM1/I part attribute looks like this:

```
EXP1 = 200E-6 * V(%IN)
```

A final step in the model is to add device parasitics. For example, a resistor can be used to give a finite output

impedance. Capacitances between the grid, cathode, and anode are also needed. The lower part of the schematic in Figure 6-13 shows a possible method for incorporating these effects. To complete the example, one could add a circuit which produces the family of I-V curves (shown in Figure 6-14).

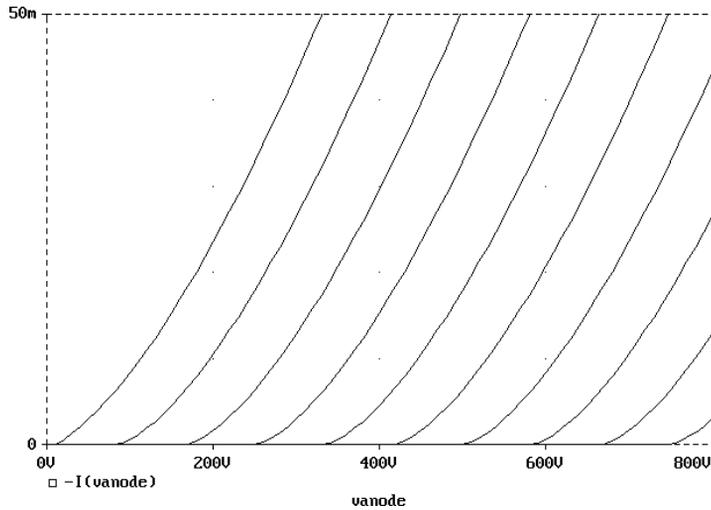


Figure 6-14 *Triode Subcircuit Producing a Family of I-V Curves*

PSpice A/D-Equivalent Parts

PSpice A/D-equivalent parts respond to a differential input and have double-ended output. These parts reflect the structure of PSpice A/D “E” and “G” devices, thus having two pins for each controlling input and the output in the symbol. [Table 6-4](#) summarizes the PSpice A/D-equivalent parts available in the symbol library.

Table 6-4 *PSpice A/D-Equivalent Parts*

Category	Symbol	Description	Attributes
Mathematical Expression	EVALUE	general purpose	EXPR
	GVALUE		
	ESUM	special purpose	(none)
	GSUM		
	EMULT		
Table Look-Up	GMULT		
	ETABLE	general purpose	EXPR
GTABLE	TABLE		
Frequency Table Look-Up	EFREQ	general purpose	EXPR
	GFREQ		TABLE
Laplace Transform	ELAPLACE	general purpose	EXPR
	GLAPLACE		XFORM

There are no equivalent “F” or “H” part types in the symbol library since PSpice A/D “F” and “H” devices do not support the ABM extensions.

PSpice A/D-equivalent ABM parts can be classified as either “E” part types or “G” part types. The E part type provides a voltage output, and the G part type provides a current output. The part’s transfer function can contain any mixture of voltages and currents as inputs. Hence, there is no longer a division between voltage-controlled and current-controlled parts. Rather the part type is dictated only by the output requirements. If a

voltage output is required, use an E part type. If a current output is necessary, use a G part type.

Each E or G part type in the `abm.s1b` symbol file is defined by a template that provides the specifics of the transfer function. Other attributes in the model definition can be edited to customize the transfer function. By default, the template cannot be modified directly using Attributes on the Edit menu in Schematics. Rather, the values for other attributes (such as the expressions used in the template) are usually edited, then these values are substituted into the template. However, the symbol editor can be used to modify the template or designate the template as modifiable from within Schematics. This way, custom symbols can be created for special-purpose application.

Implementation of PSpice A/D-Equivalent Parts

Although you generally use Schematics to place and specify PSpice A/D-equivalent ABM parts, it is useful to know the PSpice A/D command syntax for “E” and “G” devices. This is especially true when creating custom ABM symbols since symbol templates must adhere to PSpice A/D syntax.

The general forms for PSpice A/D “E” and “G” extensions are:

```
E <name> <connecting nodes> <ABM keyword> <ABM function>
G <name> <connecting nodes> <ABM keyword> <ABM function>
```

where

<i><name></i>	is the device name appended to the E or G device type character
<i><connecting nodes></i>	specifies the $\langle (+ \text{ node name}, - \text{ node name}) \rangle$ pair between which the device is connected

<i><ABM keyword></i>	specifies the form of the transfer function to be used, as one of:
VALUE	arithmetic expression
TABLE	lookup table
LAPLACE	Laplace transform
FREQ	frequency response table
CHEBYSHEV	Chebyshev filter characteristics
<i><ABM function></i>	specifies the transfer function as a formula or lookup table as required by the specified <i><ABM keyword></i>

Refer to the online *MicroSim PSpice A/D Reference Manual* for detailed information.

Modeling Mathematical or Instantaneous Relationships

The instantaneous models (using VALUE and TABLE extensions to PSpice A/D “E” and “G” devices in the symbol templates) enforce a direct response to the input at each moment in time. For example, the output might be equal to the square root of the input at every point in time. Such a device has no memory, or, a flat frequency response. These techniques can be used to model both linear and nonlinear responses.

Note *For AC analysis, a nonlinear device is first linearized around the bias point, and then the linear equivalent is used.*

EVALUE and GVALUE parts

The EVALUE and GVALUE parts allow an instantaneous transfer function to be written as a mathematical expression in standard notation. These parts take the input signal, perform the function specified by the EXPR attribute on the signal, and output the result on the output pins.

In controlled sources, EXPR may contain constants and parameters as well as voltages, currents, or time. Voltages may

be either the voltage at a net, such as V(5), or the voltage across two nets, such as V(4,5). Currents must be the current through a voltage source (V device), for example, I(VSENSE). Voltage sources with a value of 0 are handy for sensing current for use in these expressions.

Functions may be used in expressions, along with arithmetic operators (+, -, *, and /) and parentheses. Available built-in functions are summarized in [Table 3-2 on page 3-16](#).

The EVALUE and GVALUE symbols are defined, in part, by the following attributes (default values are shown):

```
EVALUE
    EXPR      V(%IN+, %IN-)

GVALUE
    EXPR      V(%IN+, %IN-)
```

Sources are controlled by expressions which may contain voltages, currents, or both. The following examples illustrate customized EVALUE and GVALUE parts.

Example 1

In the example of an EVALUE device shown in Figure 6-15, the output voltage is set to 5 volts times the square root of the voltage between pins %IN+ and %IN-.

The attribute settings for this device are as follows:

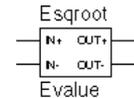
```
EXPR = 5v * SQRT(V(%IN+, %IN-))
```

Example 2

Consider the device in Figure 6-16. This device could be used as an oscillator for a PSK (Phase Shift Keyed) modulator.

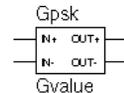
A current through a source is a sine wave with an amplitude of 15 mA and a frequency of 10 kHz. The voltage at the input pin can shift the phase by 1 radian/volt. Note the use of the TIME parameter in this expression. This is the PSpice A/D internal sweep variable used in transient analyses. For any analysis other than transient, TIME = 0. The relevant attribute settings for this device are shown below:

```
EXPR = 15ma * SIN(6.28 * 10kHz * TIME + V(%IN+, %IN-))
```



5v * SQRT(V(%IN+, %IN-))

Figure 6-15 EVALUE Part Example



15ma * SIN(6.28 * 10kHz * TIME + V(%IN+, %IN-))

Figure 6-16 GVALUE Part Example

EMULT, GMULT, ESUM, and GSUM

The EMULT and GMULT parts provide output which is based on the product of two input sources. The ESUM and GSUM parts provide output which is based on the sum of two input sources. The complete transfer function may also include other mathematical expressions.

Example 1

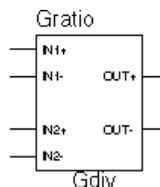
Consider the device in Figure 6-17. This device computes the instantaneous power by multiplying the voltage across pins %IN+ and %IN- by the current through VSENSE. This device's behavior is built-in to the TEMPLATE attribute as follows (appears on one line):

```
TEMPLATE=E^@REFDES %OUT+ %OUT- VALUE {V(%IN1+,%IN1-)  
*V(%IN2+,%IN2-)}
```

You can use the symbol editor to change the characteristics of the template to accommodate additional mathematical functions, or to change the nature of the transfer function itself. For example, you may want to create a voltage divider, rather than a multiplier. This is illustrated in the following example.

Example 2

Consider the device in Figure 6-18.



```
G^@REFDES %OUT+ %OUT- VALUE {V(%IN1+,%IN1-)/V(%IN2+,%IN2-)}
```

Figure 6-18 GMULT Part Example

With this device, the output is a current is equal to the ratio of the voltages at input pins 1 and input pins 2. If $V(\%IN2+, \%IN2-) = 0$, the output depends upon $V(\%IN1+, \%IN1-)$ as follows:

```
if V(%IN1+, %IN1-) = 0, output = 0  
if V(%IN1+, %IN1-) > 0, output = MAXREAL  
if V(%IN1+, %IN1-) < 0, output = -MAXREAL
```

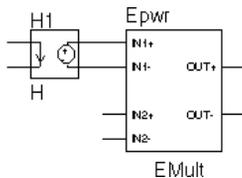


Figure 6-17 EMULT Part

where MAXREAL is a PSpice A/D internal constant representing a very large number (on the order of $1e30$). In general, the result of evaluating an expression is limited to MAXREAL. Note that the output of the symbol can also be used as part of the controlling function.

To create this device, you would first make a new symbol, GDIV, based on the GMULT part. Edit the GDIV template to divide the two input values rather than multiply them.

Lookup Tables (ETABLE and GTABLE)

The ETABLE and GTABLE parts use a transfer function described by a table. These device models are well suited for use with measured data.

The ETABLE and GTABLE symbols are defined in part by the following attributes (default values are shown):

ETABLE

TABLE	(-15, -15), (15,15)
EXPR	V(%IN+, %IN-)

GTABLE

TABLE	(-15, -15), (15,15)
EXPR	V(%IN+, %IN-)

First, EXPR is evaluated, and that value is used to look up an entry in the table. EXPR is a function of the input (current or voltage) and follows the same rules as for VALUE expressions.

The table consists of pairs of values, the first of which is an input, and the second of which is the corresponding output. Linear interpolation is performed between entries. For values of EXPR outside the table's range, the device's output is a constant with a value equal to the entry with the smallest (or largest) input. This characteristic can be used to impose an upper and lower limit on the output.

An example of a table declaration (using the TABLE attribute) would be the following:

TABLE =

+ (0, 0) (.02, 2.690E-03) (.04, 4.102E-03) (.06, 4.621E-03)
+ (.08, 4.460E-03) (.10, 3.860E-03) (.12, 3.079E-03) (.14,
+ 2.327E-03)
+ (.16, 1.726E-03) (.18, 1.308E-03) (.20, 1.042E-03) (.22,
+ 8.734E-04)
+ (.24, 7.544E-04) (.26, 6.566E-04) (.28, 5.718E-04) (.30,
+ 5.013E-04)
+ (.32, 4.464E-04) (.34, 4.053E-04) (.36, 3.781E-04) (.38,
+ 3.744E-04)
+ (.40, 4.127E-04) (.42, 5.053E-04) (.44, 6.380E-04) (.46,
+ 7.935E-04)
+ (.48, 1.139E-03) (.50, 2.605E-03) (.52, 8.259E-03) (.54,
+ 2.609E-02)
+ (.56, 7.418E-02) (.58, 1.895E-01) (.60, 4.426E-01)

Frequency-Domain Device Models

Frequency-domain models (ELAPLACE, GLAPLACE, EFREQ, and GFREQ) are characterized by output that depends on the current input as well as the input history. The relationship is therefore non-instantaneous. For example, the output may be equal to the integral of the input over time. In other words, the response depends upon frequency.

During AC analysis, the frequency response determines the complex gain at each frequency. During DC analysis and bias point calculation, the gain is the zero-frequency response. During transient analysis, the output of the device is the convolution of the input and the impulse response of the device.

Laplace Transforms (LAPLACE)

The ELAPLACE and GLAPLACE parts allow a transfer function to be described by a Laplace transform function. The ELAPLACE and GLAPLACE symbols are defined, in part, by the following attributes (default values are shown):

ELAPLACE	
EXPR	V(%IN+, %IN-)
XFORM	1/s
GLAPLACE	
EXPR	V(%IN+, %IN-)
XFORM	1/s

The LAPLACE parts use a Laplace transform description. The input to the transform is the value of EXPR, where EXPR follows the same rules as for VALUE expressions (see [EVALUE and GVALUE parts on page 6-30](#)). XFORM is an expression in the Laplace variable, s . It follows the rules for standard expressions as described for VALUE expressions with the addition of the s variable.

The output of the device depends on the type of analysis being done. For DC and bias point, the output is simply the zero

Moving back and forth between the time and frequency-domains can cause surprising results. Often the results are quite different than what one would intuitively expect. For this reason, we strongly recommend familiarity with a reference on Fourier and Laplace transforms. A good one is:

- 1 R. Bracewell, *The Fourier Transform and Its Applications*, McGraw-Hill, Revised Second Edition (1986)

We also recommend familiarity with the use of transforms in analyzing linear systems. Some references on this subject:

- 2 W. H. Chen, *The Analysis of Linear Systems*, McGraw-Hill (1962)
- 3 J. A. Aseltine, *Transform Method in Linear System Analysis*, McGraw-Hill (1958)
- 4 G. R. Cooper and C. D. McGillen, *Methods of Signal and System Analysis*, Holt, Rinehart, and Winston (1967)

Voltages, currents, and TIME cannot appear in a Laplace transform.

frequency gain times the value of EXPR. The zero frequency gain is the value of XFORM with $s = 0$. For AC analysis, EXPR is linearized around the bias point (similar to the VALUE parts). The output is then the input times the gain of EXPR times the value of XFORM. The value of XFORM at a frequency is calculated by substituting $j \cdot \omega$ for s , where ω is 2 π -frequency. For transient analysis, the value of EXPR is evaluated at each time point. The output is then the convolution of the past values of EXPR with the impulse response of XFORM. These rules follow the standard method of using Laplace transforms. We recommend looking at one or more of the references cited in [Frequency-Domain Device Models on page 6-35](#) for more information.

Example

The input to the Laplace transform is the voltage across the input pins, or $V(\%IN+, \%IN-)$. The EXPR attribute may be edited to include constants or functions, as with other parts. The transform, $1/(1+.001 \cdot s)$, describes a simple, lossy integrator with a time constant of 1 millisecond. This can be implemented with an RC pair that has a time constant of 1 millisecond.

Using the symbol editor, you would define the XFORM and EXPR attributes as follows:

```
XFORM = 1/(1+.001*s)
EXPR = V(%IN+, %IN-)
```

The default template remains (appears on one line):

```
TEMPLATE= E^@REFDES %OUT+ %OUT- LAPLACE
{@EXPR} = (@XFORM)
```

After netlist substitution of the template, the resulting transfer function would become:

$$V(\%OUT+, \%OUT-) = \text{LAPLACE } \{V(\%IN+, \%IN-)\} = (1/(1+.001 \cdot s))$$

The output is a voltage and is applied between pins $\%OUT+$ and $\%OUT-$. For DC, the output is simply equal to the input, since the gain at $s = 0$ is 1.

For AC analysis, the gain is found by substituting $j \cdot \omega$ for s . This gives a flat response out to a corner frequency of $1000/(2\pi) = 159$ Hz and a roll-off of 6 dB per octave after 159 Hz. There is also a phase shift centered around 159 Hz. In other words, the

gain has both a real and an imaginary component. The gain and phase characteristic is the same as that shown for the equivalent control system part example using the LAPLACE part (see Figure 6-7 on page 6-19).

For transient analysis, the output is the convolution of the input waveform with the impulse response of $1/(1+.001\cdot s)$. The impulse response is a decaying exponential with a time constant of 1 millisecond. This means that the output is the “lossy integral” of the input, where the loss has a time constant of 1 millisecond.

This will produce a PSpice A/D netlist declaration similar to:

```
ERC 5 0 LAPLACE {V(10)} = {1/(1+.001*s)}
```

Frequency Response Tables (EFREQ and GFREQ)

The EFREQ and GFREQ parts are described by a table of frequency responses in either the magnitude/phase domain or complex number domain. The entire table is read in and converted to magnitude in dB and phase in degrees. Interpolation is performed between entries. Phase is interpolated linearly; magnitude is interpolated logarithmically. For frequencies outside the table’s range, 0 (zero) magnitude is used.

EFREQ and GFREQ attributes are defined as follows:

EXPR	value used for table lookup; defaults to V(%IN+, %IN-) if left blank.
TABLE	series of either (input frequency, magnitude, phase) triplets, or (input frequency, real part, imaginary part) triplets describing a complex value; defaults to (0,0,0) (1Meg,-10,90) if left blank.
DELAY	group delay increment; defaults to 0 if left blank.

R_I	table type; if left blank, the frequency table is interpreted in the (input frequency, magnitude, phase) format; if defined with any value (such as YES), the table is interpreted in the (input frequency, real part, imaginary part) format.
MAGUNITS	units for magnitude where the value can be DB (decibels) or MAG (raw magnitude); defaults to DB if left blank.
PHASEUNITS	units for phase where the value can be DEG (degrees) or RAD (radians); defaults to DEG if left blank.

The DELAY attribute increases the group delay of the frequency table by the specified amount. The delay term is particularly useful when an EFREQ or GFREQ device generates a non-causality warning message during a transient analysis. The warning message issues a delay value that can be assigned to the symbol's DELAY attribute for subsequent runs, without otherwise altering the table.

The output of the device depends on the analysis being done. For DC and bias point, the output is simply the zero frequency magnitude times the value of EXPR. For AC analysis, EXPR is linearized around the bias point (similar to EVALUE and GVALUE parts). The output for each frequency is then the input times the gain of EXPR times the value of the table at that frequency. For transient analysis, the value of EXPR is evaluated at each time point. The output is then the convolution of the past values of EXPR with the impulse response of the frequency response. These rules follow the standard method of using Fourier transforms. We recommend looking at one or more of the references cited in [Frequency-Domain Device Models on page 6-35](#) for more information.

Note *The table's frequencies must be in order from lowest to highest.*

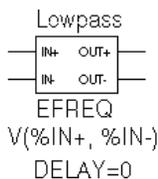


Figure 6-19 shows an EFREQ device used as a low pass filter. The input to the frequency response is the voltage across the input pins. The table describes a low pass filter with a response of 1 (0 dB) for frequencies below 5 kilohertz and a response of

Figure 6-19 *EFREQ Part*

.001 (-60 dB) for frequencies above 6 kilohertz. The output is a voltage across the output pins.

This part is defined by the following attributes:

```
TABLE = (0, 0, 0) (5kHz, 0, -5760) (6kHz, -60, -6912)
DELAY =
R_I =
MAGUNITS =
PHASEUNITS =
```

Since R_I, MAGUNITS, and PHASEUNITS are undefined, each table entry is interpreted as containing frequency, magnitude value in dB, and phase values in degrees. Delay defaults to 0.

The phase lags linearly with frequency meaning that this table exhibits a constant time (group) delay. The delay is necessary so that the impulse response is causal. That is, so that the impulse response does not have any significant components before time zero.

The constant group delay is calculated from the values for a given table entry as follows:

$$\text{group delay} = \text{phase} / 360 / \text{frequency}$$

For this example, the group delay is 3.2 msec (6912 / 360 / 6k = 5760 / 360 / 6k = 3.2m). An alternative specification for this table could be:

```
TABLE = (0, 0, 0) (5kHz, 0, 0) (6kHz, -60, 0)
DELAY = 3.2ms
R_I =
MAGUNITS =
PHASEUNITS =
```

This produces a PSpice A/D netlist declaration like this:

```
ELOWPASS 5 0 FREQ {V(10)} = (0,0,0) (5kHz,0,0) (6kHz-60,0)
+ DELAY = 3.2ms
```

Cautions and Recommendations for Simulation and Analysis

Instantaneous Device Modeling

During AC analysis, nonlinear transfer functions are handled the same way as other nonlinear parts: each function is linearized around the bias point and the resulting small-signal equivalent is used.

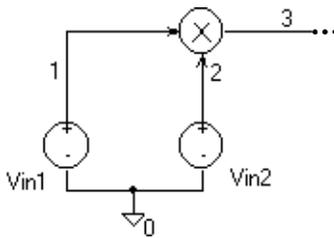


Figure 6-20 Voltage Multiplier Circuit (Mixer)

Consider the voltage multiplier (mixer) shown in Figure 6-20. This circuit has the following characteristics:

$$\begin{aligned} V_{in1}: & \quad DC=0v \quad AC=1v \\ V_{in2}: & \quad DC=0v \quad AC=1v \end{aligned}$$

where the output on net 3 is $V(1)*V(2)$.

During AC analysis, $V(3) = 0$ due to the 0 volts bias point voltage on nets 1, 2, and 3. The small-signal equivalent therefore has 0 gain (the derivative of $V(1)*V(2)$ with respect to both $V(1)$ and $V(2)$ is 0 when $V(1)=V(2)=0$). So, the output of the mixer during AC analysis will be 0 regardless of the AC values of $V(1)$ and $V(2)$.

Another way of looking at this is that a mixer is a nonlinear device and AC analysis is a linear analysis. The output of the mixer has 0 amplitude at the fundamental. (Output is nonzero at DC and twice the input frequency, but these are not included in a linear analysis.)

If you need to analyze nonlinear functions, such as a mixer, use transient analysis. Transient analysis solves the full, nonlinear circuit equations. It also allows you to use input waveforms with different frequencies (for example, V_{IN1} could be 90 MHz and V_{IN2} could be 89.8 MHz). AC analysis does not have this flexibility, but in return it uses much less computer time.

Frequency-Domain Parts

Some caution is in order when moving between frequency and time domains. This section discusses several points that are involved in the implementation of frequency-domain parts. These discussions all involve the transient analysis, since both the DC and AC analyses are straightforward.

The first point is that there are limits on the maximum values and on the resolution of both time and frequency. These are related: the frequency resolution is the inverse of the maximum time and vice versa. The maximum time is the length of the transient analysis, TSTOP. Therefore, the frequency resolution is $1/TSTOP$.

Laplace Transforms

For Laplace transforms, PSpice A/D starts off with initial bounds on the frequency resolution and the maximum frequency determined by the transient analysis parameters as follows. The frequency resolution is initially set below the theoretical limit to $(.25/TSTOP)$ and is then made as large as possible without inducing sampling errors. The maximum frequency has an initial upper bound of $(1/(RELTOL * TMAX))$, where TMAX is the transient analysis Step Ceiling value, and RELTOL is the relative accuracy of all calculated voltages and currents. If a Step Ceiling value is not specified, PSpice A/D uses the Transient Analysis Print Step, TSTEP, instead.

Note *TSTOP, TMAX, and TSTEP values are configured using Transient on the Setup menu. The RELTOL attribute is set using Options on the Setup menu.*

PSpice A/D then attempts to reduce the maximum frequency by searching for the frequency at which the response has fallen to RELTOL times the maximum response. For instance, for the transform:

$$1/(1+s)$$

the maximum response, 1.0, is at $s = j \cdot \omega = 0$ (DC). The cutoff frequency used when RELTOL=.001, is approximately $1000/$

$(2\pi) = 159$ Hz. At 159 Hz, the response is down to .001 (down by 60 db). Since some transforms do not have such a limit, there is also a limit of $10/\text{RELTOL}$ times the frequency resolution, or $10/(\text{RELTOL}\cdot\text{TSTOP})$. For example, consider the transform:

$$e^{-0.001\cdot s}$$

This is an ideal delay of 1 millisecond and has no frequency cutoff. If $\text{TSTOP} = 10$ milliseconds and $\text{RELTOL} = .001$, then PSpice A/D imposes a frequency cutoff of 10 MHz. Since the time resolution is the inverse of the maximum frequency, this is equivalent to saying that the delay cannot resolve changes in the input at a rate faster than .1 microseconds. In general, the time resolution will be limited to $\text{RELTOL}\cdot\text{TSTOP}/10$.

A final computational consideration for Laplace parts is that the impulse response is determined by means of an FFT on the Laplace expression. The FFT is limited to 8192 points to keep it tractable, and this places an additional limit on the maximum frequency, which may not be greater than 8192 times the frequency resolution.

If your circuit contains many Laplace parts which can be combined into a more complex single device, it is generally preferable to do this. This saves computation and memory since there are fewer impulse responses. It also reduces the number of opportunities for numerical artifacts that might reduce the accuracy of your transient analyses.

Laplace transforms can contain poles in the left half-plane. Such poles will cause an impulse response that increases with time instead of decaying. Since the transient analysis is always for a finite time span, PSpice A/D does not have a problem calculating the transient (or DC) response. However, you need to be aware that such poles will make the actual device oscillate.

Non-causality and Laplace transforms

PSpice A/D applies an inverse FFT to the Laplace expression to obtain an impulse response, and then convolves the impulse response with the dependent source input to obtain the output. Some common impulse responses are inherently non-causal. This means that the convolution must be applied to both past and future samples of the input in order to properly represent the inverse of the Laplace expression.

A good example of this is the expression {S}, which corresponds to differentiation in the time domain. The impulse response for {S} is an impulse pair separated by an infinitesimal distance in time. The impulses have opposite signs, and are situated one in the infinitesimal past, the other in the infinitesimal future. In other words, convolution with this corresponds to applying a finite-divided difference in the time domain.

The problem with this for PSpice A/D is that the simulator only has the present and past values of the simulated input, so it can only apply half of the impulse pair during convolution. This will obviously not result in time-domain differentiation. PSpice A/D can detect, but not fix this condition, and issues a non-causality warning message when it occurs. The message tells what percentage of the impulse response is non-causal, and how much delay would need to be added to slide the non-causal part into a causal region. {S} is theoretically 50% non-causal. Non-causality on the order of 1% or less is usually not critical to the simulation results.

One more point about {S}. You can delay it to keep it causal, but keep in mind that the separation between the impulses is infinitesimal. This means that a very small time step is needed. For this reason, it is usually better to use a macromodel to implement differentiation.

Here are some useful guidelines:

- In the case of a Laplace device (ELAPLACE), multiply the Laplace expression by e to the $(-s * \langle \text{the suggested delay} \rangle)$.
- In the case of a frequency table (EFREQ or GFREQ), do either of the following:
 - Specify the table with DELAY=*the suggested delay*.
 - Compute the delay by adding a phase shift.

Chebyshev filters

All of the considerations given above for Laplace parts also apply to Chebyshev filter parts. However, PSpice A/D also attempts to deal directly with inaccuracies due to sampling by applying Nyquist criteria based on the highest filter cutoff frequency. This is done by checking the value of TMAX. If

TMAX is not specified it is assigned a value, or if it is specified, it may be reduced.

For low pass and band pass filters, TMAX is set to $(0.5/FS)$, where FS is the stop band cutoff in the case of a low pass filter, or the upper stop band cutoff in the case of a band pass filter.

For high pass and band reject filters, there is no clear way to apply the Nyquist criterion directly, so an additional factor of two is thrown in as a safety margin. Thus, TMAX is set to $(0.25/FP)$, where FP is the pass band cutoff for the high pass case or the upper pass band cutoff for the band reject case. It may be necessary to set TMAX to something smaller if the filter input has significant frequency content above these limits.

Frequency tables

For frequency response tables, the maximum frequency is twice the highest value. It will be reduced to $10/(RELTOL \cdot TSTOP)$ or 8192 times the frequency resolution if either value is smaller.

The frequency resolution for frequency response tables is taken to be either the smallest frequency increment in the table or the fastest rate of phase change, whichever is least. PSpice A/D then checks to see if it can be loosened without inducing sampling errors.

Trading Off Computer Resources For Accuracy

It should be clear from the foregoing discussion that there is a significant trade-off between accuracy and computation time for parts modeled in the frequency domain. The amount of computer time and memory scale approximately inversely to RELTOL. Therefore, if you can use RELTOL=.01 instead of the default .001, you will be ahead. However, you should first assure yourself based on the relevant criteria described above that this will not adversely affect the impulse response. You may also wish to vary TMAX and TSTOP, since these also come into play.

Since the trade-off issues are fairly complex, it is advisable to first simulate a small test circuit containing only the frequency-domain device, and then after proper validation, proceed to incorporate it in your larger design. The PSpice A/D defaults will be appropriate most of the time if accuracy is your main concern, but it is still worth checking.

Note *Do not set RELTOL to a value above 0.01. This can seriously compromise the accuracy of your simulation.*

Basic Controlled Sources

As with basic SPICE, PSpice A/D has basic controlled sources derived from the standard SPICE E, F, G, and H devices.

Table 6-5 summarizes the linear controlled source types provided in the standard symbol library.

Table 6-5 *Basic Controlled Sources in analog.slb*

Part Type	Symbol Name
Controlled Voltage Source (PSpice A/D “E” device)	E
Current-Controlled Current Source (PSpice A/D “F” device)	F
Controlled Current Source (PSpice A/D “G” device)	G
Current-Controlled Voltage Source (PSpice A/D “H” device)	H

Creating Custom ABM Parts

When you require a controlled source that is not provided within the special purpose set, or that exhibits more elaborate behavior than is provided with the general purpose parts (with multiple controlling inputs, for example), we recommend building the functionality into a custom symbol similar to the special purpose parts introduced above.

The transfer function can be built into the symbol either of the following ways:

- directly in the TEMPLATE definition
- by defining the part’s EXPR and related attributes (if any)

Familiarity with the PSpice A/D syntax for declaring E and G devices will help you form a suitable TEMPLATE definition.

Refer to your *MicroSim Schematics User’s Guide* for a description of how to create a custom symbol.

Refer to the online *MicroSim PSpice A/D Reference Manual* for more information about E and G devices.

Part Three

Setting Up and Running Analyses

Part Three describes how to set up and run analyses and provides setup information specific to each analysis type.

[Chapter 7, Setting Up Analyses and Starting Simulation](#), explains the procedures general to all analysis types to set up and start the simulation.

[Chapter 8, DC Analyses](#), describes how to set up DC analyses, including DC sweep, bias point detail, small-signal DC transfer, and DC sensitivity.

[Chapter 9, AC Analyses](#), describes how to set up AC sweep and noise analyses.

[Chapter 10, Transient Analysis](#), describes how to set up transient analysis and optionally Fourier components. This chapter also explains how to use the Stimulus Editor to create time-based input.

[Chapter 11, Parametric and Temperature Analysis](#), describes how to set up parametric and temperature analyses, and how to run post-simulation performance analysis in Probe on the results of these analyses.

Chapter 12, Monte Carlo and Sensitivity/Worst-Case Analyses, describes how to set up Monte Carlo and sensitivity/worst-case analyses for statistical interpretation of your circuit's behavior.

Setting Up Analyses and Starting Simulation

7

Chapter Overview

This chapter provides an overview of setting up analyses and starting simulation which applies to any analysis type. The other chapters in **Part Three**, *Setting Up and Running Analyses* provide specific analysis setup information for each analysis type.

This chapter includes the following sections:

[Analysis Types on page 7-2](#)

[Setting Up Analyses on page 7-3](#)

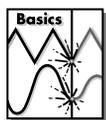
[Starting Simulation on page 7-10](#)

Analysis Types

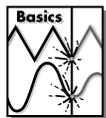
Table 7-1 provides a summary of the available PSpice analyses and the corresponding dialog box (accessed using Setup on the Analysis menu) where the analysis parameters are specified.

Table 7-1 *Classes of PSpice Analyses*

Analysis	Analysis Setup Dialog Box	Swept Variable
Standard analyses		
DC sweep	DC sweep	source parameter temperature
bias point	bias point detail	
small-signal DC transfer	transfer function	
DC sensitivity	sensitivity	
frequency response	AC sweep	frequency
noise (requires a frequency response analysis)	AC sweep	frequency
transient response	transient	time
Fourier (requires transient response analysis)	transient	time
Simple multi-run analyses		
parametric	parametric	
temperature	temperature	
Statistical analyses		
Monte Carlo	Monte Carlo/ worst-case	
Sensitivity/worst-case	Monte Carlo/ worst-case	



Note *Parametric analysis is not supported in PSpice Basics.*



Note *Monte Carlo and sensitivity/worst-case analyses are not supported by PSpice Basics.*

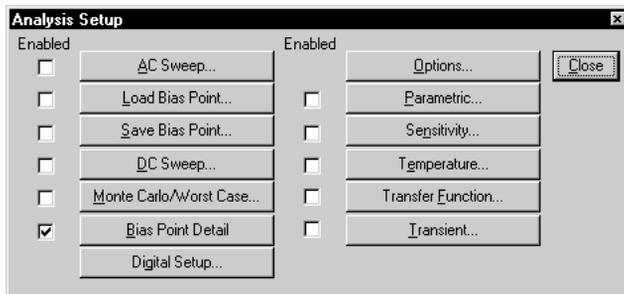
See **Part Four**, *Viewing Results*, for information about using Probe.

The Probe waveform analyzer is used to display and graphically analyze the results of PSpice simulations for swept analyses. Supplementary analysis information is generated to the simulation output file in the form of lists and tables.

Setting Up Analyses

To set up one or more analyses

- 1 In the schematic editor, from the Analysis menu, select Setup.



Specific information for setting up each type of analysis is discussed in the following chapters.

- 2 In the Analysis Setup dialog box, click an analysis button.
- 3 If a setup options dialog box is displayed for the selected analysis type, complete the specification appropriately.
- 4 If needed, select (✓) the check box next to the analysis-type button to enable the analysis.
- 5 Set up any other analyses you want to perform for the circuit.

See [Output Variables on page 7-4](#) for a description of the output variables that can be entered in the setup options dialog box displayed for an analysis type.

Execution Order for Standard Analyses

During simulation, any analyses that are enabled are performed in the order shown in [Table 7-2](#). Each type of analysis is conducted at most once per run.

Several of the analyses (small-signal transfer, DC sensitivity, and frequency response) depend upon the bias point calculation. Since so many analyses use the bias point, PSpice calculates it automatically.

Table 7-2 *Execution Order for Standard Analyses*

1. DC sweep	5. DC sensitivity
2. Bias point	6. Small-signal DC transfer
3. Frequency response	7. Transient response
4. Noise	8. Fourier components

Output Variables

Certain analyses (such as noise, Monte Carlo, sensitivity/worst-case, DC sensitivity, Fourier, and small-signal DC transfer function) require you to specify output variables for voltages and currents at specific points on the schematic. Depending upon the analysis type, you may need to specify the following:

- voltage on a net, a pin, or at a terminal of a semiconductor device
- current through a part or into a terminal of a semiconductor device
- a device name

If output variables or other information are required, a dialog box is displayed when you click on the button for the analysis type in the Analysis Setup dialog box.

Voltage

Specify voltage in the following format:

$$v[\textit{modifiers}](\textit{<out id>},[\textit{<out id>}]) \quad (1)$$

where *<out id >* is:

$$\textit{<net id>} \text{ or } \textit{<pin id>} \quad (2)$$

$$\textit{<net id>} \text{ is a fully qualified net name} \quad (3)$$

$$\textit{<pin id>} \text{ is } \textit{<fully qualified device name>:} \textit{<pin name>} \quad (4)$$

A fully qualified net name (as referred to in line 3 above) is formed by prefixing the visible net name (from a label applied to one of the segments of a wire or bus, or an offpage port connected to the net) with the full hierarchical path, separated by periods. At the top level of hierarchy, this is just the visible name.

A fully qualified device name (from line 4 above) is distinguished by specifying the full hierarchical path followed by the device's reference designator, separated by period characters. For example, a resistor with reference designator R34 inside part Y1 placed on a top-level page is referred to as Y1.R34 when used in an output variable.

A *<pin id >* (from line 4) is uniquely distinguished by specifying the full part name (as described above) followed by a colon, and the pin name. For example, the pins on a capacitor with reference designator C31 placed on a top-level page and pin names 1 and 2 would be identified as C31:1 and C31:2, respectively.

Current

Specify current in the following format:

$$i[\textit{modifiers}](\textit{<out device>}[\textit{:modifiers}])$$

where *<out device >* is a fully qualified device name.

Modifiers

The basic syntax for output variables can be modified to indicate terminals of semiconductors and/or AC specifications. The modifiers come before `<out id>` or `<out device>`. Or, when specifying terminals (such as source or drain), the modifier is the pin name contained in `<out id>`, or is appended to `<out device>` separated by a colon.

Modifiers can be specified as follows:

- For voltage:

`v[AC suffix](<out id>[, out id])`
`v[terminal]*(<out device>)`

- For current:

`i[AC suffix](<out device>[:terminal])`
`i[terminal][AC suffix](<out device>)`

where

<i>terminal</i>	specifies one or two terminals for devices with more than two terminals, such as D (drain), G (gate), S (source)
<i>AC suffix</i>	specifies the quantity to be reported for an AC analysis, such as M (magnitude), P (phase), G (group delay)
<i>out id</i>	specifies either the <i><net id></i> or <i><pin id></i> (<i><fully qualified device name></i> : <i><pin name></i>)
<i>out device</i>	specifies the <i><fully qualified device name></i>

These building blocks can be used for specifying output variables as shown in [Table 7-3](#) (which summarizes the accepted output variable formats) and Tables [7-4](#) through [7-7](#) (which list valid elements for two-terminal, three or four-terminal, transmission line devices, and AC specifications).

