

# **PSpice Help**

**Product Version 16.6**  
**October 2012**

© 1991–2012 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

---

# Contents

---

<b>About PSpice</b>	15
<u>What is PSpice?</u>	15
<u>What is Probe?</u>	16
<u>What is the Stimulus Editor?</u>	18
<u>What is the Model Editor?</u>	19
<u>Types of analyses you can run with PSpice</u>	19
<u>AC sweep and noise</u>	20
<u>DC sweep &amp; other DC calculations</u>	21
<u>Transient and Fourier</u>	21
<u>Parametric and temperature</u>	22
<u>Monte Carlo and sensitivity/worst-case</u>	22
<u>Files used by PSpice (input files)</u>	23
<u>Files Generated by Schematic Editors or Design Entry Programs</u>	23
<u>Other Input Files</u>	24
<u>Model libraries</u>	25
<u>Stimulus files</u>	25
<u>Include files</u>	26
<u>Preparing and configuring input files</u>	26
<u>Files generated by PSpice (output files)</u>	27
<u>Online documentation</u>	28
<u>Additional sources of information about PSpice</u>	28
<b><u>Preparing your design for simulation</u></b>	31
<u>Creating designs for simulation and board layout</u>	31
<u>Placing stimulus sources</u>	32
<u>Editing simulation models from Design Entry Programs</u>	33
<u>Creating a simulation netlist</u>	33
<u>Setting up analyses</u>	34
<u>Setting up an AC analysis</u>	35
<u>Setting up the loading of bias points</u>	36
<u>Setting up the saving of bias points</u>	36
<u>Setting up a DC analysis</u>	37

## PSpice Help

---

<u>Setting up a Monte Carlo/worst-case analysis</u>	38
<u>Setting the bias point detail</u>	40
<u>Setting digital options</u>	41
<u>Setting up a parametric analysis</u>	42
<u>Setting up a sensitivity analysis</u>	43
<u>Setting the temperature</u>	44
<u>Setting up a transient analysis</u>	45
<u>Simulating your circuit</u>	46
<u>Interacting with a simulation</u>	46
<u>Extending a transient analysis</u>	48
<u>Interrupting a simulation</u>	50
<u>Scheduling changes to runtime parameters</u>	53
<u>Running multiple simulations</u>	55
<u>Using the Simulation Manager</u>	56
<u>Understanding the Simulation Manager</u>	58
<u>Available functionality of the Simulation Manager</u>	60
<u>Error message handling by the Simulation Manager</u>	60
<u>Setting up multiple simulations</u>	61
<u>Starting, stopping, and pausing simulations</u>	62
<u>Attaching PSpice to a simulation</u>	63
<u>Setting options in the Simulation Manager</u>	63
<u>The Simulation Manager File menu</u>	65
<u>The Simulation Manager Edit menu</u>	66
<u>The Simulation Manager View menu</u>	67
<u>The Simulation Manager Simulation menu</u>	68
<u>The Simulation Manager Tools menu</u>	68
<u>The Simulation Manager Toolbar</u>	69
<u>Entering distributions</u>	71
<u>Using markers</u>	71
<u>Limiting waveform data file size</u>	73
<u>Setting data collection options</u>	73
<u>Suppressing data from a transient run</u>	75
<u>Assigning marker colors</u>	75
<u>Viewing results</u>	76
<u>Viewing results as you simulate</u>	76
<u>Configuring PSpice Display of Simulation Results</u>	77

## PSpice Help

---

<a href="#">Viewing Monte Carlo histograms</a>	77
<a href="#">Creating parts for existing simulation models</a>	79
<a href="#">Defining part properties needed for simulation</a>	80
<a href="#">Handling unmodeled pins</a>	81
<a href="#">Saving a copy of your project</a>	82
<a href="#">Simulating non-PSpice projects</a>	82
<b><a href="#">Setting up your design for simulation</a></b>	<b>85</b>
<a href="#">Files needed for simulation</a>	85
<a href="#">Files that Design Entry Programs generate</a>	85
<a href="#">Other files that you can configure for simulation</a>	85
<a href="#">Files that PSpice generates</a>	87
<a href="#">Checklist for simulation setup</a>	88
<a href="#">When netlisting fails or the simulation does not start</a>	90
<a href="#">Using parts that you can simulate</a>	92
<a href="#">Part naming conventions</a>	95
<a href="#">Finding the part that you want</a>	95
<a href="#">Passive parts</a>	96
<a href="#">Breakout parts</a>	97
<a href="#">Behavioral parts</a>	98
<a href="#">Defining part properties needed for simulation</a>	99
<a href="#">Specifying values for part properties</a>	106
<a href="#">Using global parameters and expressions for values</a>	107
<a href="#">Expressions</a>	108
<a href="#">Defining power supplies</a>	110
<a href="#">Defining stimuli</a>	111
<a href="#">Things to watch for</a>	114
<a href="#">Analog libraries with modeled parts</a>	120
<a href="#">Digital libraries with modeled parts</a>	121
<b><a href="#">Performing Circuit Analysis</a></b>	<b>123</b>
<a href="#">Analyzing waveforms with PSpice</a>	123
<a href="#">What are the features of PSpice simulation profiles?</a>	124
<a href="#">Creating a new simulation profile</a>	125
<a href="#">Using a simulation profile</a>	126
<a href="#">Editing a simulation profile</a>	127
<a href="#">Deleting a simulation profile</a>	128

## PSpice Help

---

<a href="#">Viewing the simulation queue</a>	128
<a href="#">General simulation settings for simulation profiles</a>	129
<a href="#">Analysis settings for simulation profiles</a>	130
<a href="#">Include files settings for simulation profiles</a>	131
<a href="#">Library settings for simulation profiles</a>	132
<a href="#">Stimulus settings for simulation profiles</a>	133
<a href="#">Options for simulation profiles</a>	134
<a href="#">Data collection options for simulation profiles</a>	139
<a href="#">Probe windows settings for simulation profiles</a>	141
<b><a href="#">Traces</a></b>	143
<a href="#">Adding traces</a>	143
<a href="#">Viewing trace information</a>	144
<a href="#">Editing trace display properties</a>	144
<a href="#">Setting grid display properties</a>	145
<a href="#">Setting plot edge properties</a>	145
<a href="#">Defining analog trace expressions</a>	145
<a href="#">Defining digital trace expressions</a>	151
<a href="#">Narrowing the list of output variables</a>	156
<a href="#">Deleting traces</a>	157
<a href="#">Setting the digital plot size</a>	157
<a href="#">Using cursors</a>	158
<a href="#">Moving cursors along a trace</a>	161
<a href="#">Changing views</a>	163
<a href="#">Creating a Fourier Transform</a>	166
<a href="#">Cautions when using FFTs</a>	167
<a href="#">Changing axis settings</a>	167
<a href="#">Adding a new Y axis</a>	171
<a href="#">Deleting a Y axis</a>	171
<a href="#">Using multiple plots</a>	171
<a href="#">Using Probe windows</a>	173
<a href="#">Toggling between display modes</a>	173
<a href="#">Keeping the Probe window visible at all times</a>	174
<a href="#">To print plots</a>	174
<a href="#">Using Display Control</a>	174
<a href="#">Using plot window templates</a>	176
<a href="#">Creating a plot window template</a>	178

## PSpice Help

---

<a href="#">Modifying a plot window template</a>	181
<a href="#">Deleting a plot window template</a>	181
<a href="#">Copying a plot window template</a>	182
<a href="#">Restoring a plot window template</a>	182
<a href="#">Viewing the properties of a plot window template</a>	184
<a href="#">Loading a plot window template</a>	185
<a href="#">Placing plot window template markers</a>	185
<a href="#">Labeling plots</a>	188
<a href="#">Editing labels</a>	191
<a href="#">Copying Probe data to other applications</a>	192
<a href="#">Loading large data file</a>	194
<a href="#">Importing traces</a>	194
<a href="#">Import Traces</a>	195
<a href="#">Using performance analysis and measurements</a>	197
<a href="#">Using Performance Analysis</a>	197
<a href="#">Using Measurement Expressions</a>	197
<a href="#">Composing Measurement Expressions</a>	198
<a href="#">Measurement Expression Example</a>	199
<a href="#">Viewing Measurement Results</a>	200
<a href="#">Evaluating a measurement</a>	200
<a href="#">Measurement Results Example</a>	201
<a href="#">Measurement Definitions Included with PSpice</a>	202
<a href="#">Creating Custom Measurement Definitions</a>	204
<a href="#">Composing a New Measurement Definition</a>	205
<a href="#">Managing Measurements</a>	206
<a href="#">Copying a Measurement Definition</a>	206
<a href="#">Editing a Measurement Definition</a>	207
<a href="#">Measurement Definition Example</a>	207
<a href="#">Measurement Definition Syntax</a>	208
<a href="#">Measurement Name Syntax</a>	209
<a href="#">Marked Point Expression Syntax</a>	210
<a href="#">Comments Syntax</a>	210
<a href="#">Search Command Syntax</a>	211
<a href="#">Limiting a Measurement Expression to a Specific Range of Data</a>	216
<a href="#">Control Elements in Dialog Boxes</a>	217
<a href="#">Introducing the Performance Analysis Wizard</a>	219

## PSpice Help

---

<u>Selecting a Measurement</u>	220
<u>Selecting Measurement Arguments</u>	220
<u>Testing the Measurement</u>	221
<b><u>Setting Options</u></b>	223
<u>Setting Probe window options</u>	223
<u>Setting large data file options</u>	224
<u>Selecting a Printer</u>	225
<u>Using Print Preview</u>	225
<u>Setting up the Page</u>	225
<u>Printing in PSpice</u>	226
<u>Setting the Width of Printed Plot Lines</u>	227
<u>Changing the Screen Colors</u>	227
<u>Header and footer codes</u>	228
<u>Customizing toolbars</u>	229
<u>Customizing Commands</u>	231
<u>Customizing keyboard shortcuts</u>	231
<u>Settings schemes</u>	233
<b><u>Reference Information</u></b>	235
<u>Using files</u>	235
<u>PSpice default keyboard shortcuts</u>	236
<u>Invalid node names</u>	237
<u>Limits in PSpice and Probe</u>	238
<b><u>Files and Commands</u></b>	241
<u>Using .PRB files</u>	241
<u>Loading .PRB files</u>	241
<u>PRB file</u>	242
<u>Moving data to other applications</u>	242
<u>Logging commands</u>	243
<u>Creating and changing macros</u>	244
<u>CSDF</u>	246
<u>Binary</u>	247
<u>Specifying default command line options</u>	247
<u>Configuring the pspice.INI file</u>	247
<b><u>Descriptions of menus</u></b>	269
<u>The File menu</u>	269



## PSpice Help

---

<u>The Edit menu</u>	271
<u>The View menu</u>	273
<u>The Simulation menu</u>	276
<u>The Trace menu</u>	276
<u>The Plot menu</u>	278
<u>The Tools menu</u>	280
<u>The Window menu</u>	281
<u>Terms used in File</u>	282
<u>Export Data</u>	282
<u>Find dialog box</u>	283
<u>Regular expressions</u>	284
<u>The Large Data File dialog box</u>	285
<u>Using the Quick Reference Card</u>	287
<u>Keyboard shortcuts</u>	287
<u>PSpice toolbars</u>	289
<u>Index of PSpice symbol and part properties</u>	299
<u>Introduction to device equations</u>	379
<u>Making device model changes</u>	379
<u>Changing a parameter name</u>	380
<u>Giving a parameter an alias</u>	381
<u>Adding a parameter</u>	381
<u>Changing the device equations</u>	382
<u>Functional subsections of the device source file</u>	383
<u>Adding a new device</u>	384
<u>Specifying new internal device structure</u>	385
<u>Recompiling and linking the device equations option</u>	388
<u>Personalizing your DLL</u>	389
<u>Simulating with the device equations option</u>	389
<u>Selecting which models to use from a Device Equations DLL</u>	389
<u>Popup Menu Items</u>	391
<u>PSpice Errors and Solutions</u>	405
<u>.PROBE and .ALIAS must agree on /CSDF</u>	405
<u>Invalid device type</u>	405
<u>Maximum number of alias nodes exceeded</u>	405
<u>Unable to open index file</u>	405
<u>Model type unknown</u>	406

## PSpice Help

---

<u>Duplicate library entry for &lt;modelname&gt;</u>	406
<u>Out of Memory</u>	406
<u>Unrecognizable command</u>	406
<u>Unable to open stimulus file</u>	407
<u>Model references form circular list. For example:</u>	407
<u>Unable to open probe file</u>	407
<u>Unable to make index for library file</u>	407
<u>Model &lt;modelname&gt; referenced by model &lt;modelname&gt;, is undefined</u>	408
<u>Subcircuit &lt;filename&gt; used by &lt;filename&gt; is undefined</u>	408
<u>Unable to open library</u>	408
<u>Making new index file for library file</u>	409
<u>Missing model name in library</u>	409
<u>Missing model type in library</u>	409
<u>Missing subcircuit name</u>	409
<u> VON - VOFF  too small for VSWITCH model</u>	410
<u>RON or ROFF less than or equal to zero for VSWITCH model</u>	410
<u>RON or ROFF greater than 1/GMIN for VSWITCH model</u>	410
<u>RON = ROFF for VSWITCH model</u>	410
<u> ION - IOFF  too small for ISWITCH model</u>	411
<u>RON or ROFF less than or equal to zero for ISWITCH model</u>	411
<u>RON or ROFF greater than 1/GMIN for ISWITCH model</u>	411
<u>RON = ROFF for ISWITCH model</u>	411
<u>&lt;param&gt; not a subcircuit param</u>	412
<u>Less than 2 connections at node</u>	412
<u>Node is floating</u>	412
<u>Invalid radix, expecting BIN (1), OCT (3), or HEX (4)</u>	412
<u>Unrecognized parameter</u>	413
<u>Tolerances on model &lt;modelname&gt; ignored due to &lt;tolerance&gt;</u>	413
<u>MC or .WCASE ignored (No &lt;analysis type&gt; command in circuit)</u>	413
<u>No models had tolerances. .MC or .WCASE ignored</u>	413
<u>The circuit matrix is singular and cannot be solved.</u>	414
<u>The circuit matrix is too close to being singular to solve.</u>	414
<u>Convergence problem</u>	414
<u>Convergence problem</u>	415
<u>Time step is too small in Transient Analysis at xxx</u>	415
<u>Missing or invalid expression</u>	415

## PSpice Help

---

<u>Missing expression</u>	416
<u>Bad radix spec</u>	416
<u>LABEL invalid in REPEAT loop</u>	416
<u>Missing goto label</u>	416
<u>GOTO invalid in REPEAT loop</u>	417
<u>HREPEAT missing FOR or FOREVER</u>	417
<u>Attempt to redefine builtin name</u>	417
<u>Must be D</u>	417
<u>Must be I or V or D</u>	418
<u>Must be I or V</u>	418
<u>Must be V</u>	418
<u>Must be I or V, D not allowed</u>	418
<u>Expression not allowed here</u>	419
<u>Unknown parameter</u>	419
<u>Probability must not be less than 0.</u>	419
<u>At least two pairs of numbers necessary</u>	419
<u>Please simplify .. distribution too complicated</u>	420
<u>Use RLGC &amp; LEN for lossy line</u>	420
<u>Use Z0 &amp; TD or F/NL for ideal line</u>	420
<u>Z0 or RLGC parameters must be specified</u>	420
<u>TD or F must be specified</u>	421
<u>BadTransferFunction</u>	421
<u>Missing REPEAT iteration count</u>	421
<u>Symbols Table overflow</u>	421
<u>Voltage Source and/or Inductor Loop Involving xxx</u>	422
<u>Convergence problem</u>	422
<u>Convergence problem</u>	422
<u>Invalid Outside of .SUBCKT</u>	423
<u>Library Index File Does Not Have the Correct Format</u>	423
<u>Unable to Find Library File</u>	423
<u>Library File Has Changed Since Index File Was Created</u>	423
<u>The Timestamp Changed from xxx to yyy</u>	424
<u>Model &lt;modelname&gt; Used by &lt;filename&gt; Is Undefined</u>	424
<u>Missing param name in library</u>	424
<u>There Are No Devices in This Circuit (This Message Will Be Printed)</u>	424
<u>Only one .TEMP value allowed with .STEP</u>	425