Table 7-3 PSpice Output Variable Formats

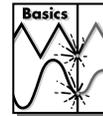
Format	Meaning
V[ac](<i>< + out id ></i>)	voltage at <i>out id</i>
V[ac](<i>< +out id >, < - out id ></i>)	voltage across + and - <i>out id</i> 's
V[ac](<i>< 2-terminal device out id ></i>)	voltage at a 2-terminal <i>device out id</i>
V[ac](<i>< 3 or 4-terminal device out id ></i>) or V<x>[ac](<i>< 3 or 4-terminal out device ></i>)	voltage at non-grounded terminal <i>x</i> of a 3 or 4-terminal <i>device</i>
V<x><y>[ac](<i>< 3 or 4-terminal out device ></i>)	voltage across terminals <i>x</i> and <i>y</i> of a 3 or 4-terminal <i>device</i>
V[ac](<i>< transmission line out id ></i>) or V<z>[ac](<i>< transmission line out device ></i>)	voltage at one end <i>z</i> of a <i>transmission line device</i>
I[ac](<i>< 3 or 4-terminal out device >:<x></i>) or I<x>[ac](<i>< 3 or 4-terminal out device ></i>)	current through non-grounded terminal <i>x</i> of a 3 or 4-terminal <i>device</i>
I[ac](<i>< transmission line out device >:<z></i>) or I<z>[ac](<i>< 3 or 4-terminal out device ></i>)	current through one end <i>z</i> of a <i>transmission line out device</i>
< DC sweep variable >	voltage or current source name

Table 7-4 *Element Definitions for 2-Terminal Devices*

Device Type	<out id> or <out device> Device Indicator	Output Variable Examples
capacitor	C	V(CAP:1) I(CAP)
diode	D	V(D23:1) I(D23)
voltage-controlled voltage source	E	V(E14:1) I(E14)
current-controlled current source	F	V(F1:1) I(F1)
voltage-controlled current source	G	V(G2:1) I(G2)
current-controlled voltage source	H	V(HSOURCE:1) I(HSOURCE)
independent current source	I	V(IDRIV:+) I(IDRIV)
inductor	L	V(L1:1) I(L1)
resistor	R	V(RC1:1) I(RC1)
voltage-controlled switch	S	V(SWITCH:+) I(SWITCH)
independent voltage source	V	V(VSRC:+) I(VSRC)
current-controlled switch	W	V(W22:-) I(W22)

Table 7-5 *Element Definitions for 3- or 4-Terminal Devices*

Device Type	<out id> or <out device> Device Indicator	<pin id>	Output Variable Examples
GaAs MESFET	B	D (Drain terminal) G (Gate terminal) S (Source terminal)	V(B11:D) ID(B11)
Junction FET	J	D (Drain terminal) G (Gate terminal) S (Source terminal)	VG(JFET) I(JFET:G)
MOSFET	M	B (Bulk, substrate terminal) D (Drain terminal) G (Gate terminal) S (Source terminal)	VDG(M1) ID(M1)
bipolar transistor	Q	B (Base terminal) C (Collector terminal) E (Emitter terminal) S (Source terminal)	V(Q1:B) I(Q1:C)
IGBT	Z	C (Collector terminal) E (Emitter terminal) G (Gate terminal)	V(Z1:C) I(Z1:C)



Note The IGBT device type is not supported by PSpice Basics.

Table 7-6 *Element Definitions for Transmission Line Devices*

Device Type	<out id> or <out device> Device Indicator	<z>	Output Variable Examples
transmission line	T	A (Port A) B (Port B)	V(T32:A+) I(T32:B-)

Table 7-7 *Element Definitions for AC Analysis Specific Elements*

<ac suffix> Device Symbol	Meaning	Output Variable Examples
(none)	magnitude (default)	V(V1) I(V1)
M	magnitude	VM(CAP1:1) IM(CAP1:1)
DB	magnitude in decibels	VDB(R1)
P	phase	IP(R1)
R	real part	VR(R1)
I	imaginary part	VI(R1)

The INOISE, ONOISE, DB(INOISE), and DB(ONOISE) output variables are predefined for use with noise (AC sweep) analysis.

Starting Simulation

Once you have used MicroSim Schematics to enter your circuit design and to set up the analyses to be performed, you can start simulation by selecting Simulate from the Analysis menu. When you enter and set up your circuit this way, the files needed for simulation are automatically created by Schematics and the simulator is started from Schematics.

There may be situations, however, when you want to run PSpice outside of Schematics. You may want to simulate a circuit that wasn't created in Schematics, for example, or you may want to run simulations of multiple circuits in batch mode.

This section includes the following:

[Starting Simulation from Schematics](#), below

[Starting Simulation Outside of Schematics on page 7-11](#)

[Setting Up Batch Simulations on page 7-12](#)

[The Simulation Status Window on page 7-14](#)

Starting Simulation from Schematics

Once you have set up the analyses for the circuit, you can start simulation from Schematics any of the following ways:

- select Simulate from the Analysis menu
- press **F11**
- click on the Simulate icon



Starting Simulation Outside of Schematics

To start PSpice outside of Schematics

- 1 Double-click on the PSpice icon in the MicroSim Program Group.
- 2 Select Open from the File menu.
- 3 Do one of the following:
 - Double-click on the circuit file name in the list box.
 - Enter the name of the circuit file to be simulated in the File Name text box.

Setting Up Batch Simulations

Multiple simulations can be run in batch mode when starting PSpice directly with circuit file input. You can use batch mode, for example, to run a number of simulations overnight without user intervention. There are two ways to do this as described below.

Multiple simulation setups within one circuit file

Multiple circuit/simulation descriptions can be concatenated into a single circuit file and simulated once with PSpice. Each circuit/simulation description in the file must begin with a title line and end with a .END statement.

The simulator reads all the circuits in the circuit file and then processes each one in sequence. The Probe data file and simulation output file contain the outputs from each circuit in the same order as they appeared in the circuit file. The effect is the same as if you had run each circuit separately and then concatenated all of the outputs.

Running simulations with multiple circuit files

You can direct PSpice to simulate multiple circuit files using one of the following methods.

Method 1

- 1 Click on the PSpice icon in the MicroSim program group.
- 2 Select Properties from the File menu.
- 3 Include the following switch in the command line:

```
/wNO_NOTIFY
```

This disables the message that pops up each time a simulation is completed.

- 4 Select Open from the File menu from the PSpice status window.
- 5 Do one of the following:
 - Type each file name in the File Name text box separated by a space.

- Use the combination keystrokes and mouse clicks in the list box as follows: **Ctrl**+click to select file names one at a time, and **Shift**+click to select groups of files.

Method 2

- 1 Click on the PSpice icon in the MicroSim program group.
- 2 Select Properties from the File menu.
- 3 Include the following switch in the command line:

```
/wNO_NOTIFY
```

This disables the message that pops up each time a simulation is completed.

- 4 Update the command line in one of the following ways:
 - Include a list of circuit file names separated by spaces.
 - Read a file of run time properties (using the *@<file name>* syntax) which contains a list of circuit file names.

Circuit file names may be fully qualified or contain the wild card characters * and ?.

The Simulation Status Window

As PSpice performs the circuit simulation, a status window is displayed so you can monitor the progress of the simulation. Figure 7-1 shows an example of the PSpice status window.

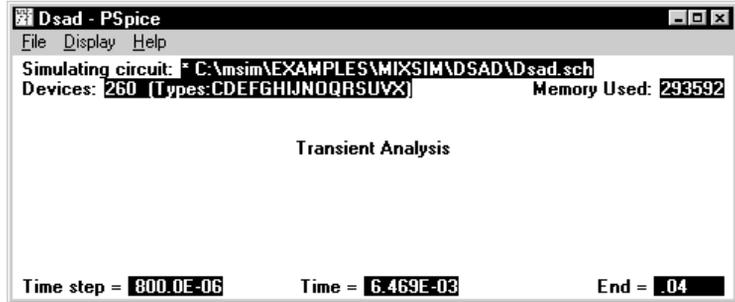


Figure 7-1 PSpice Status Window

The status window includes the following:

Title bar This area at the top of the window identifies the name of the circuit file currently being simulated, and the name of the simulation output file where audit trail information will be written.

Menus The menus accessed from the menu bar include items to control the simulator and customize the window display characteristics. These are especially useful when invoking PSpice directly.

Simulation progress display The lower portion of the window displays the progress of each simulation as it proceeds.

DC Analyses

8

Chapter Overview

This chapter describes how to set up DC analyses and includes the following sections:

[DC Sweep on page 8-2](#)

[Bias Point Detail on page 8-8](#)

[Small-Signal DC Transfer on page 8-10](#)

[DC Sensitivity on page 8-12](#)

DC Sweep

Minimum Requirements to Run a DC Sweep Analysis

Minimum circuit design requirements

Table 8-1 *DC Sweep Circuit Design Requirements*

Swept Variable Type	Requirement
voltage source	voltage source with a DC specification (VDC, for example)
temperature	none
current source	current source with a DC specification (IDC, for example)
model parameter	PSpice model (.MODEL)
global parameter	global parameter defined with a parameter block (.PARAM)

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.

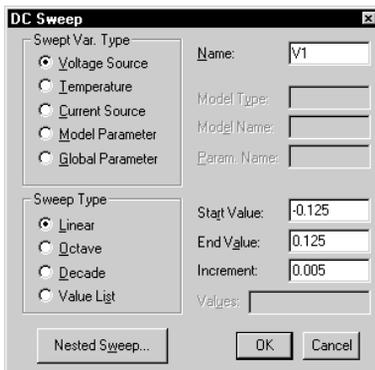


Figure 8-1 *DC Sweep Setup*

Minimum program setup requirements

- In the Analysis Setup dialog box, click the DC Sweep button. Complete the DC Sweep dialog box as needed.
- If needed, select (✓) the DC Sweep check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

Note *Do not specify a DC sweep and a parametric analysis for the same variable.*

Overview of DC Sweep

The DC sweep analysis causes a DC sweep to be performed on the circuit. DC sweep allows you to sweep a source (voltage or current), a global parameter, a model parameter, or the temperature through a range of values. The bias point of the circuit is calculated for each value of the sweep. This is useful for finding the transfer function of an amplifier.

For the DC sweep analysis specified in Figure 8-1, the voltage source V1 is swept from -0.125 volts to 0.125 volts by steps of 0.005. This means that the output has $(0.125 + 0.125)/0.005 + 1 = 51$ lines.

A source with a DC specification (such as VDC or IDC) must be used if the swept variable is to be a voltage type or current source. To set the DC attribute, select Attributes from the Edit menu.

The default DC value of V1 is overridden during the DC sweep analysis and is made to be the swept value. All of the other sources retain their values.

After running the analysis, the simulation output file (example.out for the example.sch circuit in Figure 8-2) contains a table of voltages relating V1, node OUT1, and node OUT2.

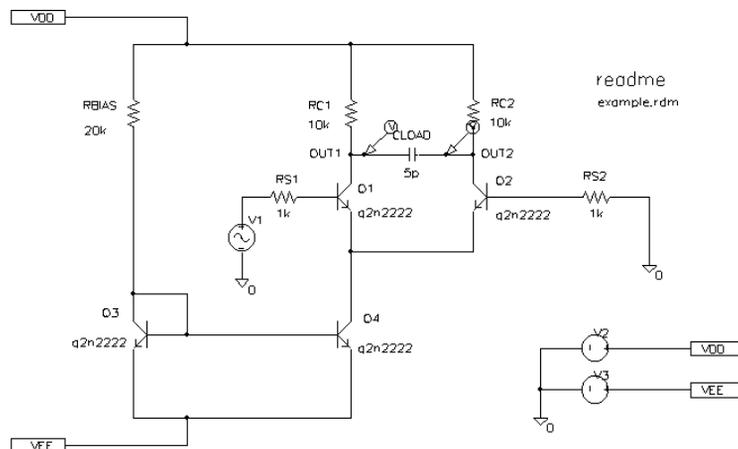


Figure 8-2 Example Schematic example.sch

To calculate the DC response of an analog circuit, PSpice removes time from the circuit. This is done by treating all

Click  or double-click the symbol.

The example circuit example.sch is provided with the MicroSim program installation.

capacitors as open circuits, all inductors as shorts, and using only the DC values of voltage and current sources.

In order to solve the circuit equations, PSpice uses an iterative algorithm using continuous equations.

Setting Up a DC Stimulus

To run a DC sweep or small-signal DC transfer analysis, you need to place and connect one or more independent sources and then set the DC voltage or current level for each source.

To set up a DC stimulus

- 1 Place and connect one of these symbols in your schematic:

If you are planning to run an AC or transient analysis in addition to a DC analysis, see the following:

- [Using time-based stimulus symbols with AC and DC attributes on page 3-20](#) for other source symbols that you can use.
- [Using VSRC or ISRC symbols on page 3-21](#) to find out how to specify the TRAN attribute for a time-based input signal when using VSRC or ISRC symbols.

For voltage input

Use this...	When you are running...
VDC	A DC sweep and/or transfer function analysis only.
VSRC	Multiple analysis types including DC sweep and/or transfer function.

For current input

Use this...	When you are running...
IDC	A DC sweep and/or transfer function analysis only.
ISRC	Multiple analysis types including DC sweep and/or transfer function.

- 2 Double-click the symbol instance. A dialog box appears listing the attribute settings for the symbol instance.

- 3 Define the DC specification as follows:

Set this attribute...	To this value...
DC	<i>DC_level</i> where <i>DC_level</i> is in volts or amps (units are optional).

Nested DC Sweeps

A second sweep variable can be selected once a primary sweep value has been specified in the DC Sweep dialog box. When you specify a secondary sweep variable, it forms the outer loop for the analysis. That is, for every increment of the second sweep variable, the first sweep variable is stepped through its entire range of values.

To set up a nested sweep

- 1 Click the Nested Sweep button in the DC Sweep dialog box.
- 2 Specify a secondary DC sweep in the DC Nested Sweep dialog box.
- 3 Select (✓) the Enable Nested Sweep check box.
- 4 Click Main Sweep to return to the DC Sweep dialog box, or click OK to return to the Analysis Setup dialog box.



Curve Families for DC Sweeps

Whenever a nested DC sweep is performed, the entire curve family is displayed. That is, the nested DC sweep is treated as a single Probe data section (or you can think of it as a single PSpice run).

For the circuit shown in Figure 8-3, you could set up a DC sweep analysis with an outer sweep of the voltage source VD and an inner sweep of the voltage source VG as listed in [Table 8-2](#).

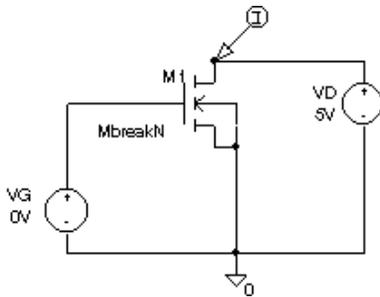


Figure 8-3 Curve Family Example Schematic

Use Mark Current Into Pin in MicroSim Schematics on the Markers menu to add a current marker.

V_VD is the hierarchical name for VD created by netlisting the schematic. This is the name used by Probe.

Table 8-2 Curve Family Example Setup

	Outer Sweep	Nested Sweep
Swept Var Type	Voltage Source	Voltage Source
Sweep Type	Linear	Linear
Name	VD	VG
Start Value	0	0
End Value	5	2
Increment	0.1	0.5

When the DC sweep analysis is run, add a current marker at the drain pin of M1 and display the simulation result in Probe. The result will look like Figure 8-4.

To add a load line for a resistor, add a trace that computes the load line from the sweep voltage. Assume that the X axis variable is the sweep voltage V_VD, which runs from 0 to 5 volts. The expression which will add a trace that is the load line for a 50 kohm resistor is:

$$(5V - V_VD) / 50K$$

This can be useful for determining the bias point for each member of a curve family as shown in Figure 8-5.

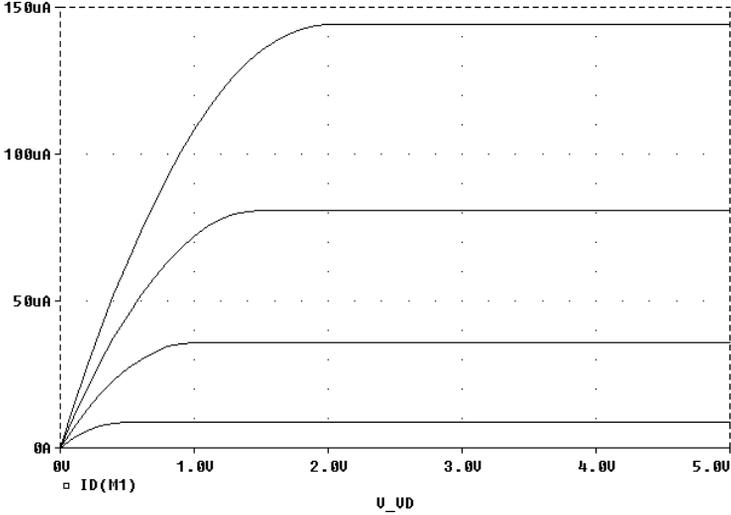


Figure 8-4 *Device Curve Family*

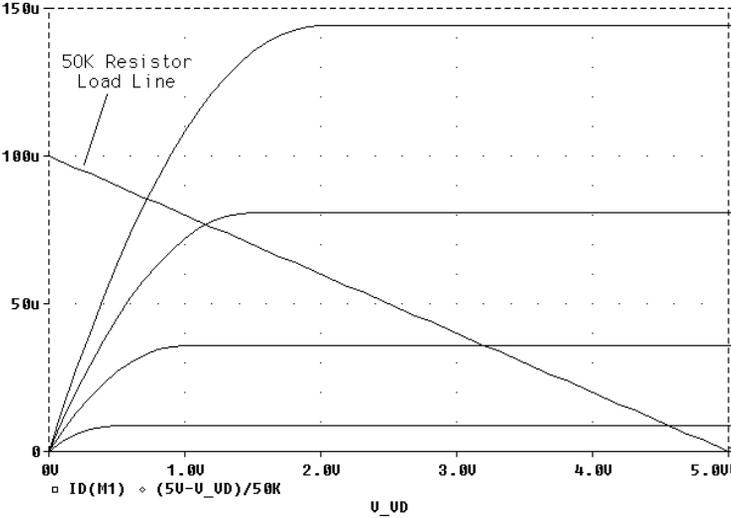


Figure 8-5 *Operating Point Determination for Each Member of the Curve Family*

Bias Point Detail

Minimum Requirements to Run a Bias Point Detail Analysis

Minimum circuit design requirements

None.

Minimum program setup requirements

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.

- In the Analysis Setup dialog box, select (✓) the Bias Point Detail check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

Also see [Save and Load Bias Point on page A-2](#).

Overview of Bias Point Detail

The bias point is calculated for any analysis whether or not the Bias Point Detail analysis is enabled in the Analysis Setup dialog box. However, additional information is reported when the Bias Point Detail analysis is enabled.

When Bias Point Detail analysis is not enabled, only analog node voltages are reported to the output file.

When the Bias Point Detail analysis is enabled, the following information is reported to the output file:

- a list of all analog node voltages
- the currents of all voltage sources and their total power
- a list of the small-signal parameters for all devices

If Bias Point Detail is enabled, you can suppress the reporting of the bias point analog node values:

- 1 Select Setup from the Analysis menu.
- 2 In the Analysis Setup dialog box, click Options.
- 3 In the Yes/No options for the Options dialog box, set NOBIAS to Y (for yes).

Small-Signal DC Transfer

Minimum Requirements to Run a Small-Signal DC Transfer Analysis

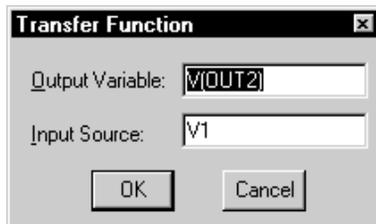
Minimum circuit design requirements

- The circuit should contain an input source, such as VSRC.

Minimum program setup requirements

- In the Analysis Setup dialog box, click the Transfer Function button. In the Transfer Function dialog box, specify the name of the input source desired. See [Output Variables on page 7-4](#) for a description of output variable formats.
- If needed, in the Analysis Setup dialog box, select (✓) the Transfer Function check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.



Overview of Small-Signal DC Transfer

The small-signal DC transfer analysis causes the small-signal transfer function to be calculated by linearizing the circuit around the bias point. The small-signal gain, input resistance, and output resistance are calculated and reported.

To calculate the small-signal gain, input resistance, and output resistance, you need to specify an output voltage or current through a voltage source in the Transfer Function dialog box.

For example, entering $V(a,b)$ as the output variable specifies that the output variable is the output voltage between two nets, a and b . Entering $I(VDRIV)$ as the output variable specifies that the output variable is the current through a voltage source $VDRIV$.

You also need to specify the input source name in the Transfer Function dialog box. The gain from the input source to the output variable is output along with the input and output resistances.

For example, if you enter $V(OUT2)$ as the output variable and $v1$ as the input source, the input resistance for $V1$ is calculated, the output resistance for $V(OUT2)$ is calculated, and the gain from $V1$ to $V(OUT2)$ is calculated. All calculations are reported to the simulation output file.

DC Sensitivity

Minimum Requirements to Run a DC Sensitivity Analysis

Minimum circuit design requirements

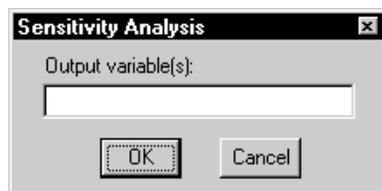
None.

Minimum program setup requirements

- In the Analysis Setup dialog box, click the Sensitivity button. In the Sensitivity Analysis dialog box, enter the output variable desired.
- If needed, in the Analysis Setup dialog box, select (✓) the Sensitivity check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.

See [Output Variables on page 7-4](#) for a description of output variables.



Overview of DC Sensitivity

DC sensitivity analysis calculates and reports the sensitivity of one node voltage to each device parameter for the following device types:

- resistors
- independent voltage and current sources
- voltage and current-controlled switches
- diodes
- bipolar transistors

The sensitivity is calculated by linearizing all devices around the bias point.

AC Analyses

9

Chapter Overview

This chapter describes how to set up AC sweep and noise analyses.

[AC Sweep Analysis on page 9-2](#) describes how to set up an analysis to calculate the frequency response of your circuit. This section also discusses how to define an AC stimulus and how PSpice treats nonlinear devices in an AC sweep.

[Noise Analysis on page 9-9](#) describes how to set up an analysis to calculate device noise contributions and total input and output noise.

AC Sweep Analysis

What You Need to Do to Run an AC Sweep

The following procedure describes the minimum set of things that you need to do to run an AC sweep analysis. For more detail on any step, go to the page referenced in the sidebar next to the step.

To find out how, see [Setting Up an AC Stimulus on page 9-3](#).

To find out how, see [Setting Up an AC Analysis on page 9-5](#).

To find out how, see [Starting Simulation on page 7-10](#).

To set up and run an AC sweep

- 1 Place and connect a voltage or current source with an AC input signal.
- 2 Set up the AC sweep simulation specification and enable the analysis by doing one of the following:
 - If you have not yet specified the simulation parameters, complete the AC Sweep and Noise Analysis dialog box as needed.
 - If you disabled the AC sweep analysis after having set up the simulation parameters, then in the Analysis Setup dialog box, select (✓) the AC Sweep check box to enable it.
- 3 Start the simulation.

What is AC Sweep?

AC sweep is a frequency response analysis. PSpice calculates the small-signal response of the circuit, linearized around the bias point, to a combination of inputs. Here are a few things to note:

To find out more, see [How PSpice Treats Nonlinear Devices on page 9-7](#).

- Nonlinear devices, such as voltage- or current-controlled switches, are linearized about their bias point value before PSpice runs the linear (small-signal) analysis.

- Because AC sweep analysis is a linear analysis, it only considers the gain and phase response of the circuit; it does not limit voltages or currents.

The best way to use AC sweep analysis is to set the source magnitude to one. This way, the measured output equals the gain, relative to the input source, at that output.



Setting Up an AC Stimulus

To run an AC sweep analysis, you need to place and connect one or more independent sources and then set the AC magnitude and phase for each source.

To set up an AC stimulus

- 1 Place and connect one of these symbols in your schematic:

For voltage input	
Use this...	When you are running...
VAC	An AC sweep analysis only.
VSRC	Multiple analysis types including AC sweep.

For current input	
Use this...	When you are running...
IAC	An AC sweep analysis only.
ISRC	Multiple analysis types including AC sweep.

Note Unlike DC sweep, the AC sweep analysis setup dialog box does not include an input source option. Instead, each independent source in your circuit contains its own AC specification for magnitude and phase.

If you are planning to run a DC or transient analysis in addition to an AC analysis, see [If you want to specify multiple stimulus types on page 3-20](#) for additional information and source symbols that you can use.

- 2 Double-click the symbol instance. A dialog box appears listing the attribute settings for the symbol instance.
- 3 Depending on the source symbol that you placed, define the AC specification as follows:

For VAC or IAC	
Set this attribute...	To this value...
ACMAG	AC magnitude in volts (for VAC) or amps (for IAC); units are optional.
ACPHASE	Optional AC phase in degrees.

For VSRC or ISRC	
Set this attribute...	To this value...
AC	<i>Magnitude_value</i> [<i>phase_value</i>] where <i>magnitude_value</i> is in volts or amps (units are optional) and the optional <i>phase_value</i> is in degrees.

If you are also planning to run a transient analysis, see [Using VSRC or ISRC symbols on page 3-21](#) to find out how to specify the TRAN attribute.

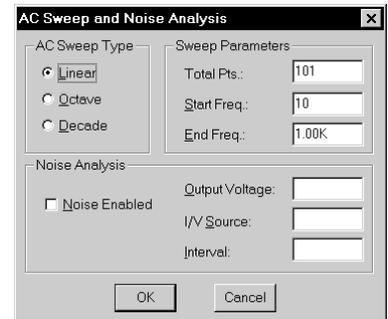
Setting Up an AC Analysis

To set up the AC analysis

- 1 From the Analysis menu, select Setup.
- 2 Click AC Sweep.
- 3 In the AC Sweep dialog box, choose the AC Sweep Type and set the number of sweep points as follows:

To sweep frequency...	Do this...
linearly	Choose Linear and set Total Pts to the total number of points in the sweep.
logarithmically by octaves	Choose Octave and set Pts/Octave to the total number of points per octave.
logarithmically by decades	Choose Decades and set Pts/Decade to the total number of points per decade.

- 4 In the Start Freq and End Freq text boxes, enter the starting and ending frequencies, respectively, for the sweep.
- 5 Click OK.



If you also want to run a noise analysis, then before clicking OK, complete the Noise Analysis frame in this dialog box as described in [Setting Up a Noise Analysis on page 9-11](#).

AC Sweep Setup in “example.sch”

If you look at the example circuit, `example.sch`, provided with your MicroSim programs, you’ll find that its AC analysis is set up as shown in Figure 9-1.

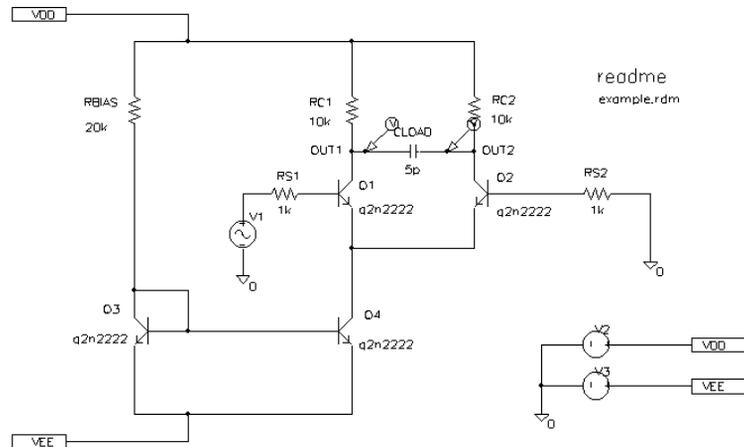
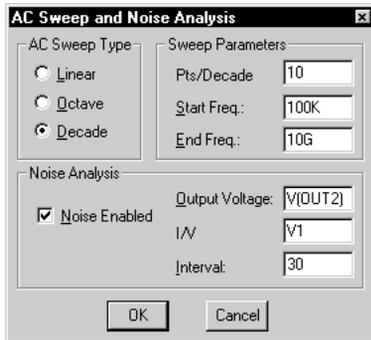


Figure 9-1 AC Analysis Setup for `example.sch`

Note The source, `V1`, is a `VSIN` source that is normally used for setting up sine wave signals for a transient analysis. It also has an AC attribute so that you can also use it for an AC analysis.

To find out more about `VSIN` and other source symbols that you can use for AC analysis, see [Using time-based stimulus symbols with AC and DC attributes on page 3-20](#).

Frequency is swept from 100 kHz to 10 GHz by decades, with 10 points per decade. The `V1` independent voltage source is the only input to an amplifier, so it is the only AC stimulus to this circuit. Magnitude equals 1 V and relative phase is left at zero degrees (the default). All other voltage sources have zero AC value.

How PSpice Treats Nonlinear Devices

An AC Sweep analysis is a linear or small-signal analysis. This means that nonlinear devices must be linearized to run the analysis.

What's required to linearize a device

If you were to manually linearize a device such as a transistor amplifier, you would need to do the following:

- 1 Compute the DC bias point for the circuit.
- 2 Compute the complex impedance and/or transconductance values for each device at this bias point.
- 3 Perform the linear circuit analysis at the frequencies of interest by using simplifying approximations.

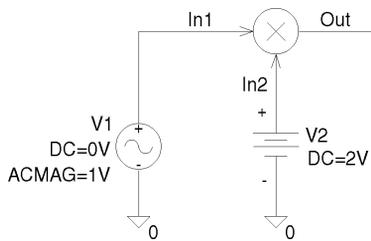
Example: Replace a bipolar transistor in common-emitter mode with a constant transconductance (collector current proportional to base-emitter voltage) and a number of constant impedances.

What PSpice does

PSpice automates this process for you. PSpice computes the partial derivatives for nonlinear devices at the bias point and uses these to perform small-signal analysis.

Example: Nonlinear behavioral modeling block

Suppose you have an analog behavioral modeling block that multiplies $V(1)$ by $V(2)$. Multiplication is a nonlinear operation. To run an AC sweep analysis on this block, the block needs to be replaced with its linear equivalent. To determine the linear equivalent block, PSpice needs a known bias point.



Using a DC source

Consider the circuit shown here. At the DC bias point, PSpice calculates the partial derivatives which determine the linear response of the multiplier as follows:

$$\begin{aligned} V(Out) &= V(In1) \cdot \frac{\partial V(Out)}{\partial V(In1)} + V(In2) \cdot \frac{\partial V(Out)}{\partial V(In2)} \\ &= V(In1) \cdot V(In2) + V(In2) \cdot V(In1) \end{aligned}$$

where the terms in bold are calculated at the DC bias point.

For this circuit, this equation reduces to:

$$V(Out) = V(In1) \cdot 2 + V(In2) \cdot 0$$

This means that the multiplier acts as an amplifier of the AC input with a gain that is set by the DC input.



Caution: Multiplying AC sources

This is exactly how a double-balanced mixer behaves, which is in practice, a simple multiplier.

Note *A double-balanced mixer with inputs at the same frequency would produce outputs at DC and at twice the input frequency, but these terms cannot be seen with a linear, small-signal analysis.*

Suppose that you replace the 2 volt DC source in the above example with an AC source with amplitude 1 and no DC value (DC=0). When PSpice computes the bias point, there are no DC sources in the circuit, so all nodes are at 0 volts at the bias point. Now the linear equivalent of the multiplier block is a block with gain 0, which means that there is no output voltage at the fundamental frequency.

Noise Analysis

What You Need to Do to Run a Noise Analysis

The following procedure describes the minimum set of things that you need to do to run a noise analysis. For more detail on any step, go to the page referenced in the sidebar next to the step.

To set up and run an AC sweep

- 1 Place and connect a voltage or current source with an AC input signal.
- 2 Set up the AC sweep simulation specification.
- 3 Set up the noise simulation specification and enable the analysis by doing one of the following.
 - If you have not yet specified the simulation parameters, complete the noise analysis parameters in the AC Sweep and Noise Analysis dialog box.
 - If you disabled the AC sweep analysis after having set up the AC and noise simulation parameters, then in the Analysis Setup dialog box, select (✓) the AC Sweep check box to enable it.
- 4 Start the simulation.

To find out how, see [Setting Up an AC Stimulus on page 9-3](#).

To find out how, see [Setting Up an AC Analysis on page 9-5](#).

To find out how, see [Setting Up a Noise Analysis on page 9-11](#).

To find out how, see [Starting Simulation on page 7-10](#).

What is Noise Analysis?

When running a noise analysis, PSpice calculates and reports the following for each frequency specified for the AC sweep analysis:

Example: Diodes have separate noise contributions from thermal, shot, and flicker noise.

- device noise, which is the noise contribution propagated to the specified output net from every resistor and semiconductor device in the circuit; for semiconductor devices, the device noise is also broken down into constituent noise contributions where applicable
- total output and equivalent input noise

This value...	Means this...
Output noise	RMS sum of all the device contributions propagated to a specified output net
Input noise	equivalent noise that would be needed at the input source to generate the calculated output noise in an ideal (noiseless) circuit

How PSpice calculates total output and input noise

To calculate total noise at an output net, PSpice computes the RMS sum of the noise propagated to the net by all noise-generating devices in the circuit.

To calculate the equivalent input noise, PSpice then divides total output noise by the gain from the input source to the output net. This results in the amount of noise which, if injected at the input source into a noiseless circuit, would produce the total noise originally calculated for the output net.

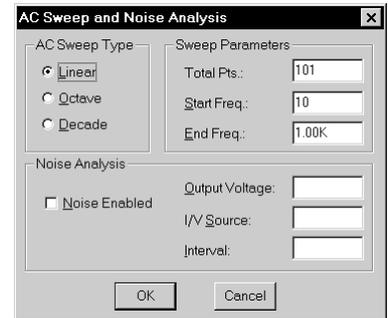
Setting Up a Noise Analysis

To set up the noise analysis

- 1 From the Analysis menu, select Setup.
- 2 Click AC Sweep.
- 3 In the AC Sweep dialog box, set up an AC sweep analysis as described on page 9-5.
- 4 In the AC Sweep dialog box, select (✓) the Noise Enabled check box.
- 5 Enter the noise analysis parameters as follows:

In this text box...	Type this...
Output Voltage	A voltage output variable of the form $V(\text{node}, [\text{node}])$ where you want the total output noise calculated.
I/V Source	The name of an independent current or voltage source where you want the equivalent input noise calculated. Note <i>If the source is in a lower level of a hierarchical schematic, separate the names of the hierarchical devices with periods (.).</i>
Interval	An integer n designating that at every n^{th} frequency, you want to see a table printed in the PSpice output file (.out) showing the individual contributions of all of the circuit's noise generators to the total noise.

- 6 Click OK.



To find out more about valid syntax, see [Output Variables on page 7-4](#).

Example: U1.V2

Note *In Probe, you can view the device noise contributions at every frequency specified in the AC sweep. The Interval parameter has no effect on what PSpice writes to the Probe data file.*

For a break down of noise output variables by supported device type, see [Table 13-9 on page 13-39](#).

Analyzing Noise in Probe

Probe supports these output variable formats, which you can use to view traces for device noise contributions and total input or output noise at every frequency in the analysis.

To view this...	Use this output variable...	Which is represented by this equation *
Flicker noise for a device	NFID(<i>device_name</i>) NFIB(<i>device_name</i>)	$\text{noise} \propto k_f \cdot \frac{I^{a_f}}{f^b}$
Shot noise for a device	NSID(<i>device_name</i>) NSIB(<i>device_name</i>) NSIC(<i>device_name</i>)	For diodes and BJTs: $\text{noise} \propto 2qI$ For GaAsFETs, JFETs, and MOSFETs: $\text{noise} \propto 4kT \cdot \frac{dI}{dV} \cdot \frac{2}{3}$
Thermal noise for the RB, RC, RD, RE, RG, or RS constituent of a device, respectively	NRB(<i>device_name</i>) NRC(<i>device_name</i>) NRD(<i>device_name</i>) NRE(<i>device_name</i>) NRG(<i>device_name</i>) NRS(<i>device_name</i>)	$\text{noise} \propto \frac{4kT}{R}$
Total noise for a device	NTOT(<i>device_name</i>)	Sum of all contributors in <i>device_name</i>
Total output noise for the circuit	NTOT(ONoise)	$\sum_{\text{device}} NTOT(\text{device})$
RMS-summed output noise for the circuit	V(ONoise)	RMS sum of all contributors ($\sqrt{NTOT(ONoise)}$)
Equivalent input noise for the circuit	V(INoise)	$\frac{V(ONoise)}{\text{gain}}$

.*

* To find out more about the equations that describe noise behavior, refer to the appropriate device type in the Analog Devices chapter in the online *MicroSim PSpice A/D Reference Manual*.

About noise units

This type of noise output variable...	Is reported in these units...
Device contribution of the form Nxxx	$(volts)^2/(Hz)$
Total input or output noise of the form V(ONoise) or V(INoise)	$(volts)/(\sqrt{Hz})$

Example

You can run a noise analysis on the circuit shown in [Figure 9-1 on page 9-6](#).

To run a noise analysis on the example:

- 1 In Schematics, open the `example.sch` circuit provided with your MicroSim programs in the `Examples\Schemat\Example` subdirectory.
- 2 From the Analysis menu, select Setup.
- 3 Clear the check boxes for the Transient and Temperature analyses.
- 4 Click AC Sweep.
- 5 Select (✓) the Noise Enabled check box.



These are the existing settings for the noise analysis parameters:

Output Voltage	V(OUT2)
I/V Source	V1
Interval	30

These settings mean that PSpice will calculate noise contributions and total output noise at net OUT2 and equivalent input noise from V1.

For a discussion of the Interval parameter, see page [9-11](#).

To find out more about Probe macros, refer to Probe online help.

Figure 9-2 shows Probe traces for Q1's constituent noise sources as well as total noise for the circuit after simulating. Notice that the trace for RMSSUM (at the top of the plot), which is a macro for the trace expression

$$\text{SQRT}(\text{NTOT}(\text{Q1}) + \text{NTOT}(\text{Q2}) + \text{NTOT}(\text{Q3}) + \dots),$$

exactly matches the total output noise, V(ONOISE), calculated by PSpice.

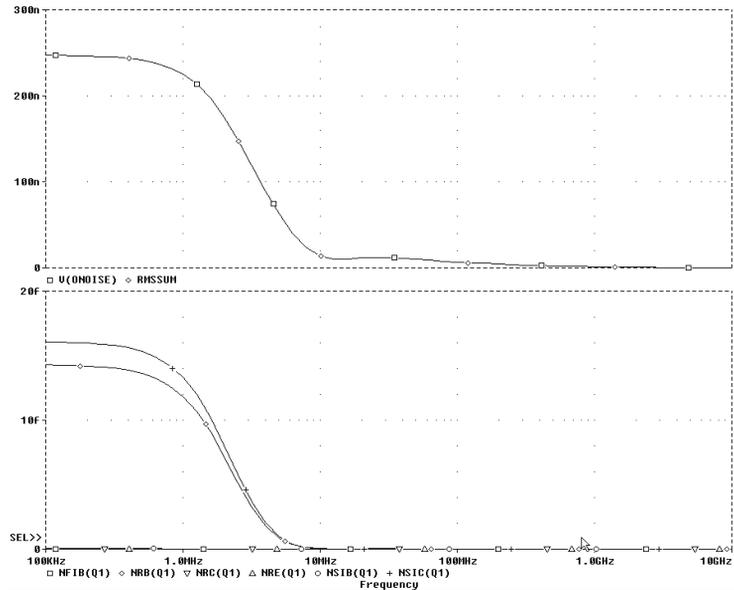


Figure 9-2 Device and Total Noise Traces for “example.sch”

Transient Analysis

10

Chapter Overview

This chapter describes how to set up a transient analysis and includes the following sections:

[Overview of Transient Analysis on page 10-2](#)

[Defining a Time-Based Stimulus on page 10-3](#)

[Transient \(Time\) Response on page 10-15](#)

[Internal Time Steps in Transient Analyses on page 10-17](#)

[Switching Circuits in Transient Analyses on page 10-17](#)

[Plotting Hysteresis Curves on page 10-18](#)

[Fourier Components on page 10-20](#)

Overview of Transient Analysis

Minimum Requirements to Run a Transient Analysis

Minimum circuit design requirements

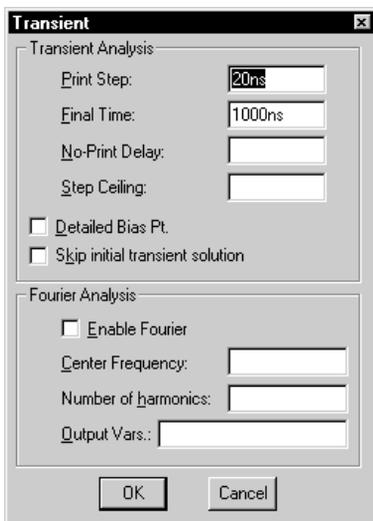
Circuit should contain one of the following:

- an independent source with a transient specification (see [Table 10-1](#))
- an initial condition on a reactive element
- a controlled source that is a function of time

Minimum program setup requirements

- In the Analysis Setup dialog box, click the Transient button. Complete the Transient dialog box as needed.
- If needed, in the Analysis Setup dialog box, select (✓) the Transient check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.



Defining a Time-Based Stimulus

Overview of Stimulus Generation

Symbols that generate input signals for your circuit can be divided into two categories:

- those whose transient behavior is characterized graphically using the Stimulus Editor
- those whose transient behavior is characterized by manually defining their attributes within Schematics



Note PSpice Basics does not include the Stimulus Editor.

Their symbols are summarized in [Table 10-1](#).

Table 10-1 Stimulus Symbols for Time-Based Input Signals

Specified by...	Symbol Name	Description
Using the Stimulus Editor	VSTIM	voltage source
	ISTIM	current source

Table 10-1 *Stimulus Symbols for Time-Based Input Signals*

Specified by...	Symbol Name	Description
Defining symbol attribute	VSRC	voltage sources
	VEXP	
	VPULSE	
	VPWL	
	VPWL_RE_FOREVER	
	VPWL_F_RE_FOREVER	
	VPWL_N_TIMES	
	VPWL_F_N_TIMES	
	VSFFM	
	VSIN	
	ISRC	current sources
	IEXP	
	IPULSE	
	IPWL	
	IPWL_RE_FOREVER	
	IPWL_F_RE_FOREVER	
	IPWL_N_TIMES	
	IPWL_F_N_TIMES	
	ISFFM	
	ISIN	

To use any of these source types, you must place the symbol in your schematic and then define its transient behavior.

Each attribute-characterized stimulus has a distinct set of attributes depending upon the kind of transient behavior it represents. For VPWL_F_xxx, IPWL_F_xxx, and FSTIM, a separate file contains the stimulus specification.

As an alternative, the Stimulus Editor utility automates the process of defining the transient behavior of stimulus devices. The Stimulus Editor allows you to create analog stimuli which generate sine wave, repeating pulse, exponential pulse, single-frequency FM, and piecewise linear waveforms. This applies to both stimulus symbols placed in your schematic as well as new ones that you might create.

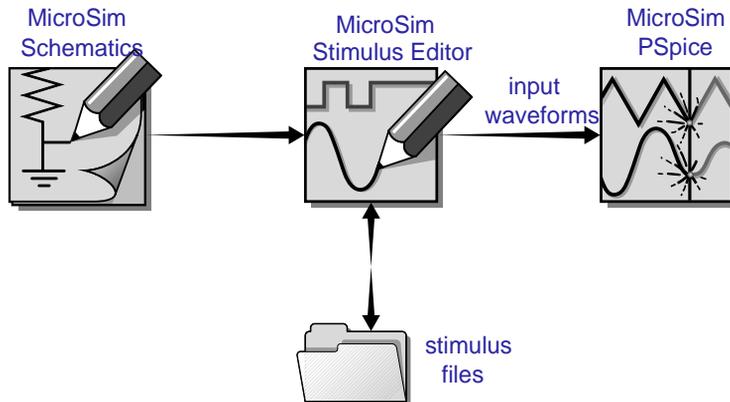


Figure 10-1 Relationship of Stimulus Editor with Schematics and PSpice

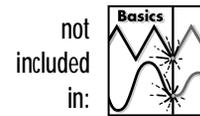
The stimulus specification created using the Stimulus Editor is saved to a file, automatically configured into the schematic, and associated with the corresponding VSTIM, ISTIM, or DIGSTIM part instance or symbol definition.

The Stimulus Editor Utility

The Stimulus Editor is a utility that allows you to quickly set up and verify the input waveforms for a transient analysis. You can create and edit voltage sources and current sources for your circuit. Menu prompts guide you to provide the necessary parameters, such as the rise time, fall time, and period of a repeating pulse. Graphical feedback allows you quickly verify the waveform.

Stimulus Files

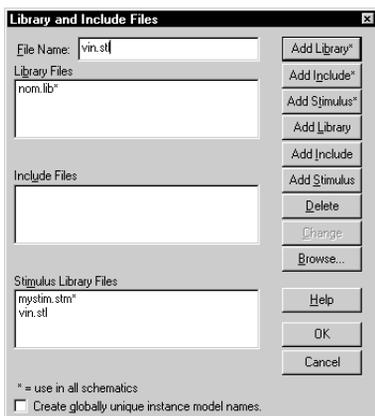
The Stimulus Editor produces a file containing the stimuli with their transient specification. These stimuli are defined as simulator device declarations using the V (voltage source) and I



MicroSim program versions without the Stimulus Editor must use the characterized-by-attribute sources listed in [Table 10-1 on page 10-3](#).

(current source) forms. Since the Stimulus Editor produces these statements automatically, you will never have to be concerned with their syntax. However, if you are interested in a detailed description of their syntax, see the descriptions for V and I devices in the *Analog Devices* chapter of the the online *MicroSim PSpice A/D Reference Manual*.

Configuring Stimulus Files



In the schematic editor, Library and Include Files on the Analysis menu allows you to view the list of stimulus files pertaining to your current schematic, or to manually add, delete, or change the stimulus file configuration. The Stimulus Library Files list box displays all of the currently configured stimulus files. One file is specified per line. Files can be configured as either global to the Schematics environment or local to the current schematic. Global files are marked with an asterisk (*) after the file name.

When starting the Stimulus Editor from Schematics, stimulus files are automatically configured (added to the list) as local to the current schematic. Otherwise, new stimulus files can be added to the list by entering the file name in the File Name text box and then clicking on the Add Stimulus (local configuration) or Add Stimulus* (global configuration) button. All other commands work as described for model and include files in [Configuring Model Libraries on page 4-41](#).

Starting the Stimulus Editor

The Stimulus Editor is fully integrated with Schematics and can be run from either the schematic editor or symbol editor.

You can start the Stimulus Editor by the following methods:

- Double-click a stimulus instance
- Select one or more stimulus instances in the schematic and select Stimulus from the Edit menu.

- Select Edit Stimulus from the Analysis menu.

When you first start the Stimulus Editor, you may need to adjust the scale settings to fit the trace you are going to add. You can use Axis Settings on the Plot menu or the corresponding toolbar button to change the displayed data, the extent of the scrolling region, and the minimum resolution for each of the axes.

Displayed Data Range parameters determine what portion of the stimulus data set will be presented on the screen. Extent of Scrolling Region parameters set the absolute limits on the viewable range. Minimum Resolution parameters determine the smallest usable increment (example: if it is set to 1 msec, then you cannot add a data point at 1.5 msec).

Defining Stimuli

- 1 Place stimulus part instances from the symbol set: VSTIM, ISTIM, interface ports (IF_IN and INTERFACE), and DIGSTIM.
- 2 Double-click the source instance to start the Stimulus Editor. When you are asked whether you want to edit the named stimulus, click OK.
- 3 Fill in the transient specification according to the dialogs and prompts.

Piecewise linear stimuli can be specified by direct manipulation of the input waveform display.
- 4 Save the edits by selecting Save from the File menu.

Example: piecewise linear stimulus

- 1 Open an existing schematic or start a new one.
- 2 Select Get New Part from the Draw menu and either browse the `source.slb` Symbol Library file for VSTIM (and select it), or type VSTIM in the Part text box.
- 3 Place the symbol. It looks like a regular voltage source with a STIMULUS attribute displayed.
- 4 Double-click the STIMULUS label and type `Vfirst`. This names the stimulus that you are going to create.
- 5 If you are working in a new schematic, use Save As from the File menu to save it. This is necessary since the schematic name is used to create the default stimulus file name.
- 6 Double-click the VSTIM symbol. This starts the Stimulus Editor and displays the New Stimulus dialog box. You can see that the stimulus already has the name of Vfirst.
- 7 Select PWL in the dialog box and click OK. The cursor looks like a pencil. The message in the status bar at the bottom of the screen lets you know that you are in the process of adding new data points to the stimulus. The left end of the bottom status bar displays the current coordinates of the cursor.

- 8 Move the cursor to (200ns, 1) and click the left mouse button. This adds the point. Notice that there is automatically a point at (0,0). Ignore it for now and continue to add a couple more points to the right of the current one.
- 9 Click-right to stop adding points.
- 10 Select Save from the File menu.

If you make a mistake or want to make any changes, reshape the trace by dragging any of the handles to a new location. The dragged handle cannot pass any other defined data point.

To delete a point, click its handle and press **[Del]**.

To add additional points, either select Add Point from the Edit menu, press **[Alt]+[A]**, or click the Add Point toolbar button.

At this point you can return to Schematics, edit the current stimulus, or go on to create another.

Example: sine wave sweep

- 1 Open an existing schematic or start a new one.
- 2 Place a VSTIM symbol on your schematic.
- 3 To name the stimulus, double-click the STIMULUS attribute and type V_{sin} .
- 4 Double-click the VSTIM symbol to start the Stimulus Editor.
- 5 Define the stimulus parameter for amplitude:
 - a Select Cancel while in the New Stimulus dialog.
 - b Select Parameters from the Tools menu.
 - c Enter $AMP=1$ in the Definition text box, and click OK.
 - d Select New from the Stimulus menu or click the New Stimulus button in the toolbar.
 - e Give the stimulus the name of V_{sin} .
 - f Select SIN as the type of stimulus to be created, and click OK.
- 6 Define the other stimulus properties:
 - a Enter 0 for Offset Value.

This example creates a 10 K sin wave with the amplitude parameterized so that it can be swept during a simulation.

- b** Enter {AMP} for Amplitude. The curly braces are required. They indicate that the expression needs to be evaluated at simulation time.
 - c** Enter 10k for Frequency and click OK.
 - d** Select Save from the File menu.
- 7** Within Schematics, place and define the PARAM symbol:
 - a** Select Get New Part from the Draw menu.
 - b** Either browse special.slb for the PARAM symbol or type in the name.
 - c** Place the symbol on your schematic and double-click it to edit the attributes.
 - d** Set the value of the NAME1 attribute to AMP (no curly braces).
 - e** Set the value of the VALUE1 attribute to 1.
- 8** Set up the parametric sweep and other analyses:
 - a** Select Setup from the Analysis menu, and click the Parametric button.
 - b** Select Global Parameter in the Swept Var. Type frame.
 - c** Select Linear in the Sweep type frame.
- 9** Enter AMP in the Name text box.
- 10** Specify values for the Start Value, End Value, and Increment text boxes.

You can now set up your usual Transient, AC, or DC analysis and run the simulation.

Creating New Stimulus Symbols

- 1** Use the symbol editor to edit or create a symbol with the following attributes:
 - STIMTYPE type of stimulus with a value of ANALOG
 - STIMULUS name of the stimulus model

- 2 Select Stimulus from the Edit menu. Schematics searches the configured list of global stimulus files. If you are creating a new stimulus symbol and the stimulus is not found, you are prompted for the name of the stimulus file in which the new definition should be saved.
- 3 From within the Stimulus Editor, edit the transient specification as indicated by the dialogs and prompts, or by direct manipulation of the input waveform display for analog piecewise linear stimuli.
- 4 Select Save from the File menu.

Editing a Stimulus

To edit an existing stimulus

- 1 Start the Stimulus Editor and select Get from the Stimulus menu.
- 2 Double-click the trace name at the bottom of the X axis. This opens the Stimulus Attributes dialog box where you can modify the attributes of the stimulus directly and immediately see the effect of the changes.

PWL stimuli are a little different since they are a series of time/value pairs.

To edit a PWL stimulus

- 1 Double click the trace name. This displays the handles for each defined data point.
- 2 Click any handle to select it. To reshape the trace, drag it to a new location. To delete the data point, press **[Del]**.
- 3 To add additional data points, either select Add from the Edit menu or click the Add Point button.
- 4 Right-click to end adding new points.

This provides a fast way to scale a PWL stimulus.

To select a time and value scale factor for PWL stimuli

- 1 Select the PWL trace by clicking on its name.
- 2 Select Attributes from the Edit menu or click the corresponding toolbar button.

Deleting and Removing Traces

To delete a trace from the displayed screen, select the trace name by clicking on its name, then press **[Del]**. This will only erase the display of the trace, not delete it from your file. The trace is still available by selecting Get from the Stimulus menu.

To remove a trace from a file, select Remove from the Stimulus menu.

Note *Once a trace is removed, it is no longer retrievable. Delete traces with caution.*

Manual Stimulus Configuration

Stimuli can be characterized by manually starting the Stimulus Editor and saving their specifications to a file. These stimulus specifications can then be associated to stimulus instances in your schematic or to stimulus symbols in the symbol library.

To manually configure a stimulus

- 1** Start the Stimulus Editor by double-clicking on the Stimulus Editor icon in the MicroSim program group.
- 2** Open a stimulus file by selecting Open from the File menu. If the file is not found in your current library search path, you are prompted for a new file name.
- 3** Create one or more stimuli to be used in your schematic. For each stimulus:
 - a** Name it whatever you want. This name will be used to associate the stimulus specification to the stimulus instance in your schematic, or to the symbol in the symbol library.
 - b** Provide the transient specification.
 - c** Select Save from the File menu.

- 4** In the schematic editor, configure the Stimulus Editor's output file into your schematic:
 - a** Select Library and Include Files from the Analysis menu.
 - b** Enter the file name specified in step [2](#).
 - c** If the stimulus specifications are for local use in the current schematic, click the Add Stimulus (or Add Include) button. For global use by a symbol in the Symbol Library or by any schematic, use Add Stimulus* (or Add Include*) instead.
 - d** Click OK.
- 5** Modify either the stimulus instances in the schematic or symbols in the symbol library to reference the new stimulus specification.
- 6** Associate the transient stimulus specification to a stimulus instance:
 - a** Place a stimulus part in your schematic from the symbol set: VSTIM, ISTIM, and DIGSTIM.
 - b** Click the VSTIM, ISTIM, or DIGSTIM instance.
 - c** Select Attributes from the Edit menu.
 - d** Click the STIMULUS= attribute, type in the name of the stimulus, and click Save Attr.
 - e** Complete specification of any VSTIM or ISTIM instances by selecting Attributes from the Edit menu and editing their DC and AC attributes.

Click the DC= attribute, type its value in the Value text box, and then click Save Attr.

Click the AC= attribute, type its value in the Value text box, and then click Save Attr.
 - f** Click OK to return to the schematic.

- 7 To change stimulus references globally for a symbol:
- a Select Edit Library from the File menu to start the symbol editor.
 - b Create or change a symbol definition, making sure to define the following attributes:

PART	symbol name (it is good practice to have the symbol name match the STIMULUS name)
STIMULUS	stimulus name as defined in the Stimulus Editor

See [Chapter 5, Creating Symbols for Models](#), for a description of how to create and edit symbols.

Transient (Time) Response

The Transient response analysis causes the response of the circuit to be calculated from $TIME = 0$ to a specified time. A transient analysis specification is shown for the circuit `example.sch` in Figure 10-2. (`example.sch` is shown in [Figure 10-3](#).) The analysis is to span the time interval from 0 to 1000 nanoseconds and values should be reported to the simulation output file every 20 nanoseconds.

During a transient analysis, any or all of the independent sources may have time-varying values. In `example.sch`, the only source which has a time-varying value is V1 (VSIN part) with attributes:

```
VOFF = 0v
VAMPL = 0.1v
FREQ = 5Meg
```

V1's value varies as a 5 MHz sine wave with an offset voltage of 0 volts and a peak amplitude of 0.1 volts. In general, more than one source has time-varying values.

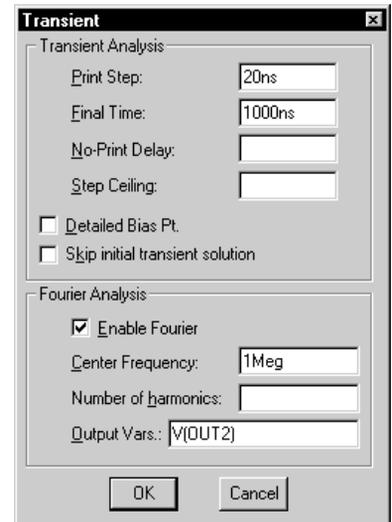
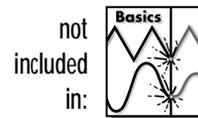


Figure 10-2 Transient Analysis Setup for `example.sch`

The example circuit `example.sch` is provided with the MicroSim program installation.

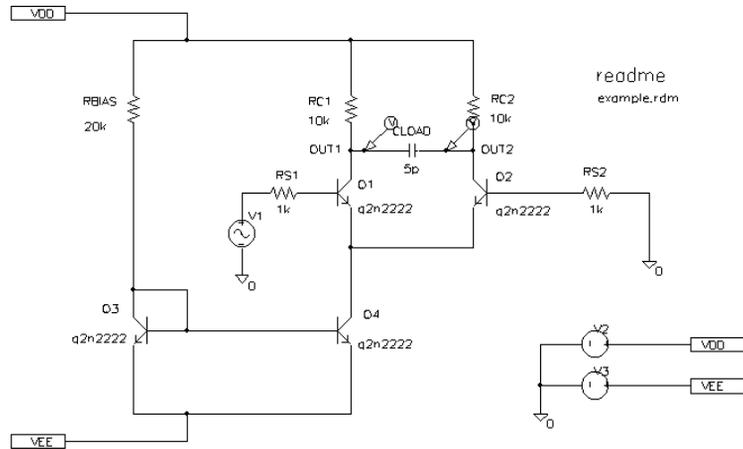


Figure 10-3 Example Schematic `example.sch`

The transient analysis does its own calculation of a bias point to start with, using the same technique as described for DC sweep. This is necessary because the initial values of the sources can be different from their DC values. If you want to report the small-signal parameters for the transient bias point, you should use the Transient command and enable Detailed Bias Point. Otherwise, if all you want is the result of the transient run itself, you should only enable the Transient command.

In the simulation output file `example.out`, the bias-point report for the transient bias point is labeled INITIAL TRANSIENT SOLUTION.

Internal Time Steps in Transient Analyses

During analog analysis, PSpice maintains an internal time step which is continuously adjusted to maintain accuracy while not performing unnecessary steps. During periods of inactivity, the internal time step is increased. During active regions, it is decreased. The maximum internal step size can be controlled by specifying so in the Step Ceiling text box in the Transient dialog. PSpice will never exceed either the step ceiling value or two percent of the total transient run time, whichever is less.

The internal time steps used may not correspond to the time steps at which information has been requested to be reported. The values at the print time steps are obtained by 2nd-order polynomial interpolation from values at the internal steps.

Switching Circuits in Transient Analyses

Running transient analysis on switching circuits can lead to long run times. PSpice must keep the internal time step short compared to the switching period, but the circuit's response extends over many switching cycles.

One method of avoiding this problem is to transform the switching circuit into an equivalent circuit without switching. The equivalent circuit represents a sort of quasi steady-state of the actual circuit and can correctly model the actual circuit's response as long as the inputs do not change too fast.

This technique is described in:
V. Bello, "Computer Program Adds SPICE to Switching-Regulator Analysis," *Electronic Design*, March 5, 1981.

Plotting Hysteresis Curves

Transient analysis can be used to look at a circuit's hysteresis. Consider, for instance, the circuit shown in Figure 10-4 (netlist in Figure 10-5).

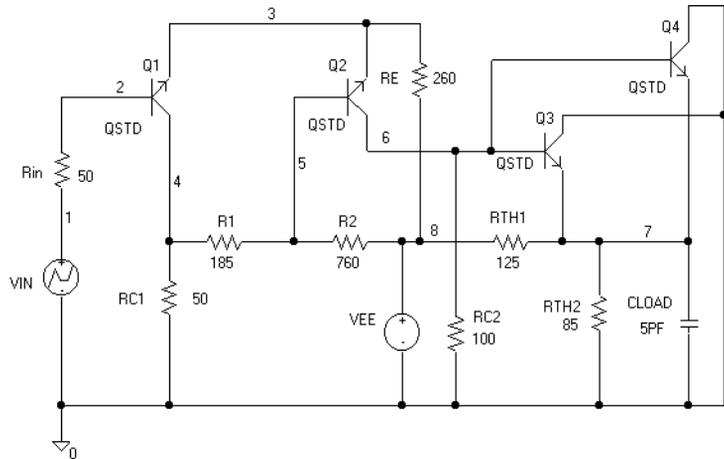


Figure 10-4 ECL Compatible Schmitt Trigger

* Schematics Netlist

```

R_RIN      1 2 50
R_RC1     0 3 50
R_R1      3 5 185
R_R2      5 8 760
R_RC2     0 6 100
R_RE      4 8 260
R_RTH2    7 0 85
C_CLOAD   0 7 5PF
V_VEE     8 0 dc -5
V_VIN     1 0
+PWL 0 -8 1MS -1.0V 2MS -1.8V
R_RTH1    8 7 125
Q_Q1      3 2 4 QSTD
Q_Q2      6 5 4 QSTD
Q_Q3      0 6 7 QSTD
Q_Q4      0 6 7 QSTD

```

Figure 10-5 Netlist for Schmitt Trigger Circuit

The QSTD model is defined as:

```
.MODEL QSTD NPN( is=1e-16 bf=50 br=0.1 rb=50
+ rc=10 tf=.12ns tr=5ns cje=.4pF pe=.8 me=.4
+ cjc=.5pF pc=.8 mc=.333 ccs=1pF va=50)
```

Instead of using the DC sweep to look at the hysteresis, we use the transient analysis, (Print Step = .01ms and Final Time = 2ms) sweeping VIN from -1.8 volts to -1.0 volts and back down to -1.8 volts, very slowly. This has two advantages:

- it avoids convergence problems
- it covers both the upward and downward transitions in one analysis

After the simulation, when we are in Probe, the X axis variable is initially set to be Time. By selecting X Axis Settings from the Plot menu and clicking on the Axis Variable button, we can set the X axis variable to be V(1). Then we can use Add on the Trace menu to display V(7), and change the X axis to a user defined data range from -1.8V to -1.0V (X Axis Settings on the Plot menu). This plots the output of the Schmitt trigger against its input, which is what we want. The result looks similar to Figure 10-6.

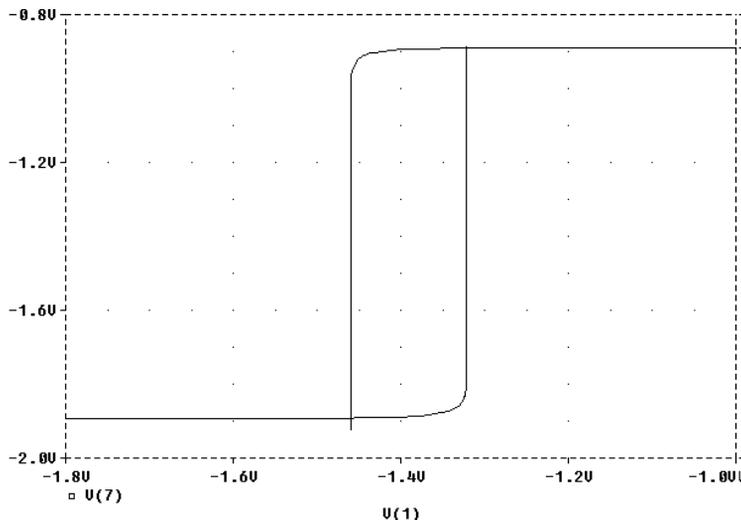


Figure 10-6 Hysteresis Curve Example: Schmitt Trigger

Fourier Components

Fourier analysis is enabled through the transient analysis setup dialog box. Fourier analysis calculates the DC and Fourier components of the result of a transient analysis. By default, the 1st through 9th components are computed, however, more can be specified.

You must do a transient analysis in order to do a Fourier analysis. The sampling interval used during the Fourier transform is equal to the print step specified for the transient analysis.

When selecting Fourier to run a harmonic decomposition analysis on a transient waveform, only a portion of the waveform is used. Using Probe, a Fast Fourier Transform (FFT) of the complete waveform can be calculated and its spectrum displayed.

In the example Fourier analysis specification shown in [Figure 10-2 on page 10-15](#), the voltage waveform at node OUT2 from the transient analysis is to be used and the fundamental frequency is to be 1 megahertz for the harmonic decomposition. The period of fundamental frequency is 1 microsecond (inverse of the fundamental frequency). Only the last 1 microsecond of the transient analysis is used, and that portion is assumed to repeat indefinitely. Since V1's sine wave does indeed repeat every 1 microsecond, this is sufficient. In general, however, you must make sure that the fundamental Fourier period fits the waveform in the transient analysis.

Parametric and Temperature Analysis

11

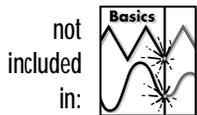
Chapter Overview

This chapter describes how to set up parametric and temperature analyses. Parametric and temperature are both simple multi-run analysis types.

This chapter includes the following sections:

[Parametric Analysis on page 11-2](#)

[Temperature Analysis on page 11-11](#)



Parametric Analysis

Minimum Requirements to Run a Parametric Analysis

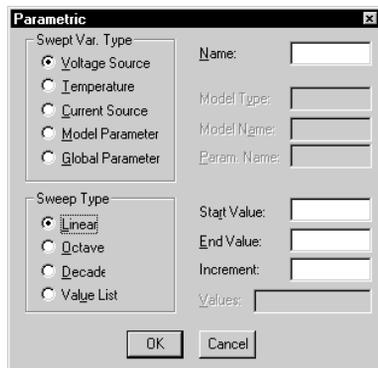
Minimum circuit design requirements

- Set up the circuit according to the swept variable type as listed in [Table 11-1](#).
- Set up a DC sweep, AC sweep, or transient analysis.

Table 11-1 *Parametric Analysis Circuit Design Requirements*

Swept Variable Type	Requirement
voltage source	voltage source with a DC specification (VDC, for example)
temperature	none
current source	current source with a DC specification (IDC, for example)
model parameter	PSpice model
global parameter	global parameter defined with a parameter block (PARAM)

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.



Minimum program setup requirements

- In the Analysis Setup dialog box, click the Parametric button. Complete the Parametric dialog box as needed.
- If needed, in the Analysis Setup dialog box, select (✓) the Parametric check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

Note Do not specify a DC sweep and a parametric analysis for the same variable.

Overview of Parametric Analysis

Parametric analysis performs multiple iterations of a specified standard analysis while varying a global parameter, model parameter, component value, or operational temperature. The effect is the same as running the circuit several times, once for each value of the swept variable.

See [Parametric Analysis on page 2-24](#) for a description of how to set up a parametric analysis.

Example: RLC Filter

This example shows how to perform a parametric sweep and how to analyze the results with performance analysis.

With performance analysis, values can be derived from a series of simulator runs and plotted versus a parameter that varies between those runs. For this example, the derived values we wish to plot are the overshoot and the rise time versus the damping resistance of the filter. Deriving these values by hand is quite involved and letting the simulator do the work for us is an appealing alternative.

Entering the schematic

The schematic for the RLC filter (`rlcfilt.sch`) is shown in Figure 11-1.

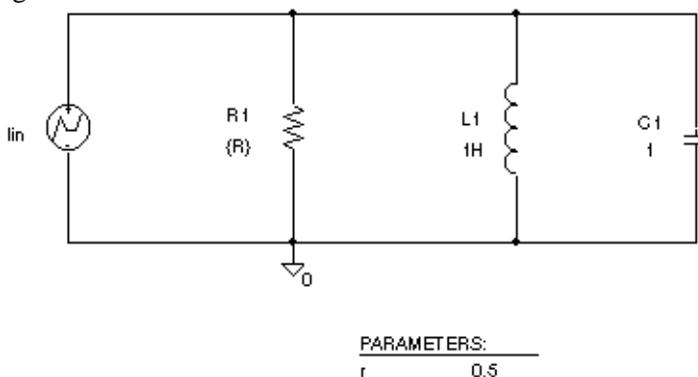


Figure 11-1 *Passive Filter Schematic*

This series of PSpice runs varies the value of resistor R1 from 0.5 to 1.5 ohms in 0.1 ohm steps. Since the time-constant of the circuit is about one second, we perform a transient analysis of approximately 20 seconds.

Create the circuit in MicroSim Schematics by placing a piecewise linear independent current source (IPWL from `source.slb`). Set the current source attributes as follows:

```
AC = 1a
T1 = 0s
I1 = 0a
T2 = 10ms
I2 = 0a
T3 = 10.1ms
I3 = 1a
```

Place an instance of a resistor and set its VALUE attribute to the expression, {R}. To define R as a global parameter, place a PARAM pseudocomponent defining its NAME1 attribute to R and VALUE1 attribute to 0.5. Place an inductor and set its value to 1H, place a capacitor and set its value to 1, and place an analog ground symbol (AGND from `port.slb`). Wire the schematic symbols together as shown in Figure 11-1.

Running the simulation

Run PSpice with the following analyses enabled:

transient	print step:	100ms
	final time:	20s
parametric	swept var. type:	global parameter
	sweep type:	linear
	name:	R
	start value:	0.5
	end value:	1.5
	increment:	0.1



or press **F11**

After setting up the analyses, start the simulation by selecting Simulate from the Analysis menu.

Using performance analysis to plot overshoot and rise time

After performing the PSpice simulation that creates the data file called `rlcfilt.dat`, you can run Probe to compute the specified performance analysis goal functions.

When Probe is started, you are presented with a list of all the sections or runs in the Probe data file produced by PSpice. To use the data from every run, select All and click OK in the Available Selections dialog box. In the case of Figure 11-2, the trace I(L1) from the ninth section was added by specifying the following in the Add Traces dialog box:

I(L1)@9

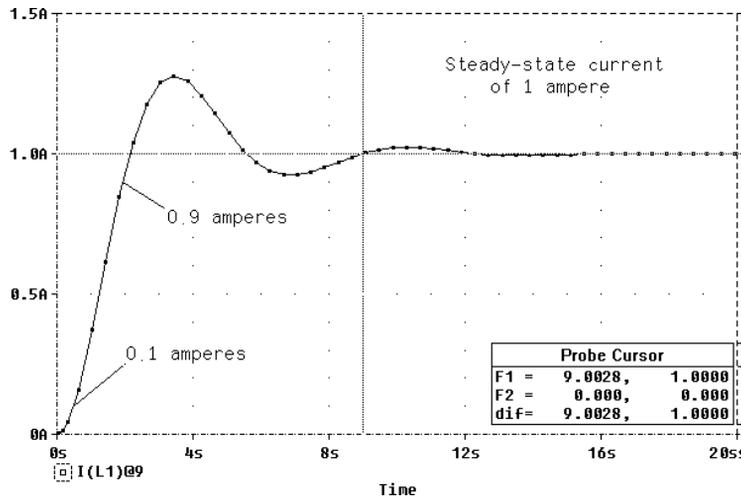


Figure 11-2 Current of L1 when R1 is 1.5 Ohms

To run performance analysis:

- 1 Select X Axis Setting from the Plot menu in Probe.
- 2 Select (✓) the Performance Analysis check box to enable performance analysis
- 3 Click OK.

Probe resets the X axis variable for the graph to be the parameter that changed between PSpice runs. In the example, this is the R parameter.

To see the rise time for the current through the inductor L1, select the Add from the Trace menu and then enter:

To access the Add Traces dialog box, in Probe, from the Add menu, select Traces.

Troubleshooting tip

More than one PSpice run or data section is required for performance analysis because one data value is derived for each waveform in a related set of waveforms. A trace of the derived values cannot be displayed if there is only one run or data section because at least two data points are required to produce a trace.

However, you can use Eval Goal Functions on the Trace menu in Probe to evaluate a goal function on a single waveform, thus producing a single data point result.

The genrise and overshoot goal functions are contained in the file `msim.prb` in the MSIM directory.

```
genrise( I(L1) )
```

In Figure 11-3, we can see how the rise time decreases as the damping resistance increases for the filter.

Another Y axis can be added to the plot for the overshoot of the current through L1 by selecting Add Y Axis from the Plot menu. The Y axis is immediately added. We now select Add from the Trace menu and enter:

```
overshoot( I(L1) )
```

Figure 11-3 shows how the overshoot increases with increasing resistance.

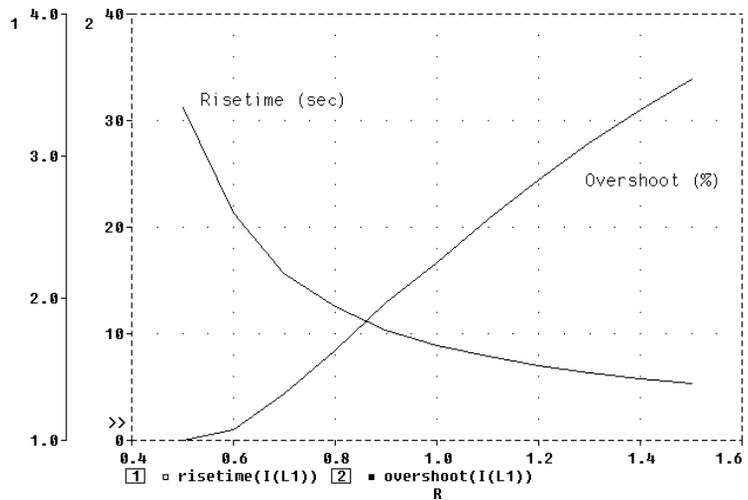


Figure 11-3 Rise Time and Overshoot vs. Damping Resistance

Now we can use the multiple X axes feature to view the original waveform family for inductor L1 current along with the derived rise time and overshoot data. We must first add a new plot by selecting Add Plot from the Plot menu. To set this plot's X axis to a unique scale, select Unsync Plot from the Plot menu. You'll notice that the new plot's X axis is now labeled with range and variable information. However, it is still set for Performance Analysis (with resistance R as the X axis label). We can toggle off the Performance Analysis feature by selecting X Axis Settings from the Plot menu and clearing the Performance Analysis check box. This affects only the current plot. (The plot marked with SEL>> is the current plot.)