## PSpice Help

---

<u>Only one .TEMP, .DC TEMP, or .STEP TEMP permitted</u>	425
<u>Unable to open file</u>	425
<u>Missing .ENDS in .SUBCKT</u>	426
<u>Name on .ENDS does not match .SUBCKT</u>	426
<u>Invalid device in subcircuit</u>	426
<u>Subcircuit &lt;filename&gt; is Undefined</u>	426
<u>Incorrect Number of Interface Nodes for &lt;filename&gt;</u>	427
<u>Digital Simulator Option not present</u>	427
<u>Cannot Open Temporary Digital File</u>	427
<u>Missing model</u>	427
<u>Missing number of nodes</u>	428
<u>Too few output nodes specified</u>	428
<u>Bad or missing parameter</u>	428
<u>Invalid value</u>	428
<u>Undefined parameter used in expression</u>	429
<u>Undefined Parameter: &lt;parameter&gt;</u>	429
<u>I(node) is not valid</u>	429
<u>Must be independent source (I or V)</u>	429
<u>Digital node table overflow</u>	430
<u>Missing parameter</u>	430
<u>Not a valid parameter for model type</u>	430
<u>Must be 'I' or 'V'</u>	430
<u>Missing node number</u>	431
<u>Missing device name</u>	431
<u>Analog simulator option not present</u>	431
<u>Invalid parameter</u>	431
<u>Inductor part of this K device</u>	432
<u>Inductor part of another core device</u>	432
<u>Transmission line part of this K device</u>	432
<u>Invalid specification</u>	432
<u>Bad value</u>	433
<u>Invalid number</u>	433
<u>No analog devices--DC sweep ignored</u>	433
<u>No analog devices--small-signal analysis Ignored</u>	433
<u>Missing value</u>	434
<u>EOF in subcircuit</u>	434

## PSpice Help

---

<u>Errors and Solutions</u> .....	435
<u>Unable to write to disk: check if disk is full</u> .....	435
<u>Unable to read from file - improper mode</u> .....	435
<u>Unable to write to file - improper mode</u> .....	435
<u>Cannot open file: filename</u> .....	435
<u>Cannot open directory for backup directory name</u> .....	436
<u>File error, fseek failed</u> .....	436
<u>File error, ftell failed</u> .....	436
<u>File error, cannot reopen</u> .....	436
<u>Previous error in opening file</u> .....	436
<u>Backup failed: Permission denied to file</u> .....	437
<u>Backup failed: Bad file number</u> .....	437
<u>Backup failed: Cannot write to file</u> .....	437
<u>Extension can only have up to three characters</u> .....	438
<u>Filenames can have only up to eight characters</u> .....	438
<u>File name contains an invalid character</u> .....	438
<u>Blanks are not allowed in file names</u> .....	438
<u>Directory in path does not exist</u> .....	438
<u>No filename?!</u> .....	438
<u>Cannot open temporary file</u> .....	439



---

# About PSpice

---

## What is PSpice?

PSpice<sup>1</sup> is a simulation program that models the behavior of a circuit containing any mix of analog and digital devices. 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.

Because the analog and digital simulation algorithms are built into the same program, PSpice simulates mixed-signal circuits with no performance degradation because of tightly coupled feedback loops between the analog and digital sections.

PSpice can perform the following types of analyses:

- AC, DC, 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 circuits behavior varies with changing component values
- Digital worst-case timing analysis to help you find timing problems that occur with only certain combinations of slow and fast signal transmissions

## Models

PSpice includes model libraries that feature over 15,000 analog and 1,600 digital models of devices manufactured in North America, Japan, and Europe. Among these libraries are numerous models with parameters that you can tweak for a given device. These include independent temperature effects.

PSpice also supports analog and digital behavioral modeling, so you can describe functional blocks of circuitry using mathematical expressions and functions.

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

---

1. Depending on the license and installation, either PSpice or AMS Simulator is installed. However, all information about PSpice provided in this manual is true for AMS Simulator.

- transmission line models, including delay, reflection, loss, dispersion, and crosstalk
- nonlinear magnetic core models, including saturation and hysteresis
- eight MOSFET models, including BSIM4 version 4.1, BSIM3 version 3.2, and EKV version 2.6
- five GasFET models, including Parker-Skellern and TriQuints TOM2 model
- IGBTs
- digital components with analog I/O models

### **Related Topics**

For information about...	<a href="#">Click this topic...</a>
What Probe is...	<a href="#">What is Probe?</a>
What the Stimulus Editor is...	<a href="#">What is the Stimulus Editor?</a>
What the Model Editor is...	<a href="#">What is the Model Editor?</a>
The types of analyses you can run with PSpice ...	<a href="#">Types of analyses you can run with PSpice</a>
More resources and training for PSpice ...	<a href="#">Additional sources of information about PSpice</a>

## **What is Probe?**

After completing the simulation, PSpice plots the waveform results so you can visualize the circuits behavior and determine the validity of your design. You can use the waveform analysis features of PSpice to visually analyze and interactively manipulate the waveform data produced by circuit simulation. This built-in waveform analyzer is referred to as Probe.

Probe uses high-resolution graphics so you can view the results of a simulation both on the screen and in printed form. On the screen, waveforms appear as plots displayed in Probe windows within the PSpice workspace.



In effect, waveform analysis is a software oscilloscope. Performing a PSpice simulation corresponds to building or changing a breadboard, and performing waveform analysis corresponds to looking at the breadboard with an oscilloscope. Taken together, simulation and waveform analysis is an iterative process. After analyzing simulation results, you can refine your design and simulation settings and then perform a new simulation and waveform analysis.

With waveform analysis you can:

- view simulation results in multiple Probe windows
- compare simulation results from multiple circuit designs in a single Probe window
- display simple voltages, currents, power, 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
- display analog and digital waveforms simultaneously with a common time base (for mixed analog/digital simulations)
- add text labels and other annotation symbols for clarification

What you can plot in Probe depends on the types of analyses you run. Bode plots, phase margin, derivatives for small-signal characteristics, waveform families, and histograms are only a few of the possibilities. You can also plot other waveform characteristics such as rise time versus temperature, or percent overshoot versus component value.

PSpice generates two forms of output: the simulation output file and the waveform 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 waveform data file is the most informative and flexible method for evaluating simulation results. The waveform data file is used by Probe to generate the waveforms displayed in the PSpice workspace.

## Related Topics

For information about...      [Click this topic...](#)

How to view simulation results...      [Viewing results](#)

How to configure the display of simulation results...      [Configuring PSpice Display of Simulation Results](#)

Using and configuring Probe windows...      [Using Probe windows](#)

## What is the Stimulus Editor?

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, current sources, and digital stimuli for your circuit. Menu prompts guide you to provide the necessary parameters, such as the rise time, fall time, and period of an analog repeating pulse, or the complex timing relations with repeating segments of a digital stimulus. Graphical feedback allows you to quickly verify the waveform.

The Stimulus Editor is a graphical waveform editor that allows you to define the shape of time-based signals used to test your circuit designs response during simulation. You can use the Stimulus Editor to set up and verify the input waveforms for a transient analysis. You can also create and edit voltage sources, current sources, and digital stimuli for your circuit design.

Using the Stimulus Editor, you can define:

- analog stimuli with sine wave, pulse, piecewise linear, exponential pulse, single-frequency FM shapes, and
- digital stimuli that range from simple clocks to complex pulse patterns and bus sequences.

The Stimulus Editor lets you draw analog piecewise linear and all digital stimuli by clicking at the points along the timeline that correspond to the input values that you want at transitions. The Stimulus Editor produces a file containing the stimuli with their transient specifications. These stimuli are defined as simulator device declarations using the V (voltage source), I (current source), and U STIM (digital stimulus generator) forms. Since the Stimulus Editor produces these statements automatically, you will never have to be concerned with their syntax.

For information about using the Stimulus Editor, see the Stimulus Editor online Help.

## What is the Model Editor?

The Model Editor is a model extractor that generates model definitions for PSpice to use during simulation. The Model Editor can generate model definitions:

- using the device information found in standard data sheets, or
- using the templates provided by PSpice.

While creating models based on device characteristic curves, as you enter the data sheet information, the Model Editor displays device characteristic curves so you can verify the model-based behavior of the device. When you are finished, the Model Editor automatically creates a part for the model so you can use the modeled part in your design immediately.

The Model Editor converts information that you enter from the device manufacturers 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.

While creating models based on PSpice provided templates, you need to specify the values of various simulation parameters defined in the template. Values entered by you overwrite the default values in the template. The models created using PSpice provided templates are always of .SUBCKT type.

For information about using the Model Editor, see the Model Editor online Help.

## Types of analyses you can run with PSpice

### Basic analyses

Click any of the following analysis types for more information:

- [AC sweep and noise](#)
- [DC sweep & other DC calculations](#)

■ Transient and Fourier

### 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.

Click either of the following analysis types for more information:

- Parametric and temperature
- Monte Carlo and sensitivity/worst-case

### AC sweep and noise

These AC analyses evaluate circuit performance in response to a small-signal alternating current source. The table below summarizes what PSpice calculates for each AC analysis type.

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:

- 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



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

## DC sweep & other DC calculations

These DC analyses evaluate circuit performance in response to a direct current source. The table below summarizes what PSpice calculates for each DC analysis type.

For this DC analysis...	PSpice computes this...
DC sweep	Steady-state voltages, currents, and digital states 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

## Transient and Fourier

These time-based analyses evaluate circuit performance in response to time-varying sources. The table below summarizes what PSpice calculates for each time-based analysis type.

For this time-based analysis...	PSpice computes this...
Transient	<p>Voltages, currents, and digital states tracked over time</p> <p>For digital devices, you can set the propagation delays to minimum, typical, and maximum. If you have enabled digital worst-case timing analysis, then PSpice considers all possible combinations of propagation delays within the minimum and maximum range</p>
Fourier	DC and Fourier components of the transient analysis results



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

## 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. The table below shows the circuit values that you can step for each kind of analysis.

**Note:** p

.	
For this analysis...	You can step one of these...
Parametric	<ul style="list-style-type: none"><li>■ global parameter</li><li>■ model parameter</li><li>■ component value</li><li>■ DC source</li><li>■ operational temperature</li></ul>
Temperature	<ul style="list-style-type: none"><li>■ operational temperature</li></ul>

## 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. The table below summarizes how PSpice runs each statistical analysis type.

**Note:**

For this statistical analysis...	PSpice does this...
Monte Carlo	For each simulation, randomly varies all device model parameters for which you have defined a tolerance

Sensitivity/worst-case

Computes the probable worst-case response of the circuit in two steps:

1. Computes component sensitivity to changes in the device model parameters. This means PSpice varies device model parameters in a non-random manner for which you have defined a tolerance, one at a time for each device and runs a simulation with each change.
2. Sets all model parameters for all devices to their worst-case values (assumed to be at one of the tolerance limits) and runs a final simulation.

## Files used by PSpice (input files)

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
- the stimulus definitions to test with

This information is provided in various data files. Some of these are generated by schematic editors<sup>1</sup>, others come from libraries (which can also be generated by other programs like the Stimulus Editor and the Model Editor), and still others are user-defined.

## Files Generated by Schematic Editors or Design Entry Programs

Capture and Design Entry HDL are design entry programs 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
- marking the points in the circuit where you want to see simulation results

1. Schematic editor or design entry programs refer to OrCAD Capture or Design Entry HDL.

For more information about designing circuits with Capture or Design Entry HDL, see their respective online helps.

When you begin the simulation process, the design entry programs first generate 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.

The netlist file contains a list of the device names and their values, and the connections between the devices. The name that design entry programs generate for this file is `DESIGN_NAME.net`. The netlist file is located in the directory:

```
<project_directory>\worklib\<design_name>
```

```
sp_sim_1\
```

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 the design entry programs generate for this file is `PROFILE_name.cir`.

## Other Input Files

Before starting the simulation, PSpice needs to read other files that contain simulation information for your circuit. These are model files, and if required, stimulus files and include files. You can create these files using programs like the Stimulus Editor and the Model Editor. These programs automate file generation and provide graphical ways to verify the data. You can also use the Model Text view in the Model Editor (or another text editor like Notepad) to enter the data manually.



## Related Topics

For information about	<a href="#">Click this topic...</a>
What model libraries are	<a href="#">Model libraries</a>
What stimulus files are	<a href="#">Stimulus files</a>
What include files are	<a href="#">Include files</a>
Preparing and configuring input files	<a href="#">Preparing and configuring input files</a>

## Model libraries

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 PSpice 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 by using the Model Text view in the Model Editor (or another text editor like Notepad), or
- automatically by using the Model Editor

## Stimulus files

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

- manually by using the Model Text View of the Model Editor (or a standard text editor) to create the definition (a typical file extension is .STM), or
- automatically by using the Stimulus Editor (which generates a .STL file extension)

**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 properties.

## Include files

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

You can create an include file using any text editor, such as Notepad. Typically, include file names have a .INC extension.

## Preparing and configuring input files

You must first prepare the circuit design using either Capture or Design Entry HDL as the primary design entry program. Entering the design in either Capture or Design Entry HDL is the most efficient way to draw up the circuit and define the various parameters required for simulation. Once the circuit is entered in the design entry programs, you can configure the input files for analysis by PSpice.

Along with the netlist and circuit files generated by the design entry programs, 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.

How you configure your model libraries and other files determines the way PSpice uses those 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 profile only, one design only (local) or to any design (global)
- change the search order

For more detailed information about designing circuits and configuring files, see the Capture or Design Entry HDL online Help.

## **Files generated by PSpice (output files)**

After 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 waveform data file and the PSpice output file.

### **Waveform data file**

The waveform data file contains simulation results that can be displayed graphically. PSpice 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 design so PSpice displays the results as the simulation progresses or after the simulation completes.

There are two ways to add waveforms to the display:

- from within PSpice, by specifying trace expressions
- from within the design entry programs, by placing markers for cross-probing

After PSpice reads the data file and displays the initial set of results, you can add more waveforms and 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 (such as the enabled analyses)
- simulation results
- 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
- the simulation control symbols (like VPRINT1 and VLOT1) that you place and connect to nets in your design

## Online documentation

To access online documentation, you must open the Cadence Help window.

1. From the programs folder in the Windows Start menu, choose the OrCAD release and then the Cadence Help shortcut.
2. Click the PSpice category to show the documents in the category.
3. Double-click a document title to open that document.

You can also open PSpice specific documents by choosing Help - Documentation from the PSpice window. You can either follow the links in the displayed content or press F2 to open a list of all documents in the Cadence installation.

## Additional sources of information about PSpice

### Recommended textbooks

Many textbooks and technical articles have been written in several languages about how to use PSpice when doing circuit analysis. The following is a brief list of some useful resources.

- Goody, Roy W., PSpice for Windows, A Circuit Simulation Primer, Prentice-Hall, 1995
- Goody, Roy W., PSpice for Windows, Vol. II, Operational Amplifiers & Digital Circuits, Prentice-Hall, 1996.
- Herniter, Marc E., Schematic Capture with PSpice, Macmillan, 1994.
- Kielkowski, Ron, Inside SPICE, Overcoming the Obstacles of Circuit Simulation, McGraw-Hill, 1994.
- Kielkowski, Ron, SPICE, Practical Device Modeling, McGraw-Hill, 1995.
- Tuinenga, Paul W., Spice, A Guide to Circuit Simulation & Analysis Using PSpice, 2nd Ed., Prentice-Hall, 1992.

### Training courses

In addition to books and articles, training programs are available in Capture and PSpice using the latest version of the products. For the latest schedule of courses, go to the Cadence Web site at <http://www.cadence.com/education/>.

Other training courses in PSpice are provided across the U.S. at various colleges and technical schools. In other countries, certain Cadence distributors offer training courses in the native language.

### **Internet resources**

<http://www.cadence.com/orcad/index.html>

### **OrCAD community site**

This site contains many technical articles, application notes and other technical tips about how to use PSpice.

<http://www.cadence.com/orcad/index.html>

### **PCB home page**

This site provides information about all Cadence tools in the PCB design space. You can get information about the tools that can be used with PSpice. The PCB home page also provides demos of various PCB tools and loads of other downloads.

### **Vendor-supplied models**

Many manufacturers of electronic devices provide PSpice models for simulating circuits that use their components. In most cases, and with little or no special modification, these vendor-supplied models can be used directly with PSpice. To obtain a PSpice model for a specific device that is not included with the standard PSpice libraries, contact the manufacturer directly.



---

# Preparing your design for simulation

---

PSpice is a mixed analog/digital electrical circuit design simulator that can calculate the behavior of analog-only, mixed analog/digital, and digital-only circuit designs with speed and accuracy. PSpice simulates mixed analog/digital circuit designs, calculates voltages and currents of the analog devices and nodes, and calculates the states of digital nodes (nodes connected to digital devices only).

## Creating designs for simulation and board layout

In order to simulate a design with PSpice that is created in the design entry programs, Capture or Design Entry HDL, you must begin the project as an analog type intended for simulation. Existing projects in the design entry programs cannot be simulated without special modifications. To learn how to simulate an existing project with PSpice, see [Simulating non-PSpice projects](#).

### To create a new project for simulation

1. From the File menu in Capture's Project Manager, point to New and select Project.

The New Project dialog box appears.

2. In the Name text box, enter the name for the new project.
3. Under the Create a New Project Using frame, select Analog or Mixed-Signal Circuit Wizard.

**Note:** You must create a project (not a design) and select the Analog or Mixed-Signal Circuit Wizard option in order to be able to simulate the new design with PSpice.

4. In the Location text box, enter the path where you want the new project files to be stored, or use the Browse button to locate the directory.
5. Click OK.
6. Enter any special libraries to be included, if necessary, and click Finish to create the new project directory and open the schematic page editor.

A design that is targeted for simulation has:

- parts for which there are simulation models available and configured

■ sources of stimulus to the circuit

The part libraries for PSpice are in the PSpice subfolder in the `tools\Capture\Library` directory in your main installation directory. Each part must have a `PSPICETEMPLATE` property in order to netlist correctly for use with PSpice.

When creating designs for both simulation and printed circuit board layout, some of the parts you use are for simulation only (that is, simulation stimulus parts like voltage sources), and some of the parts you use have simulation models that only model some of the pins of a real device.

The parts that are to be used for simulation, but not for board layout, have a `PSPICEONLY=TRUE` property.

You can add this (or any) property to your own custom parts to make them simulation-only.

## Placing stimulus sources

Parts for stimulus sources for simulation are in the `SOURCE.OLB` part library. You can place a source in the design as you would any other part.

If you have the Stimulus Editor, you can place one of the following from (`SOURCSTM.OLB`):

- `VSTIM`—voltage stimulus source
- `ISTIM`—current stimulus source
- `DIGSTIMn`—digital stimulus source

and use the Stimulus Editor to define the stimulus waveform for transient analysis.

If you do not have the Stimulus Editor, use the source parts from `SOURCE.OLB` and edit the properties of the source to define stimulus parameters.

For information on using the Stimulus Editor, refer to the Stimulus Editor online Help.

## To place a stimulus source

From Capture's Place menu, choose Part. In Design Entry HDL, use the Component Browser.

1. Click the Add Library button.
2. Select `SOURCSTM.OLB` (from the PSpice library sub-directory) and click Open.
3. From the list of parts, select the source part you want to use in your design.



4. Click OK to place the source on the schematic page.

To edit a stimulus source with the Stimulus Editor

1. On the schematic page, select the stimulus source or sources to be edited.
2. From the Edit menu, choose PSpice Stimulus to start the Stimulus Editor

## Editing simulation models from Design Entry Programs

You can define and edit simulation models directly from Capture using the PSpice Model Editor.

The PSpice Model Editor is useful for characterizing specific models from data sheet curves.

When editing a part on the design, Capture makes an individual copy of a model or subcircuit representation, allowing you to modify the copy. These changes only apply to the specific part instance in the current design. By default, these individual instances are saved in a file with the same name as the originating design file and a .LIB extension.

Before you can edit a model for a part on the design, it must have a MODEL or IMPLEMENTATION property.

### To edit the model for a part on the design

1. Select the part on the schematic page.
2. From the Edit menu, choose PSpice Model.

The Model Editor starts with the model loaded for editing.



***The Model Editor does not support the use of AKOs in the .MODEL statement. A model statement containing an AKO will not simulate in PSpice.***

## Creating a simulation netlist

A netlist is the connectivity description of a circuit, showing all of the components, their interconnections, and their values. When you create a simulation netlist from Capture, that netlist describes the current design.

The flat netlist is generated for all levels of hierarchy, starting from the top, regardless of whether you are pushed into any level of the hierarchy. Flat netlists are most commonly used as input to PCB layout tools. The flat simulation netlist format for PSpice contains device entries for all parts on a subcircuit (child) schematic multiple times, once for each instance of the hierarchical part or block used.

## Creating the netlist

You can generate a simulation netlist in one of two ways:

- from Capture's Project Manager by using the Create Netlist command under the Tools menu. Similarly, from the Tools - Options menu of Design Entry HDL.

or

- directly from within Capture or Design Entry HDL by using the Create Netlist command under the PSpice menu

During the netlist process, Capture creates files with different extensions: the .NET file contains the netlist; the .ALS file contains alias information for cross-probing.

**Note:** For more detailed information about generating simulation netlists from design entry programs, refer to the User Guide or the online Help.

## Setting up analyses

Unless you intend to run the simulation using a circuit (.CIR) file, you must create a simulation profile (or edit an existing one) before you can set up a PSpice simulation. See [Creating a new simulation profile](#) for more information.

To set up a PSpice simulation

1. From design entry program PSpice menu, choose New Simulation Profile.

**Note:** In order to access the PSpice menu and set up simulations, you must be working with a PSpice project in the design entry program. The project type is defined when you begin a new project.

2. In the Simulation Settings dialog box, click the Analysis tab.
3. Make your selections. The options are:
  - ☐ [Setting up an AC analysis](#)
  - ☐ [Setting up a parametric analysis](#)

- ☐ [Setting up the loading of bias points](#)
- ☐ [Setting up a sensitivity analysis](#)
- ☐ [Setting up the saving of bias points](#)
- ☐ [Setting the temperature](#)
- ☐ [Setting up a DC analysis](#)
- ☐ [Setting the bias point detail](#)
- ☐ [Setting up a Monte Carlo/worst-case analysis](#)
- ☐ [Setting digital options](#)
- ☐ [Setting up a transient analysis](#)

4. Click OK.

## Setting up an AC analysis

The AC analysis calculates the small-signal frequency response of the circuit (linearized around the bias point) over a range of frequencies.

### To set up an AC analysis

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select AC Sweep/Noise.
3. From the Options list, select General Settings.
4. In the AC Sweep Type frame, choose either [Linear](#) or Logarithmic. (If Logarithmic, also select [Decade](#) or [Octave](#).) Enter the start frequency, end frequency, and points in the text boxes.
5. To have noise analysis enabled, under Noise Analysis, select the Enabled check box. Enter the output voltage, I/V source, and interval in the text boxes.
6. Click OK.

### To run the active simulation

1. From the Simulation menu, choose Run.

## Setting up the loading of bias points

The Load Bias Point analysis option includes a .LOADBIAS statement in the circuit file and loads the contents of the bias point file. Normally the bias point file is produced by a previous circuit simulation using the Save Bias Points option.

This option is available for the Time Domain (Transient), DC Sweep, and Bias Point analyses.

### To set the bias load point

In the Simulation Settings dialog box, click the Analysis tab.

1. From the Analysis type list, select Time Domain (Transient), DC Sweep, or Bias Point.
2. From the Options list, select Load Bias Point.
3. In the Load Bias Information from filename text box, enter the name of a file that contains a .LOADBIAS statement.
4. Click OK.

### To run the active simulation

1. From the Simulation menu, choose Run.

## Setting up the saving of bias points

The Save Bias Point analysis option inserts a .SAVEBIAS statement into the circuit file, and the bias point node voltages for the specified analysis (DC, OP, or TRAN) is saved to the file you specify.

### To set the save bias load point

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select Time Domain (Transient), DC Sweep, or Bias Point.
3. From the Options list, select Save Bias Point.
4. In the Save Bias Information in Filename text box, enter a file name in which to save the bias point information.
5. Under Options, type values in the text boxes for when to save bias information during the analysis:

- ☐ Primary Sweep value
  - ☐ Secondary Sweep value
  - ☐ Parametric Sweep value
  - ☐ Monte Carlo options
  - ☐ Temperature Sweep temperature
6. Under Options, select Do Not Save Subcircuit Voltages and Currents if you do not want to save the node voltages and currents for sub-circuits.
  7. Click OK.

### **To run the active simulation**

1. From the Simulation menu, choose Run.

### **Setting up a DC analysis**

The DC analysis performs a DC sweep. The DC sweep analysis calculates the circuit's bias point over a range of values.

### **To set up the DC Sweep**

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select DC Sweep.
3. From the Options list, select Primary Sweep.
4. Under Sweep variable, choose Voltage source, Current source, Global parameter, Model parameter, or Temperature.
5. Under Sweep type, choose Linear, Logarithmic, or Value list. (If Logarithmic, also select Decade or Octave.)
6. Click OK.

### **To run the active simulation**

1. From the Simulation menu, choose Run.

## Setting up a Monte Carlo/worst-case analysis

Monte Carlo/Worst Case analyses vary the lot or device tolerances of devices between multiple runs of an analysis (DC sweep, AC sweep, or transient).

You can run either a Monte Carlo or a worst-case analysis, but not both at the same time. Before running either analysis, you must set up the device and lot tolerances of the model parameters to be investigated.

### To set the Monte Carlo/Worst Case options

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select DC Sweep, AC Sweep/Noise, or Time Domain (Transient).
3. From the Options list, select Monte Carlo/Worst Case.
4. Choose either Monte Carlo or Worst-case/Sensitivity.
5. In the Output Variable text box, type the output variable, using the following format:

`V(<net name> [, <net name>])`

where <net name> must be a fully qualified net name. For example,  $V(sense)$  represents the voltage at a net, and  $V(a,b)$  represents the output voltage across two nets a and b.

6. Enter the Monte Carlo or Worst-case/Sensitivity options as described below.
7. Click OK.

### Monte Carlo options

You can set the following options:

- ☐ Number of runs
- ☐ Use distribution and custom distributions with the Distributions button
- ☐ In the Random number seed text box, type an odd integer ranging from 1 to 32767.
- ☐ In the Save data from list, select one of the following:

<none>

All	Forces all output to be generated, including the nominal run.
First	Generates output only during the first n runs. Type the value for n in the Runs text box.
Every	Generates output every nth run. Type the value for n in the Runs text box.
Runs (list)	Performs an analysis and generates output only for listed runs. Up to 25 values can be specified in the Runs text box. Prints out at the beginning of each run the model parameter values actually used for each component during that run.

### **Worst-case/Sensitivity options**

1. From the Vary Devices That Have list, select Vary both DEV and LOT, Vary DEV, Vary LOT.
2. In the Limit devices to type(s) text box, type a list of devices to include in the analysis.
3. Select the Save data from each sensitivity run check box to save data from each sensitivity run.

### **Output file options**

1. Click the More Settings button.
2. From the Find list, select one of the following collating functions:

YMAX	Finds the greatest difference in each waveform from the nominal run.
MAX	Finds the maximum value of each waveform.
MIN	Finds the minimum value of each waveform.
RISE_EDGE	Finds the first occurrence of the waveform crossing above the threshold value. Type a threshold value in the Threshold value text box.
FALL_EDGE	Finds the first occurrence of the waveform crossing below the threshold value. Type a threshold value in the Threshold value text box.

3. Under Worst-Case direction, choose either Hi or Low.

4. Select the List model parameter values check box to produce a list of the model parameters actually used for each run.

### History support options

1. Click the MC Load/Save button.
2. To enable saving of the randomly generated model parameter values for each run, complete the following steps.
  - a. In the Load/Save Monte Carlo Parameter File dialog box, select the Save parameter values in the filename check box.
  - b. In the text box that is enabled, specify the name and the location of the file in which the parameter data is to be saved. If required, you can use the Browse button to navigate to the required location.

The model parameter values are saved in a Monte Carlo parameter ( `.mcp` ) file. When you simulate the design, a `.mcp` file with the complete history of variation of parameter values with in the tolerance range will be generated.

3. To reuse model parameter values generated and saved during a previous Monte Carlo analysis, complete the following sequence of steps.
  - a. In the Load/Save Monte Carlo Parameter File dialog box, select the Load parameter values in filename check box.
  - b. In the text box that is enabled, specify the name and the location of the `.mcp` file from which the parameter data is to be read. You can use the Browse button to navigate to the required location.

When you now simulate the circuit, all the parameter values stored in the `.mcp` file will be reused during the simulation.

4. Click *OK* to save your settings.

### To run the active simulation

1. From the Simulation menu, choose *Run*.

### Setting the bias point detail

The Bias Point analysis saves detailed bias point information to the simulation output file.

The information reported to the output file includes the following:



- ☐ a list of all the analog node voltages
- ☐ a list of all the digital node voltages
- ☐ the currents through all the voltage sources, and their total power
- ☐ a list of the small-signal parameters for all the devices

### **To save detailed bias point information to the output file**

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select Bias Point.
3. Under Output File Options, select any of the following that you want saved to the output file:
  - ☐ Include detailed bias point information for nonlinear controlled sources and semiconductors
  - ☐ Perform Sensitivity analysis
  - ☐ Calculate small-signal DC gain
4. Click OK.

### **To run the active simulation**

1. From the Simulation menu, choose Run.

## **Setting digital options**

Set digital options for DC analyses.

### **To set the digital options**

1. In the Simulation Settings dialog box, click the Options tab.
2. From the Category list, select Gate-level Simulation.
3. Under Timing Mode, choose Minimum, Typical, Maximum, or Worst-case (min/max).
4. Select Suppress simulation error messages to not include error messages in the waveform data file generated for this simulation.
5. From the Initialize All Flip-Flops To list, select X, 0, or 1.

6. In the Default I/O level for Interfaces box, enter a default propagation delay mode.
7. Click OK.

### **To run the active simulation**

1. From the Simulation menu, choose Run.

### **Setting up a parametric analysis**

A parametric analysis performs a sweep analysis while varying a global parameter. The simulator performs a series of simulations; there is one for each value of the parameter. All expressions in the circuit are re-evaluated with the new parameter value at the beginning of each run.

### **To set the parametric options**

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select Time Domain (Transient), DC Sweep, or AC Sweep/ Noise to use as the basic analysis.
3. Under Options, select Parametric Sweep.
4. Under Sweep variable, choose a variable to sweep during the analysis:
  - ☐ Voltage source
  - ☐ Current source
  - ☐ Global parameter
  - ☐ Model parameter
  - ☐ Temperature
5. Under Sweep type, choose one of the following:
  - ☐ Linear
  - ☐ Logarithmic: Decade or Octave
  - ☐ Value list
6. Under Sweep type, the values in the Start Value and End Value text boxes vary depending upon which sweep variable type you select. The Start Value can be greater or less than the End Value.

7. Click OK.

### **To run the active simulation**

1. From the Simulation menu, choose Run.

### **Setting up a sensitivity analysis**

Performs a DC sensitivity analysis. One or more output variables can be specified. The <output variable>, if it is a current, is restricted to be current through a voltage source.

Device sensitivities are provided for the following device types only:

- ☐ resistors
- ☐ independent voltage and current sources
- ☐ voltage and current-controlled switches
- ☐ diodes
- ☐ bipolar transistors

### **To set the sensitivity analysis**

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select DC Sweep.
3. From the Options list, select Monte Carlo/Worst Case.
4. Choose Worst-case/Sensitivity.
5. In the Output variable text box, type an output variable, using the following format:

`V(<net name> [, <net name>])`

where <net name> must be a fully qualified net name. It has the form such as: V(sense), the voltage at a net; or a form such as: V(a,b), the output voltage across two nets a and b.