Now we can add the trace for I(L1) as we've done before (Add on the Trace menu), changing the Y axis range to 0A - 1.5A (Y Axis Settings on the Plot menu), and the X axis range to 0s - 20s (X Axis Settings on the Plot menu). This produces the display shown in Figure 11-4.

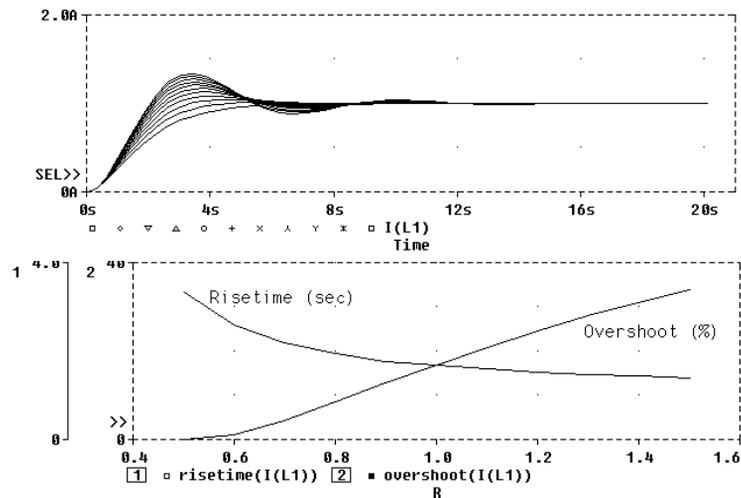


Figure 11-4 Inductor Waveform Data Viewed with Derived Rise Time and Overshoot Data

Example: Frequency Response vs. Arbitrary Parameter

A common request is to view a plot of the linear response of a circuit at a specific frequency as one of the circuit parameters varies (such as the output of a band pass filter at its center frequency vs. an inductor value). In this example, the value of a nonlinear capacitance is measured using a 10 kHz AC signal and plotted vs. its bias voltage. The capacitance is in parallel with a resistor, so a Probe expression is used to calculate the capacitance from the complex admittance of the R-C pair.

This technique for measuring branch capacitances works well in both simple and complex circuits.

Setting up the circuit

Enter the circuit in Schematics as shown in Figure 11-5

To create the capacitor model in the schematic editor:

- 1 Place a Cbreak symbol.
- 2 Select it so that it is highlighted.
- 3 Select Model from the Edit menu.
- 4 Select Edit Instance Model Text. Enter the following:


```
.model Cnln CAP(C=1 VC1=-0.01 VC2=0.05)
```

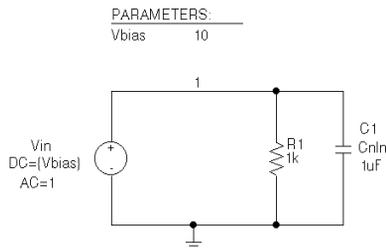


Figure 11-5 RLC Filter Example Circuit

Set up the circuit for a parametric AC analysis (sweep Vbias), and run PSpice. Include only the frequency of interest in the AC sweep.

Displaying results in Probe

Use Probe to display the capacitance calculated at the frequency of interest vs. the stepped parameter.

After analyzing the circuit with PSpice:

- 1 Run Probe.
- 2 Load all AC analysis sections.
- 3 Select Add from the Trace menu.
- 4 Add the following trace expression:

$$\text{IMG}(-I(V_{in})/V(1,0)) / (2 * 3.1416 * \text{Frequency})$$

Or add the expression:

$$\text{CvF}(-I(V_{in})/V(1,0))$$

Where CvF is a macro which measures the effective capacitance in a complex conductance. Macros are defined using Macro on the Trace menu. The CvF macro should be defined as:

$$\text{CvF}(G) = \text{IMG}(G) / (2 * 3.1416 * \text{Frequency})$$

Note that $-I(V_{in})/V(1)$ is the complex admittance of the R-C branch; the minus sign is required for correct polarity.

To use performance analysis to plot capacitance vs. bias voltage

- 1 In Probe, select Performance Analysis from the Trace menu.
- 2 Click Wizard.
- 3 Click Next>.
- 4 Click YatX in the Choose a Goal Function list, and then click Next>.
- 5 In the Name of Trace text box, type the following:

$$\text{CvF}(-I(V_{in})/V(1))$$

- 6 In the X value text box, type 10K.
- 7 Click Next>. The wizard displays the gain trace for the first run to text the goal function (YatX).
- 8 Click Finish. The resultant Probe plot is shown in Figure 11-6.

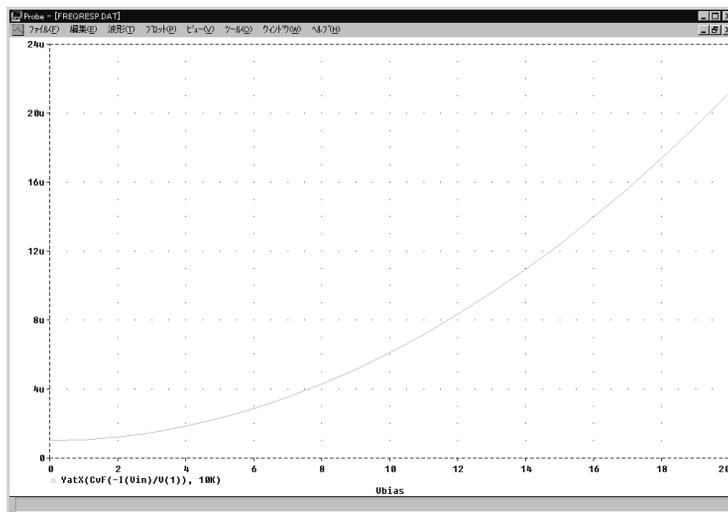


Figure 11-6 Probe Plot of Capacitance vs. Bias Voltage

Temperature Analysis

Minimum Requirements to Run a Temperature Analysis

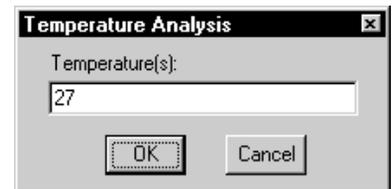
Minimum circuit design requirements

None.

Minimum program setup requirements

- In the Analysis Setup dialog box, click the Temperature button. Specify the temperature or list of temperatures in the Temperature Analysis dialog box.
- If needed, in the Analysis Setup dialog box, select (✓) the Temperature check box to enable it.
- Start the simulation as described in [Starting Simulation on page 7-10](#).

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.



Overview of Temperature Analysis

When a temperature analysis is run, PSpice reruns standard analyses enabled in the Analysis Setup dialog box at different temperatures.

Temperature analysis allows zero or more temperatures to be specified. If no temperature is specified, the circuit is run at 27°C. If more than one temperature is listed, the effect is the same as running the simulation several times, once for each temperature in the list.

Setting the temperature to a value other than the default results in recalculating the values of temperature-dependent devices. In `example.sch` (see Figure 11-7), the temperature for all of the analyses is set to 35°C. The values for resistors RC1 and RC2

Running multiple analyses for different temperatures can also be achieved using parametric analysis (see [Parametric Analysis on page 11-2](#)). With parametric analysis, the temperatures can be specified either by list, or by range and increments within the range.

are recomputed based upon the CRES model which has parameters TC1 and TC2 reflecting linear and quadratic temperature dependencies.

Likewise, the Q3 and Q4 device values are recomputed using the Q2N2222 model which also has temperature-dependent parameters. In the simulation output file, these recomputed device values are reported in the section labeled TEMPERATURE ADJUSTED VALUES.

The example circuit `example.sch` is provided with the MicroSim program installation.

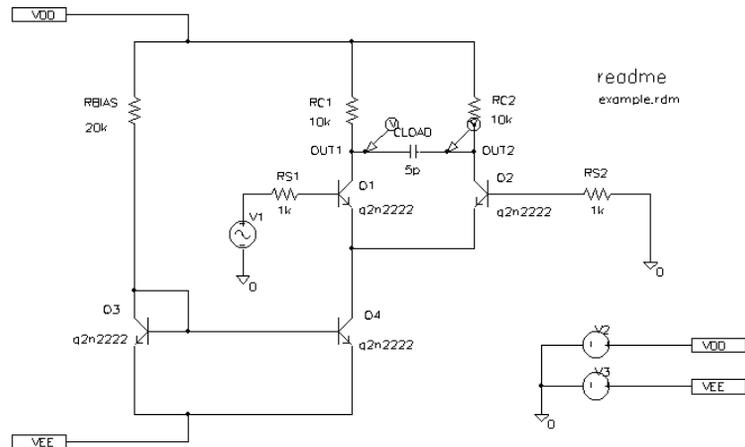


Figure 11-7 Example Schematic `example.sch`

Monte Carlo and Sensitivity/ Worst-Case Analyses

12

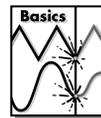
Chapter Overview

This chapter describes how to set up Monte Carlo and sensitivity/worst-case analyses and includes the following sections:

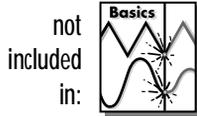
[Statistical Analyses on page 12-2](#)

[Monte Carlo Analysis on page 12-7](#)

[Worst-Case Analysis on page 12-25](#)



This entire chapter describes features that are not included in PSpice Basics.



Statistical Analyses

Monte Carlo and sensitivity/worst-case are statistical analyses. This section describes information common to both types of analyses.

See [Monte Carlo Analysis on page 12-7](#) for information specific to Monte Carlo analyses, and see [Worst-Case Analysis on page 12-25](#) for information specific to sensitivity/worst-case analyses.

Generating statistical results for Probe

As the number of Monte Carlo or worst-case runs increase, simulation takes longer and the Probe data file gets larger. Large Probe data files may be slow to open and slow to draw traces.

One way to avoid this problem is to set up an overnight batch job to run the simulation and execute Probe commands. You can even set up the batch job to produce a series of plots on paper which are ready for you in the morning.

Overview of Statistical Analyses

The Monte Carlo and worst-case analyses vary the lot or device tolerances of devices between multiple runs of an analysis (DC, AC, or transient). Before running the analysis, you must set up the model and/or lot tolerances of the model parameter to be investigated.

A Monte Carlo analysis causes a Monte Carlo (statistical) analysis of the circuit to be performed.

A worst-case analysis causes a sensitivity and worst-case analysis of the circuit to be performed.

Sensitivity/worst-case analyses are different from Monte Carlo analysis in that they compute the parameters using the sensitivity data rather than random numbers.

You can run either a Monte Carlo or a worst-case analysis, but you *cannot* run both at the same time. Multiple runs of the selected analysis are done while parameters are varied. You can select only one analysis type (AC, DC, or transient) per run. The analysis selected is repeated in subsequent passes of the analysis.

Output Control for Statistical Analyses

Monte Carlo and sensitivity/worst-case analyses can generate the following types of reports:

- model parameter values used for each run (that is, the values with tolerances applied)
- waveforms from each run, as a function of specifying data collection, or by specifying output variables in the analysis set up
- summary of all the runs using a collating function

Output is saved to the Probe data file for use by the Probe graphical waveform analyzer. For Monte Carlo analyses, Probe offers a special facility through the performance analysis feature to produce histograms of derived data.

For information about performance analysis, see [Example: RLC Filter on page 11-3](#).

For information about histograms, see [Creating histograms on page 12-20](#).

Model Parameter Values Reports

The List option in the MC Options section of the Monte Carlo or Worst Case dialog box produces a list of the model parameters actually used for each run.

This list is written to the simulation output file at the beginning of the run and contains the parameters for each device, as opposed to the parameters for each .MODEL statement. This is because devices can have different parameter values when using a model statement containing a DEV tolerance.

Note that for medium and large circuits, the List option can produce a large output file.

Waveform Reports

For Monte Carlo analyses, there are four variations of the output which can be specified in the Output section of the Monte Carlo or Worst Case dialog. These options are:

All	forces all output to be generated (including nominal run)
First*	generates output only during the first n runs
Every*	generates output for every n th run
Runs*	does specified analysis and generates outputs only for the listed runs (up to 25 values can be specified in the list)

In excess of about 10 runs, the Probe display tends to become more of a band than a set of individual waveforms. This can be useful for seeing the typical spread for a particular output variable. As the number of runs becomes larger, the spread more closely approximates the actual worst-case limits for the circuit.

The * indicates that you can specify runs in the Value text box.

Values for the output variables specified in the selected analyses are saved to the simulation output file and Probe data file. Note that even a modest number of runs can produce large output files.

Collating Functions

You may want to compress the results of Monte Carlo and worst-case analyses further. Using the collating function specified in the Function section of the Monte Carlo or Worst Case dialog box, each run can be represented by a single number. A table of deviations per run is reported in the simulation output file.

Collating functions are listed in [Table 12-1](#).

Table 12-1 *Collating Functions Used in Statistical Analyses*

Function	Description
YMAX	find the greatest difference in each waveform from the nominal
MAX	find the maximum value of each waveform

Table 12-1 *Collating Functions Used in Statistical Analyses*

Function	Description
MIN	find the minimum value of each waveform
RISE_EDGE	find the first occurrence of the waveform crossing above a specified threshold value
FALL_EDGE	find the first occurrence of the waveform crossing below a specified threshold value

Temperature Considerations in Statistical Analyses

Refer to *Temperature Effects on Monte Carlo Analysis* in the *Application Notes* manual for more information on this topic.

The statistical analyses perform multiple runs, as does the temperature analysis. Conceptually, the Monte Carlo and worst-case loops are inside the temperature loop. However, since both temperature and tolerances affect the model parameters, you can quickly get into detailed questions about how the two interact. Therefore, we recommend not enabling temperature analysis when using the Monte Carlo or worst-case analyses.

Also, it will not work to sweep the temperature in a DC sweep analysis while performing one of these statistical analyses, or to put tolerances on temperature coefficients. You will notice in `example.sch` how the temperature value is fixed at 35 °C.

The example circuit `example.sch` is provided with the MicroSim software installation.

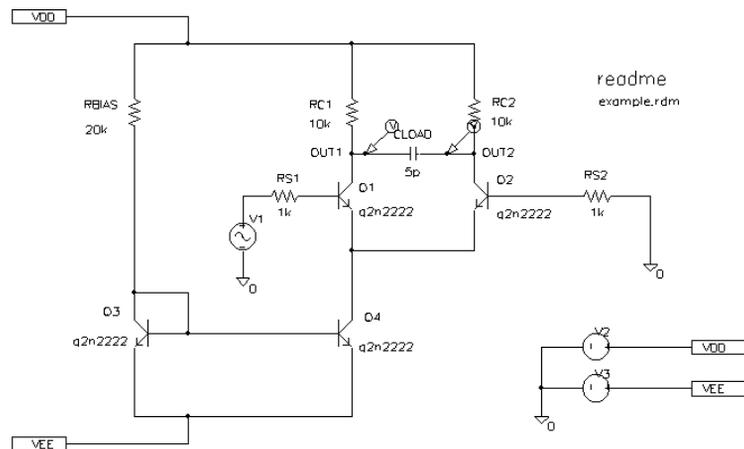


Figure 12-1 Example Schematic `example.sch`

Monte Carlo Analysis

The Monte Carlo analysis computes the circuit response to changes in component values by randomly varying all of the device model parameters for which a tolerance is specified. This provides statistical data on the impact of a device parameter's variance.

With Monte Carlo analysis model parameters are given tolerances, and multiple analyses (DC, AC, or transient) are run using these tolerances. A typical application of Monte Carlo analysis is predicting yields on production runs of a circuit.

For `example.sch` in Figure 12-1 on page 12-6, effects due to variances in resistors RC1 and RC2 values can be analyzed by assigning a model description to these resistors that includes a 5% device tolerance on the multiplier parameter R. Then, you can run a Monte Carlo analysis that runs a DC analysis first with the nominal R multiplier value for RC1 and RC2, then the specified number of additional runs with the R multiplier varied independently for RC1 and RC2 within 5% tolerance.

To modify `example.sch` and set up simulation

- 1 Replace RC1 and RC2 with RBREAK symbols, setting attribute values to match the resistors that are being replaced (VALUE=10k) and reference designators to match previous names.
- 2 Select Model from the Edit menu, then select Edit Instance Model. Create the model CRES as follows:

```
.MODEL CRES RES( R=1 DEV=5% TC1=0.02 TC2=0.0045 )
```

By default, MicroSim Schematics saves the definition to the model file `example.lib` and automatically configures the file for local use with the current schematic.

- 3 Click on the resistor instance. Select Model from the Edit menu, then select Change Model Reference to change the model reference to CRES.
- 4 Set up a new Monte Carlo analysis as shown in Figure 12-2. The analysis specification instructs PSpice to do one nominal run and four Monte Carlo runs, saving the DC analysis output from those five runs.

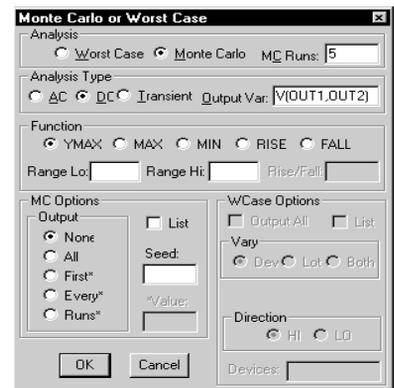
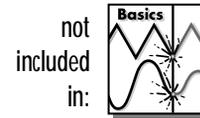


Figure 12-2 Monte Carlo Analysis Setup for `example.sch`

PSpice starts as usual by running *all* of the analyses enabled in the Analysis Setup dialog with all parameters set to their nominal values. However, with Monte Carlo enabled, the DC sweep analysis results are saved for later reference and comparison.

After the nominal analyses are finished, more of the specified analysis runs are performed (DC sweep in this example). Subsequent runs use the same analysis specification as the nominal with one major exception. **Instead of using the nominal parameter values, the tolerances are applied to set new parameter values and thus, new component values.**

The summary report generated in this example specifies that the waveform generated from V(OUT1, OUT2) should be the subject of the collating function YMAX. In each of the last four runs, the new V(OUT1, OUT2) waveform is compared to the nominal V(OUT1, OUT2) waveform for the first run, calculating the maximum deviation in the Y direction (YMAX collating function). The deviations are printed in order of size along with their run number (see Figure 12-3).

```

****      SORTED DEVIATIONS OF V(OUT1,OUT2) TEMPERATURE = 35.000 DEG C
              MONTE CARLO SUMMARY
*****

Mean Deviation =  -.2477
Sigma           =  .3035

  RUN              MAX DEVIATION FROM NOMINAL
Pass   3              .5729 (1.89 sigma) lower at U_U1 =  -.02
              ( 94.885% of Nominal)
Pass   4              .3549 (1.17 sigma) lower at U_U1 =  -.02
              ( 96.832% of Nominal)
Pass   2              .3122 (1.03 sigma) lower at U_U1 =  -.02
              ( 97.212% of Nominal)
Pass   5              .2493 ( .82 sigma) higher at U_U1 =  -.02
              ( 102.23% of Nominal)

```

Figure 12-3 Summary of Monte Carlo Runs for example.sch

With the List option enabled, a report is also generated showing the parameter value used for each device in each run. In this case (see Figure 12-4), run 3 exhibits the highest deviation.

```
* C:\MSI\EX\EXAMPLE.SCH

****   UPDATED MODEL PARAMETERS   TEMPERATURE = 35.000 DEG C
      MONTE CARLO PASS 3
*****

**** CURRENT MODEL PARAMETERS FOR DEVICES REFERENCING cres
      R_RC1      R_RC2
R      1.0304E+00  9.5053E-01
```

Figure 12-4 *Parameter Values for Monte Carlo Pass 3*

There is a trade-off in choosing the number of Monte Carlo runs. More runs provide better statistics, but take proportionally more computer time. The amount of computer time scales directly with the number of runs: 20 transient analyses take 20 times as long as one transient analysis. During Monte Carlo runs, the PSpice status display includes a line showing the run number and the total number of runs to be done. This gives an idea of how far the program has progressed.

Probe offers a facility to generate histograms of data derived from Monte Carlo waveform families through the performance analysis feature.

For information about performance analysis, see [Example: RLC Filter on page 11-3](#).

For information about histograms, see [Creating histograms on page 12-20](#).

Tutorial: Monte Carlo Analysis of a Pressure Sensor

In this tutorial, you will see how the performance of a pressure sensor circuit with a pressure-dependent resistor bridge is affected by manufacturing tolerances. You will use Monte Carlo analysis features provided in Schematics and PSpice to explore these effects.

Drawing the schematic

To begin, construct the bridge as shown in the schematic in Figure 12-5.

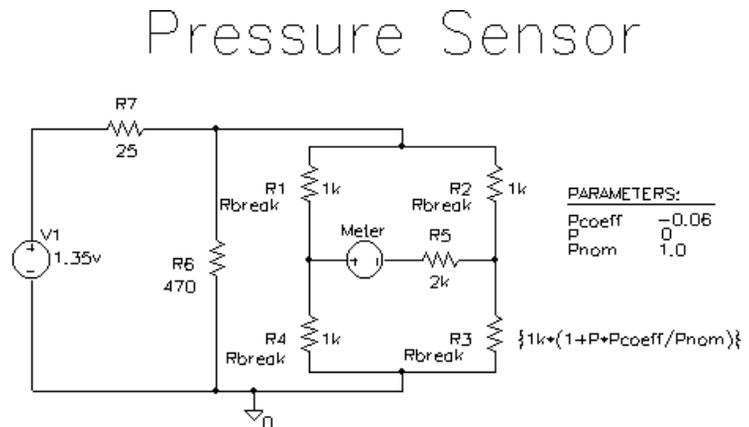


Figure 12-5 Pressure Sensor Circuit

Here are a few things you should know when placing and connecting the symbols:

 or press **Ctrl+G**

- To get the symbol you want to place, select Get New Part from the Draw menu, and enter the symbol name.
- To rotate a symbol before placing it, press **Ctrl+R**.
- For V1 and Meter, place a generic voltage source using the VSRC symbol. When you place the source for the meter, change its name by double-clicking the symbol's reference designator and typing `Meter` in the Edit Reference Designator dialog box.
- For R1-R7, place a resistor using the R symbol.

- Place the analog ground using the AGND symbol.
- To connect the symbols, use Wire from the Draw menu.
- To move values and/or reference designators, click the value or reference designator to select it, then drag it to the new location.



or press **Ctrl+W**

Defining component values

Define the component values as shown in Figure 12-5. For the pressure sensor, you need to do the following:

- Change the resistor values for R3, R5, R6, and R7 from their 1K default.
- Set the DC value for the V1 voltage source.

To change resistor values

- 1 Double-click the value label for a resistor symbol.
- 2 Type the new value. Depending on the resistor you are changing, set its value to one of the following (refer to Figure 12-5).

If you are changing this resistor...	Type this...
R3	$\{1k * (1 + P * Pcoeff / Pnom)\}$
R5	2k
R6	470
R7	25

- 3 Repeat steps [1-2](#) for each resistor symbol in your schematic.

To set the DC value for the V1 source and make it visible

- 1 Double-click the V1 source symbol.
- 2 In the Edit Attributes dialog box, double-click DC=.
- 3 In the Value text box, type 1.35v.

Note Because the Meter source is used to measure current, it has no DC value and can be left unchanged.

Note The value for R3— $\{1k * (1 + P * Pcoeff / Pnom)\}$ —is an expression that represents linear dependence of resistance on pressure. To complete the definition for R3, you will create and define global parameters for Pcoeff, P, and Pnom later on in this tutorial.

- press  **4** Click Save Attr to accept the changes.
- 5** Click Change Display.
- 6** In the What to Display frame, choose the Value Only option to make the DC value (1 . 35v) visible on the schematic.
- 7** Click OK to accept the change, then click OK again to exit the attributes dialog box.

Setting up the parameters

To complete the value specification for R3, define the global parameters: Pcoeff, P, and Pnom.

To define and initialize Pcoeff, P, and Pnom

- 1 Place a PARAM symbol on the schematic.
- 2 Double-click the PARAM symbol.
- 3 For each parameter:
 - a Double-click the NAME*n* or VALUE*n* attribute to specify the parameter name and corresponding value as follows.

For this attribute...	Type this...
NAME1	Pcoeff
VALUE1	-0.06
NAME2	P
VALUE2	0
NAME3	Pnom
VALUE3	1.0

- press  **b** Click Save Attr.
- 4** Click OK.

Using resistors with models

To explore the effects of manufacturing tolerances on the behavior of this circuit, you will set device (DEV) and (LOT) tolerances on the model parameters for resistors R1, R2, R3, and R4 in a later step (see page [12-14](#)). This means you need to use resistor symbols that have model associations.

Because R symbols do not have an associated model (and therefore no model parameters), change the resistor symbols to Rbreak symbols which do have a model.

When PSpice runs a Monte Carlo analysis, it uses tolerance values to determine how to vary model parameters during the simulation.

To replace R1, R2, R3, and R4 with the RBREAK symbol

- 1 Click R1 to select it.
- 2 +click R2, R3, and R4 to add them to the selection set.
- 3 From the Edit menu, select Replace.
- 4 In the Replacement text box, type Rbreak.
- 5 Choose the Selected Parts Only option.
- 6 Click OK.

Notice that the reference designator changes to the next consecutive set of identifiers.

- 7 To change each resistor's reference designator back to its original identifier, do the following for each resistor that you just replaced:
 - a Double-click the resistor's reference designator label.
 - b Type in its original identifier as shown in Figure 12-5 on page 12-10.
 - c Click OK.

Saving the schematic

Before editing the models for the Rbreak resistors, save the schematic.

To save the schematic

- 1 From the File menu, select Save As.
- 2 In the File Name text box, type `psensor.sch`.
- 3 Click OK.

Defining tolerances for the resistor models

Assign device (DEV) and lot (LOT) tolerances to the model parameters for resistors R1, R2, R3, and R4 using the model editor.

To assign 2% device and 10% lot tolerances to the resistance multiplier for R1

- 1 Select R1.
- 2 From the Edit menu, select Model.

The Edit Model dialog box appears with these choices:

- **Change Model Reference**, which lets you change the model association for the symbol instance; if you had a model named MyModel, you could change R1 to reference MyModel.
 - **Edit Instance Model (Text)**, which starts the model editor and loads a *copy* of the model definition; for this tutorial, this means the PSpice command syntax for the .MODEL Rbreak definition.
 - **Edit Instance Model (Parts)**, which starts the Parts utility and loads the device information for the corresponding model; this selection lets you change the behavioral properties of the device, but not device and lot tolerances.
- 3 Click Edit Instance Model (Text).

Schematics searches the libraries for the Rbreak model definition, makes a copy to create an instance model, and

The model editor lets you change the .MODEL or .SUBCKT syntax for a model definition. To find out more about the model editor, see [Using the Model Editor on page 4-29](#).

names it *<old model name>-X*, which in this tutorial, is Rbreak-X. In the model editor, you can change this name to whatever you want.

- 4 To change the instance model name from Rbreak-X to Rmonte1, do the following:
 - a In the model editor window, double-click Rbreak-X in the `.model Rbreak-X RES` line.
 - b Type RMonte1.
- 5 To add a 2% device tolerance and a 10% lot tolerance to the resistance multiplier, do the following:
 - a Change the R=1 line following the `.MODEL` statement to:

```
R=1 DEV=2% LOT=10%
```

The model editing window should look something like Figure 12-6.

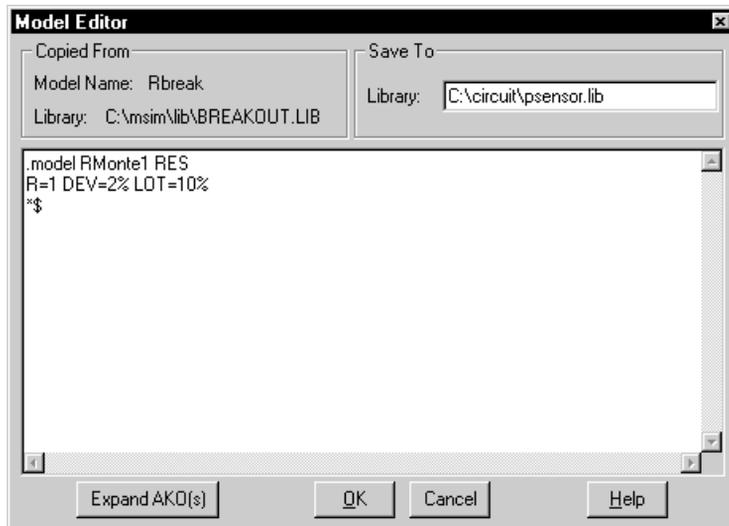


Figure 12-6 Model Definition for RMonte1

- 6 Click OK to accept the changes.

By default, Schematics saves the RMonte1 `.MODEL` definition to the `schematic_name.lib` library, which is `psensor.lib`. Schematics also automatically configures the library for local use.

To find out more about adding model libraries to the configuration, see [Configuring Model Libraries on page 4-41](#).

To have resistors R2 and R4 use the same tolerances as R1

- 1 Select R2.
- 2 From the Edit menu, select Model.
- 3 Click Change Model Reference.
- 4 Type `RMonte1`.
- 5 Repeat steps **1-4** for R4.

To assign 5% device tolerance to the resistance multiplier for R3

- 1 Select R3.
- 2 From the Edit menu, select Model.
- 3 Click Edit Instance Model (Text).
- 4 Change the instance model name in the `.MODEL` statement to `RTherm`.
- 5 Change the `R=1` line following the `.MODEL` statement to:
`R=1 DEV=5%`
- 6 Click OK to accept the changes.

Your schematic should look like Figure 12-7.

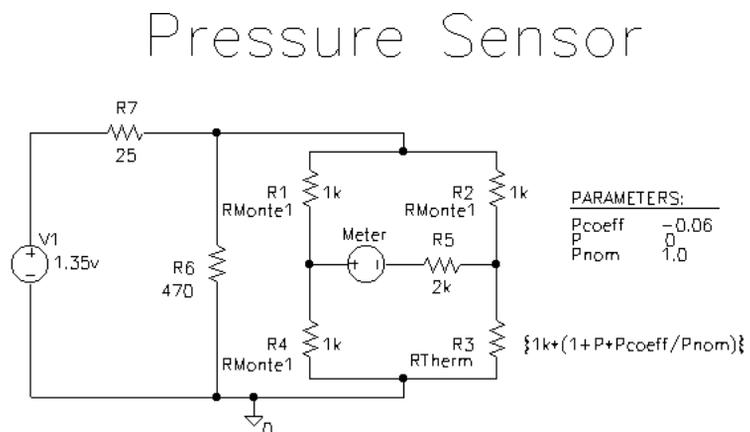


Figure 12-7 Pressure Sensor Circuit with `RMonte1` and `RTherm` Model Definitions

Setting up the analyses

Define and enable a DC analysis that sweeps the pressure value, and a Monte Carlo analysis that runs the DC sweep with each change to the resistance multipliers.

To set up the DC sweep

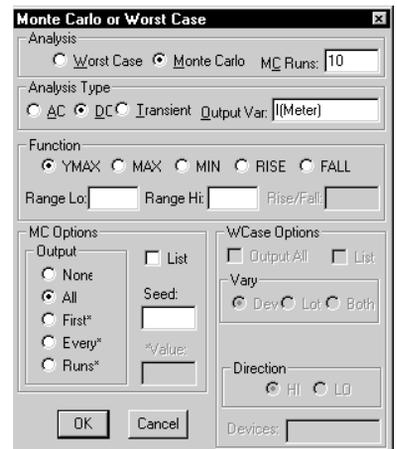
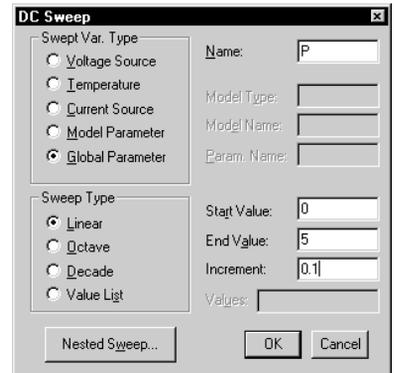
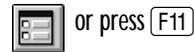
- 1 From the Analysis menu, select Setup.
- 2 Click DC Sweep.
- 3 In the Swept Var. Type frame, select Global Parameter.
- 4 Type values in the relevant text boxes as follows.

In this text box...	Type this...
Name	P
Start Value	0
End Value	5.0
Increment	0.1

- 5 Click OK.

To set up the Monte Carlo analysis

- 1 In the Analysis Setup dialog box, click Monte Carlo/Worst Case.
- 2 In the Analysis frame, do the following:
 - a Choose the Monte Carlo option.
 - b In the MC Runs text box, type 10.
- 3 In the Analysis Type frame, do the following:
 - a Choose the DC option.
 - b In the Output Var text box, type I(Meter).
- 4 In the MC Options frame, choose the All option.
- 5 Click OK.



To verify that the DC sweep and Monte Carlo analyses are enabled

- 1 In the Analysis Setup dialog box, make sure that the check box next to the DC Sweep and to the Monte Carlo/Worst Case buttons are selected (✓). If not, click them to enable the analyses.
- 2 Click OK.

Running the analysis and viewing the results

Run the simulation and analyze the results in Probe.

To complete setup, simulate, and view results

- 1 In Schematics, from the Analysis menu, select Probe setup.
 - a On the Probe Startup tab, make sure the Automatically Run Probe after Simulation option and the Show All Markers options are chosen.



or press **F11**

- 2 From the Analysis menu, select Simulate.

When complete, Probe automatically starts. Because PSpice ran a Monte Carlo analysis, PSpice saved multiple runs or sections of data. These are listed in the Available Sections dialog box.

- 3 In Probe, in the Available Sections dialog box, click All, and then OK.
- 4 To display current through the Meter voltage source, do the following:

- a In Schematics, from the Markers menu, select Mark Current into Pin.
- b Place a current marker on the left-hand pin of the Meter source.

- 5 Return to the Probe window to see the family of curves for I(Meter) as a function of P.

Note For more on analyzing Monte Carlo results in Probe, see the next section on Monte Carlo histograms.



Another way to view the family of curves without using schematic markers is as follows:

- 1 In Probe, from the Trace menu, select Add.
- 2 In the Simulation Output Variables list, double-click I(Meter).

Monte Carlo Histograms

A typical application of Monte Carlo analysis is predicting yields on production runs of a circuit. Probe can be used to display data derived from Monte Carlo waveform families as histograms, part of Probe's performance analysis feature.

To illustrate this feature, we will simulate a fourth order Chebyshev active filter, running a series of 100 AC analyses while randomly varying resistor and capacitor values for each run. Then, having defined performance analysis goal functions for bandwidth and center frequency, we will observe the statistical distribution of these quantities for the 100 runs.

For information about performance analysis, see [Example: RLC Filter on page 11-3](#).

Chebyshev filter example

The Chebyshev filter is designed to have a 10 kHz center frequency and a 1.5 kHz bandwidth. The schematic for the filter is shown in Figure 12-8. The stimulus specifications for V1, V2, and V3 are:

```
V1:  DC=-15
V2:  DC=+15
V3:  AC=1
```

The components were rounded to the nearest available 1% resistor and 5% capacitor value. In our analysis, we are concerned with how the bandwidth and the center frequency vary when 1% resistors and 5% capacitors are used in the circuit.

Creating models for Monte Carlo analysis

Since we are interested in varying the resistors and capacitors in the filter circuit, we will need to create models for these components on which we can set some device tolerances for Monte Carlo analysis. The `breakout.slb` Symbol Library file contains generic devices for this purpose. The resistors and capacitors in this schematic are the `Rbreak` and `Cbreak` symbols from `breakout.slb`. Using the model editor, we must modify the models for these components as follows:

```
.model RMOD RES(R=1 DEV=1%)
.model CMOD CAP(C=1 DEV=5%)
```

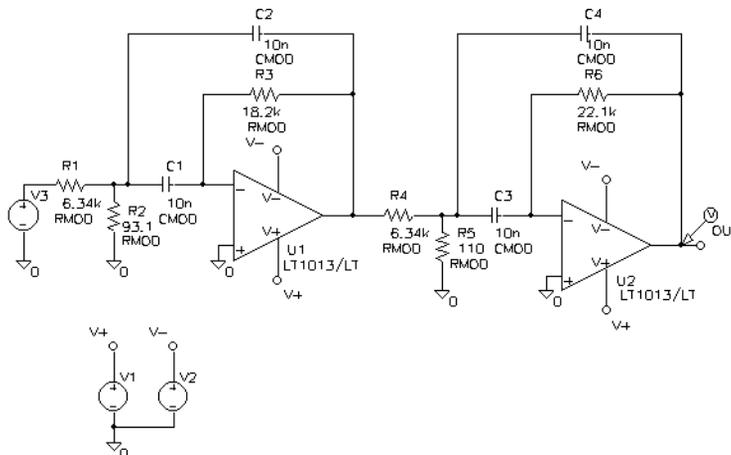


Figure 12-8 Chebyshev Filter

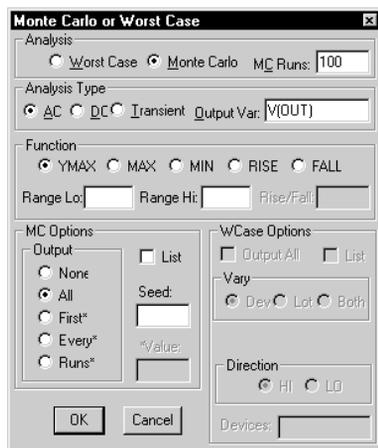


Figure 12-9 Monte Carlo Analysis Setup Example

Setting up the analysis

To analyze our filter, we will set up both an AC analysis and a Monte Carlo analysis. The AC analysis sweeps 50 points per decade from 100 Hz to 1 MHz. The Monte Carlo analysis is set to take 100 runs (see Figure 12-9). The analysis type is AC and the output variable that we are interested in is V(OUT). We will select All in the MC Options box.