6. Click OK.

### **To run the active simulation**

1. From the Simulation menu, choose Run.

## Setting the temperature

Set the temperature to specify the temperature or list of temperatures at which all analyses are performed. The temperatures are in degrees Centigrade. If more than one temperature is given, then all analyses are done for each temperature.

You can type either a single value for the Temperature box, or a list of temperatures. When a list is typed, the circuit is simulated multiple times, once for each temperature in the list. Running an analysis at multiple temperatures can also be done as a parametric analysis. With parametric analysis, the temperatures can be specified either by list or by range and increments within the range.

The default temperature for simulation is 27 degrees Celsius.

**Note:** The statistical analyses perform multiple runs, as does the Temperature analysis when a temperature range is typed. Conceptually, the Monte Carlo and worst case loops are inside the Temperature loop. However, since both temperature and tolerances effect the model parameters, the interaction of the two can become complicated.

Therefore, it is recommended that you should not use the Temperature analysis option to sweep multiple temperatures when using Monte Carlo or worst case analyses in a circuit. For the same reason, sweeping the temperature with a DC Sweep analysis while performing one of these statistical analyses is not recommend. In addition, putting tolerances on temperature coefficients is not recommended.

### To set the temperature

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis type list, select a basic analysis type.
3. From the Options list, select Temperature (Sweep).
4. Choose one of the following:
  - ☐ Run the simulation at temperature, to run the simulation at a constant temperature. Enter a value in the text box.
  - ☐ Repeat the simulation for each of the temperatures, to repeat the simulation at different temperatures. Type a list of temperatures in the text box.
5. Click OK.

### To run the active simulation

1. From the Simulation menu, choose Run.

### Setting up a transient analysis

A transient analysis calculates the behavior of the circuit over time.

### To set the transient option

1. In the Simulation Settings dialog box, click the Analysis tab.
2. From the Analysis Type list, select *Time Domain (Transient)*.
3. In the Run to Time text box, type the length of the transient analysis.
4. Select *Run in resume mode* if you want to pause the simulation after running for a specific time. You can enter the time to run in the RunFor text box of the PSpice toolbar. After the simulation pauses, you can change parameters and restart the simulation.
5. To perform a Fourier Analysis, click the Output File Options button, then select (?) Enable Fourier.

A Fourier Analysis performs a decomposition into Fourier components of the transient analyses results.

6. Click OK.

### To run the active simulation

1. From the Simulation menu, choose Run.

### To check if a part has a simulation model defined

1. In schematic page editor, double-click a part on the schematic page. If a simulation model is available for a part, the part has:
  - ☐ a `PSPICETEMPLATE` property specifying the PSpice simulation netlisting syntax for the part
  - ☐ an Implementation type PSpice model and an Implementation property specifying the name of the model or subcircuit
2. The `PSPICETEMPLATE` contains `@MODEL` somewhere along the line.

The simulation model specified by the Implementation property must be contained in a model library that is configured.

## Simulating your circuit

Simulating performs a PSpice circuit analysis on the current design. This command automatically performs an Electrical Rule Check (DRC), and netlist generation.

You must create a simulation profile (or edit an existing one) before you can set up a PSpice simulation. See [Creating a new simulation profile](#) for more information.

### To simulate your circuit from within PSpice

1. From PSpice's Simulation menu, choose Run.

The simulation creates output files with a .OUT extension, and if the simulation completes successfully, produces a file with a .DAT extension. The output (.OUT) file contains bias point information, model parameter values, and so on. The .DAT file is the waveform data file containing the simulation results to be displayed by PSpice. All viewpoint and sensor displays are automatically updated.

**Note:** Waveform data is only produced if you run an AC, DC, or a transient analysis.

2. A Probe window appears and displays the results of the simulation (if you have this option enabled in the [Probe windows settings](#) of your simulation profile).
3. If there are errors during the simulation, from the View menu, choose Output File.

### To simulate your circuit from within Design Entry Programs

1. From the PSpice menu, choose Run.

## Interacting with a simulation

### Overview

PSpice includes options for interacting with a simulation by changing certain runtime parameters in the course of the analysis. With the interactive simulation feature, you can do the following:

- ☐ Extend a transient analysis after TSTOP has been reached in order to achieve the desired results.

## PSpice Help

### Preparing your design for simulation

---

- ☐ Interrupt a bias or transient analysis, change certain runtime parameters, and then resume the simulation with the new settings.
- ☐ Schedule changes to certain runtime parameters so that they are made automatically during a simulation.

**Note:** You cannot interact with a simulation while it is running. You must first pause an active simulation in order to change a parameter, then resume it for the changes to take effect.

For more details about interactive simulation, click on the Related Topics below.



***The ability to interact with a simulation only applies to bias point and transient analyses. You cannot interact with other analysis types.***

### What the various versions of PSpice support

The following table identifies what interactive functionality is available with each version of PSpice.

PSpice version	Interactive simulation functionality
PSpice Lite	<input type="checkbox"/> extend transient analysis
PSpice	<input type="checkbox"/> extend transient analysis
	<input type="checkbox"/> interrupt a simulation, change parameters, and resume the simulation
	<input type="checkbox"/> schedule automatic changes to parameters during simulation

## Related Topics

For information about

Click this topic...

Extending a transient analysis

[Extending a transient analysis](#)

Interrupting a simulation and  
changing runtime parameters

[Interrupting a simulation](#)

Scheduling changes to runtime  
parameters

[Scheduling changes to runtime parameters](#)

## Extending a transient analysis

### Overview

Often, a long transient analysis will run to the completion time (*TSTOP*) without achieving the desired simulation results (achieving a steady state, for instance). To achieve better results, the value for *TSTOP* would have to be increased and the entire simulation would have to be rerun from the beginning. This was time-consuming and inefficient for large simulations.

You can set up a transient analysis so that it will pause automatically when it reaches the *TSTOP* value. Once paused, you can review the results and determine if the simulation should run longer. If desired, you can increase the value of *TSTOP* and resume the transient analysis from the point at which it paused, thus saving a good deal of processing time.

**Note:** For more details about using *TSTOP*, see the online PSpice Reference Guide.

To help clarify under what conditions simulations will either be terminated or paused, the following table explains the different behaviors of PSpice for particular simulation scenarios:

Simulation scenario	Behavior of PSpice
Running a single transient simulation using a profile or a circuit file containing one circuit.	PSpice will stop (terminate) after a successful simulation if the <i>RunFor</i> text box is blank. -or- PSpice will pause if there is a value for <i>RunFor</i> , or if a convergence error occurs, allowing you to change certain runtime parameters and resume the analysis.



## PSpice Help

### Preparing your design for simulation

---

Running a single AC/DC simulation using a profile or a circuit file containing one circuit.

PSpice will stop (terminate) after a successful simulation.

-or-

PSpice will pause if a convergence error occurs, allowing you to change certain runtime parameters and resume the analysis.

Running a single simulation with a profile or a circuit file containing outer loops.

PSpice will stop (terminate) after a successful simulation, or if a convergence error occurs.

Running a queued simulation.

PSpice will stop (terminate) after a successful simulation, or if a convergence error occurs.

Launching a new simulation when another one is already active in PSpice.

If the old simulation has completed, PSpice will load the new simulation and run it.

-or-

If the old simulation is running or paused, PSpice will prompt you to choose whether to run the new simulation instead, place it in the queue or cancel it.

### How the RunFor value for TSTOP controls the simulation

When you enter a value for TSTOP in the *RunFor* text box on the simulation toolbar in PSpice, you can control the simulation in various ways:

- ☐ If the *RunFor* text box is blank, the original value for *TSTOP* in the Simulation Profile will be used. The simulation will run to completion and then stop.
- ☐ If you enter a value in the *RunFor* text box, the *RunFor* value will override any value for *TSTOP* in the Simulation Profile. The simulation will run until it reaches the *RunFor* value and then pause.
- ☐ If the simulation is paused before *RunFor* (*TSTOP*) is reached, you can enter a new value in the *RunFor* text box, then click the Run toolbar button. PSpice will resume the simulation, run for the additional time specified (*RunFor*), and then pause again.
- ☐ If you pause a simulation, and want to restart it using the original value for TSTOP in the Simulation Profile, clear the RunFor text box and press the Pause button again.

**Note:** For more information about running and managing multiple simulations, click [Running multiple simulations](#).

### To extend a transient analysis

1. Click in the *RunFor* text box on the PSpice toolbar and enter a value for TSTOP.



2. Click on the Run toolbar button to run the simulation.

The simulation will run and pause when it reaches the value you entered for TSTOP in the *RunFor* text box. (The original value for *TSTOP* in the simulation profile is overridden and ignored if you enter a value in the *RunFor* text box.)

3. Change any parameters you need to adjust. (See [Interrupting a simulation](#).)
4. Change the value of TSTOP in the RunFor text box, if needed.
5. Click on the Run toolbar button to resume the simulation.

The simulation will resume from the point at which it last paused, and then run for the amount of time specified in the RunFor text box, at which point it will pause again.

6. Repeat Steps 3 – 5 as needed.

## Interrupting a simulation

### Overview

You can interrupt (pause) a simulation, change certain runtime parameters, and then resume the simulation from the point at which it was paused using the new parameters.



***The new parameters are temporary values and are not saved in the simulation profile. However, they are logged in the output file so that you can refer to them later.***

After you pause a simulation, you can change the following runtime parameters using the PSpice Runtime Settings dialog box:

- ☐ RELTOL
- ☐ ABSTOL
- ☐ VNTOL
- ☐ GMIN
- ☐ TSTOP
- ☐ TMAX
- ☐ ITL1
- ☐ ITL2
- ☐ ITL4
- ☐ Autoconvergence
- ☐ Enable Advanced Convergence Algorithm

**Note:** For more details about using these runtime parameters, see the online PSpice Reference Guide.

The PSpice Runtime Settings dialog box will appear automatically whenever a simulation fails to converge. (In such cases, the simulation will be paused automatically.) It will also appear if you attach PSpice to a simulation that was paused in the background. (For more information about managing background simulations, click [Using the Simulation Manager](#).)



***In cases where you have paused a simulation and intend to resume it, PSpice will only recognize changes you make in the PSpice Runtime Settings dialog box. Any changes you make to the Simulation Profile will not be applied until the simulation is restarted again from the beginning.***

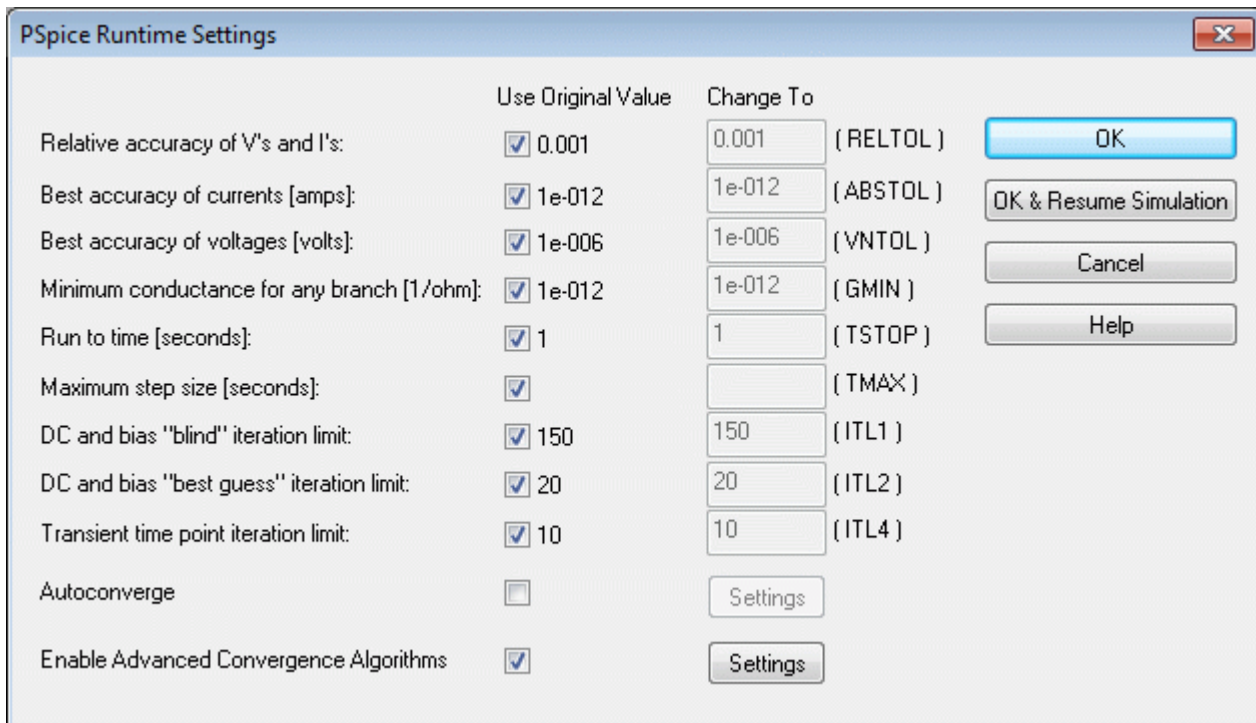
### To interrupt a simulation and change parameters

1. In PSpice, from the Simulation menu, choose Edit Runtime Settings.

The PSpice Runtime Settings dialog box appears.

## PSpice Help

### Preparing your design for simulation



The PSpice Runtime Settings dialog box is shown. It has a title bar with a close button (X). The dialog contains a table with three columns: a description of the parameter, a 'Use Original Value' checkbox, and a 'Change To' section with a text box and a label in parentheses. On the right side of the dialog are four buttons: 'OK', 'OK & Resume Simulation', 'Cancel', and 'Help'.

	Use Original Value	Change To
Relative accuracy of V's and I's:	<input checked="" type="checkbox"/> 0.001	0.001 (RELTOL)
Best accuracy of currents [amps]:	<input checked="" type="checkbox"/> 1e-012	1e-012 (ABSTOL)
Best accuracy of voltages [volts]:	<input checked="" type="checkbox"/> 1e-006	1e-006 (VNTOL)
Minimum conductance for any branch [1/ohm]:	<input checked="" type="checkbox"/> 1e-012	1e-012 (GMIN)
Run to time [seconds]:	<input checked="" type="checkbox"/> 1	1 (TSTOP)
Maximum step size [seconds]:	<input checked="" type="checkbox"/>	(TMAX)
DC and bias "blind" iteration limit:	<input checked="" type="checkbox"/> 150	150 (ITL1)
DC and bias "best guess" iteration limit:	<input checked="" type="checkbox"/> 20	20 (ITL2)
Transient time point iteration limit:	<input checked="" type="checkbox"/> 10	10 (ITL4)
Autoconverge	<input type="checkbox"/>	Settings
Enable Advanced Convergence Algorithms	<input checked="" type="checkbox"/>	Settings

2. If you want to use the original value for a particular parameter, click the Use Original Value check box for that parameter.

The original parameter values are derived from the simulation profile. By default, the Use Original Value check boxes are checked (enabled).

3. If you want to change one or more parameters, enter new values for each of the runtime parameters you want to change in the text boxes under the column Change To.

If a Change To text box is grayed out, uncheck the Use Original Value check box.

4. Select *Autoconverge* to specify that PSpice should try to converge the simulation.
5. Click OK & Resume Simulation to resume the simulation with the new parameters.

**Note:** If you do not want to resume the simulation, but merely want to exit this dialog box and preserve the values you entered, click OK. If you run the simulation later, the new parameters will be applied. If you want to exit this dialog box without preserving the values, click Cancel.

## Scheduling changes to runtime parameters

### Overview

In certain situations, you may want to predefine a set of values for a parameter and schedule these values to take effect at various time intervals during a long simulation. For instance, you may want to use a smaller time step value during periods where the input stimulus changes rapidly, but otherwise use a larger value.

You can set up automatic changes to certain runtime parameters that will occur at scheduled times during a simulation. By scheduling the changes, you don't have to interrupt the simulation manually, and can even run it in a batch mode in the background.

The following runtime parameters can be changed at scheduled times during a simulation. Note that these only apply to transient analysis; you cannot interact with other analysis types.