Creating histograms

Because the data file can become quite large when running a Monte Carlo analysis and because we are only interested in the output of the filter, we will place a voltage marker at the output of the filter.

To collect data for the marked node only

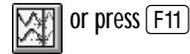
- 1 From the Analysis menu, select Probe Setup.
- 2 On the Probe Startup tab, choose the Automatically Run Probe after Simulation option.
- 3 On the Data Collection tab, choose the At Markers Only option.
- 4 Click OK.

To run the simulation and load Probe with data

- 1 From the Analysis menu, select Simulate.

When complete, Probe automatically starts. Because PSpice ran a Monte Carlo analysis, PSpice saved multiple runs or sections of data. These are listed in the Available Sections dialog box.

- 2 In Probe, in the Available Sections dialog box, click All, and then OK.



To display a histogram for the 1 dB bandwidth

- 1 In Probe, from the Plot menu, select X Axis Settings.

- 2 In the Processing Options frame, select () the Performance Analysis check box.

The display changes to the histogram display where the Y axis is the percent of samples.

- 1 From the Trace menu, select Add.

- 2 In the Goal Functions list, select:

`Bandwidth(1, db_level).`

- 3 In the Simulation Output Variables list, select V(OUT).

- 4 In the Trace Command text box, position the cursor after the V in `Bandwidth(V(OUT),)` and type `DB`.

The text box should now read like this:

`Bandwidth(VDB(OUT),)`

- 5 In the Trace Command text box, position the cursor after the comma and type `1` for the 1 dB level.

The text box should now read like this:

`Bandwidth(VDB(OUT), 1)`

- 6 Click OK to view the histogram.

For information about performance analysis, see [Example: RLC Filter on page 11-3](#).

You can also display this histogram by using the performance analysis wizard to display `Bandwidth(VDB(OUT), 1)`.

To change the number of histogram divisions

- 1 From the Tools menu, select Options.

- 2 In the Number of Histogram Divisions text box, replace 10 with 20.

3 Click Save and then OK.

The histogram for 1 dB bandwidth is shown in Figure 12-10.

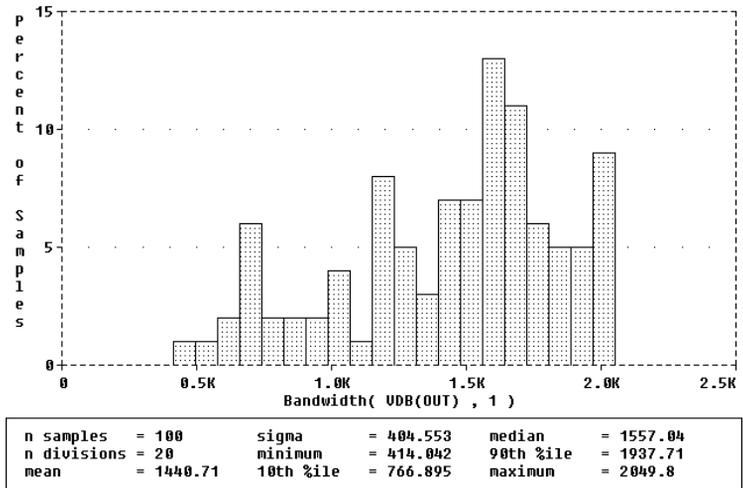


Figure 12-10 1 dB Bandwidth Histogram

The statistics for the histogram are displayed along the bottom of the display. The statistics show the number of Monte Carlo runs, the number of divisions or vertical bars that make up the histogram, mean, sigma, minimum, maximum, 10th percentile, median, and 90th percentile. Ten percent of the goal function values are less than or equal to the 10th percentile number, and 90% of the goal function values are greater than or equal to that number.

If there is more than one goal function value that satisfies this criteria, then the 10th percentile is the midpoint of the interval between the goal function values that satisfy the criteria. Similarly, the median and 90th percentile numbers represent goal function values such that 50% and 90% (respectively) of the goal function values are less than or equal to those numbers. Sigma is the standard deviation of the goal function values.

We can also show the distribution of the center frequency of our filter.

To display the center frequency

- 1 From the Trace menu, select Add.
- 2 In the Goal Functions list, select:
 - CenterFreq(1, db_level).
- 3 In the Simulation Output Variables list, select V(OUT).
- 4 In the Trace Command text box, position the cursor after the V in CenterFreq(V(OUT) ,) and type DB.

The text box should now read like this:

```
CenterFreq(VDB(OUT) , )
```

- 5 In the Trace Command text box, position the cursor after the comma and type 1 for the 1 dB level.

The text box should now read like this:

```
CenterFreq(VDB(OUT) , 1)
```

- 6 Click OK to view the histogram.

The new histogram replaces the previous histogram. To display both histograms at once, use Add Plot on the Plot menu before selecting Add from the Trace menu. The histogram of the center frequency is as shown in Figure 12-11.

If needed, you can turn off the statistical data display as follows:

- 1 From the Tools menu, select Options.
- 2 Clear the Display Statistics check box.
- 3 Click Save, and then OK.

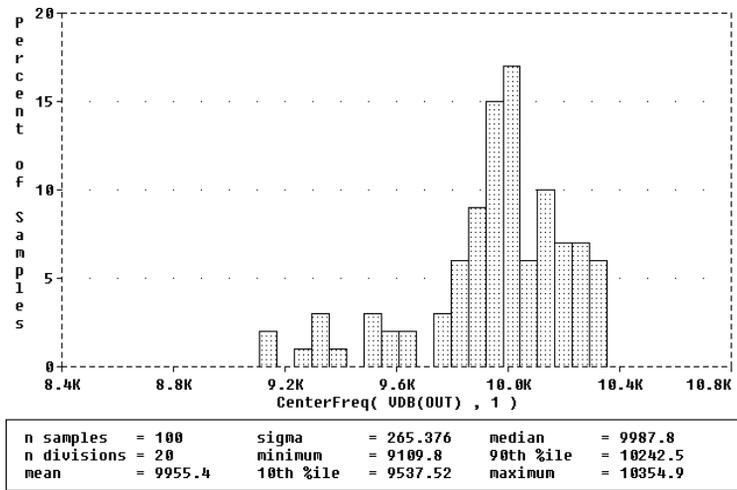
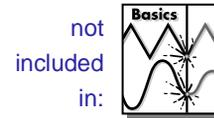


Figure 12-11 Center Frequency Histogram

Worst-Case Analysis

This section discusses the analog worst-case analysis feature of PSpice. The information provided in this section will help you to apply it properly and with realistic expectations.



Overview of Worst-Case Analysis

Worst-case analysis is used to find the worst probable output of a circuit or system given the restricted variance of its parameters. For instance, if the values of R1, R2, and R3 can vary by $\pm 10\%$, then the worst-case analysis attempts to find the combination of possible resistor values which result in the worst simulated output. As with any other analysis, there are three important parts: inputs, procedure, and outputs.

Inputs

Besides the circuit description, two forms of information are required from the user:

- parameter tolerances
- a definition of what *worst* means

PSpice allows tolerances to be set on any number of the parameters that characterize a model. Models can be defined for nearly all primitive analog circuit components, such as resistors, capacitors, inductors, and semiconductor devices. PSpice reads the standard model parameter tolerance syntax specified in the .MODEL statement. For each model parameter, PSpice uses the nominal, minimum, and maximum probable values, and the DEV and/or LOT specifiers; the probability distribution type (such as UNIFORM or GAUSS) is ignored.

The criterion for determining the *worst* values for the relevant model parameters is defined in the .WC statement as a function of any standard output variable in a specified range of the sweep. In a given range, the measurement must be reduced to a single value by one of these five collating functions:

Analog behavioral models can be used to measure waveform characteristics other than those detected by the available collating functions, such as rise time or slope. Analog behavioral models can also be used to incorporate several voltages and currents into one output variable to which a collating function may be applied. See [Chapter 6, Analog Behavioral Modeling](#), for more information.

MAX	maximum output variable value
MIN	minimum output variable value
YMAX	output variable value at the point where it differs the most with the nominal run
RISE_EDGE (value)	sweep value where the output variable value crosses <i>above</i> a given threshold value
FALL_EDGE (value)	sweep value where the output variable value crosses <i>below</i> a given threshold value

Worst is user-defined as the highest (HI) or lowest (LO) possible collating function relative to the nominal run.

Procedure

To establish the initial value of the collating function, worst-case analysis begins with a nominal run with all model parameters at their nominal values. Next, multiple sensitivity analyses determine the individual effect of each model parameter on the collating function. This is accomplished by varying model parameters, one at a time, in consecutive simulations. The direction (*better* or *worse*) in which the collating function changes with a small increase in each model parameter is recorded.

Finally, for the worst-case run, each parameter value is taken as far from its nominal as allowed by its tolerance, in the direction which should cause the collating function to be its worst (given by the HI or LO specification).

This procedure saves time by performing the minimum number of simulations required to make an educated guess at the parameter values which produce the worst results. It also has some limitations, which will be discussed in the following sections.

Outputs

A summary of the sensitivity analysis is printed in the PSpice output file (.out). This summary shows the percent change in the collating function corresponding to a small change in each

model parameter. If a .PROBE statement is included in the circuit file, then the results of the nominal and worst-case runs are saved for viewing with Probe.

An important condition for correct worst-case analysis



Worst-case analysis is not an optimization process; it does not *search* for the set of parameter values which result in the worst result. It assumes that the worst case occurs when each parameter has been either pushed to one of its limits or left at its nominal value as indicated by the sensitivity analysis. **It shows the true worst-case results when the collating function is *monotonic* within all tolerance combinations.** Otherwise, there is no guarantee. Usually you cannot be certain if this condition is true, but insight into the operation of the circuit may alert you to possible anomalies.

Worst-Case Analysis Example

The schematic shown in Figure 12-12 is for an amplifier circuit which is a biased BJT. This circuit is used to demonstrate how a simple worst-case analysis works. It also shows how *non-monotonic* dependence of the output on a single parameter can adversely affect the worst-case analysis. Since an AC (small-signal) analysis is being performed, setting the input to unity means that the output, $V_m([OUT])$, is the magnitude of the gain of the amplifier. The only variable declared in this circuit is the resistance of $Rb2$. Since the value of $Rb2$ determines the bias on the BJT, it also affects the amplifier's gain.

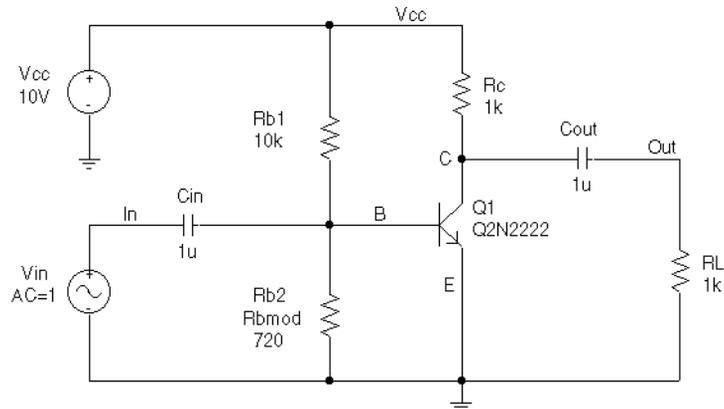


Figure 12-12 Simple Biased BJT Amplifier

Figure 12-14 is the circuit file used to run *either* a parametric analysis (.STEP, shown enabled in the circuit file) that sets the value of resistor $Rb2$ by stepping model parameter R through values spanning the specified DEV tolerance range, *or* a worst-case analysis (shown disabled in the circuit file) that allows PSpice to determine the worst-case value for parameter R based upon a sensitivity analysis. PSpice allows only one of these analyses to be run in any given simulation. Note that the AC and worst-case analysis specifications (.AC and .WC statements) are written so that the worst-case analysis tries to minimize $V_m([OUT])$ at 100 kHz.

The netlist and circuit file in Figure 12-14 is set up to run either a parametric (.STEP) or worst-case (.WC) analysis of the specified AC analysis. These simulations demonstrate the

conditions under which worst-case analysis works well and those that can produce misleading results when output is not monotonic with a variable parameter (see Figure 12-15 and Figure 12-16).

For demonstration, the parametric analysis is run first, generating the curve shown in Figure 12-15 and Figure 12-16. This curve, derived using the YatX goal function shown in Figure 12-13, illustrates the non-monotonic dependence of gain on *Rb2*. To do this yourself, place the goal function definition in a `probe.gf` file in the circuit directory. Then run Probe, load all of the AC sweeps, set up the X axis for performance analysis, and add the following trace:

```
YatX(Vm([OUT]),100k)
```

Next, the parametric analysis is commented out and the worst-case analysis is enabled. Two runs are made using the two versions of the *Rbmod* .MODEL statement shown in the circuit file. The model parameter, R, is a multiplier which is used to scale the nominal value of any resistor referencing the *Rbmod* model (*Rb2* in this case).

The first .MODEL statement leaves the nominal value of *Rb2* at 720 ohms. The sensitivity analysis increments R by a small amount and checks its effect on Vm([OUT]). This slight increase in R causes an increase in the base bias voltage of the BJT, and increases the amplifier's gain, Vm([OUT]). The worst-case analysis correctly sets R to its minimum value for the lowest possible Vm([OUT]) (see Figure 12-15).

The second .MODEL statement scales the nominal value of *Rb2* by 1.1 to approximately 800 ohms. The gain still increases with a small increase in R, but a larger increase in R increases the base voltage so much that it drives the BJT into saturation and nearly eliminates the gain. The worst-case analysis is fooled by the sensitivity analysis into assuming that *Rb2* must be minimized to degrade the gain, but maximizing *Rb2* is much worse (see Figure 12-16). Note that even an optimizer, which checks the local gradients to determine how the parameters should be varied, is fooled by this circuit.

Consider a slightly different scenario: *Rb2* is set to 720 ohms so that maximizing it is not enough to saturate the BJT, but *Rb1* is variable also. The true worst case occurs when *Rb2* is

```
YatX(1, X_value)=y1
{
  1|sfxv(X_value)!1;
}
```

Figure 12-13 *YatX Goal Note* The YatX goal function is used on the simulation results for the parametric sweep (.STEP) defined in Figure 12-14. The resulting curves are shown in Figure 12-15 and Figure 12-16.

```

* Worst-case analysis comparing monotonic and non-monotonic
* output with a variable parameter

.lib

***** Input signal and blocking capacitor *****
Vin In      0      ac      1
Cin In      B      1u

***** "Amplifier" *****
* gain increases with small increase in Rb2, but
* device saturates if Rb2 is maximized.
Vcc Vcc     0      10
Rc  Vcc     C      1k
Q1  C       B      0      Q2N2222
Rb1 Vcc     B      10k
Rb2 B       0      Rbmod  720
.model Rbmod res(R=1 dev 5%)      ; WC analysis results
                                   ; are correct
* .model Rbmod res(R=1.1 dev 5%)  ; WC analysis misled
                                   ; by sensitivity

***** Load and blocking capacitor *****
CoutC      Out      1u
Rl  Out     0      1k

* Run with either the .STEP or the .WC, but not both.
* This circuit file is currently set up to run the .STEP
* (.WC is commented out)

**** Parametric Sweep—providing plot of Vm([OUT]) vs. Rb2 ****
.STEP Res Rbmod(R)  0.8  1.2  10m

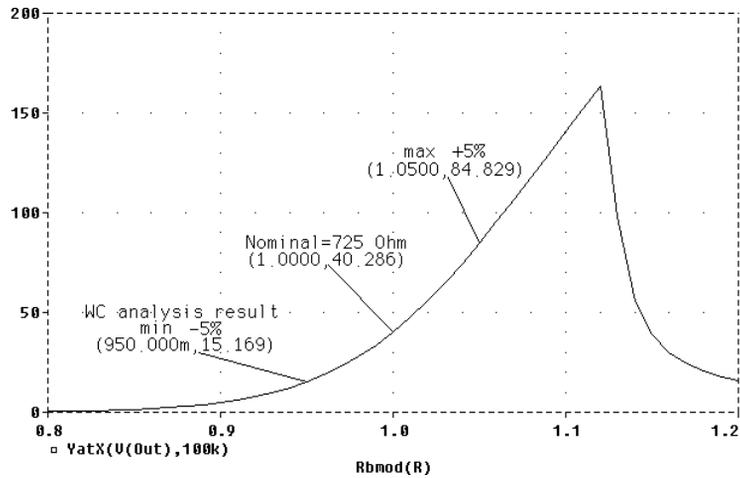
***** Worst-case analysis *****
* run once for each of the .model definitions stated above)
* WC AC Vm([Out]) min range 99k 101k list output all

.AC Lin 3 90k 110k
.probe
.end

```

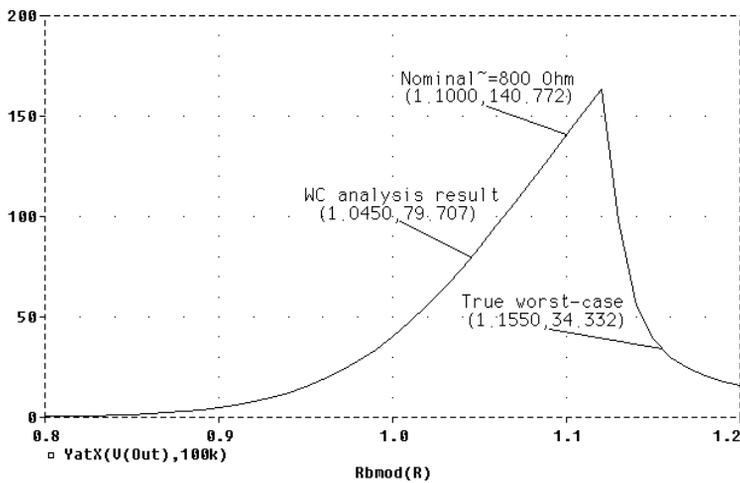
Figure 12-14 *Amplifier Netlist and Circuit File*

maximized and *Rb1* is minimized. Checking their individual effects is not sufficient, even if the circuit were simulated four times with each resistor in turn set to its extreme values.



Output is monotonic within the tolerance range. Sensitivity analysis correctly points to the minimum value.

Figure 12-15 Correct Worst-Case Results



Output is non-monotonic within the tolerance range, thus producing incorrect worst-case results.

Figure 12-16 Incorrect Worst-Case Results



Hints and Other Useful Information

VARY BOTH, VARY DEV, and VARY LOT

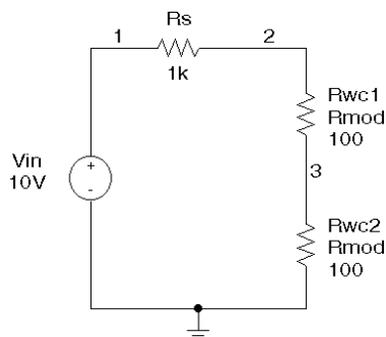


Figure 12-17 Schematic Demonstrating Use of VARY

When VARY BOTH is specified in the .WC statement and a model parameter is specified with both DEV and LOT tolerances defined, the worst-case analysis may produce unexpected results. The sensitivity of the collating function is only tested with respect to LOT variations of such a parameter; for example, during the sensitivity analysis, the parameter is varied once affecting all devices referring to it and its effect on the collating function is recorded. For the worst-case analysis, the parameter is changed for all devices by LOT + DEV in the determined direction. Consider the example schematic in Figure 12-17 and circuit file in Figure 12-18.

```
WCASE VARY BOTH Test
Vin 1 0 10V
Rs 1 2 1K
Rwc12 3 Rmod 100
Rwc23 0 Rmod 100
.MODEL Rmod RES(R=1 LOT 10% DEV 5%)
.DC Vin LIST 10
.WC DC V(3) MAX VARY BOTH LIST OUTPUT ALL
.ENDS
```

Figure 12-18 Circuit File Demonstrating Use of VARY BOTH

In this case, $V(3)$ is maximized if:

- $Rwc1$ and $Rwc2$ are both increased by 10% per the LOT tolerance specification, and
- $Rwc1$ is decreased by 5% and $Rwc2$ is increased by 5% per the DEV tolerance specification.

The final values for $Rwc1$ and $Rwc2$ should be 105 and 115, respectively. However, because $Rwc1$ and $Rwc2$ are varied together during the sensitivity analysis, it is assumed that both must be increased to their maximum for a maximum $V(3)$. Therefore, both are increased by 15%.

Here again, the purpose of the technique is to reduce the number of simulations. For a more accurate worst-case analysis, you should first perform a worst-case analysis with VARY LOT,

manually adjust the nominal model parameter values according to the results, then perform another analysis with VARY DEV specified.

Gaussian distributions

Parameters using Gaussian distributions are changed by 3σ (three times sigma) for the worst-case analysis.

YMAX collating function

The purpose of the YMAX collating function is often misunderstood. This function does not try to maximize the deviation of the output variable value from nominal. Depending on whether HI or LO is specified, it tries to maximize or minimize the output variable value itself at the point where maximum deviation occurred during sensitivity analysis. This may result in maximizing or minimizing the output variable value over the entire range of the sweep. This collating function is usually useful when you know the direction in which the maximum deviation occurs.

RELTOL

During the sensitivity analysis, each parameter is varied (multiplied) by $1 + \text{RELTOL}$ where RELTOL is specified in a .OPTIONS statement, or defaults to 0.001.

Sensitivity analysis

The sensitivity analysis results are printed in the output file (.out). For each varied parameter, the percent change in the collating function and the sweep variable value at which the collating function was measured are given. The parameters are listed in worst output order; for example, the collating function was its worst when the first parameter printed in the list was varied.

When the YMAX collating function is used, the output file also lists mean deviation and sigma values. These are based on the changes in the output variable from nominal at every sweep point in every sensitivity run.

Manual optimization

Worst-case analysis can be used to perform manual optimization with PSpice. The monotonicity condition is usually met if the parameters have a very limited range. Performing worst-case analysis with tight tolerances on the parameters yields sensitivity and worst-case results (in the output file) which can be used to decide how the parameters should be varied to achieve the desired response. You can then make adjustments to the nominal values in the circuit file, and perform the worst-case analysis again for a new set of gradients. Parametric sweeps (.STEP), like the one performed in the circuit file shown in Figure 12-14, can be used to augment this procedure.

Monte Carlo analysis

Monte Carlo (.MC) analysis may be helpful when worst-case analysis cannot be used. Monte Carlo analysis can often be used to verify or improve on worst-case analysis results. Monte Carlo analysis randomly selects possible parameter values, which can be thought of as randomly selecting points in the parameter space. The worst-case analysis assumes that the worst results occur somewhere on the surface of this space, where parameters (to which the output is sensitive) are at one of their extreme values.

If this is not true, the Monte Carlo analysis may find a point at which the results are worse. To try this, simply replace .WC in the circuit file with .MC <#runs>, where <#runs> is the number of simulations you are willing to perform. More runs provide higher confidence results. To save disk space, do not specify any OUTPUT options. The Monte Carlo summary in the output file lists the runs in decreasing order of collating function value.

Now add the following option to the .MC statement, and simulate again.

```
OUTPUT LIST RUNS <worst_run#>
```

This performs only two simulations: the nominal and the worst Monte Carlo run. The parameter values used during the worst run are written to the output file, and the results of both simulations are saved.

Using Monte Carlo analysis with YMAX is a good way to obtain a conservative guess at the maximum possible deviation from nominal, since worst-case analysis usually cannot provide this information.

Part Four

Viewing Results

Part Four describes the ways to view simulation results.

[Chapter 13, Analyzing Waveforms in Probe](#), describes how to perform graphical waveform analysis of simulation results.

[Chapter 14, Viewing Results on the Schematic](#), explains how to view bias point voltages and currents directly on your schematic to help you quickly debug your circuit.

[Chapter 15, Other Output Options](#), describes the special symbols you can place on your schematic to generate additional information to the PSpice output file and PSpice window.

Analyzing Waveforms in Probe

13

Chapter Overview

This chapter describes how to use Probe to perform graphical waveform analysis of simulation results. This chapter includes the following:

[Overview of Probe on page 13-2](#)

[Setting Up Probe on page 13-5](#)

[Running Probe on page 13-9](#)

[Example on page 13-21](#)

[User Interface Features on page 13-24](#)

[Probe Trace Expressions on page 13-32](#)

Overview of Probe

MicroSim Probe is the waveform analyzer for PSpice simulations. In Probe, you can visually analyze and interactively manipulate the waveform data produced by circuit simulation.

Probe uses high-resolution graphics so you can view the results of a simulation both on the screen and in hard copy. In effect, Probe is a software oscilloscope. Running PSpice corresponds to building or changing a breadboard, and running Probe corresponds to looking at the breadboard with an oscilloscope.

With Probe you can:

- view simulation results in multiple plot windows
- compare simulation results from multiple circuit designs, including checkpoint schematics, in a single plot window
- display simple voltages, currents, and noise data
- display complex arithmetic expressions that use the basic measurements
- display Fourier transforms of voltages and currents, or of arithmetic expressions involving voltages and currents
- add text labels and other annotation symbols for clarification

PSpice generates two forms of output: the PSpice output file and the Probe data file. The calculations and results reported in the simulation output file act as an audit trail of the simulation.

However, the graphical analysis of information in the Probe data file is the most informative and flexible method for evaluating simulation results.

Elements of a Plot Window

A plot window is a separately managed waveform display area. A plot window can include multiple plots. Figure 13-1 shows the Probe window with two plot windows displayed (toolbars disabled).

Because a plot window is a window object, you can minimize and maximize the window or move and scale the window within the Probe window area. A toolbar can be displayed in the Probe window and applies to the active plot window.

From the View menu, select Toolbar to display or hide the toolbar.

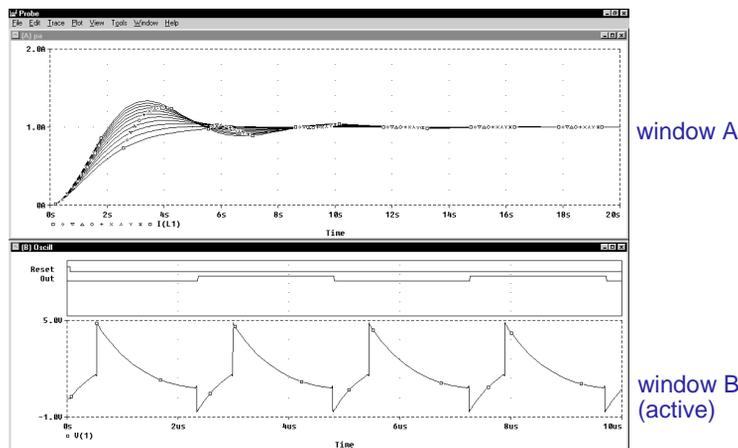


Figure 13-1 Probe Window with Two Plot Windows

You can have one or more Probe data files open in one plot window. Do one of the following:

- Using the Design Journal feature in Schematics, set up Probe to automatically load open working schematics and checkpoint files.
- After the first file is loaded, load other files into the same plot window by manually appending them in Probe.

You can use the Design Journal feature in Schematics to create checkpoint schematics, allowing you to create an electronic record of design development and to perform what-if analyses on your original schematic. See the online Help in Schematics for more information.

Managing Multiple Plot Windows

Any number of plot windows can be opened. Each plot window is an independent window.

The same Probe data file can be displayed in more than one plot window. Only one plot window is active at any given time, identified by a highlighted title bar. Menu, keyboard, and cursor operations affect only the active plot window. Another plot window can be made active by clicking anywhere in the window.

Printing multiple windows

You can print all or selected plot windows, with up to nine windows on a single page. When you select Print from the File menu, a list of all open plot windows is displayed. Each plot window is identified by the unique identifier in parentheses in its title bar.

The arrangement of plot windows on the page can be customized using the Page Setup dialog box. You can print in either portrait (vertical) or landscape (horizontal) orientation. You can also use Print Preview to view all of the plot windows as they will appear when printed.

Setting Up Probe

Configuring Probe Colors

You can configure Probe display and print colors in:

- the configuration file, `msim.ini`, and
- the Probe Options dialog box.

Editing display and print colors in the `msim.ini` file

In the `msim.ini` file, you can control the following print and display color settings:

- the colors that Probe uses to display traces
- the colors that Probe uses for the plot window foreground and background
- the order that Probe uses colors to display traces
- the number of colors that Probe uses to display traces

To edit display and print colors in the `msim.ini` file

Note *After editing the `msim.ini` file, you must restart Probe before your changes will take effect.*

- 1 Using MicroSim Text Editor (or any other text editor), open the `msim.ini` file.
- 2 Scroll to the [PROBE DISPLAY COLORS] or [PROBE PRINTER COLORS] section of the file.
- 3 Add or modify a color entry. See [Table 13-1 on page 13-6](#) for a description of color entries and their default values. Valid item names include:
 - BACKGROUND
 - FOREGROUND
 - TRACE_1 through TRACE_12

You can over-ride the color configuration you set here for traces by assigning colors to markers in Schematics. For more information, see [Using Schematic Markers to Add Traces on page 13-12](#).

For information on how to use the available colors and color order in a Probe plot window, see [Configuring trace color schemes on page 13-7](#).

Colors for all items are specified as `<item name>=<color>`. The item names and what they represent are listed in Table 13-1.

Here are the color names you can specify:

black	blue	brightblue
brightcyan	brightgreen	brightmagenta
brightred	brightwhite	brightyellow
brown	cyan	darkblue
darkcyan	darkgray	darkgreen
darkmagenta	darkpink	darkred
green	lightblue	lightgray
lightgreen	magenta	mustard
orange	pink	purple

- 4 If you added or deleted trace number entries, set `NUMTRACECOLORS=n` to the new number of traces ($1 \leq n \leq 12$). This item represents the number of trace colors displayed on the screen or printed before the color order repeats.
- 5 Save the file.

Table 13-1 *Default Probe Item Colors*

Item Name	Description	Default
BACKGROUND	specifies the color of window background	BLACK
FOREGROUND	specifies the default color for items not explicitly specified	WHITE
TRACE_1	specifies the first color used for trace display	BRIGHTGREEN
TRACE_2	specifies the second color used for trace display	BRIGHTRED
TRACE_3	specifies the third color used for trace display	BRIGHTBLUE
TRACE_4	specifies the fourth color used for trace display	BRIGHTYELLOW
TRACE_5	specifies the fifth color used for trace display	BRIGHTMAGENTA
TRACE_6	specifies the sixth color used for trace display	BRIGHTCYAN



When you want to copy Probe plots to the clipboard and then paste them into a black and white document, try these color settings:

```
BACKGROUND =
BRIGHTWHITE
FOREGROUND = BLACK
```

Configuring trace color schemes

In the Probe Options dialog box, you can set options for how the available colors and the color order specified in the `msim.ini` file are used to display the traces in a Probe plot window. You can use:

- a different color for each trace
- the same color for all the traces that belong to the same y-axis
- the available colors in sequence for each y-axis
- the same color for all the traces that belong to the same data file, including data files for checkpoint schematics

To configure trace color schemes in the Probe Options dialog box

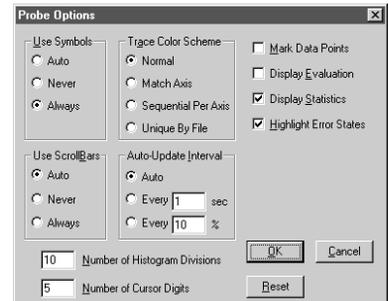
- 1 From the Tools menu, select Options to display the Probe Options dialog box.
- 2 In the Trace Color Scheme frame, choose one of the following options:

Choose this option...	To do this...
Normal	Use a different color for each trace (for up to 12 traces, depending on the number of colors set in the <code>msim.ini</code> file).
Match Axis	Use the same color for all the traces that belong to the same y-axis. The title of the axis (by default, 1, 2, etc.) is the same color as its traces.
Sequential Per Axis	Use the available colors in sequence for each y-axis.
Unique by File	Use the same color for all the traces in one plot window that belong to the same data file.

- 3 Click OK.

For information on what the default available colors and color order are and how to change them, see [Editing display and print colors in the `msim.ini` file on page 13-5](#).

Use this option, in conjunction with the Design Journal feature, to see the differences between a working schematic and its checkpoints. See Schematics online Help for more information.



Probe saves the selected color scheme for future Probe sessions.

Customizing the Probe Command Line

A `.prb` file is an ASCII text file that contains display configurations, goal functions, and macros. A command file is an ASCII text file that contains a list of commands. For more information about `.prb` and command files, see the online Help in Probe.

For a listing and description of command line options, refer to the online *MicroSim PSpice Reference Manual*.

Command files, `.prb` files, and options can be specified in the Probe command line. Probe recognizes these options when you start it automatically after simulation or when you start it from Schematics by selecting Run Probe from the Analysis menu.

To edit the Probe command line

- 1 In Schematics, from the Option menu, select Editor Configuration.
- 2 In the Editor Configuration dialog box, select App Settings.
- 3 In the Simulate Command frame, edit the Command text box.

This command line is saved to `msim.ini`.

Configuring Update Intervals

You can define the frequency at which Probe updates the waveform display as follows:

- At fixed time intervals (every n sec)
- According to the percentage of simulation completed (every n %), where n is user-defined

The default setting (Auto) allows Probe to update traces each time it gets new data from a simulation.

To change the update interval

- 1 From the Tools menu, select Options.
- 2 In the Auto-Update Interval frame, choose the interval type (sec or %), then type the interval in the text box.

Running Probe

Starting Probe

If you are using Schematics, you can automatically start Probe after a simulation is run, or you can start Probe separately from Windows 95 or NT.

When you start Probe, you can use the default `.prb` file or you can use a custom `.prb` file.

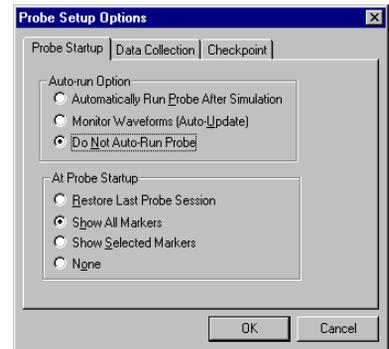
To automatically start Probe after simulation

- 1 In Schematics, from the Analysis menu, select Probe Setup.
- 2 In the Probe Setup Options dialog box, select the Probe Startup tab.
- 3 In the Auto-Run Option frame, select Automatically Run Probe After Simulation.
- 4 Select any other options that you want to use.
- 5 Click OK.

To start Probe and monitor results during a simulation

- 1 In Schematics, from the Analysis menu, select Probe Setup.
- 2 In the Probe Setup Options dialog box, select the Probe Startup tab.
- 3 In the Auto-Run Option frame, select Monitor Waveforms (Auto-Update). If this option is unavailable, do the following:
 - a Select the Data Collection tab.
 - b Clear the Text Data File Format (CSDF) check box. The Monitor Waveforms (Auto-Update) option should now be available from the Probe Startup tab.
- 4 Click OK.

You do not need to quit Probe if you are finished examining the simulation results for one circuit and want to begin a new simulation. However, when you set up Probe to run automatically after simulation, Probe unloads the old data file for a circuit each time that you run a new simulation of the circuit. After the simulation is complete, the new or updated Probe data file is loaded for viewing.



If you open a new Probe window (from the Window menu, select New Window) while monitoring the data, the new window also starts in monitor mode because it is associated with the same Probe data file.



During a multi-run simulation (such as Monte Carlo, parametric, or temperature), Probe displays only the data for the current run in the plot window.



or press **Insert**

For more information, see [Using Schematic Markers to Add Traces on page 13-12](#).

press **F12**

- 5 From the Analysis menu, select Simulate to start the simulation. Probe starts automatically and displays one window in monitor mode.
- 6 Do one of the following to select the waveforms to be monitored:
 - In Probe, from the Trace menu, select Add, and enter one or more trace expressions.
 - In Schematics, from the Markers menu, select and place one or more markers (and marker color, as needed).

To start Probe from Schematics

- 1 From the Analysis menu, select Run Probe.

To start Probe in Windows 95

- 1 From the Windows Start menu, select the MicroSim program folder and then the Probe shortcut.

To start Probe manually using the Windows Run command

- 1 From the Windows Start menu, select Run.
- 2 Type the path to the Probe command file and command line options or a data file (.dat).

The command for starting Probe using the Windows Run command is:

probe options data_file*

This option...	Means this...
<i>options</i>	one or more command line options for running Probe with a command file (C option), log file (L option), and so on, where options are preceded with / or -
<i>data_file</i>	name of the Probe data file generated by PSpice

See the online *MicroSim PSpice A/D Reference Manual* for a complete list of Probe command line options.

Other Ways to Run Probe

Starting Probe during a simulation

Once a simulation is in progress, you can monitor the results for the data section currently being written by PSpice. This function is only available when Monitor Waveforms (Auto-Update) is not enabled in Schematics in the Probe Setup Options dialog box.

To start Probe during a simulation

- 1 Start the simulation as described in [Starting Simulation on page 7-10](#).
- 2 In Schematics, from the Analysis menu, select Run Probe. Probe automatically opens the data file and the data section that PSpice is currently writing.

When started during a simulation, the Probe window monitors the waveforms for as long as the current data section is being written. After the data section is finished, the window reverts to manual mode.

Pausing a simulation and then running Probe

You can pause a simulation to analyze waveforms before the simulation is finished. Once you pause the simulation, you can either resume the simulation or terminate it.

To pause a simulation and then run Probe

- 1 In the PSpice simulation status window, from the File menu, select Pause Simulation.
- 2 From the File menu, select Run Probe and verify that the analysis is proceeding correctly.
- 3 Do one of the following:
 - In the PSpice simulation status window, from the File menu, select Pause Simulation to clear the check mark.
 - In the PSpice simulation status window, from the File menu, select Terminate Simulation.

Interacting with Probe while in monitor mode

All of the Probe functionality is available when in monitor mode. However, functions that change the x-axis domain (set a new x-axis variable) pause monitoring and place the window in manual mode until the x-axis is reverted to its original domain. The following table describes how to enable the functions that change the x-axis domain:

	Enable this function...	By doing this...
	Fast Fourier transforms	<ol style="list-style-type: none"> 1 From the Plot menu, select X Axis Settings. 2 In the Processing Options frame, choose the Fourier option.
	Performance analysis	<ol style="list-style-type: none"> 1 From the Plot menu, select X Axis Settings. 2 In the Processing Options frame, choose the Performance Analysis option.
	New x-axis variable	<ol style="list-style-type: none"> 1 From the Plot menu, select X Axis Settings, and click Axis Variable. 2 In the X Axis Variable dialog box, specify a new x-axis variable.
	Goal function evaluation	<ol style="list-style-type: none"> 1 From the Trace menu, select Eval Goal Function. 2 In the Evaluate Goal Function(s) dialog box, specify a goal function.
	Load a completed data section	<ol style="list-style-type: none"> 1 From the File menu, select Append. 2 Specify a .dat file to append.

Using Schematic Markers to Add Traces

See [Probe Trace Expressions on page 13-32](#) for ways to add traces within Probe.

You can place markers on a schematic to identify the points where you want to see the waveform results displayed in Probe. Markers can be placed:

- Before simulation to limit results written to the Probe data file and automatically display those traces in Probe.

- During or after simulation, with Probe running, to automatically display traces in the active plot window.

You can also control the color of each marker you place. The color you choose for a marker will also be the color of its trace in Probe.

The Markers menu provides additional selections for controlling display of marked results in Probe, after initial marker placement, and during or after simulation.

To place markers on a schematic

- 1 In Schematics, from the Markers menu, select the marker type you want to place.

Waveform	Markers menu selection	Symbol selection
voltage	Mark Voltage/Level	not required
voltage differential	Mark Voltage Differential	not required
current	Mark Current into Pin	not required
dB*	Mark Advanced	VDB (voltage) IDB (current)
phase*	Mark Advanced	VPHASE (voltage) IPHASE (current)
group delay*	Mark Advanced	VGROUPDELAY (voltage) IGROUPDELAY (current)
real*	Mark Advanced	VREAL (voltage) IREAL (current)
imaginary*	Mark Advanced	VIMAGINARY (voltage) IIMAGINARY (current)

*. You can use these markers instead of the built-in Probe functions provided in output variable expressions (see [Table 13-10 on page 13-40](#)). Note that these markers are only useful during or after AC analyses.

- 2 If you want to select a color for the marker and its Probe trace, choose one of the colors from the list in the Simulation toolbar.
- 3 Point to wires or pins and click to place markers.
- 4 Right-click to quit placing markers.



marker color list

The color you choose for a marker (and its trace) over-rides the Probe trace color configuration described in [Configuring Probe Colors on page 13-5](#).

The colors available in the marker color list are the same as the colors available for trace display in Probe. This color set is configured in the msim.ini file as described in [Editing display and print colors in the msim.ini file on page 13-5](#).

- 5 If you have not simulated the circuit yet, from the Analysis menu, select Simulate.

To control the display of marked results in Probe

- 1 In Schematics, from the Markers menu, select one of the following:

Choose this option...	To do this...
Show All	Display traces for all markers placed on any page or level of the schematic in Probe.
Show Selected	Display traces only for selected markers (highlight markers of interest and select Show Selected).
Clear All	Remove all markers from the schematic and all corresponding traces from the Probe display.

Limiting Probe Data File Size

When PSpice runs, it creates a Probe data file. The size of this file for transient analyses is roughly equal to:

(# transistors)·(# simulation time points)·24 bytes

The size for other analyses is about 2.5 times smaller. For long runs, especially transient runs, this can generate Probe data files that are several megabytes in size. Even if this does not cause a problem with disk space, large Probe data files take longer to read in and longer to display traces on the screen.

You can limit Probe data file size by:

- placing markers on your schematic before simulation and having PSpice restrict the saved data to these markers only
- excluding data for internal subcircuits
- suppressing simulation output

Limiting file size using markers

One reason that Probe data files are large is that, by default, PSpice stores *all net voltages and device currents* for each step (for example, time or frequency points). However, if you have placed markers on your schematic prior to simulation, PSpice saves only the results for the marked wires and pins.

To limit file size using markers

- 1 In Schematics, from the Analysis menu, select Probe Setup.
- 2 In the Probe Setup Options dialog box, select the Data Collection tab.
- 3 In the Data Collection frame, select At Markers Only and click OK.
- 4 In Schematics, from the Markers menu, select the marker type you want to place.
- 5 If desired, on the Simulation toolbar, from the marker color list, select a color for one or more of the markers (and its Probe trace).
- 6 Click wires or pins to place markers.





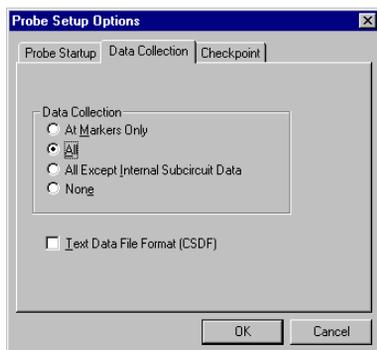
- 7 Right-click to quit placing markers.
- 8 From the Analysis menu, select Simulate.

Limiting file size by excluding internal subcircuit data

By default, PSpice writes data to the Probe file for all internal nodes and devices in subcircuit models on a schematic. You can choose to exclude data for internal subcircuit nodes and devices.

To limit file size by excluding data for internal subcircuits

- 1 In Schematics, from the Analysis menu, select Probe Setup.
- 2 In the Probe Setup Options dialog box, select the Data Collection tab.
- 3 In the Data Collection frame, select All Except Internal Subcircuit Data and click OK.
- 4 From the Analysis menu, select Simulate.



Suppressing part of the data run also limits the size of the PSpice output file.

Limiting file size by suppressing the first part of simulation output

Long transient simulations create large Probe data files because PSpice stores many data points. You can suppress a part of the data from a transient run by setting the simulation analysis to start the output at a time later than 0. This does not affect the transient calculations themselves—these always start at time 0. This delay only suppresses the output for the first part of the simulation.

To limit file size by suppressing the first part of transient simulation output

- 1 In Schematics, from the Analysis menu, select Setup.
- 2 In the Analysis Setup dialog box, click the Transient button.
- 3 In the No-Print Delay text box, type a delay time and click OK.
- 4 From the Analysis menu, select Simulate. No data will be stored until the delay has elapsed.



or press **F11**

Using Simulation Data from Multiple Files

You can load simulation data from multiple files into the same Probe plot in two ways:

- By setting up Probe for automatic loading of checkpoint and working schematic data files.
- By appending data files.

When more than one data file is loaded, you can add traces using all loaded data, data from only one file, or individual data sections from one or more files.

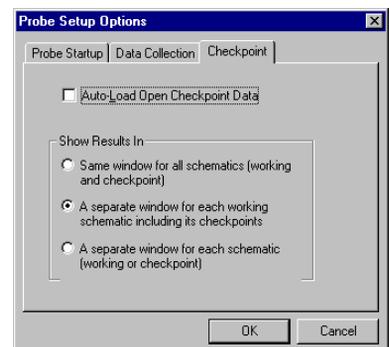
Setting up Probe for automatic loading of data files

In Schematics, you can set up Probe so data from open checkpoint and working schematics simulations is automatically loaded.

To set up Probe for automatic loading of data files

- 1 In Schematics, from the Analysis menu, select Probe Setup.
- 2 In the Probe Setup Options dialog box, select the Checkpoint tab.
- 3 Select the following option: Automatically load data for open checkpoints.

You can use the Design Journal feature in Schematics to create checkpoint schematics, which are copies of your working (or current development) schematic at any point of development up to the present. Checkpoint schematics allow you to create an electronic record of design development and to perform what-if analyses on your original schematic. See the online Help in Schematics for more information.



- 4 In the Show Results In frame, choose one of the following options:

Choose this option...	To do this...
Same window for all schematics (working and checkpoint)	Load the data sets for all open working and checkpoint schematics in one plot window.
Separate windows for each working schematic including its checkpoints	Load the data sets for each open working schematic and its checkpoints in one plot window.
A separate window for each schematic (working or checkpoint)	Load all data sets in separate plot windows.

- 5 Click OK.

Appending data files

You can manually load data sets one at time into a plot window using the Append command.

To append a data file



- 1 In Probe, from the File menu, select Append.
- 2 Select a `.dat` file to append, and click OK.
- 3 If the file has multiple sections of data for the selected analysis type, the Available Sections dialog box appears. Do one of the following:
 - Click the sections you want to use.
 - Click the All button to use all sections.
- 4 Click OK.

Adding traces from specific loaded data files

If two or more data files have identical simulation output variables, trace expressions that include those variables will plot traces for each file. However, you can specify which data file to use in the trace expression. You can also determine which data file was used to generate a specific trace.

To add a trace from a specific loaded data file

- 1 In Probe, from the Trace menu, select Add to display the Add Traces dialog box.
- 2 In the Trace Expression text box, type an expression using the following syntax:

trace_expression@fn

where *n* is the numerical order (from left to right) of the data file as it appears in the Probe title bar, or

trace_expression@s@fn

where *s* is a specific data section of a specific data file.

- 3 Click OK.

To identify the source file for an individual trace

- 1 In the trace legend, double-click the symbol for the trace you want to identify (Figure 13-2).

The Section Information dialog box appears, containing the trace name and, if there is more than one data file loaded in the plot, the full path for the file from which the trace was generated. Also listed is information about the simulation that generated the data file and the number of data points used. If the trace is from a checkpoint data file, the dialog box includes the checkpoint description (Figure 13-3).



or press **Insert**

The Simulation Output Variables list in the Add Traces dialog box contains the output variables for all loaded data files.

Example: To plot the V(1) output for data section 1 from the second data file loaded, type the following trace expression:

V(1)@1@f2

You can alternately use the name of the loaded data file to specify it. For example, to plot the V(1) output for all data sections of a loaded data file, MyFile.dat, type the following trace expression:

V(1)@"MyFile.dat"



trace symbols

Figure 13-2 Trace Legend Symbols

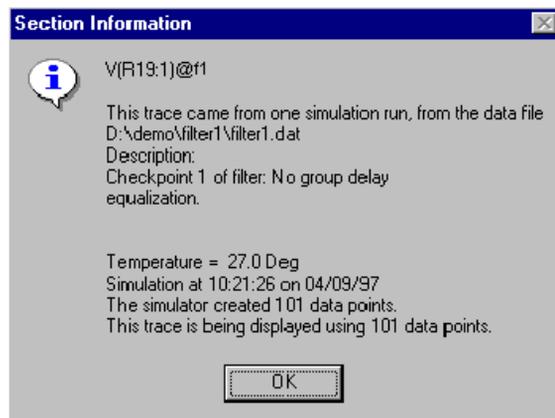
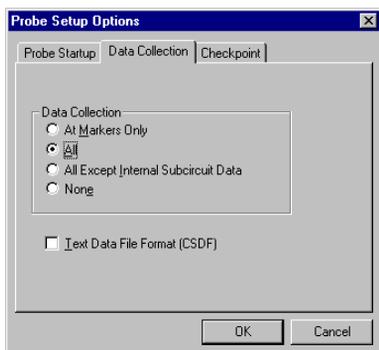


Figure 13-3 *Section Information Message Box*

Warning: Data files saved in the CSDF format are two or more times the size of binary files.

When you first open a CSDF data file, Probe converts it back to the Probe .dat format. This conversion takes two or more times as long as opening a .dat Probe file. Probe saves the new .dat file for future use.



The default Probe data file format is binary. However, you can save the Probe data file in the Common Simulation Data Format (CSDF) instead.

To save simulation results in ASCII format

- 1 In Schematics, from the Analysis menu, select Probe Setup.
- 2 In the Probe Setup Options dialog box, select the Data Collection tab.
- 3 Select Text Data File Format (CSDF).
- 4 Click OK.

PSpice will write simulation results to the Probe data file in ASCII format (as .csd instead of .dat) following the CSDF convention.

Example

In this section, basic techniques for operating Probe are demonstrated using the circuit `Example.sch`.

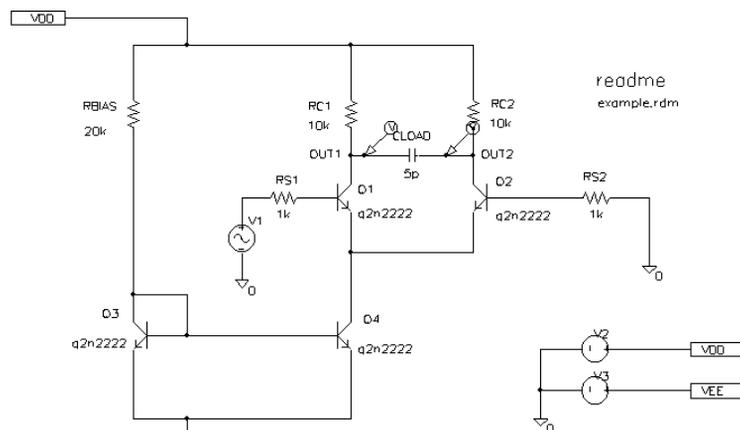


Figure 13-4 Example Schematic `Example.sch`

The example circuit

`Example.sch` is provided with your MicroSim programs.

When shipped, `Example.sch` is set up with multiple analyses. For this example, the AC sweep, DC sweep, Monte Carlo/worst-case, and small-signal transfer function analyses have been disabled. The specification for each of these disabled analyses remains intact. You can run them in the future by selecting Setup from the Analysis menu in Schematics and selecting (✓) the check boxes next to the analyses.

Running the Simulation

The simulation is run with the Bias Point Detail, Temperature, and Transient analyses enabled. The temperature analysis is set to 35 degrees. The transient analysis is setup as follows:

Print Step	20ns
Final Time	1000ns
Enable Fourier	selected
Center Frequency	1Meg
Output Vars	V(OUT2)



or press **F11**

To start the simulation

- 1 Start Schematics. If Schematics is already running, be sure you are in the schematic editor.
- 2 Open the following file from the directory where you installed your MicroSim programs:
`Examples\Schemat\Example\Example.sch`
- 3 From the Analysis menu, select Simulate to start the simulation.

By default, Probe is set to automatically run. However, if Probe does not run automatically after a simulation is complete, then from the Analysis menu, select Run Probe.

If Probe is set to show traces for all markers on startup, you will see the V(OUT1) and V(OUT2) traces when the Probe window displays. To clear these traces from the plot, do the following: from the Trace menu, select Delete All.

PSpice generates a binary Probe data file containing the results of the simulation. The Probe screen displays with the data file, `Example.dat`, already loaded (Figure 13-5).

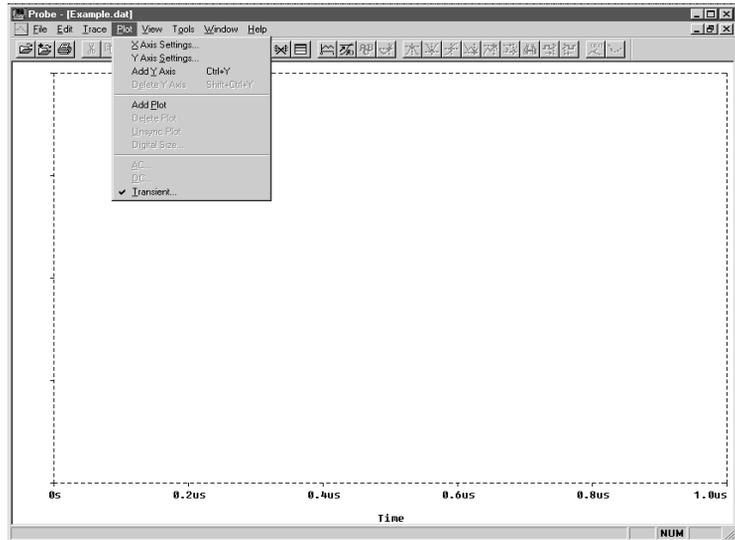


Figure 13-5 *Probe Main Window with Loaded Example.dat and Open Plot Menu*

The name of the data file, `Example.dat`, is shown in the title bar. All Probe commands are available through the menu items. Notice that the Transient command on the Plot menu is selected, indicating that the data currently loaded are the transient analysis results.

Displaying voltages on nets and currents into pins

Having selected an analysis, voltages on nets and currents into device pins can be displayed in the Probe plot using either the schematic marker method or by explicitly specifying Probe output variables (as will be demonstrated in this example).

To display the voltages at the OUT1 and OUT2 nets using output variables

- 1 From the Trace menu, select Add. Probe displays a list of valid output variables in the Simulation Output Variables frame.
- 2 Click V(OUT1) and V(OUT2), then click OK. The plot window should look similar to Figure 13-6.

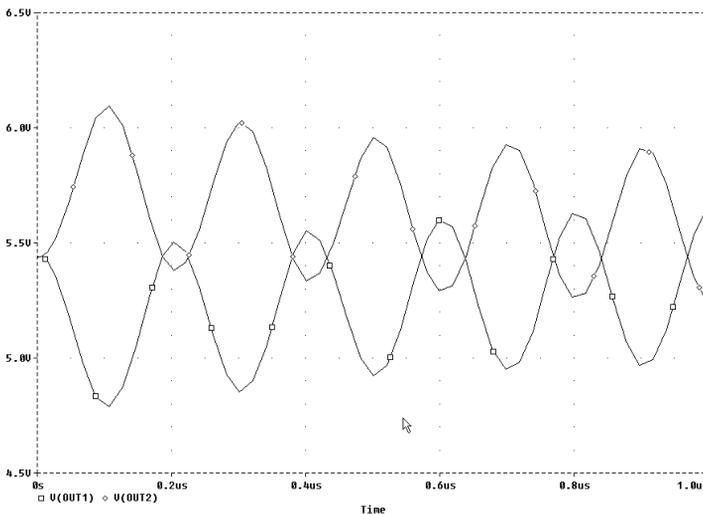


Figure 13-6 Output from Transient Analysis: Voltage at OUT1 and OUT2

User Interface Features

Probe offers a number of direct manipulation techniques and shortcuts to analyze the waveform data. These techniques are described in this section.

Shortcut keys

Many of the menu functions in Probe have matching keystrokes. For instance, having placed a selection rectangle in the plot, you can type **Ctrl+A** instead of selecting Area from the View menu. For a list of shortcut keys, from the Help menu, select Keyboard Shortcuts.



Click anywhere on the plot to remove the selection rectangle without zooming.

Zoom Regions

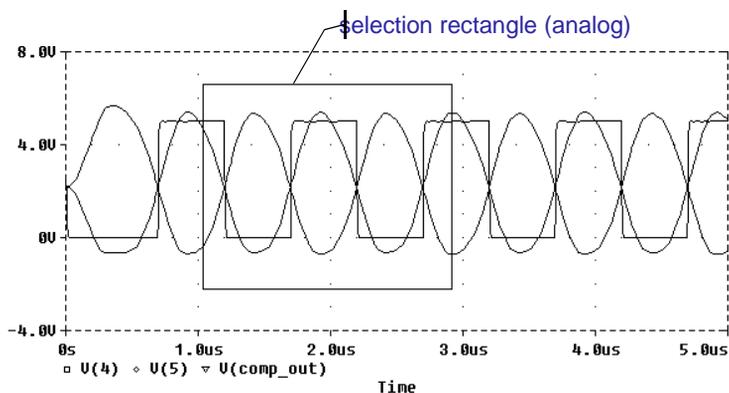
Probe provides a direct manipulation method for marking the zoom region in the plot.

To zoom in or out

- 1 Do one of the following on the toolbar:
 - Click View In to zoom in by a factor of 2 around the point you specify.
 - Click View Out buttons to zoom out by a factor of 2 around the point you specify.

To zoom in using the mouse

- 1 Drag the mouse to make a selection rectangle as shown below.



- 2 From the View menu, select Area.

Scrolling Traces

By default, when a plot is zoomed, standard scroll bars appear to the right or at the bottom of the plot area as necessary. These can be used to pan through the data. You can configure scroll bars so they are always present or are never displayed.

To configure scroll bars

- 1 In Probe, from the Tools menu, select Options.
- 2 In the Use Scroll Bars frame, choose the appropriate scroll bars option.

Choose this option...	To do this...
Auto	Have scroll bars appear when a plot is zoomed or additional traces are displayed in the plot but are not visible (default).
Never	Never display scroll bars. This mode provides maximum plot size and is useful on VGA and other low resolution displays.
Always	Display scroll bars at all times. However, they are disabled if the corresponding axis is full scale.

For information about adding labels (including text, line, poly-line, arrow, box, circle, ellipse, and mark), see the online Help in Probe.

You can also double-click the trace name to modify the trace expression.

You can also double-click a text or ellipse label to modify it.

Modifying Trace Expressions and Labels

You can modify trace expressions, text labels, and ellipse labels that are currently displayed within the plot window, thus eliminating the need to delete and recreate any of these objects.

To modify trace expressions

- 1 Click the trace name to select it (selection is indicated by a color change).
- 2 From the Edit menu, select Modify Object.
- 3 In the Modify Trace dialog box, edit the trace expression just as you would when adding a trace.

To modify text and ellipse labels

- 1 Click the text or ellipse to select it (selection is indicated by a color change).
- 2 From the Edit menu, select Modify Object.
- 3 Edit the label by doing one of the following:
 - In the Ellipse Label dialog box, change the inclination angle.
 - In the Text Label dialog box, change the text label.

Moving and Copying Trace Names and Expressions

Trace names and expressions can be selected and moved or copied, either within the same plot window or to another plot window.

To copy or move trace names and expressions

- 1 Click one or more (**Shift**+click) trace names. Selected trace names are highlighted.
- 2 From the Edit menu, select Copy or Cut to save the trace names and expressions to the clipboard. Cut removes trace names and traces from the plot window.
- 3 In the plot window where traces are to be added, do one of the following:
 - To add trace names to the end of the currently displayed set, select Paste from the Edit menu.
 - To add traces before a currently displayed trace name, select the trace name and then select Paste from the Edit menu.

Here are some considerations when copying or moving trace names and expressions into a different plot window:

- If the new plot window is reading the same Probe data file, the copied or moved trace names and expressions display traces that are identical to the original selection set.
- If the new plot window is reading a *different* Probe data file, the copied or moved names and expressions display different traces generated from the new data.

For example, suppose two data files, `mysim.dat` and `yoursim.dat` each contain a V(2) waveform. Suppose also that two plot windows are currently displayed where window A is loaded with `mysim.dat`, and window B is loaded with `yoursim.dat`.

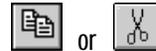
When V(2) is copied from window A to window B, the trace looks different because it is determined by data from `yoursim.dat` instead of `mysim.dat`.

When adding a trace to a plot window, you can make the trace display name different from the trace expression:

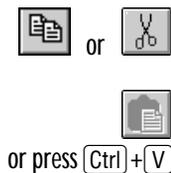
- 1 From the trace menu, select Add.
- 2 In the Trace Expression text box, enter a trace expression using the syntax:

```
trace_expression[;display_name]
```

- 3 Click OK.



For information about adding labels (including text, line, poly-line, arrow, box, circle, ellipse, and mark), see the online Help in Probe.



Copying and Moving Labels

Labels can be selected and moved or copied, either within the same plot window or to another plot window.

To copy labels

- 1 Select one or more (**Shift**+click) labels, or select multiple labels by drawing a selection rectangle. Selected labels are highlighted.
- 2 From the Edit menu, select Copy or Cut to save the labels to the clipboard. Cut will remove labels from the plot window.
- 3 From the plot window where labels are to be added, select Paste from the Edit menu.
- 4 Click on the new location to place the labels.

To move labels

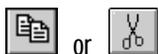
- 1 Select one or more (**Shift**+click) labels, or select multiple labels by drawing a selection rectangle. Selected labels are highlighted.
- 2 Move the labels by dragging them to a new location.

Tabulating Trace Data Values

You can generate a table of data points reflecting one or more traces in the plot window and use this information in a document or spreadsheet.

To view the trace data values table

- 1 Select one or more (**Shift**+click) traces. Selected traces are highlighted.
- 2 From the Edit menu, select Copy or Cut to save the trace data point values to the Clipboard. Cut will remove traces from the plot window.
- 3 From within Clipboard Viewer, select either Text or OEM Text from the Display menu.



To export the data points to other Windows 95 or NT programs

- 1 Select one or more (**Shift**+click) traces. Selected traces are highlighted.
- 2 From the Edit menu, select Copy or Cut to save the trace data point values to the Clipboard. Cut will remove traces from the plot window.
- 3 Paste the data from the Clipboard into the MicroSim Text Editor (or any other text editor), a spreadsheet program, or a technical computing program (such as Mathcad).
- 4 Save the file.

Saving the data directly to a file from Clipboard Viewer can create superfluous data at the beginning of the file.



Cursors

When one or more traces are displayed, Probe provides cursors that can be used to display the exact coordinates of two points on the same trace, or points on two different traces. In addition, differences are shown between the corresponding coordinate values for the two cursors.

To display both cursors

- 1 From the Tools menu, point to Cursor, then select Display.

The Probe Cursor box appears on the screen, showing the current position of the cursor on the x-axis and y-axis. As you move the cursors, the values in the cursor box change.

Both cursors are initially placed on the trace listed first in the trace legend. The corresponding trace symbol is outlined with a dashed line.



or press **Ctrl**+**Shift**+**C**

You can move the cursor box any where over the plot window by dragging the box to another location.

To move cursors along a trace using menu commands

- 1 From the Tools menu, point to Cursor, then select Peak, Trough, Slope, Min, Max, Point, or Search.

For information about cursors commands, see the online Help in Probe.

For a family of curves (such as from a nested DC sweep), you can use the mouse or the arrow keys to move the cursor to one of the other curves in the family. You can also click the desired curve.

To move cursors along a trace using the mouse

- 1 Use the right and left mouse buttons as described in [Table 13-2](#).

Table 13-2 *Mouse Actions for Cursor Control*

Category	Action	Function
cursor assignment	Left-click the trace symbol.	Associates the first cursor with the selected trace.
	Right-click the trace symbol.	Associates the second cursor with the selected trace.
cursor movement	Left-click in the display area.	Moves the first cursor to the closest trace segment at the X position.
	Right-click in the display area.	Moves the second cursor to the closest trace segment at the X position.

To move cursors along a trace using the keyboard

1 Use key combinations as described in [Table 13-3](#).

Table 13-3 *Key Combinations for Cursor Control*

Key Combination	Function
Ctrl + ← and Ctrl + →	Changes the trace associated with the first cursor.
Shift + Ctrl + ← and Shift + Ctrl + →	Changes the trace associated with the second cursor.
← and →	Moves the first cursor along the trace.
Shift + ← and Shift + →	Moves the second cursor along the trace.
Home	Moves the first cursor to the beginning of the trace.
Shift + Home	Moves the second cursor to the beginning of the trace.
End	Moves the first cursor to the end of the trace.
Shift + End	Moves the second cursor to the end of the trace.

Probe Trace Expressions

Traces are referred to by Probe output variable names. Probe output variables are similar to the PSpice output variables specified in the Schematics Analysis Setup dialog boxes for noise, Monte Carlo, worst-case, transfer function, and Fourier analyses. However, there are additional alias forms that are valid within Probe. Both forms are discussed here.

To add traces using Probe output variables

- 1 From the Trace menu, select Add to display the Add Traces dialog box.
- 2 Construct a trace expression using any combination of these controls:
 - In the Simulation Output Variables frame, click output variables.
 - In the Functions or Macros frame, select operators, functions, constants, or macros.
 - In the Trace Expression text box, type in or edit output variables, operators, functions, constants, or macros.
- 3 If you want to change the name of the trace expression as it displays in the plot window, use the following syntax:
trace expression;display name
- 4 Click OK.

You can display a subset of the available simulation output variables by clearing or selecting the variable type check boxes in the Simulation Output Variables frame. Variable types not generated by the circuit simulation are dimmed.

For more information about trace expressions, see [Analog Trace Expressions on page 13-40](#).

Basic Output Variable Form

This form is representative of those used for specifying some PSpice analyses.

```
<output>[AC suffix](<name>[,name])
```

This placeholder...	Means this...
<output>	type of output quantity: V for voltage or I for current
[AC suffix]*	quantity to be reported for an AC analysis. For a list of valid AC suffixes, see Table 13-6 on page 13-37
<name>[,name]	<p>specifies either the <i>net</i> or (+ <i>net</i>, - <i>net</i>) pair for which the voltage is to be reported, or the <i>device</i> for which a current is reported, where:</p> <ul style="list-style-type: none"> • <i>net</i> specifies either the <i>net</i> or <i>pin id</i> (<fully qualified device name>:<pin name>) • <i>device name</i> specifies the fully qualified device name; for a list of device types, see Table 13-7 on page 13-37 and Table 13-8 on page 13-38

A fully qualified device name consists of the full hierarchical path followed by the device's reference designator. For information about syntax, see the voltage output variable naming rules on page [7-5](#).

Output Variable Form for Device Terminals

This form can only be specified in Probe. The primary difference between this and the basic form is that the terminal symbol appears before the *net* or *device name* specification (whereas the basic form treats this as the *pin name* within the *pin id*).

`<output>[terminal]*[AC suffix](<name>[,name])`

This placeholder...	Means this...
<code><output></code>	type of output quantity: V for voltage, I for current, or N for noise
<code>[terminal]*</code>	one or more terminals for devices with more than two terminals; for a list of terminal IDs, see Table 13-8 on page 13-38
<code>[AC suffix]*</code>	quantity to be reported for an AC analysis; for a list of valid AC suffixes, see Table 13-6 on page 13-37
<code><name>[,<name>]</code>	<i>net</i> , <i>net</i> pair, or fully qualified <i>device name</i> ; for a list of device types, see Table 13-7 on page 13-37 and Table 13-8 on page 13-38

[Table 13-4 on page 13-35](#) summarizes the allowed Probe output formats. [Table 13-5 on page 13-36](#) provides examples of equivalent output variables. Note that some of the output variable formats are unique to Probe.

Table 13-4 Probe Output Variable Formats

Format	Meaning
Voltage Variables	
V[ac](<i>< +analog net ></i> [<i>< -analog net ></i>])	Voltage between + and - <i>analog net</i> ids
V<pin name>[ac](<i>< device ></i>)	Voltage at <i>pin name</i> of a <i>device</i>
V< x >[ac](<i>< 3 or 4-terminal device ></i>)	Voltage at non-grounded terminal <i>x</i> of a <i>3 or 4-terminal device</i>
V< z >[ac](<i>< transmission line device ></i>)	Voltage at end <i>z</i> of a <i>transmission line device</i> (<i>z</i> is either A or B)
Current Variables	
I[ac](<i>< device ></i>)	Current into a <i>device</i>
I< x >[ac](<i>< 3 or 4-terminal device ></i>)	Current into terminal <i>x</i> of a <i>3 or 4-terminal device</i>
I< z >[ac](<i>< transmission line device ></i>)	Current into end <i>z</i> of a <i>transmission line device</i> (<i>z</i> is either A or B)
Sweep Variables	
< DC sweep variable >	name of any variable used in the DC sweep analysis
FREQUENCY	AC analysis sweep variable
TIME	transient analysis sweep variable

Table 13-4 *Probe Output Variable Formats (continued)*

Format	Meaning
Noise Variables	
V[db](ONoise)	total RMS-summed noise at output net
V[db](INoise)	total equivalent noise at input source
NTOT(ONoise)	sum of all noise contributors in the circuit
N< noise type >(< device name >)	contribution from <i>noise type</i> of <i>device name</i> to the total output noise*

*. See [Table 13-9 on page 13-39](#) for a complete list of noise types by device type. For information about noise output variable equations, the units used to represent noise quantities in Probe, and a noise analysis example, see [Analyzing Noise in Probe on page 9-12](#).

Table 13-5 *Examples of Probe Output Variable Formats*

A Basic Form	An alias equivalent	Meaning
V(NET3,NET2)	(same)	voltage between analog nets labeled NET3 and NET2
V(C1:1)	V1(C1)	voltage at pin1 of capacitor C1
VP(Q2:B)	VBP(Q2)	phase of voltage at base of bipolar transistor Q2
V(T32:A)	VA(T32)	voltage at port A of transmission line T32
I(M1:D)	ID(M1)	current through drain of MOSFET device M1
VIN	(same)	voltage source named VIN
FREQUENCY	(same)	AC analysis sweep variable
NFID(M1)	(same)	flicker noise from MOSFET M1

Table 13-6 *Output Variable AC Suffixes*

Suffix	Meaning of Output Variables
none	magnitude
DB	magnitude in decibels
G	group delay ($-dPHASE/dFREQUENCY$)
I	imaginary part
M	magnitude
P	phase in degrees
R	real part

Table 13-7 *Device Names for Two-Terminal Device Types*

Two-Terminal Device Type*	Device Type Letter
capacitor	C
diode	D
voltage-controlled voltage source**	E
current-controlled current source**	F
voltage-controlled current source**	G
current-controlled voltage source**	H
independent current source	I
inductor	L
resistor	R
voltage-controlled switch**	S
independent voltage source	V
current-controlled switch**	W

*. The pin name for two-terminal devices is either 1 or 2.

**.. The controlling inputs for these devices are not considered terminals.

Table 13-8 *Terminal IDs by Three & Four-Terminal Device Type*

Three & Four-Terminal Device Type	Device Type Letter	Terminal IDs
GaAs MESFET	B	D (drain) G (gate) S (source)
Junction FET	J	D (drain) G (gate) S (source)
MOSFET	M	D (drain) G (gate) S (source) B (bulk, substrate)
Bipolar transistor	Q	C (collector) B (base) E (emitter) S (substrate)
transmission line	T	A (<i>near</i> side) B (<i>far</i> side)
IGBT	Z	C (collector) G (gate) E (emitter)

Table 13-9 *Noise Types by Device Type*

Device Type	Noise Types*	Meaning
B (GaAsFET)	FID	flicker noise
	RD	thermal noise associated with RD
	RG	thermal noise associated with RG
	RS	thermal noise associated with RS
	SID	shot noise
	TOT	total noise
D (diode)	FID	flicker noise
	RS	thermal noise associated with RS
	SID	shot noise
	TOT	total noise
J (JFET)	FID	flicker noise
	RD	thermal noise associated with RD
	RG	thermal noise associated with RG
	RS	thermal noise associated with RS
	SID	shot noise
	TOT	total noise
M (MOSFET)	FID	flicker noise
	RB	thermal noise associated with RB
	RD	thermal noise associated with RD
	RG	thermal noise associated with RG
	RS	thermal noise associated with RS
	SID	shot noise
	TOT	total noise
Q (BJT)	FIB	flicker noise
	RB	thermal noise associated with RB
	RC	thermal noise associated with RC
	RE	thermal noise associated with RE
	SIB	shot noise associated with base current
	SIC	shot noise associated with collector current
	TOT	total noise
R (resistor)	TOT	total noise
Iswitch	TOT	total noise
Vswitch	TOT	total noise

*. These variables report the contribution of the specified device's noise to the total output noise in units of V^2/Hz . This means that the sum of all device noise contributions is equal to the total output noise in V^2/Hz , $NTOT(ONoise)$.

Analog Trace Expressions

Trace expression aliases

Analog trace expressions in Probe vary from the output variables used in PSpice analyses because net values can be specified by:

$$\langle \text{output variable} \rangle [; \text{display name}]$$

as opposed to the $\langle \text{output variable} \rangle$ format used by PSpice. With this format, the trace expression can be displayed in the legend with an optional alias.