- ☐ RELTOL
- ☐ ABSTOL
- ☐ VNTOL
- ☐ GMIN
- ☐ ITL4

**Note:** For more details about using these runtime parameters, please see the online PSpice Reference Guide.

### PSpice command syntax for scheduling parameter changes

You can schedule parameter changes by entering them either in the Simulation Profile or in a text file using the new expression SCHEDULE, and then including that file in the simulation profile settings.

The expression SCHEDULE is a piecewise constant function (from time x forward use y) and takes the form:

`SCHEDULE (x1,y1,x2,y2...xn,yn)`

where x is the time value, which must be > 0, and y is the value of the associated parameter. You must include an entry for time=0.

When used with the .OPTION command, the syntax is as follows:

## PSpice Help

### Preparing your design for simulation

---

```
.OPTIONS <Parameter Name>={SCHEDULE(<time-value>, <parameter value>, <time-value>,  
    <parameter value>, ...)}
```

For example,

```
.OPTIONS RELTOL={SCHEDULE( 0s, .001, 2s, .005) }
```

indicates that RELTOL should have a value of 0.001 from time 0 up to time 2s, and a value of 0.005 from time 2s and beyond (that is: RELTOL=.001 for t, where  $0 < t < 2s$ , and RELTOL=.005 for t, where  $t < 2s$ ).

#### To schedule changes to runtime parameters

1. Open a standard text editor (such as Notepad) and create a text file with the command syntax shown above, using the appropriate values for the different parameters.
2. In design entry program, open the design you want to simulate.
3. From the PSpice menu, choose Edit Simulation Profile.
4. Click on the Configuration Files tab.
5. Click Include in the Category field.
6. Under the Filename text box, enter the name of the text file you created in Step 1, or click the Browse button to locate the file and enter the full path and filename.
7. Click the Add to Design button to include the file as part of the circuit.
8. Click OK.

When you run the simulation, the scheduled parameter changes will be included as part of the circuit file and the simulation will run to completion automatically.

## Related Topics

For information about

Click this topic...

Including files

[Include files settings for simulation profiles](#)

Pausing a simulation manually to change parameters

[Interrupting a simulation](#)

Running multiple simulations in the background

[Running multiple simulations](#)

## Running multiple simulations

### Overview

PSpice includes a Simulation Manager that provides enhanced control over how multiple simulations are processed. With the Simulation Manager, you can now control when particular simulations in a batch queue will actually be run. You can also preempt the current simulation to run another one first. Or, you can use the Simulation Manager to monitor the progress of a set of batch simulations that were set up and launched earlier.

None of the earlier functionality of batch processing has been lost. With the Simulation Manager, you now have even greater control and flexibility in setting up multiple simulations.

## PSpice Help

### Preparing your design for simulation

---

#### Related Topics

For information about...

Using the Simulation Manager...

Setting up multiple simulations...

Starting, stopping, and pausing simulations...

Attaching PSpice to a simulation...

Click this topic...

[Using the Simulation Manager](#)

[Setting up multiple simulations](#)

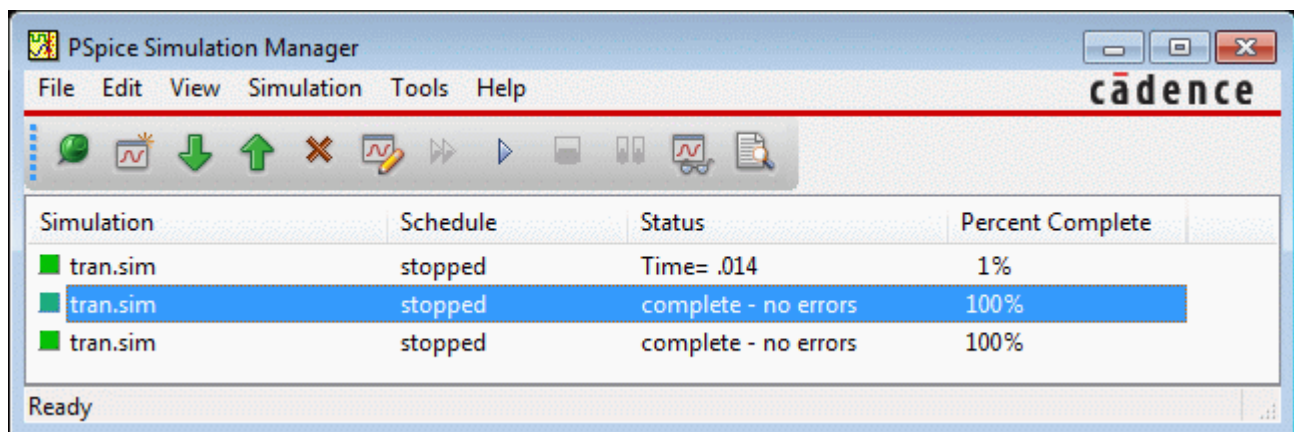
[Starting, stopping, and pausing simulations](#)

[Attaching PSpice to a simulation](#)

## Using the Simulation Manager

### Overview

The PSpice Simulation Manager provides a familiar, easy-to-use interface for controlling how multiple simulations are processed.



The Simulation Manager allows you to do the following:

- ☐ add or delete simulations
- ☐ start, stop or pause simulations
- ☐ rearrange the order of the simulations in the queue
- ☐ attach PSpice to a simulation to make it the active display
- ☐ view the status and progress of simulations running in the background



## PSpice Help

### Preparing your design for simulation

---

You can accomplish most of these functions by selecting the desired simulation in the list, then clicking on the appropriate toolbar button to execute the command. For detailed procedures on performing these tasks, see the Related Topics section below.

**Note:** For simulations that are queued in the Simulation Manager, the setting in the Simulation Profile to start Probe automatically is ignored. When a queued simulation runs to completion and finishes, it will not be loaded into Probe. You must do this manually if you want to see the results of that simulation.

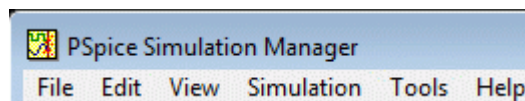
### Accessing the Simulation Manager

The Simulation Manager is invoked whenever you start a new simulation, either from PSpice or from a front-end design entry tool. Since it is active as long as a simulation is running in the background, you can also call up the Simulation Manager from the Windows system tray.

You can also launch the Simulation Manager by itself from the Windows Start menu. You do not need to have PSpice running in order to work with the Simulation Manager.

### Understanding the menu commands

The main menu bar for the Simulation Manager is shown below. For a detailed description of a particular menu, click on the name of that menu in the image.






## Related Topics

For information about	Click this topic...
Understanding the status of jobs in the queue	<a href="#"><u>Understanding the Simulation Manager</u></a>
What the various versions of PSPice support	<a href="#"><u>Available functionality of the Simulation Manager</u></a>
How the Simulation Manager handles errors	<a href="#"><u>Error message handling by the Simulation Manager</u></a>
Setting up multiple simulations	<a href="#"><u>Setting up multiple simulations</u></a>
Starting, stopping, and pausing simulations	<a href="#"><u>Starting, stopping, and pausing simulations</u></a>
Attaching PSPice to a simulation	<a href="#"><u>Attaching PSPice to a simulation</u></a>
Setting the default options for the Simulation Manager	<a href="#"><u>Setting options in the Simulation Manager</u></a>

## Understanding the Simulation Manager

Understanding the information presented by the Simulation Manager

Every job listed in the Simulation Manager will have a specific entry for Schedule, Status and Percent Complete. In addition, certain color-coded icons are shown to the left of each simulation file name to indicate their current state. A quick glance over the list of jobs will tell you immediately where any particular job is and how it will be processed. The following tables explain the meanings of the various categories and states.

Icon	Explanation
	The simulation is either in the queue and has not been run yet, or has been run to completion.
	The simulation is currently running.
	The simulation has been paused and is on hold, waiting to either be continued or stopped.

## PSpice Help

### Preparing your design for simulation

---



The simulation has been stopped and is not completed.

#### Schedule column

#### Explanation

queued	The simulation is in the queue. It will be run in the order in which it is listed in the queue. (This is the default setting.)
running	The simulation is currently running and ongoing status information is displayed.
on hold	The simulation has been paused.
stopped	The simulation has been run completely, or was stopped because of an error.

**Note:** You must manually restart a stopped simulation if you want it to run again at a later time.

#### Status column

#### Explanation

not run	The simulation has not been started yet. (This is the default setting.)
<status information>	Basic status information about the progress of the analysis will be displayed for a simulation that is currently running.
paused	<p>The simulation has been paused either manually or automatically by the Simulation Manager.</p> <p>If you change the default option that automatically resumes paused simulations in the queue, then you must remember to manually resume a paused simulation if you want it to continue at a later time.</p>
complete - no errors	The simulation has run to completion and no errors were encountered.
errors	The simulation ran partially but stopped automatically because errors were encountered.

**Percent Complete  
column**

**Explanation**

<percentage>

The percentage of completion for a simulation. This number increases as a simulation progresses.

## **Available functionality of the Simulation Manager**

### **What the various versions of PSpice support**

The following table identifies what functionality in the Simulation Manager is available with each version of PSpice.

PSpice version	Functionality of Simulation Manager
PSpice Lite	<ul style="list-style-type: none"><li>■ Only one simulation may be running or paused at a time.</li><li>■ The queue is run sequentially.</li></ul>
PSpice	<ul style="list-style-type: none"><li>■ One simulation may be running and multiple simulations may be paused.</li><li>■ The queue is run sequentially.</li></ul>

## **Error message handling by the Simulation Manager**

### **How the Simulation Manager handles errors during simulation**

Since each simulation that runs in the background runs independently, an error that occurs during one simulation will not prevent the remaining jobs in the queue from running subsequently, in order. The following common error conditions may arise, but these will not prevent the Simulation Manager from running the remaining simulations pending in the queue.

- **Simulation crash:** If a simulation crashes for whatever reason, the Simulation Manager will stop receiving progress updates. After a certain period, the Simulation Manager will stop that simulation and will automatically start the next job in the queue.
- **Simulation pause:** If a simulation pauses automatically and requires manual intervention to continue, the Simulation Manager will automatically start the next job in the queue.

- **Simulation stop:** If a simulation stops automatically, the Simulation Manager will automatically start the next job in the queue.



**Caution**

***You must manually restart a stopped or paused simulation if you want it to run again at a later time. You will not be able to shut down the Simulation Manager until all stopped and paused simulations have been deleted.***

## Setting up multiple simulations

### Overview

With the Simulation Manager, you can set up any number of batch simulations to be run sequentially in the background while you do other work in PSpice. Each new simulation that you set up will be added to the bottom of the simulation queue and will be assigned the schedule category “queued”. It will be run after all other queued jobs ahead of it have been run.

Once a job has been added, you can change its position in the queue, start, stop or pause it, or make other modifications to its status. See the Related Topics below for more details on modifying jobs in the Simulation Manager.

### To add a simulation to the queue

1. From the File menu, choose Add Simulation or click the Add Simulation button on the tabular.
2. Locate the file (.SIM, .CIR) you wish to add to the queue.

Alternately, you can add a simulation to the queue by starting the PSpice simulation directly from within the front-end tool you are using, such as Capture.



**Caution**

***If one simulation is already running in the Simulation Manager and you start another one, you will be prompted to direct the Simulation Manager in how to proceed with the new simulation. For more information about the different ways to handle this situation, click Setting options in the Simulation Manager.***

### Related Topics

For information about...

Click this topic...

Starting, stopping, and pausing simulations...

[Starting, stopping, and pausing simulations](#)

Attaching PSpice to a simulation...

[Attaching PSpice to a simulation](#)

Setting the default options for the Simulation Manager...

[Setting options in the Simulation Manager](#)

## Starting, stopping, and pausing simulations

### Overview

In the Simulation Manager, you can easily manage the various batch simulations in the queue. The most fundamental controls that are provided are the ability to start a simulation, stop it, or pause it temporarily.

### To start a simulation from the Simulation Manager

1. Select a simulation in the list.
2. From the Simulation menu, choose Run or click the Run Selected button on the toolbar.

### To stop a simulation from the Simulation Manager

1. Select the simulation that is currently running.
2. From the Simulation menu, choose Stop or click the Stop Selected button on the toolbar.

### **To pause a simulation from the Simulation Manager**

1. Select the simulation that is currently running.
2. From the Simulation menu, choose Pause or click the Pause Selected button on the toolbar.

### **Attaching PSPice to a simulation**

A simulation that is running in the Simulation Manager will not be loaded into PSPice or displayed in Probe while it is running. This allows you to work on a different design in the PSPice application while a simulation is running in the Simulation Manager.

**Note:** If you start a new simulation from within PSPice while another is running in the queue in the Simulation Manager, the Simulation Manager must decide how to treat the new job. You will be prompted to choose whether you want the new job to preempt the current simulation and start running immediately. For more details, click [Setting options in the Simulation Manager](#).

If you want to display a different simulation in PSPice by choosing from the list of jobs in the Simulation Manager, you can attach PSPice to a particular job in the queue.

### **To attach PSPice to a simulation**

1. In the Simulation Manager, select the simulation you want to attach to PSPice.
2. From the View menu, choose Simulation Results.

The PSPice program will activate and the results of the simulation you selected will become the current display in Probe. If the simulation is currently running, you will be able to view the marching waveforms.

### **Setting options in the Simulation Manager**

Each time you add a new simulation while another one is running, the Simulation Manager must decide how to treat the new job. The default setting is to add the new simulation to the bottom of the queue and continue running whatever job is currently being simulated.

You can change this default so that the Simulation Manager will start each new simulation immediately and either stop or pause whatever job is currently running. The options you can choose from are explained in the procedure below.

## PSpice Help

### Preparing your design for simulation

---

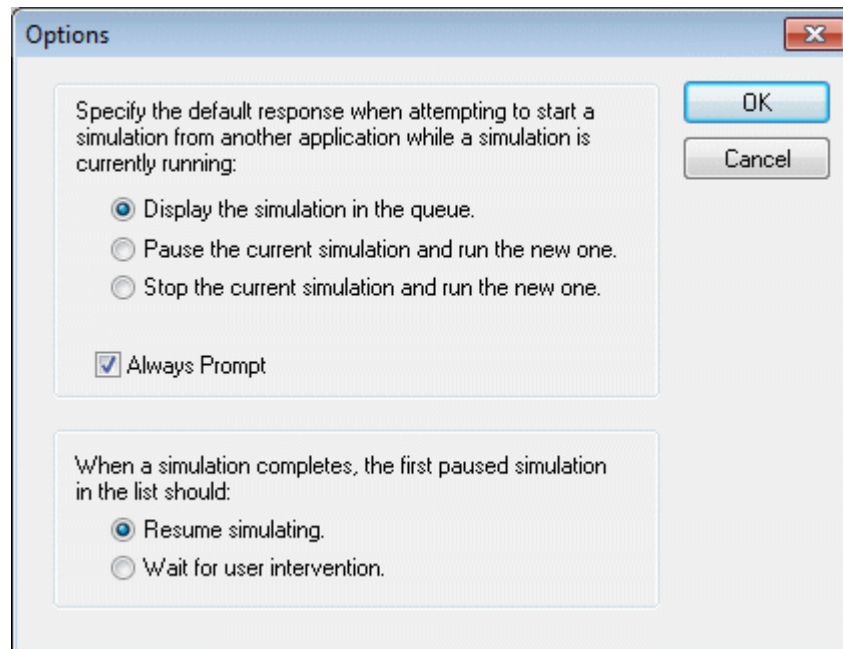
You can also choose to have the Options dialog box display each time you add a new simulation, or not show this anymore. If you disable the prompting, you can always enable it again using the following procedure.

In addition, you can define how paused simulations should be handled by the Simulation Manager. You can configure them to be resumed automatically after the previous simulation stops, or you can choose to leave them in a paused state until you manually resume them.

#### To set the default options for the Simulation Manager

1. From the Tools menu, choose Options.

The Options dialog box appears.



2. In the top frame dealing with simulations that are already running, click the appropriate radio button for the option you wish to set.

Radio button...

Function...

Display the simulation in the queue.

The simulation that is currently running will be displayed in PSpice. The new simulation will be added to the bottom of the queue and will be run after all other jobs in the queue have been run. (This is the default setting.)



## PSpice Help

### Preparing your design for simulation

---

Pause the current simulation and run the new one.

The simulation that is currently running will be paused. The new simulation will be started immediately.

Stop the current simulation and run the new one.

The simulation that is currently running will be stopped. The new simulation will be started immediately. You must remember to restart the stopped simulation later if you want it to run again.

3. If you want the Options dialog box to appear as a reminder each time you add a new simulation, be sure to check the Always Prompt box. (The default setting is to enable this feature.)
4. In the bottom frame dealing with paused simulations, click the appropriate radio button for the option you wish to set.

Radio button...

Function...

Resume simulating.

The first paused simulation in the list will automatically resume after the previous simulation has stopped. (This is the default setting.)

Wait for user intervention.

The Simulation Manager will not resume any paused simulations automatically.

**Note:** If you enable this radio button, you must remember to intervene manually if you want paused simulations to resume later.

5. Click OK to save the settings.

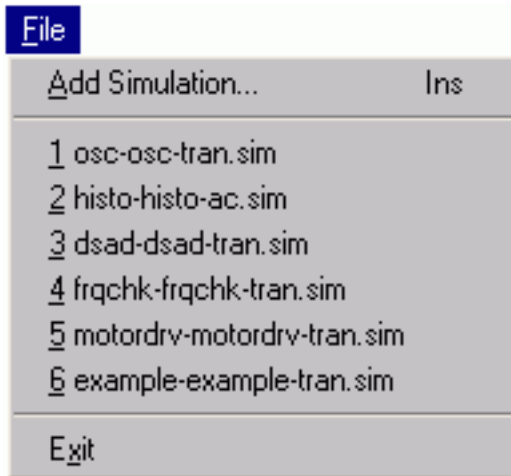
## The Simulation Manager File menu

The File menu provides basic file management functions.

## PSpice Help

### Preparing your design for simulation

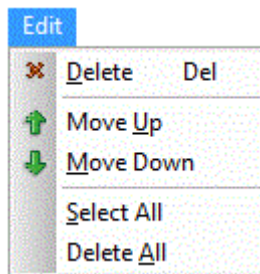
---



Menu command...	Function...
Add Simulation	Opens a simulation file (.SIM) or circuit file (.CIR) and adds it to the queue. Shortcut: INS
<MRU list>	Lists the most recently used file(s).
Exit	Exits the Simulation Manager.

### The Simulation Manager Edit menu

The Edit menu provides functions for modifying the list of jobs in the queue. Most of these commands are reproduced in the toolbar as well. For a description of the toolbar commands, click [The Simulation Manager Toolbar](#).



Menu command...	Function...
-----------------	-------------

## PSpice Help

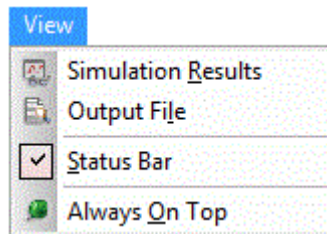
### Preparing your design for simulation

---

Delete	Deletes the selected file from the queue. Shortcut: DEL
Move Up	Moves the selected file up one position in the queue.
Move Down	Moves the selected file down one position in the queue.
Select All	Selects all files in the queue.
Delete All	Deletes all files in the queue.

## The Simulation Manager View menu

The View menu provides controls for what is displayed in the Simulation Manager.

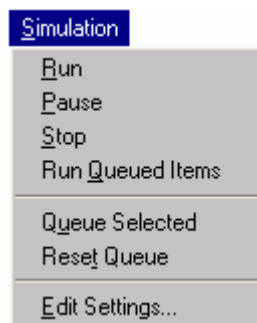


Menu command...	Function...
Simulation Results	Displays the simulation results in PSpice for the selected simulation.
Output File	Opens the output file for the selected simulation and displays it in PSpice.
Toolbar	When checked, this enables the display of the Toolbar. (The default setting is enable the display.)
Status Bar	When checked, this enables the display of the Status Bar. (The default setting is enable the display.)
Always On Top	Keeps the Simulation Manager on top of all other open applications.

## The Simulation Manager Simulation menu

### Overview

The Simulation menu provides controls for how the different simulations are processed by the Simulation Manager.



Menu command...	Function...
Run	Runs the selected simulation(s).
Pause	Pauses the selected simulation(s).
Stop	Stops the selected simulation(s).
Run Queued Items	Runs all of the simulations that are queued. The simulations will run in the order in which they are listed in the queue.
Queue Selected	Changes all selected simulations to "queued".
Reset Queue	Resets any "done" simulations to "queued".
Edit Settings	Opens the simulation profile for the selected simulation and allows you to change the analysis settings.

## The Simulation Manager Tools menu

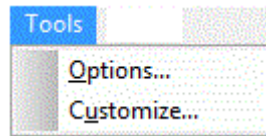
### Overview

The Tools menu allows you to change certain default settings used by the Simulation Manager when starting a simulation.

## PSpice Help

### Preparing your design for simulation

---



Menu command...	Function...
Options	Allows you to change certain default settings.
Customize	Allows you to customize toolbars and commands.


### Related Topics

For information about...	Click this topic...
Options that can be set in the Simulation Manager...	<a href="#">Setting options in the Simulation Manager</a>

## The Simulation Manager Toolbar

The Toolbar provides quick access to the most commonly used functions in the Simulation Manager. All of the buttons on the Toolbar have ToolTips to help remind you of what they do – just pass the cursor over the button to see the Tooltip.



Toolbar button	Function
	Keeps the Simulation Manager on top of all other open applications.
Always On Top	

## PSpice Help

### Preparing your design for simulation

---



Add Simulation

Adds a file to the queue. Shortcut: INS



Move Down arrow

Moves the selected file down one position in the queue.



Move Up arrow

Moves the selected file up one position in the queue.



Delete

Deletes the selected file from the queue. Shortcut: DEL



Edit Simulation  
Settings

Opens the simulation profile for the selected simulation and allows you to change the analysis settings.



Run Queued Items

Runs all of the simulations that are queued. The simulations will run in the order in which they are listed in the queue.



Run Selected

Runs the selected simulation(s).



Stop Selected

Stops the selected simulation(s).



Pause Selected

Pauses the selected simulation(s).



Displays the simulation results in PSpice for the selected simulation.

View Simulation  
Results



Opens the output file for the selected simulation and displays it in PSpice.

View Output File

## Entering distributions

### To enter your own distribution

1. Under Monte Carlo options, click the Distributions button.
2. In the Distribution Name text box, type a name for the distribution, then click Save to save the distribution with the current simulation profile.
3. In the Distribution Curve Values text box, type distribution curves, using the format:  
( $\langle$ deviation $\rangle$ , $\langle$ probability $\rangle$ )
4. To remove a distribution from the current simulation profile, under Existing Distributions, select the distribution name and click the Delete button.
5. Click OK to close the Distributions dialog box and return to the Simulation Settings dialog box.

For more information on distributions, refer to the .DISTRIBUTIONS section of the Commands chapter of the PSpice Reference Manual.

## Using markers

You can place markers in your design to indicate the points for which you want to see simulation waveforms displayed in PSpice. You can place markers before or after simulation is done.

When placed before simulation, markers can be used to limit results written to the waveform data file and to automatically display those traces in PSpice. After simulation results appear in PSpice, placing additional markers on the design automatically displays traces in the current Probe window.

## PSpice Help

### Preparing your design for simulation

---

Power markers allow you to measure the power dissipation of a particular device. You can use these markers in the same way you use current and voltage markers. Power markers are annotated with "W" and are placed on devices that have PSpice models. The corresponding power dissipation waveforms for the devices will be calculated and displayed in Probe.

Markers can be placed on subcircuit nodes as well. This allows you to perform cross-probing between the front-end design entry tool and PSpice at the lower level circuits of a hierarchical design.

The available markers are as follows:

Waveform	Markers menu command	Advanced submenu command
voltage	Voltage Level	(not required)
digital signal	Voltage Level	(not required)
voltage differential	Voltage Differential	(not required)
current	Current Into Pin	(not required)
dB	Advanced	dB Magnitude of Voltage dB Magnitude of Current
phase	Advanced	Phase of Voltage Phase of Current
group delay	Advanced	Group Delay of Voltage Group Delay of Current
real	Advanced	Real Part of Voltage Real Part of Current
imaginary	Advanced	Imaginary Part of Voltage Imaginary Part of Current
power	Power Dissipation	(not required)



***Quiescent power information will be shown only for devices with analog interface pins. It is not currently possible to determine the exact power consumption for devices with digital interface pins.***



## Limiting waveform data file size

When PSpice performs a simulation, it creates a waveform data file. The size of this file for a transient analysis is roughly equal to:

```
(# transistors) * (# simulation time points) * 24 bytes
```

The size for other analysis types is about 2.5 times smaller. For long runs, especially transient runs, this can generate waveform data files that are several megabytes in size. Even if this does not cause a problem with disk space, large waveform data files take longer to read in and take longer to display traces on the screen. You can limit the file size by suppressing part of the data from a transient run. For more details, click [Suppressing data from a transient run](#).

You can also limit the waveform data file size by setting options that determine how much data is collected. For more details on these data collection options, click [Setting data collection options](#).

## Setting data collection options

One reason that waveform 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.

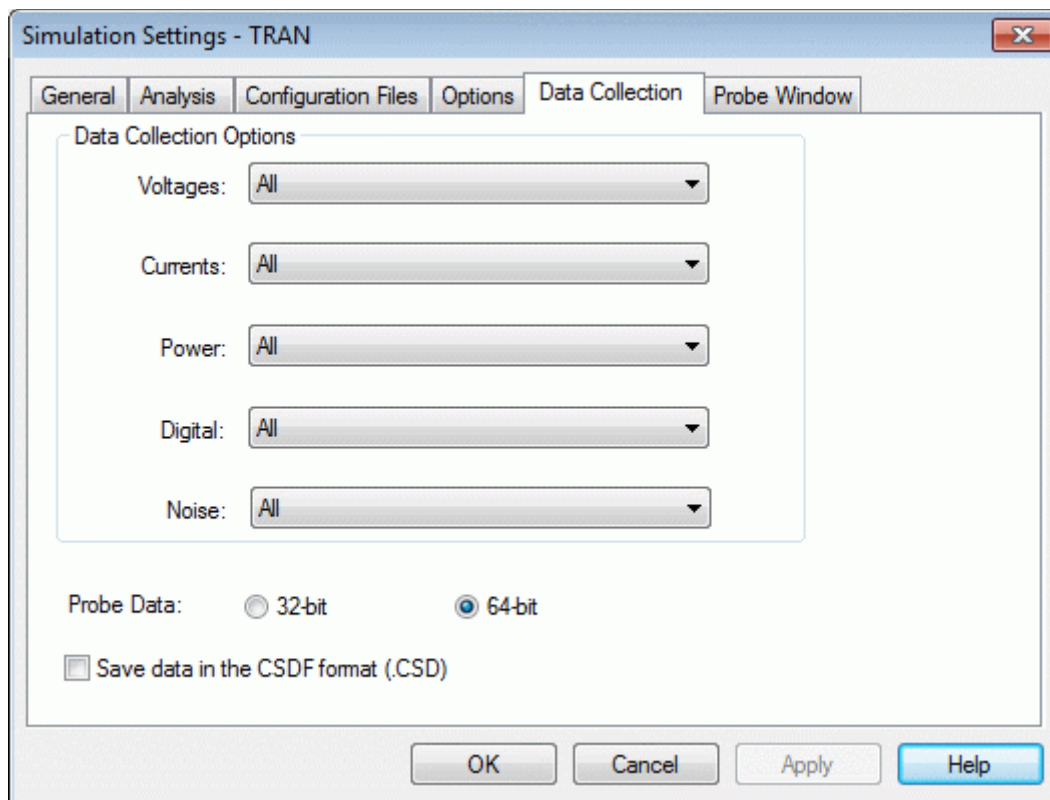
### To limit file size by setting collection options

1. Choose *PSpice – Edit Simulation Profile* to display the Simulation Settings dialog box.

## PSpice Help

### Preparing your design for simulation

---



2. Click the Data Collection tab.

3. In the Data Collection Options frame, choose the desired option for each type of marker (Voltages, Currents, Power, Digital, Noise).

Option	Description
All	All data will be collected and stored. (This is the default setting.)
All but Internal Subcircuits	All data will be collected and stored except for internal sub-circuits of hierarchical designs (top level data only).
At Markers Only	Data will only be collected and stored where markers are placed.
None	No data will be collected.

4. Check the Save data in the CSDF format (.CSD) if you want the data to be stored in this format.

**Note:** By default, the probe data has an accuracy of *64-bit*. You can change this to *32-bit*.

5. Click OK to close the Simulation Settings dialog box.
6. From the PSpice menu, point to Markers, then choose the marker type you want to place.
7. Point to the wires, pins or devices you wish to mark and click to place the chosen markers.
8. Right-click and select End Mode to stop placing markers.
9. From the PSpice menu, choose Run to start the simulation.

When the simulation is complete, the corresponding waveforms for the marked nodes or devices will be displayed in Probe.

## Suppressing data from a transient run

Long transient simulations create large waveform 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. From PSpice menu, choose Edit Simulation Settings to display the Simulation Settings dialog box.
2. Click the Analysis tab.
3. From the Analysis type list, select the Time Domain (Transient) option.
4. In the Start saving data after text box, type a delay time.
5. Click OK to close the Simulation Settings dialog box.
6. From the PSpice menu, choose Run to start the simulation.

The simulation begins, but no data is stored until after the delay has elapsed.

## Assigning marker colors

When you place markers, they are initially grey. Once PSpice completes the simulation and displays the marked traces, colors are automatically assigned to them. The color assigned to a marker will also be the color of its trace in PSpice.

To assign a new marker color

1. In PSpice, right-click on a trace and choose Properties.
2. Select the color you want for the trace from the drop-down Color list.
3. Click OK.

The color you assign for the trace in PSpice will then be associated with the corresponding marker in schematic page.

## Viewing results

Use PSpice to view and perform waveform analysis of the simulation results.

### To view results for the current design

1. From the PSpice menu, choose View Simulation Results.

### To automatically start PSpice after simulation

1. In PSpice menu, choose Edit Simulation Settings to display the Simulation Settings dialog for the currently active profile.
2. Click the Probe Window tab, then select Display Probe window after simulation has completed.
3. Select any other options you want to use.
4. Click OK.

## Viewing results as you simulate

You can configure PSpice to run automatically when the simulation has finished, or to monitor waveforms as the simulation progresses.

### To start PSpice and monitor results during a simulation

1. To enable waveform monitoring, do the following:
  - ☐ From the PSpice menu, choose Edit Simulation Settings.
  - ☐ Click the Probe Window tab, then select Display Probe window during simulation.
  - ☐ Click OK.

2. From the PSpice menu, choose Run to start the simulation. PSpice starts automatically and displays one window in monitor mode.
3. In PSpice, select the waveforms to be monitored by using the Add Trace command on the Trace menu or by placing markers.
4. During a multiple run simulation (such as Monte Carlo, parametric or temperature), only the data for the first run is displayed. To view the curves for several runs:
  - ☐ To close the data file, choose Close from the File menu, then choose Open from the File menu to reload it.
  - ☐ Specify the data sections (runs) to load.
  - ☐ Select the traces to monitor. Waveforms for all loaded sections are displayed.

## Configuring PSpice Display of Simulation Results

To configure what PSpice displays when it is started, choose Edit Simulation Settings from the PSpice menu, click on the Probe Window tab, and then select one of the following options under the Show frame:

- ☐ All markers on open schematics  
Displays the waveforms at the points on the design which have markers.
- ☐ Last plot  
Restores the display characteristics from the last session of PSpice.

## Viewing Monte Carlo histograms

Monte Carlo analysis is frequently used to predict yields on production runs of a circuit. You can display data derived from Monte Carlo waveform families as histograms. This is part of the performance analysis feature of PSpice.

The data file generated by a Monte Carlo analysis can become quite large. You can limit what is displayed and view just a particular node by placing a voltage probe marker at the desired node in the circuit, and then collect data for only that node.