Arithmetic functions

Arithmetic expressions of output variables use the same operators as those used in PSpice (by means of symbol attribute definitions in Schematics). You can also include intrinsic functions in expressions. The intrinsic functions available in Probe are similar to those available for PSpice math expressions, but with some differences, as shown in [Table 13-10](#). A complete list of PSpice arithmetic functions can be found in [Table 3-2 on page 3-16](#).

Table 13-10 *Probe Arithmetic Functions*

Probe Function	Description	Available in PSpice?
ABS(x)	x	YES
SGN(x)	+1 (if x>0), 0 (if x=0), -1 (if x<0)	YES
SQRT(x)	$x^{1/2}$	YES
EXP(x)	e^x	YES
LOG(x)	$\ln(x)$	YES
LOG10(x)	$\log(x)$	YES
M(x)	magnitude of x	YES
P(x)	phase of x (degrees)	YES
R(x)	real part of x	YES
IMG(x)	imaginary part of x	YES
G(x)	group delay of x (seconds)	NO

Table 13-10 *Probe Arithmetic Functions (continued)*

Probe Function	Description	Available in PSpice?
PWR(x,y)	$ x ^y$	YES
SIN(x)	$\sin(x)$	YES
COS(x)	$\cos(x)$	YES
TAN(x)	$\tan(x)$	YES
ATAN(x)	$\tan^{-1}(x)$	YES
ARCTAN(x)		
d(x)	derivative of x with respect to the x-axis variable	YES*
s(x)	integral of x over the range of the x-axis variable	YES**
AVG(x)	running average of x over the range of the x-axis variable	NO
AVGX(x,d)	running average of x from X_axis_value(x)-d to X_axis_value(x)	NO
RMS(x)	running RMS average of x over the range of the x-axis variable	NO
DB(x)	magnitude in decibels of x	NO
MIN(x)	minimum of the real part of x	NO
MAX(x)	maximum of the real part of x	NO

*. In PSpice, this function is called DDT(x).

** . In PSpice, this function is called SDT(x).

Note *For AC analysis, Probe uses complex arithmetic to evaluate expressions. If the result of the expression is complex, then its magnitude is displayed.*

Rules for numeric values suffixes

Explicit numeric values are entered in the same form as PSpice (by means of symbol attributes in Schematics), with the following exceptions:

- Suffixes M and MEG are replaced with m (milli, 1E-3) and M (mega, 1E+6), respectively.
- MIL and mil are not supported.
- With the exception of the m and M scale suffixes, Probe is not case sensitive, therefore upper and lower case characters are equivalent.

Example: V(5) and v(5) are equivalent in Probe.

Example: The quantities 2e-3, 2mV, and .002v all have the same numerical value. For axis labeling purposes, Probe recognizes that the second and third forms are in volts, whereas the first is dimensionless.

Probe also knows that $W=V \cdot A$, $V=W/A$, and $A=W/V$. So, if you add this trace:

V(5)*ID(M13)

the axis values will be labeled with W.

For a demonstration of trace presentation, see [Example on page 13-21](#).

Unit suffixes are *only* used to label the axis; they never affect the numerical results. Therefore, it is always safe to leave off a unit suffix.

The units that Probe recognizes are shown in [Table 13-11](#).

Table 13-11 *Output Units Recognized by Probe*

Symbol	Unit
V	volt
A	amps
W	watt
d	degree (of phase)
s	second
Hz	hertz

Viewing Results on the Schematic

14

Chapter Overview

This chapter describes how to view bias point information directly on your schematic after running a simulation.

[Viewing Bias Point Voltages and Currents on page 14-2](#) explains how to use the interactive bias information display feature from the Analysis menu to dynamically show and hide voltages on wire segments and currents on device pins without having to run extra simulations.

[Other Ways to View Bias Point Values on page 14-11](#) explains the old way of using VIEWPOINT and IPROBE symbols to view bias information.

Note *These symbols are superseded by the interactive method described in [Viewing Bias Point Voltages and Currents on page 14-2](#).*



If you used IPROBE and VIEWPOINT symbols in earlier designs...

...you don't need to anymore. The bias information is always available and saves you steps.

When using symbols like IPROBE and VIEWPOINT, you typically have to:

- 1 Place IPROBE symbols to detect the currents that you are interested in. Each IPROBE symbol introduces a new net into the circuit.
- 2 Run one simulation and use VIEWPOINT symbols to look at bias point voltages.
- 3 Connect IPROBE symbols to any new device pins where you think there may be problems.
- 4 Then run another simulation to actually view the bias point currents on problem devices.

Because you no longer need to place these symbols to view bias information, you can minimize the number of simulations and symbols that you need to debug your circuit.

If you are using hierarchical symbols or blocks...

Schematics automatically displays pin currents for symbols representing PSpice (.SUBCKT and .MODEL) models, but not for hierarchical symbols and blocks. If you want to see currents on the pins of hierarchical symbols, then you need to use IPROBE symbols. To find out more, see [If You Have Hierarchical Symbols or Blocks on Your Schematic on page 14-10](#).

Viewing Bias Point Voltages and Currents

After simulating, you can display bias point information on your schematic so you can find problem areas in your design.

How it works

When simulating, PSpice calculates and saves the bias point voltages and currents. By default, Schematics reads all of this information and displays voltages for every net in your schematic; currents on pins are not visible, but are available for display.

After this is done, it is up to you to decide where you want to show currents and voltages. Schematics handles each separately so that you can:

- show or hide voltages on selected wire segments
- show or hide currents on selected device pins
- turn off all of the displayed voltages, and then redisplay the same voltages later
- turn off all of the displayed currents, and then redisplay the same currents later

If you run more than one analysis type

If you have set up more than one analysis, here are a few things you should know about:

- Schematics always shows bias information for the analysis that you last ran. PSpice runs analyses in this order: DC, AC, transient.

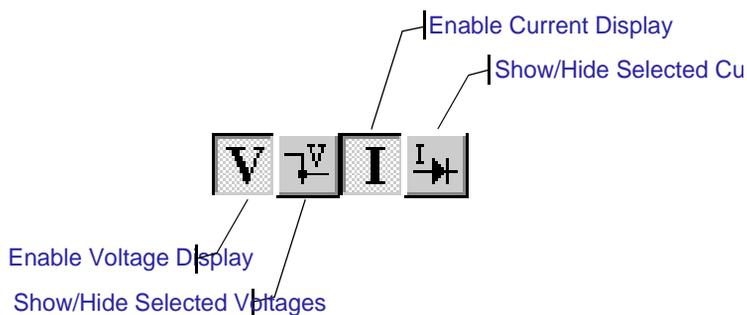
This means if you perform a multi-run analysis like parametric, Monte Carlo, sensitivity/worst-case, or temperature, you will see bias values for the last run only.

- A given voltage or current source can have a different DC value and initial transient value at TIME=0. This means the initial transient bias calculation can be different from the DC (small-signal) bias point.

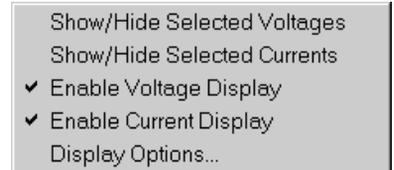
Note For a transient analysis, only the initial transient bias values are available for display.

The Bias Information Toolbar Buttons

The easiest way to control bias information display is to use the buttons on the simulation toolbar. In the following diagram, the description of each toolbar button corresponds to a command in the Display Results on Schematic submenu that you can access from the Analysis menu.



Display Results on Schematic submenu



The Enable Display buttons

The Enable Voltage Display and Enable Current Display buttons are push buttons that let you toggle the display functions. When you enable display by clicking the buttons, they appear *pushed in* as shown above.

The corresponding submenu commands are checked.

The Show/Hide buttons

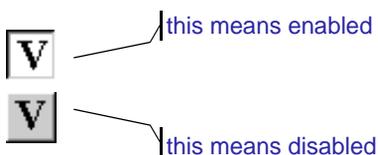
The Show/Hide Selected Voltages and Show/Hide Selected Currents buttons let you toggle the voltage and current display for selected wires and device pins. The buttons are unavailable unless you enable the corresponding display and make appropriate selections on your schematic.

The corresponding submenu commands are dimmed.

Showing Voltages

You can also display the voltage more than once for a net by using the Show/Hide Selected Voltages command on different wire segments.

On the Toolbar



By default, voltage display is initially enabled for all nets in your schematic. This means bias voltage values appear once for each net on each schematic page or level of hierarchy, either under the user-defined label for the net or next to the middle of the longest wire segment.

To enable/disable voltage display

- 1 From the Analysis menu, point to Display Results on Schematic, and then select Enable Voltage Display.

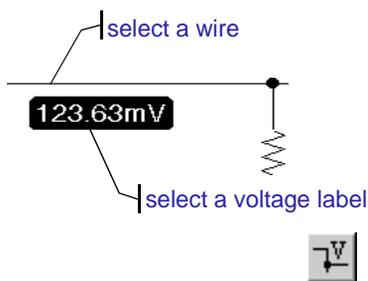
A ✓ means that voltage display is enabled; no check means the voltage display is disabled and the dependent command, Show/Hide Selected Voltages, is dimmed on the submenu and on the toolbar.

Clearing and adding selected voltage values

To clear selected voltages from the existing display

- 1 Do one of the following to select one or more voltage values:
 - Select wire segments that have a voltage value next to them.
 - Select voltage value text (labels).
- 2 From the Analysis menu, point to Display Results on Schematic, and then select Show/Hide Selected Voltages to clear voltages for the selected wire segments.

Note *If your selection includes a mix of cleared and displayed voltage values, selecting Show/Hide Selected Voltages the first time will clear the display for the selected wire segments. To display them, select Show/Hide Selected Voltages again.*



To view a subset of the voltages that are already displayed

- 1 From the Edit menu, select Select All.
- 2 From the Analysis menu, point to Display Results on Schematic, and then select Show/Hide Selected Voltages to hide all of the voltage values. 
- 3 Click to deselect everything.
- 4 Click one or more (shift-click) wire segments or select an area where you want to see voltages on several nets.
- 5 From the Analysis menu, point to Display Results on Schematic, and then select Show/Hide Selected Voltages to display voltages for the selected wire segments. 

To add selected voltages to the existing display

- 1 Select one or more wire segments that do not have a voltage value appearing next to them.
- 2 From the Analysis menu, point to Display Results on Schematic, and then select Show/Hide Selected Voltages to display voltages for the selected wire segments. 

Showing Currents

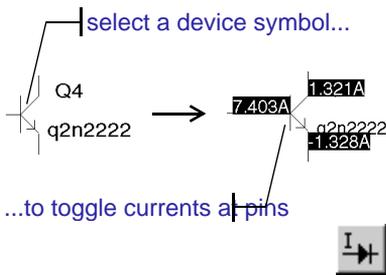
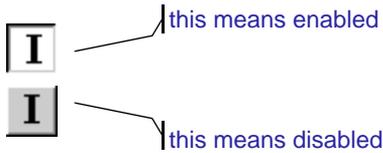
By default, current display is initially disabled for all device pins in your schematic. When you enable current display, the values you see are the currents flowing *into* the pins.

To enable/disable current display

- 1 From the Analysis menu, point to Display Results on Schematic, and then select Enable Current Display.

A ✓ means that current display is enabled; no check means that current display is disabled and the dependent command, Show/Hide Selected Currents, is dimmed on the submenu and on the toolbar.

On the Toolbar



Clearing and adding selected current values

You can add and clear current values using procedures similar to those for voltage values. Keep in mind these differences:

- Current values correspond to pins on part symbols.
- To show or hide current values, select a part symbol that has pins you are interested in.
- Use the Show/Hide Selected Currents command to toggle current display for the selected device's pins.

Note *If your selection includes a mix of devices with cleared and displayed current values, selecting Show/Hide Selected Currents the first time will clear the display for the selected devices' pins. To display them, select Show/Hide Selected Currents again.*

Changing the Precision of Displayed Data

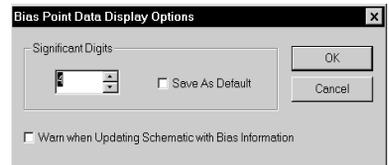
By default, Schematics displays voltage and current values with four significant digits. This means up to a total of four digits on the left and right of the decimal point.

If you want to see more or fewer significant digits, you can change the display setting. Schematics immediately updates the existing display, and uses the new precision value for any voltages and currents that you subsequently display on the same schematic.

Note *An additional simulation is not required.*

To change the precision

- 1 From the Analysis menu, point to Display Results on Schematic, and then select Display Options.
- 2 In the Significant Digits frame, select the number that you want to view.
- 3 If you want to save this as the default for future sessions, select (✓) the Save As Default check box.
- 4 Click OK.



Moving Voltage and Current Labels

To improve readability, you can relocate voltage and current labels for the active editing session. If you later move sections of the schematic, the labels maintain their relative position to the wire (for voltage labels) or pin (for current labels).

To move a voltage or current label to a new location

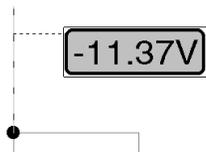
- 1 Click the voltage or current label to select it. A box appears around the label.
- 2 Drag it to the new location.

Note *If you flip or rotate a portion of a schematic, Schematics positions the voltage and current labels at their default position near the relevant wire segment or pin, whether or not you had previously repositioned the label yourself.*

Note Schematics remembers the position of voltage labels, but not current labels. This means if you close (after having saved changes) and then reopen your schematic, the voltage labels appear where you last positioned them; however, Schematics places each current label at its default location near the corresponding pin.

Verifying Label Associations

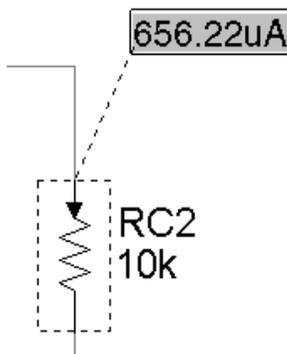
Schematics provides visual cues to help you see what piece of the schematic a voltage or current label describes.



To see which wire segment a voltage value belongs to

- 1 Click the voltage label.

Schematics highlights the wire segment that the voltage label describes and draws a dotted line from the label to the wire segment.



To see which pin a current value belongs to

- 1 Click the current label.

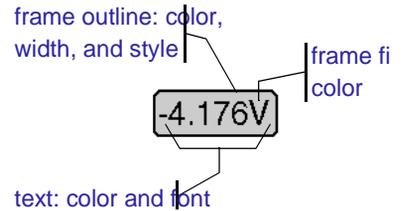
Schematics displays two things:

- an arrow positioned on the corresponding device pin; the arrow points in the direction of the current
- a dotted line from the current label to the device pin that it describes

Changing Display Colors

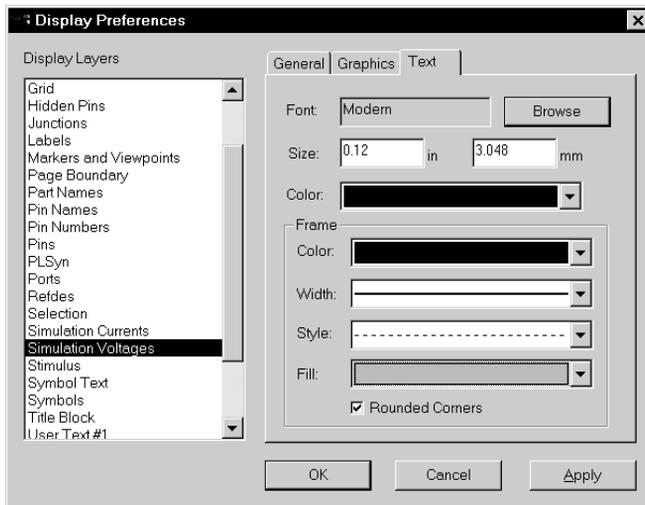
You can change the appearance of the voltage and current labels using Display Preferences in Schematics.

Labels are constructed of a frame and text. The frame defines the label's outline and fill (background) properties. Text defines the color and font of the label's numeric data.



To change display colors for bias information

- 1 In Schematics, from the Options menu, select Display Preferences.



- 2 Select the Display Layer that you want to change.

To change these...	Select this display layer...
--------------------	------------------------------

Voltage labels	Simulation Voltages
----------------	---------------------

Current labels	Simulation Currents
----------------	---------------------

- 3 Click the Text tab.
- 4 Change the frame and text properties as needed.
- 5 Click Apply if you want to leave the dialog box displayed or OK to exit the dialog box.



If you want a transparent frame (which means you see only the text portion of the label), set the frame color and fill to None.

You can also check the title bar in the Schematics window for the word `Stale`.

If you want obsolete voltage and current labels to change appearance

When you change your schematic, the voltage and current values displayed from a previous simulation may no longer be valid. As a reminder, you can have Schematics change the color of the “stale” voltage and current labels.

The procedure to change the label color is the same as that described for active voltage and current labels on page [14-9](#), except that you need to set the colors for a different display layer as follows.

To set display color for this...	Select this display layer...
Stale voltage values	Obsolete Sim. Voltages
Stale current values	Obsolete Sim. Currents

Note *If you push to the lowest level subschematic of a hierarchical symbol, then you can use the automatic bias information display feature described earlier in this chapter.*

If You Have Hierarchical Symbols or Blocks on Your Schematic

Automatic bias information display for pin currents works only with part instances that are defined by PSpice models (built-in or provided in the libraries). Because a hierarchical symbol is defined by a subschematic—not a PSpice model—you need to use an IPROBE symbol connected in series with the symbol’s pin to view current on that pin. See [Using the IPROBE Symbol to Display Current on page 14-11](#).

Other Ways to View Bias Point Values

For the analog portion of your circuit, Schematics provides two special symbols to display bias point information: VIEWPOINT and IPROBE.

Using the VIEWPOINT Symbol to Display Voltage

The VIEWPOINT symbol displays the bias point voltage on a net in your schematic.

To use a VIEWPOINT

- 1 Place the VIEWPOINT symbol and connect it to the net that you are interested in.
- 2 If you haven't yet simulated, run the simulation.

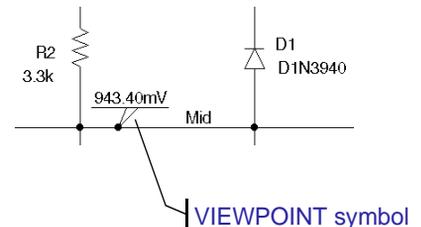
When the simulation completes, Schematics displays the bias point value for the net that the VIEWPOINT connects to. After simulating, you can add VIEWPOINT symbols to display bias point values on other nets.



There's an easier way to view bias information

The IPROBE and VIEWPOINT symbols have been superseded with the bias information display feature that is available from the Analysis menu. To find out more, see [Viewing Bias Point Voltages and Currents on page 14-2](#).

Note *In the case of hierarchical symbols and blocks, you still need to use the IPROBE symbol to view currents.*

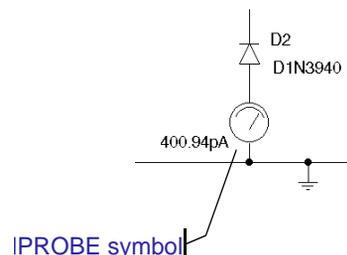


Using the IPROBE Symbol to Display Current

The IPROBE symbol displays the bias point current into a pin. Using an IPROBE symbol is like placing a zero-valued voltage source on your schematic.

To use an IPROBE

- 1 Before simulating, connect the IPROBE symbol in series to the pin you are interested in.



2 Run the simulation.

When the simulation completes, Schematics displays the bias point current next to the IPROBE symbol.

Other Output Options

15

Chapter Overview

This chapter describes how to output results in addition to those normally written to the Probe data file or PSpice output file.

[Viewing Analog Results in the PSpice Window on page 15-2](#) explains how to monitor the numerical values for voltages or currents on up to three nets in your circuit as the simulation proceeds.

[Writing Additional Results to the PSpice Output File on page 15-3](#) explains how to generate additional line plots and tables of voltage and current values to the PSpice output file.

Viewing Analog Results in the PSpice Window

Schematics provides a special WATCH1 symbol that lets you monitor voltage values for up to three nets in your schematic as a DC sweep, AC sweep or transient analysis proceeds. Results display in the PSpice window.

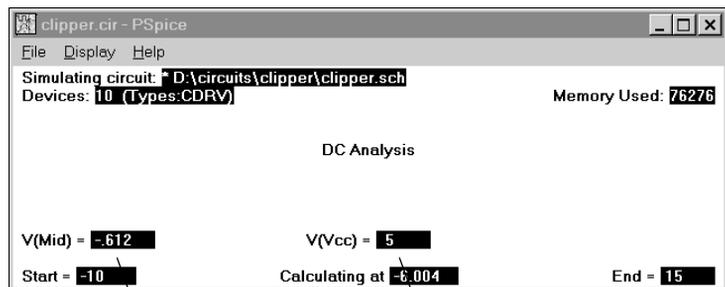
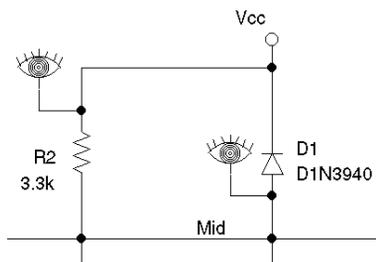
To display voltage values in the PSpice window



If the results move outside of the specified bounds, PSpice A/D pauses the simulation so that you can investigate the behavior.

- 1 Place and connect a WATCH1 symbol on an analog net.
- 2 Double-click the WATCH1 symbol instance.
- 3 Set the ANALYSIS attribute to DC, AC, or TRAN (transient) for the type of analysis results you want to see.
- 4 Set the LO and HI attributes to define the lower and upper bounds, respectively, on the values you expect to see on this net.
- 5 Repeat steps **1** through **4** for up to two more WATCH1 instances.
- 6 Start the simulation.

Example: In the schematic fragment shown below at left, WATCH1 symbols are connected to the Mid and Vcc nets. After starting the simulation, PSpice A/D displays voltages on the Mid and Vcc nets in the PSpice window.



voltage on Mid

voltage on Vcc

Writing Additional Results to the PSpice Output File

Schematics provides special symbols that let you save additional simulation results to the PSpice output file as either line-printer plots or tables.

To view the PSpice output file after having run a simulation:

- 1 From the Analysis menu, select Examine Output.

Generating Plots of Voltage and Current Values

You can generate voltage and current line-printer plots for any DC sweep, AC sweep, or transient analysis.

To generate plots of voltage or current to the output file

- 1 Place and connect any of the following symbols.

Use this symbol...	To plot this...	
VPLOT1	Voltage on the net that the symbol terminal is connected to.	
VPLOT2	Voltage differential between the two nets that the symbol terminals are connected to.	
IPLOT	Current through a net. (Insert this symbol in series, like a current meter.)	

- 2 Double-click the symbol instance.
- 3 Click the attribute name for the analysis type that you want plotted: DC, AC, or TRAN.
- 4 In the Value text box, type any non-blank value such as Y, YES, or 1.

If you do not enable a format, PSpice A/D defaults to MAG.

- 5 If you selected the AC analysis type, enable an output format:
 - a Click the attribute name for one of the following output formats: MAG (magnitude), PHASE, REAL, IMAG (imaginary), or DB.
 - b In the Value text box, type any non-blank value.
 - c Repeat the previous steps (a) and (b) for as many AC output formats as you want to see plotted.
- 6 Repeat steps 2 through 5 for any additional analysis types you want plotted.

Note *If you do not enable an analysis type, PSpice A/D reports the transient results.*

Generating Tables of Voltage and Current Values

You can generate tables of voltage and current values on nets for any DC sweep, AC sweep, or transient analysis.

To generate tables of voltage or current to the output file

- 1 Place and connect any of the following symbols.

	Use this symbol...	To tabulate this...
	VPRINT1	Voltage on the net that the symbol terminal is connected to.
	VPRINT2	Voltage differential between the two nets that the symbol terminals are connected to.
	IPRINT	Current through a cut in the net. (Insert this symbol in series, like a current meter.)

- 2 Double-click the symbol instance.
- 3 Click the attribute name for the analysis type that you want tabulated: DC, AC, or TRAN.

- 4 In the Value text box, type any non-blank value such as Y, YES, or 1.
- 5 If you selected the AC analysis type, enable an output format.
 - a Click the attribute name for one of the following output formats: MAG (magnitude), PHASE, REAL, IMAG (imaginary), or DB.
 - b In the Value text box, type any non-blank value.
 - c Repeat the previous steps (a) and (b) for as many AC output formats as you want to see tabulated.
- 6 Repeat steps 2 through 5 for any additional analysis types you want plotted.

If you do not enable a format, PSpice A/D defaults to MAG.

Note *If you do not enable an analysis type, PSpice A/D reports the transient results.*

Setting Initial State

A

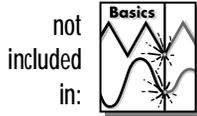
Appendix Overview

This appendix includes the following sections:

[Save and Load Bias Point on page A-2](#)

[Setpoints on page A-4](#)

[Setting Initial Conditions on page A-6](#)



If the circuit uses high gain components, or if the circuit's behavior is nonlinear around the bias point, this feature is not useful.

Save and Load Bias Point

Save Bias Point and Load Bias Point are used to save and restore bias point calculations in successive PSpice simulations. Saving and restoring bias point calculations can decrease simulation times when large circuits are run multiple times and can aid convergence.

Save/Load Bias Point affect the following types of analyses:

- transient
- DC
- AC

Save Bias Point

Save bias point is a simulation control function that allows you to save the bias point data from one simulation for use as initial conditions in subsequent simulations. Once bias point data is saved to a file, you can use the load bias point function to use the data for another simulation.

To use save bias point

- 1 In the Analysis Setup dialog box, click Load Bias Point.
- 2 Complete the Save Bias Point dialog box as described in the *Schematics User's Guide*. Click OK when finished.
- 3 Make sure the check box next to Save Bias Point is selected (✓).

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.

Load Bias Point

Load bias point is a simulation control function that allows you to set the bias point as an initial condition. A common reason for giving PSpice initial conditions is to select one out of two or more stable operating points (set or reset for a flip-flop, for example).

To use load bias point

- 1 Run a simulation using Save Bias Point for the Analysis Setup dialog box.
- 2 Prior to running another simulation, click Load Bias Point.
- 3 Specify a bias point file to load. Include the path if the file is not located in your working directory. Click OK when finished.
- 4 Make sure the check box next to Load Bias Point is selected (✓).

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.

Setpoints

Pseudocomponents that specify initial conditions are called setpoints. These apply to the analog portion of your circuit.

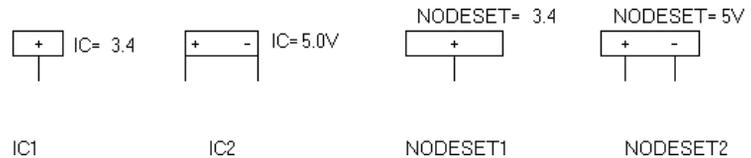


Figure A-1 Setpoints

The example in Figure A-1 includes the following:

- IC1** a one-pin symbol that allows you to set the initial condition on a net for both small-signal and transient bias points
- IC2** a two-pin symbol that allows you to set initial condition between two nets

Using IC symbols sets the initial conditions for the bias point only. It does not affect the DC sweep. If the circuit contains both an IC symbol and a NODESET symbol for the same net, the NODESET symbol is ignored.

To specify the initial condition, edit the value of the VALUE attribute to the desired initial condition. PSpice attaches a voltage source with a 0.0002 ohm series resistance to each net to which an IC symbol is connected. The voltages are clamped this way for the entire bias point calculation.

NODESET1 is a one-pin symbol which helps calculate the bias point by providing an initial guess for some net. NODESET2 is a two-pin symbol which helps calculate the bias point between two nets. Some or all of the circuit's nets may be given an initial guess. NODESET symbols are effective for the bias point (both small-signal and transient bias points) and for the first step of the DC sweep. It has no effect during the rest of the DC sweep or during the transient analysis itself.

Unlike the IC pseudocomponents, NODESET provides only an initial guess for some net voltages. It does not clamp those nodes to the specified voltages. However, by providing an initial guess, NODESET symbols may be used to break the tie (in a flip-flop, for instance) and make it come up in a desired state. To guess at the bias point, enter the initial guess in the Value text box for the VALUE attribute. PSpice attaches a voltage source with a 0.0002 ohm series resistance to each net to which an IC symbol is connected.

These pseudocomponents are netlisted as PSpice .IC and .NODESET commands. Refer to these commands in the online *MicroSim PSpice A/D Reference Manual* for more information. Setpoints can be created for inductor currents and capacitor voltages using the IC attribute described in [Setting Initial Conditions on page A-6](#).

Setting Initial Conditions

The IC attribute allows initial conditions to be set on capacitors and inductors. These conditions are applied during all bias point calculations. However, if you select (✓) the Skip Initial Transient Solution check box in the Transient Analysis Setup dialog box, the bias point calculation is skipped and the simulation proceeds directly with transient analysis at TIME=0. Devices with the IC attribute defined start with the specified voltage or current value; however, all other such devices have an initial voltage or current of 0.

Note *Skipping the bias point calculation can make the transient analysis subject to convergence problems.*

See [Setpoints on page A-4](#) for more information about IC1 and IC2.

Applying an IC attribute for a capacitor has the same effect as applying one of the pseudocomponents IC1 or IC2 across its nodes. PSpice attaches a voltage source with a 0.002 ohm series resistance in parallel with the capacitor. The IC attribute allows the user to associate the initial condition with a device, while the IC1 and IC2 pseudocomponents allow the association to be with a node or node pair.

In the case of initial currents through inductors, the association is only with a device, and so there are no corresponding pseudocomponents. The internal implementation is analogous to the capacitor. PSpice attaches a current source with a 1 Gohm parallel resistance in series with the inductor.

If you want IC attributes to be ignored when Skip Initial Transient Solution is not enabled in the Transient Analysis Setup dialog box:

- 1 Click Options in the Setup Analysis dialog box.
- 2 Set NOICTRANSLATE to Y.

See [Setting Up Analyses on page 7-3](#) for a description of the Analysis Setup dialog box.

Convergence and “Time Step Too Small Errors”

B

Appendix Overview

This appendix discusses common errors and convergence problems in PSpice.

[Introduction on page B-2](#)

[Bias Point and DC Sweep on page B-7](#)

[Transient Analysis on page B-10](#)

[Diagnostics on page B-15](#)

Introduction

In order to calculate the bias point, DC sweep and transient analysis for analog devices PSpice must solve a set of nonlinear equations which describe the circuit's behavior. This is accomplished by using an iterative technique - the Newton-Raphson algorithm - which starts by having an initial approximation to the solution and iteratively improves it until successive voltages and currents converge to the same result.

In a few cases PSpice cannot find a solution to the nonlinear circuit equations. This is generally called a “convergence problem” because the symptom is that the Newton-Raphson repeating series cannot converge onto a consistent set of voltages and currents. The following discussion gives some background on the algorithms in PSpice and some guidelines for avoiding convergence problems.

The transient analysis has the additional possibility of being unable to continue because the time step required becomes too small from something in the circuit moving too fast. This is also discussed below.

The AC and noise analyses are linear and do not use an iterative algorithm. The following discussion does not apply to them.

Newton-Raphson Requirements

The Newton-Raphson algorithm has the very nice property that *it is guaranteed to converge to a solution*. However, this nice property has some serious strings attached:

- 1 The nonlinear equations must have a solution.
- 2 The equations must be continuous.
- 3 The algorithm needs the equations' derivatives.
- 4 The initial approximation must be close enough to the solution.

Each of these can be taken in order. One must keep in mind that PSpice's algorithms are used in computer hardware that has

finite precision and finite dynamic range which produce these limits:

- voltages and currents in PSpice are limited to +/-1e10 volts and amps,
- derivatives in PSpice are limited to 1e14, and
- the arithmetic used in PSpice is double precision and has 15 digits of accuracy.

Is There a Solution?

The answer is yes for any physically realistic circuit. However, it is not difficult to set up a circuit which does not have a solution within the limits of PSpice's numerics. Consider, for example, a voltage source of one megavolt connected to a resistor of one micro-ohm. This circuit does not have a solution within the dynamic range of currents (+/- 1e10 amps). Here is a sneakier example:

```
V1      1,      0      5v
D1      1,      0      DMOD
.MODEL          DMOD( IS=1e-16 )
```

The problem here is that the diode model has no series resistance. It can be shown that the current through a diode is:

$$I = IS * e^{V/(N * k * T)}$$

N defaults to one and $k * T$ at room temperature is about .025 volts. So, in this example the current through the diode would be:

$$I = 1e-16 * e^{200} = 7.22e70 \text{ amps}$$

This circuit also does not have solution within the limits of the dynamic range of PSpice. In general, you should be careful of components without limits built into them. Extra care is needed when using the expressions for controlled sources (i.e., behavioral modeling). It is easy to write expressions whose values can be very large.

To find out more about the diode equations, refer to the *Analog Devices* chapter in the the online *MicroSim PSpice A/D Reference Manual*.

Are the Equations Continuous?

The device equations built into PSpice are continuous. The functions available for behavioral modeling are also continuous (there are several functions, such as $\text{int}(x)$, which cannot be added because of this). So, for physically realistic circuits the equations can also be continuous. Exceptions that come are usually from exceeding the limits of the numerics in PSpice. Consider the following attempt to approximate an ideal switch using the diode model:

```
.MODEL DMOD (IS=1e-16 N=1e-6)
```

The current through this diode is:

$$I = 1e-16 * e^{V/(N * 0.025)} = 1e-16 * e^{V/25e-9}$$

Because the denominator in the exponential is so small, the current I is essentially zero for $V < 0$ and almost infinite for $V > 0$. Even if there are external components that limit the current the “knee” of the diode’s I-V curve is so sharp that it is almost a discontinuity. The caution again is to avoid unrealistic model parameters. Behavioral modeling expressions need extra care.

Are the derivatives correct?

The device equations built into PSpice include the derivatives and these are correct. Depending on the device, the physical meaning of the derivatives is small-signal conductance, transconductance or gain. Unrealistic model parameters can exceed the limit of $1e14$, but it requires some effort. The main thing to look at is the behavioral modeling expressions, especially those having denominators.

Is the Initial Approximation Close Enough?

It seems like a Catch-22: Newton-Raphson is guaranteed to converge only if the analysis is started close to the answer. Worse yet, there is no measurement that can tell how close is close enough.

PSpice gets around this by making heavy use of continuity. Each analysis starts from a known solution and uses a variable step size to find the next solution. If the next solution does not converge PSpice reduces the step size, falls back and tries again.

Bias point

The hardest part of the whole process is getting started. That is, finding the bias point. PSpice first tries with the power supplies set to 100%. A solution is not guaranteed, but most of the time the PSpice algorithm finds one. If not, then the power supplies are cut back to almost zero. They are cut to a level small enough that *all nonlinearities are turned off*. When the circuit is linear a solution can be found (very near zero, of course). Then, PSpice works its way back up to 100% power supplies using a variable step size.

Once a bias point is found the transient analysis can be run. It starts from a known solution (the bias point) and steps forward in time. The step size is variable and is reduced as needed to find further solutions.

DC sweep

The DC sweep uses a hybrid approach. It uses the bias point algorithm (varying the power supplies) to get started. For subsequent steps it uses the previous solution as the initial approximation. The sweep step is not variable, however. If a solution cannot be found at a step then the bias point algorithm is used for that step.

The whole process relies heavily on continuity. It also requires that the circuit be linear when the supplies are turned off.

STEPGMIN

An alternative algorithm is GMIN stepping. This is not obtained by default, and is enabled by specifying the circuit analysis option STEPGMIN (either using `.OPTION STEPGMIN` in the netlist, or by making the appropriate choice from the Analysis/Setup/Options menu). When enabled, the GMIN stepping algorithm is applied after the circuit fails to converge with the power supplies at 100 percent, and if GMIN stepping also fails, the supplies are then cut back to almost zero.

GMIN stepping attempts to find a solution by starting the repeating cycle with a large value of GMIN, initially $1.0e10$ times the nominal value. If a solution is found at this setting it then reduces GMIN by a factor of 10, and tries again. This continues until either GMIN is back to the nominal value, or a repeating cycle fails to converge. In the latter case, GMIN is restored to the nominal value and the power supplies are stepped.

Bias Point and DC Sweep

Power supply stepping

As previously discussed, PSpice uses a proprietary algorithm which finds a continuous path from zero power supplies levels to 100%. It starts at almost zero (.001%) power supplies levels and works its way back up to the 100% levels. The minimum step size is $1e-6$ (.0001%). The first repeating series of the first step *starts at zero for all voltages*.

Semiconductors

Model parameters

The first consideration for semiconductors is to avoid physically unrealistic model parameters. Remember that as PSpice steps the power supplies up it has to step carefully through the turn on transition for each device. In the diode example above, for the setting $N=1e-6$, the knee of the I-V curve would be too sharp for PSpice to maintain its continuity within the power supply step size limit of $1e-6$.

Unguarded p-n junctions

A second consideration is to avoid “unguarded” p-n junctions (no series resistance). The above diode example also applies to the p-n junctions inside bipolar transistors, MOSFETs (drain-bulk and source-bulk), JFETs and GaAsFETs.

No leakage resistance

A third consideration is to avoid situations which could have an ideal current source pushing current into a reverse-biased p-n junction without a shunt resistance. Since p-n junctions in PSpice have (almost) no leakage resistance and would cause the junction's voltage to go beyond 1e10 volts.