### To display a histogram

1. From PSpice's Plot menu, choose Axis Settings.
2. Select the X Axis tab.

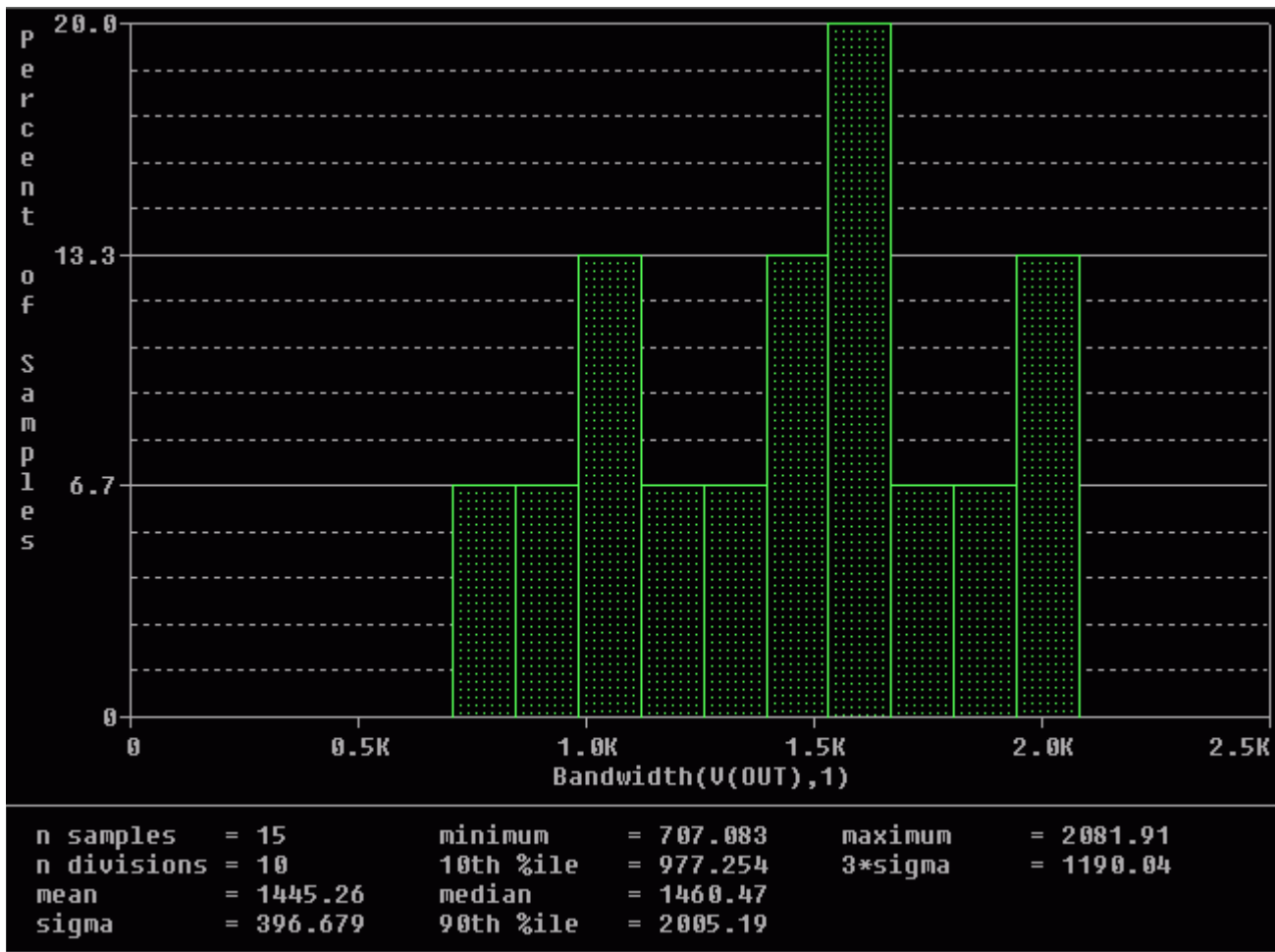
## PSpice Help

### Preparing your design for simulation

---

3. In the Processing Options frame, select the Performance Analysis check box.
4. Click OK.
5. From the Trace menu, select a measurement definition and a trace.

The histogram display appears.



The Y axis is the percent of samples. The statistics for the histogram are shown 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.

## Copying histogram data

You can use the copy function to transfer the raw histogram data points for a particular trace to the Windows clipboard. This allows you to save the data as a standard ASCII text file, or paste it directly into a report or other document for later reference.

### To copy histogram data to the clipboard

1. Select the trace name in the histogram.
2. From the Edit menu, choose Copy (or press Ctrl + C).

The histogram data points for the trace will be transferred to the Windows clipboard.

### To copy the histogram display to the clipboard

1 From the Window menu, choose Copy to Clipboard.

The histogram graph will be transferred to the Windows clipboard.

## Creating parts for existing simulation models

To simulate parts from your design, you can create parts for model definitions.

## Using Parts for Simulation

A part 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, which can include hidden pin connections or propagation delay level (for digital parts)

## Creating Parts for Models

If you want      Then do this...  
to...

Refer to the following section in  
your PSPice User's Guide ...

## PSpice Help

### Preparing your design for simulation

---

Create parts for a set of vendor or user defined models saved in a model library.

Change the graphic standard for an existing model library.

Automatically create one part each time you extract a new model.

Use the Model Editor to create parts from a model library.

Run the Model Editor and enable automatic creation of parts.

Basing new parts on a custom set of parts

Using the Model Editor to Create Parts

Using the Model Editor to Edit Models

Basing New Parts on a Custom Set of Parts



#### Caution

***The Model Editor does not support the use of AKOs in the .MODEL statement. A model statement containing an AKO will not simulate in PSpice.***

For more information on defining part properties, see [Defining part properties needed for simulation](#).

**Note:** For a list of device types that the Model Editor supports, refer to the section on Model Editor-Supported Device Types in your PSpice User's Guide.

## Defining part properties needed for simulation

The part properties Implementation and *PSPICETEMPLATE* (for simulation) may already be defined for your parts, or you may have to edit them yourself, depending on which method you used to create the parts. In addition, you can add other simulation-specific properties: *PSPICEONLY*, *IO\_Level*, *MNTYMXDLY*, and *PSpiceDefaultNet*.

This property...

Defines this...



## PSpice Help

### Preparing your design for simulation

---

Implementation	the name of the model that PSpice must use for simulation. Implementation type must be set to PSpice Model.
PSPICEONLY	an indicator that the part or special part applies only to simulation with PSpice.
PSPICETEMPLATE	the PSpice syntax for the symbol's netlist entry.
IO_Level	what level of interface subcircuit model PSpice uses for a digital part that is connected to an analog part.
MNTYMXDLY	the digital propagation delay level that PSpice must use for a digital part.
PSpiceDefaultNet	<div>the pin property specifying the net name to which a power (invisible) pin is connected.<ul style="list-style-type: none"><li>■ Whenever you define a hidden pin for a part, the part editor automatically creates an PSPICEDEFAULTNET property.</li></ul></div>

**Note:** Refer to the Defining Part Properties Needed for Simulation section of your PSpice User's Guide for further information regarding part properties.

For a complete index of all the part properties and what they are used for, see [Index of PSpice symbol and part properties](#) on page 299

## Handling unmodeled pins

There are parts that have some pins that are not modeled. To see an example of this, place an instance of the PM-741 part from the OPAMP.OLB part library. The OS1 and OS2 pins are not modeled, so only the +, -, V+, V-, and OUT pins are netlisted for simulation.

For the simulator, these pins are treated as a large resistor connected to ground. They have a pin property of *FLOAT=Unmodeled*.

Double-click the part to see the Property Editing dialog box. Note that the *PSPICETEMPLATE* property for the part only calls out the +, -, V+, V-, and OUT pins. The OS1 and OS2 pins are not called out in the *PSPICETEMPLATE* because those two pins are not modeled in the simulation model for the PM-741 part. You can view the simulation model definition for the PM-741 part.

## To view a part's simulation model

1. Click the part to select it.

2. From the Edit menu, choose PSpice Model.
3. The Model Editor will appear and display the model definition.
4. Click Cancel to exit the Model Editor without saving.

## **Saving a copy of your project**

Use the Project Manager in Capture to save all of your simulation settings and analysis setup, as well as your schematic. Similarly, save the project in Design Entry HDL, depending on the design entry program you are using.

### **To save a copy of a project**

1. In the Project Manager in Capture, from the File menu, choose Archive Project.
2. Select Library files, Output files and/or Referenced projects, depending on what types of files you want to archive. Typically, you will want to include all file types to be sure you save everything related to the project.
3. In the Archive directory text box, enter the path where you want the new archive copy to be stored, or use the Browse button to locate the directory.
4. Click OK to create the archive of your project in the specified directory.

**Note:** The .DAT and .ALS files are not saved with the project.

## **Simulating non-PSpice projects**

If you have a design in Capture that was not set up initially as a project that can be simulated by PSpice, you need to do the following modifications in order to be able to simulate it. The basic process involves creating a new project intended for simulation and then copying the existing design into it.

1. From the File menu in Capture's Project Manager, point to New and select Project.

The New Project dialog box appears.

2. In the Name text box, enter the name for the new project. (Be sure to use a different name than the name of the existing project.)
3. Under the Create a New Project Using frame, select Analog or Mixed-Signal Circuit Wizard.

## PSpice Help

### Preparing your design for simulation

---

4. In the Location text box, enter the path where you want the new project files to be stored, or use the Browse button to locate the directory.
5. Click OK.
6. Enter any special libraries to be included, if necessary, and click Finish to create the new project directory and open the schematic page editor.

Once you have created the new PSpice project, use the Copy and Paste commands in Capture to copy the entire pre-existing design into the new project. After copying, be sure to setup the simulation profile and prepare the parts for simulation as described in [Checklist for simulation setup](#) and [Setting up analyses](#).

**Note:** You must use parts in the design that come from the PSpice set of part libraries, or create them with the appropriate PSpice properties (such as *PSPICETEMPLATE*, Implementation type = PSpice Model, etc.) in order for them to be simulated with PSpice.

## **PSpice Help**

### Preparing your design for simulation

---

---

# Setting up your design for simulation

---

## 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 the design entry program such as Capture or Design Entry HDL, others come from libraries (which can also be generated by other programs like the Stimulus Editor and the Model Editor), and still others are user-defined.

## Files that Design Entry Programs generate

When you begin the simulation process, the design entry programs first generate 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 design entry program generate for this file is `DESIGN_NAME-DESIGN_NAME.NET`. The netlist file is located in the directory:

```
<project_directory>\worklib\<design_name>\cfg_analog\
```

## Other files that you can configure for simulation

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

The simulation profile contains references to the other user-configurable files that PSpice needs to read.

You can create these files using PSpice programs like the Stimulus Editor and the Model Editor. These programs automate file generation and provide graphical ways to verify the data. You can also use the Model Text view in the Model Editor (or another text editor like Notepad) to enter the data manually.

## **Model library**

PSpice uses this information in a model library 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 PSpice 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 by:

- manually using the Model Text view in the Model Editor (or another text editor like Notepad),

or

- automatically using the Model Editor.

## **Stimulus file**

You can create a stimulus file either by:

- manually using the text editor in PSpice (or a standard text editor) to create the definition (a typical file extension is .STM)

- or -

- automatically using the Stimulus Editor (which generates an .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.

You can create an include file using any standard text editor. Typically, include file names have a .INC extension.



*Tip*

An include file can contain definitions, using the PSpice .FUNC command, for functions that you want to use in numeric expressions elsewhere in your design.

## **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 a profile only, a design only (local) or to any design (global).
- Change the search order.

To configure these, edit the simulation profile by using the Configuration Files tab in the Simulation Settings dialog box.

## **Files that PSpice generates**

After 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 data file and the PSpice output file.

## Probe data file

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

After PSpice has read the data file and displays the initial set of results, you can add more waveforms to perform post-simulation analysis of the data.

There are two ways to add waveforms to the display:

- From within PSpice, by specifying trace expressions.
- From within the design entry program, by cross-probing.

## 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 parts (like VPRINT1 and VPLOT1) that you place and connect to nets in your design.

## Checklist for simulation setup

This section describes 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.



## PSpice Help

### Setting up your design for simulation

---

**Note:** You must create a new project (not a design) and select the Analog or Mixed-Signal Circuit Wizard option in order to be able to simulate the new design with PSpice.

#### Typical simulation setup steps

For more information on this step...	See this...	To find out this...
Set component values and other properties.	Using parts that you can simulate	An overview of vendor, passive, breakout, and behavioral parts.
	Specifying values for part properties	Things to consider when specifying values for part properties
	Using global parameters and expressions for values	How to define values using variable parameters, functional calls, and mathematical expressions.
Define power supplies.	Defining power supplies	An overview of DC power for analog circuits and digital power for mixed-signal circuits.
Define input waveforms.	Defining stimuli	An overview of DC, AC, and time-based stimulus parts.
Set up one or more analyses.	Setting up analyses	Procedures, general to all analysis types, to set up and start the simulation.  Detailed information about DC, AC, transient, parametric, temperature, Monte Carlo, sensitivity/worst-case, and digital analyses.
Place markers.	Using Markers	How to display results in PSpice by picking nets in your design.
Limiting data file size	Data collection options for simulation profiles	How to limit the data file size.

## PSpice Help

### Setting up your design for simulation

---

#### Advanced design entry and simulation setup steps

For more information on this step...	See this...	To find out how to...
Create new models.	PSpice User's Guide	Define models using the Model Editor or Create Subcircuit command.
Analog behavioral modeling	PSpice User's Guide	Define the behavior of a block of analog circuitry as a mathematical function or lookup table.
Digital device modeling	PSpice User's Guide	Define the functional, timing, and I/O characteristics of a digital part.
Create new parts.	Creating Parts for Existing Simulation Models	Create parts either automatically for models using the Parts utility, or by manually defining AKO parts; define simulation-specific properties.
	Capture User Guide	Create and edit part graphics, pins, and properties in general.
	Design Entry HDL User Guide	

### When netlisting fails or the simulation does not start

If you have problems starting the simulation, there may be problems with the design or with system resources. If there are problems with the design, PSpice displays errors and warnings in the Simulation Status window. You can use the Simulation Status window to get more information quickly about the specific problem.

To get online information about an error or warning shown in the Simulation Status window

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.

## **PSpice Help**

### Setting up your design for simulation

---

#### Things to check in your design

Make sure that...	To find out more, see this...
The model libraries, stimulus files, and include files are configured.	Configuring model libraries
The parts you are using have models.	Unmodeled parts and Defining part properties needed for simulation
You are not using unmodeled pins.	Unmodeled pins
You have defined the grounds.	Missing ground
Every analog net has a DC path to ground.	Missing DC path to ground
The part template is correct.	Defining part properties needed for simulation
Hierarchical parts, if used, are properly defined.	The Capture User Guide
Ports that connect to the same net have the same name.	The Capture User Guide

#### Things to check in your system configuration

Make sure that...	To find out more, see this...
The directory containing your design 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 part libraries for PSpice supply numerous parts designed for simulation. The PSpice libraries are located in the PSpice sub-folder in the *tools\capture\library* directory under your main installation directory. They include:

- vendor-supplied parts
- passive parts
- breakout parts
- behavioral parts
- special simulation-only 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 part (for some kinds of analog behavioral parts).
- A part with modeled pins to form electrical connections on your schematic.
- A translation from design part 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. The libraries contained in the PSpice subfolder are the only ones set up for simulation.

**Note:** You must use the '0' (zero) ground part in designs intended to be simulated by PSpice. If you have used other ground parts, you can rename them to '0' so that they will be accepted by PSpice.

## Special simulation-only parts

The PSpice part libraries also include special parts that you can use for simulation only. These include:

- stimulus parts to generate input signals to the circuit (see Defining stimuli)
- ground parts required by all analog and mixed-signal circuits, which need reference to ground
- simulation control parts to do things like set bias values

- output control parts to do things like generate tables and line-printer plots to the PSpice output file

## **Vendor-supplied parts**

The PSpice libraries provide an extensive selection of manufacturers' analog and digital parts. Typically, the library name reflects the kind of parts contained in the library and the vendor that provided the models. For example, MOTOR\_RF.OLB and MOTOR\_RF.LIB contain parts and models, respectively, for Motorola-made RF bipolar transistors.