The model libraries which are part of PSpice follow these guidelines.

Typos can cause unrealistic device parameters. The following MOSFET:

```
M1 3, 2, 1, 0 MMOD L=5 W=3
```

has a length of five meters and a width of three meters instead of micrometers. It should have been:

```
M1 3, 2, 1, 0 MMOD L=5u W=3u
```

PSpice flags an error for L too large, but cannot for W because power MOSFETs are so interdigitated (a zipper-like trace) that their effective W can be very high. The LIST option can show this kind of problem. When the devices are listed in the output file their values are shown in scientific notation making it easy to spot unusual values.

Switches

PSpice switches have gain in their transition region. If several are cascaded then the cumulative gain can easily exceed the derivative limit of 1e14. This can happen when modeling simple logic gates using totem-pole switches and there are several gates in cascaded in series. Usually a cascade of two switches works but three or more can cause trouble.

Behavioral Modeling Expressions

Range limits

Voltages and currents in PSpice are limited to the range $\pm 1e10$. Care must be taken that the output of expressions fall within this range. This is especially important when one is building an electrical analog of a mechanical, hydraulic or other type of system.

Source limits

Another consideration is that the controlled sources must turn off when the supplies are almost 0 (.001%). There is special code in PSpice which “squelsches” the controlled sources in a continuous way near 0 supplies. However, care should still be taken using expressions that have denominators. Take, for example, a constant power load:

```
GLOAD 3, 5 VALUE = {2Watts/V(3,5)}
```

The first repeating series starts with $V(3,5) = 0$ and the current through GLOAD would be infinite (actually, the code in PSpice which does the division clips the result to a finite value). The “squelsching” code is required to be a smooth and well-behaved function.

Note *The “squelsching” code cannot be “strong” enough to suppress dividing by 0.*

The result is that GLOAD does not turn off near 0 power supplies. A better way is described in the application note Modeling Constant Power Loads. The “squelsching” code is sufficient for turning off all expressions except those having denominators. In general, though, it is good practice to constrain expressions having the LIMIT function to keep results within physically realistic bounds.

Example: A first approximation to an opamp that has an open loop gain of 100,000 is:

```
VOPAMP 3, 5 VALUE = {V(in+,in-)*1e5}
```

This has the undesirable property that there is no limit on the output. A better expression is:

```
VOPAMP 3, 5 VALUE =  
+ {LIMIT(V(in+,in-)*1e5,15v,-15v)}
```

where the output is limited to +/- 15 volts.

Transient Analysis

The transient analysis starts using a known solution - the bias point. It then uses the most recent solution as the first guess for each new time point. If necessary, the time step is cut back to keep the new time point close enough that the first guess allows the Newton-Raphson repeating series to converge. The time step is also adjusted to keep the integration of charges and fluxes accurate enough.

In theory the same considerations which were noted for the bias point calculation apply to the transient analysis. However, in practice they show up during the bias point calculation first and, hence, are corrected before a transient analysis is run.

The transient analysis can fail to complete if the time step gets too small. This can have two different effects:

- 1 The Newton-Raphson iterations would not converge even for the smallest time step size, or
- 2 Something in the circuit is moving faster than can be accommodated by the minimum step size.

The message PSpice puts into the output file specifies which condition occurred.

Skipping the Bias Point

The SKIPBP option for the transient analysis skips the bias point calculation. In this case the transient analysis has no known solution to start from and, therefore, is not assured of converging at the first time point. Because of this, its use is not recommended. Its inclusion in PSpice is to maintain compatibility with UC Berkeley SPICE. SKIPBP has the same meaning as UIC in Berkeley SPICE. UIC is not needed in order to specify initial conditions.

The Dynamic Range of TIME

TIME, the simulation time during transient analysis, is a double precision variable which gives it about 15 digits of accuracy. The dynamic range is set to be 15 digits minus the number of digits of accuracy required by RELTOL. For a default value of $RELTOL = .001$ (.1% or 3 digits) this gives $15 - 3 = 12$ digits. This means that the minimum time step is the overall run time (TSTOP) divided by $1e12$. The dynamic range is large but finite.

It is possible to exceed this dynamic range in some circuits. Consider, for example, a timer circuit which charges up a 100uF capacitor to provide a delay of 100 seconds. At a certain threshold a comparator turns on a power MOSFET. The overall simulation time is 100 seconds. For default RELTOL this gives us a minimum time step of 100 picoseconds. If the comparator and other circuitry has portions that switch in a nanosecond then PSpice needs steps of less than 100 picoseconds to calculate the transition accurately.

Failure at the First Time Step

If the transient analysis fails at the first time point then usually there is an unreasonably large capacitor or inductor. Usually this is due to a typographical error. Consider the following capacitor:

```
C 1 3, 0 10uf
```

“1O” (has the letter O) should have been “10.” This capacitor has a value of one farad, not 10 microfarads. An easy way to catch these is to use the LIST option (on the .OPTIONS command).

LIST

The LIST option can echo back all the devices into the output file *that have their values in scientific notation*.

That makes it easy to spot any unusual values. This kind of problem does not show up during the bias point calculation because capacitors and inductors do not participate in the bias point.

Similar comments apply to the parasitic capacitance parameters in transistor (and diode) models. These are normally echoed to the output file (the NOMOD option suppresses the echo but the default is to echo). As in the LIST output, the model parameters are echoed in scientific notation making it easy to spot unusual values. A further diagnostic is to ask for the detailed operating bias point (.TRAN/OP) information.

.TRAN/OP

This lists the small-signal parameters for each semiconductor device including the calculated parasitic capacitances.

Parasitic Capacitances

It is important that switching times be nonzero. This is assured if devices have parasitic capacitances. The semiconductor model libraries in PSpice have such capacitances. If switches and/or controlled sources are used, then care should be taken to assure that no sections of circuitry can try to switch in zero time. In practice this means that if any positive feedback loops exist (such as a Schmidt trigger built out of switches) then such loops should include capacitances.

Another way of saying all this is that during transient analysis the circuit equations must be continuous over time (just as during the bias point calculation the equations must be continuous with the power supply level).

Inductors and Transformers

While the impedance of capacitors gets lower at high frequencies (and small time steps) the impedance of inductors gets higher.

Note *The inductors in PSpice have an infinite bandwidth.*

Real inductors have a finite bandwidth due to eddy current losses and/or skin effect. At high frequencies the effective inductance drops. Another way to say this is that physical inductors have a frequency at which their Q begins to roll off. The inductors in PSpice have no such limit. This can lead to very fast spikes as transistors (and diodes) connected to inductors turn on and off. The fast spikes, in turn, can force PSpice to take unrealistically small time steps.

Note *MicroSim recommends that all inductors have a parallel resistor (series resistance is good for modeling DC effects but does not limit the inductor's bandwidth).*

The parallel resistor gives a good model for eddy current loss and limits the bandwidth of the inductor. The size of resistor should be set to be equal to the inductor's impedance at the frequency at which its Q begins to roll off.

Example: A common one millihenry iron core inductor begins to roll off at no less than 100KHz. A good resistor value to use in parallel is then $R = 2 * \pi * 100e3 * .001 = 628$ ohms. Below the roll-off frequency the inductor dominates; above it the resistor does. This keeps the width of spikes from becoming unreasonably narrow.

Bipolar Transistors Substrate Junction

The UC Berkeley SPICE contains an unfortunate convention for the substrate node of bipolar transistors. The collector-substrate p-n junction has *no DC component*. If the capacitance model parameters are specified (e.g., CJS) then the junction has (voltage-dependent) capacitance but no DC current. This can lead to a sneaky problem: if the junction is inadvertently forward-biased it can create a very large capacitance. The capacitance goes as a power of the junction voltage. Normal junctions cannot sustain much forward voltage because a large current flows. The collector-substrate junction is an exception because it has no DC current.

If this happens it usually shows up at the first time step. It can be spotted turning on the detailed operating point information (.TRAN/OP) and looking at the calculated value of CJS for bipolar transistors. The whole problem can be prevented by using the PSpice model parameter ISS. This parameter “turns on” DC current for the substrate junction.

Diagnostics

If PSpice encounters a convergence problem it inserts into the output file a message that looks like the following.

```
ERROR -- Convergence problem in transient analysis at Time = 7.920E-03
```

```
Time step = 47.69E-15, minimum allowable step size = 300.0E-15
```

```
These voltages failed to converge:
```

```
V(x2.23) = 1230.23 / -68.4137
V(x2.25) = -1211.94 / 86.6888
```

```
These supply currents failed to converge:
```

```
I(X2.L1) = -36.6259 / 2.25682
I(X2.L2) = -36.5838 / 2.29898
```

```
These devices failed to converge:
```

```
X2.DCR3 X2.DCR4 x2.ktr X2.Q1 X2.Q2
```

```
Last node voltages tried were:
```

NODE	VOLTAGE	NODE	VOLTAGE	NODE	VOLTAGE	NODE	VOLTAGE
(1)	25.2000	(3)	4.0000	(4)	0.0000	(6)	25.2030
(x2.23)	1230.2000	(X2.24)	9.1441	(x2.25)	-1211.9000	(X2.26)	256.9700
(X2.28)	-206.6100	(X2.29)	75.4870	(X2.30)	-25.0780	(X2.31)	26.2810
(X3.34)	1.771E-06	(X3.35)	1.0881	(X3.36)	.4279	(X2.XU1.6)	1.2636

The message always includes the banner (ERROR -- convergence problem...) and the trailer (Last node voltages tried were...). It cannot include all three of the middle blocks.

The `Last node voltages tried...` trailer shows the voltages tried at the last Newton-Raphson iteration. If any of the nodes have unreasonable large values this is a clue that these nodes are related to the problem. “These voltages failed to converge” lists the specific nodes which did not settle onto consistent values. It also shows their values for the last two iterations. “These supply currents failed converge” does the same for currents through voltage sources and inductors. If any of the listed numbers are +/- 1e10 then that is an indication that the value is being clipped from an unreasonable value. Finally, “These devices failed to converge” shows devices whose terminal currents or core fluxes did not settle onto consistent values.

The message gives a clue as to the part of the circuit which is causing the problem. Looking at those devices and/or nodes for the problems discussed above is recommended.

Index

A

ABM

- ABM part templates, 6-6
- abm.slb, 6-3
- basic controlled sources, 6-46
- cautions and recommendations for simulation, 6-40
- control system parts, 6-7
- custom parts, 6-46
- frequency domain device models, 6-35
- frequency domain parts, 6-35, 6-41
- instantaneous models, 6-30, 6-40
- overview, 6-2
- placing and specifying ABM parts, 6-4
- PSpice A/D-equivalent parts, 6-28, 6-29
- signal names, 6-1
- simulation accuracy, 6-45
- syntax, 6-29
- triode modeling example, 6-25

AC stimulus attribute, 9-4

AC sweep analysis, 7-2, 9-1, 9-2

- about, 9-2
- displaying results in Probe, 2-22
- example, 2-20, 9-6
- introduction, 1-4
- noise analysis, 7-2, 9-9

setup, 2-20, 9-2, 9-5

stimulus, 9-3

treatment of nonlinear devices, 9-7

ACMAG stimulus attribute, 9-4

ACPHASE stimulus attribute, 9-4

adding a stimulus, 2-16

AGND ground symbol, 3-26

AKO (A Kind Of)

model, 4-30

symbol, 5-8

analog behavioral modeling, *see* ABM

analog parts

basic components (ABM), 6-7, 6-9

basic controlled sources (ABM), 6-46

behavioral, 3-12

bipolar transistor, 13-38, 13-39

bipolar transistors, 4-12, 7-9

breakout, 3-11

capacitors, 7-8

Chebyshev filters, 6-7, 6-11, 6-43, 12-19

diodes, 4-12, 7-8, 13-39

expression parts (ABM), 6-8, 6-21

frequency table parts (ABM), 6-28, 6-37, 6-44

GaAsFET, 7-9, 13-38, 13-39

IGBT, 4-12, 7-9, 13-38

inductors, 7-8

- integrators and differentiators (ABM), 6-7, 6-14
- JFET, 4-12, 7-9, 13-38, 13-39
- Laplace transform (ABM), 6-8, 6-18, 6-28, 6-35, 6-41
- limiters (ABM), 6-7, 6-10
- math functions (ABM), 6-8, 6-21
- mathematical expressions (ABM), 6-28
- MOSFET, 4-12, 7-9, 13-38, 13-39
- nonlinear magnetic core, 4-12
- opamp, 4-12
- passive, 3-10
- PSpice A/D-equivalent parts (ABM), 6-28
- resistors, 7-8, 13-39
- switch, 13-39
- table look-up (ABM), 6-7, 6-14, 6-28, 6-33
- transmission lines, 7-9, 13-38
- vendor-supplied, 3-7
- voltage comparator, 4-12
- voltage reference, 4-12
- voltage regulator, 4-12

analyses

- AC sweep, 2-20, 7-2, 9-1, 9-2
- bias point detail, 2-6, 7-2, 8-8
- DC sensitivity, 7-2, 8-12
- DC sweep, 2-10, 7-2, 8-2
- execution order, 7-4
- Fourier, 7-2
- frequency response, 7-2
- Monte Carlo, 7-2, 12-7
- noise, 7-2, 9-9
- overview, 1-3
- parametric, 2-24, 7-2, 11-2
- performance analysis, 2-30
- sensitivity/worst-case, 7-2, 12-25
- setup, 7-3
- small-signal DC transfer, 7-2, 8-10
- temperature, 7-2, 11-11
- transient, 2-16, 7-2
- types, 7-2

approximation, problems, B-5

attributes (symbol) for simulation, 5-18

B

- basic components (ABM), 6-7, 6-9
- basic controlled sources (ABM), 6-46
- behavioral modeling expressions, B-9
- behavioral parts, 3-12
- bias information display (schematic)

- changing display colors, 14-9
- changing precision, 14-7
- example, 2-7
- how it works, 14-2
- moving labels, 14-7
- replacing VIEWPOINT and IPROBE symbols, 14-2
- showing/hiding selected currents, 14-6
- showing/hiding selected voltages, 14-4
- toolbar, 14-3
- verifying label associations, 14-8

bias point

- convergence analysis, B-11
- display on schematic, 14-2
- save/restore, A-2
- skipping, B-11

bias point detail analysis, 7-2, 8-8

- example, 2-6
- introduction, 1-3

bipolar transistor, 13-38, 13-39

bipolar transistors, 4-12, 7-9

- problems, B-14

Bode plot, 1-4, 2-22

bounding box, symbol, 5-16

C

- capacitors, 7-8
- Chebyshev filters, 6-7, 6-11, 6-43, 12-19
- circuit file (.cir), 1-10
 - simulating multiple circuits, 7-12
- comparator, 4-12
- continuous equations
 - problems, B-4
- control system parts (ABM), 6-7
- controlled sources, 6-28, 6-46
- convergence analysis
 - bias point, B-11
- convergence problems, B-1
 - approximations, B-5
 - behavioral modeling expressions, B-9
 - bias point, B-7
 - bipolar transistors, B-14
 - continuous equations, B-4
 - DC sweep, B-7
 - derivatives, B-4
 - diagnostics, B-15
 - dynamic range of time, B-11
 - inductors and transformers, B-13
 - Newton-Raphson requirements, B-2

parasitic capacitances, [B-13](#)

semiconductors, [B-7](#)

switches, [B-8](#)

transient analysis, [B-10](#)

Create Subcircuit command, [4-7](#), [4-37](#)

current source, controlled, [6-28](#), [6-46](#)

cursors, Probe, [13-29](#)

custom symbol creation for models, [5-13](#)

using the Parts utility, [4-16](#), [5-11](#)

using the symbol wizard, [5-6](#)

D

DC analyses

displaying results in Probe, [2-11](#)

see also DC sweep analysis, bias point detail analysis, small-signal DC transfer analysis, DC sensitivity analysis

DC sensitivity analysis, [7-2](#), [8-12](#)

introduction, [1-3](#)

DC stimulus attribute, [8-5](#)

DC sweep analysis, [7-2](#), [8-2](#)

about, [8-3](#)

curve families, [8-6](#)

example, [2-10](#)

introduction, [1-3](#)

nested, [8-5](#)

setting up, [2-10](#)

stimulus, [8-4](#)

derivative

problems, [B-4](#)

device noise, [9-10](#), [9-12](#)

total, [9-12](#)

diagnostic problems, [B-15](#)

differentiators (ABM), [6-7](#), [6-14](#)

digital parts

vendor-supplied, [3-7](#)

digital simulation

Probe waveform display, [13-40](#)

diode, [13-39](#)

diodes, [4-12](#), [7-8](#)

dynamic range of time, [B-11](#)

E

EGND ground symbol, [3-26](#)

examples and tutorials

AC sweep, [9-6](#)

AC sweep analysis, [2-20](#)

bias point detail analysis, [2-6](#)

DC sweep analysis, [2-10](#)

example circuit creation, [2-2](#)

frequency response vs. arbitrary parameter, [11-8](#)

modeling a triode (ABM), [6-25](#)

Monte Carlo analysis, [12-10](#)

parametric analysis, [2-24](#)

performance analysis, [2-30](#), [11-3](#)

transient analysis, [2-16](#)

using Probe (analog waveform analysis), [13-20](#)

using the model editor, [4-35](#)

using the Parts utility, [4-23](#)

worst-case analysis, [12-28](#)

expression parts (ABM), [6-8](#), [6-21](#)

expressions, [3-14](#), [3-15](#)

see also parameters

ABM, [6-28](#)

functions, [3-16](#)

Probe, [13-40](#)

specifying, [3-14](#)

system variables, [3-18](#)

F

files

generated by Schematics, [1-10](#)

user-configurable, [1-11](#)

with simulation results, [1-14](#)

flicker noise, [9-12](#)

Fourier analysis, [7-2](#)

introduction, [1-5](#)

FREQUENCY output variable, [13-35](#)

frequency response vs. arbitrary parameter, [11-8](#)

frequency table parts (ABM), [6-28](#), [6-37](#), [6-44](#)

functions

Probe, [13-40](#)

PSpice A/D, [3-16](#)

G

GaAsFET, [7-9](#), [13-38](#), [13-39](#)

goal functions, [11-5](#)

in performance analysis, [11-5](#)

single data point, [11-5](#)

grid spacing

symbol graphics, [5-17](#)

symbol pins, [5-17](#)

ground

missing, [3-26](#)

- missing DC path to, 3-27
- symbols, 3-6
- group delay, 13-37

H

- histograms, 12-19
- hysteresis curves, 10-18

I

- IAC stimulus symbol, 9-3
- IC (attribute), A-6
- ICn initial conditions symbol, A-4
- IDC stimulus symbol, 3-18, 8-4
- IGBT, 4-12, 7-9, 13-38
- imaginary part, 13-37
- include files, 1-11
 - configuring, 1-13, 4-41
 - with model definitions, 4-43
- inductors, 7-8
 - problems, B-13
- initial conditions, A-2, A-6
- input noise, total, 9-12
- instance models
 - changing model references, 4-38
 - editing, 4-22
 - model editor, 4-33
 - Parts utility, 4-20
 - reusing, 4-39
 - saving for global use instead
 - using the model editor, 4-34
 - using the Parts utility, 4-21
- integrators (ABM), 6-7, 6-14
- IPLOT (write current plot symbol), 15-3
- IPRINT (write current table symbol), 15-4
- IProbe view bias point current symbol, 14-11
- ISRC stimulus symbol, 3-18, 8-4, 9-3
 - how to use, 3-21
- ISTIM stimulus symbol, 3-20

J

- JFET, 4-12, 7-9, 13-38, 13-39

L

- Laplace transform parts (ABM), 6-8, 6-18, 6-28, 6-35, 6-41, 6-42
- libraries
 - configuring, 4-41
 - footprint, 1-13
 - model, 4-4
 - package, 1-13
 - searching for models, 4-43
 - symbol, 1-13
 - see also* model libraries
- Library List, using, 3-9
- limiters (ABM), 6-7, 6-10

M

- magnetic core, nonlinear, 4-12
- magnitude, 13-37
- markers, 2-11, 13-15
 - for limiting Probe data file size, 13-15
 - for Probe trace display, 13-12
- math function parts (ABM), 6-8, 6-21
- mathematical expressions (ABM), 6-28
- mixed analog/digital circuits
 - Probe waveform display, 13-40
- MODEL attribute, 4-3, 5-18
- model editor
 - about, 4-29
 - changing
 - .MODEL definitions, 4-30
 - .SUBCKT definitions, 4-31
 - model names, 4-31
 - compared to the Parts utility, 4-7
 - example, 4-35
 - running from the
 - schematic editor, 4-33
 - symbol editor, 4-31
- model libraries, 1-11, 4-4
 - adding to the configuration, 4-44
 - analog list of, 3-23
 - and duplicate model names, 4-43
 - configuration, 4-5
 - configured as include files, 4-43
 - configuring, 1-13, 3-24, 4-41
 - directory search path, 4-46
 - global vs. local, 4-5, 4-45
 - how PSpice searches them, 4-43
 - MicroSim-provided, 4-6

nested, 4-6
 preparing for symbol creation, 5-5
 search order, 4-43, 4-45

models

- built-in, 1-2
- changing associations to symbols, 4-38
- converting AKOs to non-AKOs, 4-30
- creating symbols for, 5-6
 - custom, 5-13
 - using the Parts utility, 4-16, 5-11
- creating with the
 - using the model editor, 4-29
 - using the Parts utility, 4-10
- defined as
 - parameter sets, 4-3
 - subcircuits, 4-3, 4-37
- global vs. local, 4-5
- instance, 4-20, 4-33, 4-38, 4-39
- organization, 4-4
- preparing for symbol creation, 5-5
- saving as global
 - using the model editor, 4-31
 - using the Parts utility, 4-18
- saving as local
 - using the model editor, 4-33
 - using the Parts utility, 4-20
- testing/verifying (Parts-created), 4-13
- tools to create, 4-7
- ways to create/edit, 4-8

Monte Carlo analysis, 7-2, 12-7

- collating functions, 12-4
- histograms, 12-19
- introduction, 1-6
- model parameter values reports, 12-3
- output control, 12-3
- tutorial, 12-10
- using the model editor, 4-35
- waveform reports, 12-4
- with temperature analysis, 12-6

MOSFET, 4-12, 7-9, 13-38, 13-39

msim.ini file, editing, 13-5

multiple Y axes, Probe, 11-6

N

netlist

- failure to netlist, 3-4
- file (.net), 1-10

Newton-Raphson requirements, B-2

NOISESETn initial conditions symbol, A-4

NOICTRANSLATE (simulation option), A-6

noise analysis, 7-2, 9-9

- about, 1-4, 9-10
- device noise, 9-10
- flicker noise, 9-12
- noise equations, 9-12
- Probe output variables, 9-12, 13-39
- setup, 9-9, 9-11
- shot noise, 9-12
- thermal noise, 9-12
- total output and input noise, 9-10
- units of measure, 9-13
- viewing results in Probe, 9-12, 13-39
 - noise units, 9-13

non-causality, 6-42

nonlinear

- magnetic core, 4-12

nonlinear devices

- in AC sweep analysis, 9-7

O

opamp, 4-12

operators in expressions, 3-15

options

- RELTOL, 6-45

origin, symbol, 5-16

output control symbols, 3-6, 15-3

output file (.out), 2-9

- control symbols, 15-3
- tables and plots, 15-3

output noise, total, 9-12

output variables

- noise (Probe), 9-12, 13-39
- Probe, 13-32
- Probe arithmetic expressions, 13-40
- Probe functions, 13-40
- PSpice A/D, 7-4

P

PARAM global parameter symbol, 2-25, 3-13

parameters, 3-12

parametric analysis, 7-2, 11-2

- analyzing waveform families in Probe, 2-27
- example, 2-24
- frequency response vs. arbitrary parameter, 11-8
- introduction, 1-5

- performance analysis, 11-3
 - setting up, 2-25
 - temperature analysis, 7-2, 11-11
- parasitic capacitance, B-13
- parts
 - behavioral, 3-12
 - breakout, 3-11
 - finding, 3-8
 - passive, 3-10
 - unmodeled, 3-22
 - vendor-supplied, 3-7
- Parts utility
 - about, 1-9, 4-10
 - analyzing model parameter effects, 4-14
 - compared to the model editor, 4-7
 - creating symbols for models, 4-16, 5-11
 - custom, 5-13
 - fitting models, 4-14
 - running
 - from the schematic editor, 4-20
 - from the symbol editor, 4-18
 - stand-alone, 4-16
 - supported devices, 4-12
 - testing and verifying models, 4-13
 - tutorial, 4-23
 - using data sheet information, 4-13
 - viewing performance curves, 4-15
 - ways to use, 4-11
- performance analysis, 11-3
 - example, 2-30
 - goal functions, 11-5
- phase, 13-37
- PLogic
 - simulation status window, 7-14
 - starting, 7-10
- power supplies
 - analog, 3-18
- Probe, 13-2
 - about, 1-9
 - adding traces, 2-11
 - arithmetic expressions, 13-40
 - color configuration, 13-5
 - cursors, 13-29
 - data file (.dat), 1-14, 13-20
 - displaying results, 2-11, 2-22
 - expressions, 13-40
 - functions, 13-40
 - hysteresis curves, 10-18
 - limiting data file size, 13-15
 - multiple Y axes, 11-6

- output variables, 13-32
 - for noise, 9-12, 13-39
- performance analysis, 2-30, 11-3
- placing a cursor on a trace, 2-13
- plot update methods, 13-26
- plot windows, 13-3
- printing plot windows, 13-4
- scrolling, 13-25
- shortcut keys, 13-24
- startup, 13-9
- trace data tables, 13-28
- traces, 13-15
- traces, displaying, 2-11, 13-24
- traces, using output variables, 13-32
- using markers, 13-12
- waveform families, 2-27, 8-6
- zoom regions, 13-24

PSpice A/D

- about, 1-2
- expressions, 3-14
- functions, 3-16
- output file (.out), 1-14, 2-9, 15-3
- output variables, 7-4
- PSpice A/D-equivalent parts, 6-28, 6-29
- simulation status window, 7-14, 15-2
- starting, 7-10
- using with other programs, 1-7
- viewing in-progress output values, 15-2

R

- real part, 13-37
- regulator, 4-12
- RELTOL (simulation option), 6-45
- resistor, 13-39
- resistors, 7-8

S

- schematic
 - preparing for simulation, 1-8, 3-2
 - viewing bias point voltages and currents, 2-7, 14-2
- schematic editor
 - starting other tools from
 - model editor, 4-34
 - Parts utility, 4-20, 4-33
- scrolling, Probe, 13-25
- semiconductor
 - problems, B-7

- shot noise, 9-12
- simulation
 - about, 1-2
 - analysis
 - execution order, 7-4
 - setup, 7-3
 - types, 7-2
 - batch jobs, 7-12
 - bias point, A-2
 - failure to start, 3-4
 - initial conditions, A-2, A-6
 - output file (.out), 2-9
 - setup checklist, 3-2
 - starting, 7-10
 - status window, 7-14
 - troubleshooting checklist, 3-4
- simulation control symbols, 3-6
 - ICn, A-4
 - NODESETn (initial conditions), A-4
 - PARAM, 2-25, 3-13
- SIMULATION ONLY attribute, 5-19
- small-signal DC transfer analysis, 7-2, 8-10
 - introduction, 1-3
- Stimulus Editor, 2-17, 10-5
 - about, 1-8
 - creating new stimulus symbols, 10-10
 - defining analog stimuli, 3-20
 - defining stimuli, 10-8
 - editing a stimulus, 10-12
 - manual stimulus configuration, 10-13
 - starting, 10-6
 - stimulus files, 10-5, 10-6
- stimulus files, 1-11
 - configuring, 1-13, 4-41
- stimulus generation, 10-3
 - manually configuring, 10-13
- stimulus, adding, 2-16
 - AC sweep, 9-3
 - DC sweep, 8-4
 - for multiple analysis types, 3-20
 - transient (analog/mixed-signal), 10-3
- subcircuits, 4-3
 - creating .SUBCKT definitions from schematics, 4-7, 4-37
 - tools to create, 4-7
 - ways to create/edit, 4-8
 - see also* models
- switch, 13-39
 - problems, B-8
- symbol editor
 - starting other tools from
 - model editor, 4-31
 - Parts utility, 4-18
 - starting the
 - symbol wizard, 5-6
- symbol wizard, 5-6
 - using custom symbols, 5-13
- symbols
 - attributes for simulation, 5-18
 - base vs. AKO, 5-8
 - bounding box, 5-16
 - creating for models
 - using the Parts utility, 4-16, 5-11
 - using the symbol wizard, 5-6
 - creating new stimulus symbols, 10-10
 - editing graphics, 5-15
 - grid spacing
 - graphics, 5-17
 - pins, 5-17
 - ground, 3-6
 - non-simulation, 5-19
 - origin, 5-16
 - output control, 3-6
 - pins, 3-25, 5-17
 - preparing model libraries for symbol creation, 5-5
 - saving as global
 - using the Parts utility, 4-16, 5-11
 - using the symbol wizard, 5-6
 - simulation control, 3-6
 - simulation properties, 5-3
 - stimulus, 3-6
 - ways to create for models, 5-4
- AGND (ground), 3-26
- BBREAK (GaAsFET), 3-11
- C (capacitor), 3-10
- CBREAK (capacitor), 3-11
- creating for models
 - custom symbols, 5-13
 - using the Parts utility, 5-11
- CVAR (capacitor), 3-10
- D (diode), 3-10
- DBREAK (diode), 3-11
- EGND (ground), 3-26
- IAC (AC stimulus), 9-3
- ICn (initial conditions), A-4
- IDC (DC stimulus), 3-18, 8-4
- IPROBE (bias point current display), 14-11
- ISRC (analog stimulus), 3-18, 3-21, 8-4, 9-3
- ISTIM (transient stimulus), 3-20
- JBREAK (JFET), 3-11

K_LINEAR (transformer), 3-10
 KBREAK (inductor coupling), 3-11
 KCOUPLEn (coupled transmission line), 3-10
 LBREAK (inductor), 3-11
 MBREAK (MOSFET), 3-11
 MODEL attribute, 5-18
 NODESETn, A-4
 PARAM (global parameter), 2-25, 3-13
 QBREAK (bipolar transistor), 3-11
 R (resistor), 3-10
 RBREAK (resistor), 3-11
 RVAR (resistor), 3-10
 SBREAK (voltage-controlled switch), 3-11
 SIMULATION ONLY attribute, 5-19
 T (ideal transmission line), 3-10
 TBREAK (transmission line), 3-11
 TEMPLATE attribute, 5-19
 TLOSSY (Lossy transmission line), 3-10
 TnCOUPLEDx (coupled transmission line), 3-10
 VAC (AC stimulus), 3-19, 9-3
 VDC (DC stimulus), 3-18, 3-19
 VDC (DC stimulus)l, 8-4
 VEXP (transient stimulus), 3-19
 VIEWPOINT (bias point voltage display), 14-11
 VPULSE (transient stimulus), 3-19
 VPWL (transient stimulus), 3-19
 VPWL_F_N_TIMES (transient stimulus), 3-19
 VPWL_F_RE_FOREVER (transient stimulus), 3-19
 VPWL_N_TIMES (transient stimulus), 3-19
 VPWL_RE_FOREVER (transient stimulus), 3-19
 VSFFM (transient stimulus), 3-19
 VSIN (transient stimulus), 3-19
 VSRC (analog stimulus), 3-18, 3-19, 3-21, 8-4, 9-3
 VSTIM (analog stimulus), 3-19
 VSTIM (transient stimulus), 3-20
 WBREAK (current-controlled switch), 3-11
 XFRM_LINEAR (transformer), 3-10
 XFRM_NONLINEAR (transformer), 3-11
 ZBREAK (IGBT), 3-11
 ABMn and ABMn/I (ABM), 6-8, 6-22
 ABS (ABM), 6-8, 6-21
 ARCTAN (ABM), 6-8, 6-21
 ATAN (ABM), 6-8, 6-21
 BANDPASS (ABM), 6-7, 6-12
 BANDREJ (ABM), 6-7, 6-13
 CONST (ABM), 6-7, 6-9
 COS (ABM), 6-8, 6-21
 DIFF (ABM), 6-7, 6-9

DIFFER (ABM), 6-7, 6-14
 E (ABM controlled analog source), 6-46
 EFREQ (ABM), 6-28, 6-37
 ELAPLACE (ABM), 6-28, 6-35
 EMULT (ABM), 6-28, 6-32
 ESUM (ABM), 6-28, 6-32
 ETABLE (ABM), 6-28, 6-33
 EVALUE (ABM), 6-28, 6-30, 6-31
 EXP (ABM), 6-8, 6-21
 F (ABM controlled analog source), 6-46
 FTABLE (ABM), 6-7, 6-15
 G (ABM controlled analog source), 6-46
 GAIN (ABM), 6-7, 6-9
 GFREQ (ABM), 6-28, 6-37
 GLAPLACE (ABM), 6-28, 6-35
 GLIMIT (ABM), 6-7, 6-10
 GMULT (ABM), 6-28, 6-32
 GSUM (ABM), 6-28, 6-32
 GTABLE (ABM), 6-28, 6-33
 GVALUE (ABM), 6-28, 6-30, 6-31
 H (ABM controlled analog source), 6-46
 HIPASS (ABM), 6-7, 6-12
 ICn (initial condition), A-4
 INTEG (ABM), 6-7, 6-14
 LAPLACE (ABM), 6-8, 6-18
 LIMIT (ABM), 6-7, 6-10
 LOG (ABM), 6-8, 6-21
 LOG10 (ABM), 6-8, 6-21
 LOPASS (ABM), 6-7, 6-11
 MULT (ABM), 6-7, 6-9
 NODESETn (initial bias point), A-4
 PWR (ABM), 6-8, 6-21
 PWRS (ABM), 6-8, 6-21
 SIN (ABM), 6-8, 6-21
 SOFTLIM (ABM), 6-7, 6-10
 SQRT (ABM), 6-8, 6-21
 SUM (ABM), 6-7, 6-9
 TABLE (ABM), 6-7, 6-14
 TAN (ABM), 6-8, 6-21
 system variables in expressions, 3-18

T

table look-up parts (ABM), 6-7, 6-14, 6-28, 6-33
 temperature analysis, 7-2, 11-11
 introduction, 1-5
 with statistical analyses, 12-6
 TEMPLATE attribute, 5-19
 and non-simulation symbols, 5-19

- examples, 5-23
- naming conventions, 5-20
- regular characters, 5-20
- special characters, 5-21
- thermal noise, 9-12
- TIME (Probe output variable), 13-35
- total noise, 9-10
 - circuit, 9-12
 - per device, 9-12
- traces
 - adding, 2-11
 - direct manipulation, 13-24
 - displaying, 2-11, 2-18
 - markers, 13-15
 - output variables, 13-32
 - placing a cursor on, 2-13
- transformer
 - problems, B-13
- transient analysis, 7-2
 - example, 2-16
 - Fourier analysis, 7-2
 - hysteresis curves, 10-18
 - internal time steps, 10-17
 - introduction, 1-5
 - overview, 10-2
 - problems, B-10
 - setting up, 2-18
 - Stimulus Editor, 10-5
 - stimulus generation, 10-3
 - switching circuits, 10-17
 - transient response, 10-15
- transmission lines, 13-38
- triode, 6-25
- troubleshooting
 - checklist, 3-4
 - missing DC path to ground, 3-27
 - missing ground, 3-26
 - unconfigured libraries and files, 3-24
 - unmodeled parts, 3-22
 - unmodeled pins, 3-25
- tutorials, *see* examples and tutorials

U

- unmodeled
 - parts, 3-22
 - pins, 3-25

V

- VAC stimulus symbol, 3-19, 9-3
- variables in expressions, 3-18
- VDC stimulus symbol, 8-4
- VDC stimulus symbol, 3-18, 3-19
- vendor-supplied parts, 3-7
- VEXP stimulus symbol, 3-19
- VIEWPOINT view bias point voltage symbol, 14-11
- voltage comparator, 4-12
- voltage reference, 4-12
- voltage regulator, 4-12
- voltage source, controlled, 6-28, 6-46
- VPLoTn (write voltage plot symbol), 15-3
- VPRINTn (write voltage table symbol), 15-4
- VPULSE stimulus symbol, 3-19
- VPWL stimulus symbol, 3-19
- VPWL_F_N_TIMES stimulus symbol, 3-19
- VPWL_F_RE_FOREVER stimulus symbol, 3-19
- VPWL_N_TIMES stimulus symbol, 3-19
- VPWL_RE_FOREVER stimulus symbol, 3-19
- VSFFM stimulus symbol, 3-19
- VSIN stimulus symbol, 3-19
- VSRC stimulus symbol, 3-18, 3-19, 8-4, 9-3
 - how to use, 3-21
- VSTIM stimulus symbol, 2-16, 3-19, 3-20

W

- WATCH1 (view output variable symbol), 15-2
- waveform analysis, *see* Probe
- waveform families, displaying in Probe, 2-27
- worst-case analysis, 7-2, 12-25
 - collating functions, 12-4
 - example, 12-28
 - hints, 12-32
 - introduction, 1-6
 - model parameter values reports, 12-3
 - output control, 12-3
 - overview, 12-25
 - waveform reports, 12-4
 - with temperature analysis, 12-6

Z

- zoom regions, Probe, 13-24