Two types of libraries are provided with PSpice:

- Standard PSpice libraries
- PSpice Advanced Analysis libraries

## **Standard PSpice libraries**

The standard PSpice libraries feature over 16,000 analog and 1,600 digital and mixed-signal models of devices manufactured in North America, Japan, and Europe.

Use parts from standard PSpice libraries or PSpice Advanced Analysis libraries if you want to analyze the part with PSpice.

The standard PSpice libraries are installed in the following locations in the installation directory:

- Capture symbols for standard PSpice libraries at

`\tools\capture\library\pspice`

- Standard PSpice model libraries at

`\tools\pspice\library\`

The parts in the standard PSpice libraries are listed in the online PSpice Library List. For information on finding parts using the online PSpice Library List, see Finding the part that you want. To find out more about each model library, read the comments in the .LIB file header.

## **PSpice Advanced Analysis libraries**

The PSpice Advanced Analysis libraries contain over 4,300 analog parts. The Advanced Analysis libraries contain parameterized and standard parts. The majority of the parts are parameterized. The parametrized parts have tolerance, distribution, optimizable and smoke

## PSpice Help

### Setting up your design for simulation

---

parameters that are required by the PSpice Advanced Analysis tools. Standard parts in the Advanced Analysis libraries are similar to parts in the standard PSpice libraries.

The parametrized parts are associated with template-based PSpice models. An important advantage of using the template-based PSpice models is that you can pass simulation parameters as properties from the schematic editor. For example, if a template-based model is associated with a part, the simulation parameters that you specify on an instance of the part in your design will be passed to the model. There is no need to edit the model itself to change a parameter value. This is unlike the standard PSpice parts that are associated with device characteristic curve-based PSpice models, where you need to edit the model to change a simulation parameter. For more information on template-based and device characteristic curve-based PSpice models, see Chapter 4, “Creating and editing models”, in the online PSpice User’s Guide.

Use parametrized parts from Advanced Analysis libraries if you want to analyze the part with an Advanced Analysis tool.

This Advanced Analysis tool...	Uses these part parameters...
Sensitivity	Tolerance parameters
Optimizer	Optimizable parameters
Smoke	Smoke parameters
Monte Carlo	Tolerance parameters, Distribution parameters (default parameter value is Flat / Uniform)

**Note:** You may use a mixture of standard and parameterized parts in your design, but Advanced Analysis is performed on only the parameterized components.

The Advanced Analysis libraries are installed in the following locations in the installation directory:

- Capture symbols for Advanced Analysis libraries at

`\tools\Capture\Library\PSpice\AdvAnls\`

- PSpice Advanced Analysis model libraries at

`\tools\PSpice\Library`

The parts in the Advanced Analysis libraries are listed in the online PSpice Advanced Analysis Library List. For information on finding parts using the online PSpice Advanced Analysis Library List, see Finding the part that you want. To find out more about each model library, read the comments in the .LIB file header.

## Part naming conventions

The part names in the PSpice 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.

## 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 Capture and restricting the parts list to those names that match a specified wildcard text string, or
- the online PSpice Library List or the PSpice Advanced Analysis Library List and searching for the generic part name using capabilities of the Adobe Acrobat Reader.

### To find parts using the parts browser

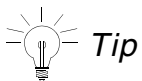
1. In Capture, from the Place menu, choose Part.
2. In the Part Name text box, type a text string with wildcard characters that approximates the part name that you want to find. Use this syntax:

`<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.



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

## To find parts using the online PSpice Library List

If you want to find out more about a part supplied in the PSpice libraries, such as manufacturer or whether you can simulate it, then search the online library lists.

Separate library lists are provided for standard PSpice libraries and Advanced Analysis libraries. The parts in the standard PSpice libraries are listed in the online PSpice Library List. The parts in the Advanced Analysis libraries are listed in the online PSpice Advanced Analysis Library List.



### *Tip*

This method finds only parts that Cadence supplies that have models.

1. Do one of the following:

- ☐ From the Windows Start menu, choose the OrCAD release programs folder and then the Cadence Help shortcut.
- ☐ From the Help menu in PSpice, choose Documents.

The Cadence Help window appears.

2. Press F2 or choose View - Navigation - Show to open the list of documents in the hierarchy if it is not already shown.
3. Click the PSpice category to show the documents in the category.
4. Double-click PSpice Library List or PSpice Advanced Analysis Library List.

This opens the library list. Click the PDF Viewer icon on the Cadence Help toolbar if required.

5. Follow the instructions in the second page of the library list to use the library list.



### *Tip*

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 Adobe Acrobat manuals.

## Passive parts

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



## PSpice Help

### Setting up your design for simulation

---

To find out more about how to use these parts and define their properties, look up the corresponding PSpice device letter in the Analog Devices chapter in the online PSpice Reference Manual, and then see the Capture Parts sections.

These parts are available..	For this device 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 <sup>1</sup> KCOUPLE <sup>1</sup>	coupled transmission line	T and K

1. For these device types, the PSpice libraries supply several parts. Refer to the online PSpice Reference Manual for the available parts.

## Breakout parts

The PSpice 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.

## PSpice Help

### Setting up your design for simulation

---

To find out more about how to use these parts and define their properties, look up the corresponding PSpice device letter in the Analog Devices chapter in the online PSpice Reference Manual, and then see the Capture Parts sections.

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

\* For this device type, the PSpice libraries supply several breakout parts. Refer to the online PSpice Reference Manual for the available parts.

## Behavioral parts

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

## Analog behavioral parts

These parts use analog behavioral modeling (ABM) to define each part's behavior as a mathematical expression or lookup table. The PSpice 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.

### Digital behavioral parts

These parts use special behavioral primitives to define each part's functional and timing behavior. These primitives are:

LOGICEXP	to define logic expressions
PINDLY	to define pin-to-pin delays
CONSTRAINT	to define constraint checks

Many of the digital parts provided in the PSpice libraries are modeled using these primitives. You can also create your own digital behavioral parts using these primitives.

### Defining part properties needed for simulation

If you want to use a part for simulation, then your part should have the PSPICETEMPLATE property defined for it:

You can also add other simulation-specific properties for digital parts: IO\_LEVEL, MNTYMXDLY, and PSPICEDEFAULTNET.

Here are the things to check when editing part properties:

- Does the PSPICETEMPLATE specify the correct number of pins/ nodes?
- Are the pins/ nodes in the PSPICETEMPLATE specified in the proper order?
- Do the pin/ node names in the PSPICETEMPLATE match the pin names on the part?

For examples of how to use the PSPICETEMPLATE property, see PSPICETEMPLATE examples.

### Editing simulation properties

To edit a property needed for simulation

1. In the schematic page editor, select the part to edit.
2. From the Edit menu, choose Properties to display the Parts spreadsheet of the Property Editor.

3. Click in the cell of the column you want to change (for example, PSPICETEMPLATE), or click the New button to add a property (and type the property name in the Name text box).
4. If needed, type a value in the Value text box.
5. Click Apply to update the design, then close the spreadsheet.

### **PSPICETEMPLATE property**

The PSPICETEMPLATE property defines the PSpice syntax for the part's netlist entry. When creating a netlist, Capture substitutes actual values from the circuit into the appropriate places in the PSPICETEMPLATE syntax, then saves the translated statement to the netlist file.

Any part that you want to simulate must have a defined PSPICETEMPLATE property. These rules apply:

- The pin names specified in the PSPICETEMPLATE property must match the pin names on the part.
- The number and order of the pins listed in the PSPICETEMPLATE property must match those for the associated .MODEL or .SUBCKT definition referenced for simulation.
- The first character in a PSPICETEMPLATE must be a PSpice device letter appropriate for the part (such as Q for a bipolar transistor).

### **PSPICETEMPLATE syntax**

The PSPICETEMPLATE contains:

1. regular characters that the schematic page editor interprets verbatim, and
2. property names and control characters that the schematic page editor translates.

### **Regular characters in templates**

Regular characters include the following:

- alphanumerics
- any keyboard part except the special syntactical parts used with attributes (@ & ? ~ #).
- white space

An identifier is a collection of regular characters of the form:

alphabetic character [any other regular character]\*.

### Property names in templates

Property names are preceded by a special character as follows:

[@|?|~|#|&]<identifier>

The schematic page editor processes the property 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> property 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.
~<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.



#### *Tip*

You can use different separator characters to nest conditional property clauses.

### The ^ character in templates

The schematic page editor replaces the ^ character with the complete hierarchical path to the device being netlisted.

Caution—Recommended scheme for netlist templates

Templates for devices in the part library start with a PSpice device letter, followed by the hierarchical path, and then the reference designator (REFDES) property.

It is recommended that you adopt this scheme when defining your own netlist templates.

Example: R^@REFDES for a resistor

### ***The \n character sequence in templates***

The part 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.

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 page 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. To avoid name conflicts in PSpice, the schematic page editor translates the following characters contained in pin names.

<b>This pin name character...</b>	<b>Is replaced with this...</b>
<	l (L)
>	g
=	e
\XXX\	XXXbar

**Note:** To include a literal % character into the netlist output, type %% in the template.

## **PSPICETEMPLATE examples**

### ***Simple resistor (R) template***

The R part has:

- two pins: 1 and 2
- two required properties: *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 part has:

- two properties: 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 page, treat G as an property in the template.

### 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.

### ***Digital stimulus parts with variable width pins template***

For a digital stimulus device template (such as that for a DIGSTIM part), a pin name can be preceded by a \* character. This signifies that the pin can be connected to a bus and the width of the pin is set to be equal to the width of the bus.

### Template

```
U^@REFDES STIM(%#PIN, 0) %*PIN  
  
\n+ STIMULUS=@STIMULUS
```

where #PIN refers to a variable width pin.

### Sample translation

```
U_U1 STIM(4,0) 5PIN1 %PIN2 %PIN3 %PIN4  
  
+ STIMULUS=mystim
```



where the stimulus is connected to a four-input bus, `a[0-3]`.

### ***Pin callout in subcircuit templates***

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.

### **IO\_LEVEL property**

The `IO_LEVEL` property defines what level of interface subcircuit model PSpice must use for a digital part that is connected to an analog part.

If you are creating a digital part, you need to

1. Add the `IO_LEVEL` property to the part and assign a value shown in the table below.

<b>Assign this value...</b>	<b>To use this interface subcircuit (level)...</b>
0	circuit-wide default
1	AtoD1 and DtoA1
2	AtoD2 and DtoA2
3	AtoD3 and DtoA3
4	AtoD4 and DtoA4

2. Use this property in the `PSPICETEMPLATE` property definition (`IO_LEVEL` is also a subcircuit parameter used in calls for digital subcircuits).

### **MNTYMXDLY property**

The `MNTYMXDLY` property defines the digital propagation delay level that PSpice must use for a digital part.

If you are creating a digital part, you need to do the following

1. Add the `MNTYMXDLY` property to the part and assign a value shown in the table below.

<b>Assign this value...</b>	<b>To use this propagation delay...</b>
0	circuit-wide default
1	minimum

## PSPice Help

### Setting up your design for simulation

---

2	typical
3	maximum
4	worst-case (min/max)

2. Use this property in the `PSPICETEMPLATE` property definition (`MNTYMXDLY` is also a subcircuit parameter used in calls for digital subcircuits).

### PSPICEDEFAULTNET property

The `PSPICEDEFAULTNET` pin property defines the net name to which a power or ground (invisible) pin is connected.

For example, if the power and ground pins on a digital part are connected to the digital nets `$G_DPWR` and `$G_DGND` respectively, then the properties are defined as follows:

```
PSPICEDEFAULTNET=$G_DPWR
```

```
PSPICEDEFAULTNET=$G_DGND
```

If you are creating a digital part, you need to do the following

1. For each power pin, create a `PSPICEDEFAULTNET` property and assign the name of the digital net to which the pin is connected.
2. Use the appropriate pin name in the `PSPICETEMPLATE` property definition.

### Specifying values for part properties

Note the following when specifying values for part properties:

- Do not leave a space between the value and its unit, if the unit is a scale symbol. For example, specify `5K` instead of `5 K`.  
  
For a listing of the scale symbols, see Numeric value conventions in the Before you begin chapter of the online PSPice Reference Guide.
- Do not use the European notation for specifying values. For example, if you specify `3K3` (the European notation for `3.3K`), PSPice reads the value as `3K`. Use `3.3K` instead.
- Specify tolerance values as percentages. If you specify an absolute value, the tolerance value will be read as an absolute number. For example, if you specify the value of the `POSTOL` property as a percentage, say `10%`, on a `10K` resistor, the distribution values will

be taken in the range of  $10K \pm 1K$ . If you specify the tolerance value as an absolute number, say 10, the distribution values will be taken in the range of  $10K \pm 10$ .

## 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

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).

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.

For 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`.

### Declaring and using a global parameter

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

- define the parameter using a `PARAM` part from *SPECIAL.OLB*
- use the parameter in place of a literal value somewhere in your design.

To declare a global parameter

1. Place a `PARAM` part in your design.
2. Double-click the `PARAM` part to display the Parts spreadsheet, then click New.
3. Do the following for each global parameter:

## PSpice Help

### Setting up your design for simulation

---

- ❑ Click *New*, then enter *NAMEn* in the Property Name text box, then click *OK*. This creates a new property for the PARAM part, *NAMEn* in the spreadsheet.
- ❑ Click in the cell below the *NAMEn* column and enter a default value for the parameter.
- ❑ While this cell is still selected, click *Display*. In the Display format frame, select *Name* and *Value*, then click *OK*.



***System variables have reserved parameter names. Do not use these parameter names when defining your own parameters.***

- ❑ Click *Apply* to update all the changes to the PARAM part, and then close the spreadsheet.

For example, to declare the global parameter VSUPPLY that will set the value of an independent voltage source to 14 volts, place the PARAM part, and then create a new property named VSUPPLY with a value of 14v.

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 property 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.

## 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.

For example, a parameter that changes with each step of a DC sweep or parametric analysis.

## Specifying expressions

To use an expression in your circuit

1. Find the numeric or boolean value you want to replace: a component value, model parameter value, other property value, or logic in an IF function test.
2. Replace the value with an expression using the following syntax:

```
{ expression }
```

where {expression} can contain any of the following:

- ☐ standard operators (listed in the Table below)
- ☐ built-in functions (listed in the PSpice User's Guide)
- ☐ user-defined functions
- ☐ system variables (listed in the PSpice User's Guide)
- ☐ user-defined global parameters
- ☐ literal operands

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

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; for digital parts, logical operators are used in Boolean expressions.

For lists of system variables and functions in arithmetic expression, refer to your PSpice User's Guide.

## Defining power supplies

### For the analog portion of your circuit

If the analog portion of 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 parts.

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

### For interfaces in mixed-signal circuits

#### ***Default digital power supplies***

Every digital part supplied in the PSpice libraries has a default digital power supply defined for its A-to-D or D-to-A interface subcircuit. This means that if you are designing a mixed-signal circuit, then you have a default 5 volt digital power supply built-in to the circuit at every interface.

#### ***Custom digital power supplies***

If needed, you can customize the power supply for different logic families.

For this logic family...	Use this part...
--------------------------	------------------

CD4000	CD4000_PWR
TTL	DIGIFPWR
ECL 10K	ECL_10K_PWR
ECL 100K	ECL_100K_PWR

## Defining stimuli

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

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

- the kind of analysis you are running,
- whether you are connecting to the analog or digital portion of your circuit, and
- how you want to define the stimulus: using the Stimulus Editor, using a file specification, or by defining part property values.

## Analog stimuli

Analog stimuli include both voltage and current sources. The following table shows the part names for voltage sources.

If you want this kind of input...	Use this part for voltage...
DC bias	VDC or VSRC
AC magnitude and phase	VAC or VSRC
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

\*\* VPWL\_F\_RE\_FOREVER and VPWL\_F\_N\_TIMES are file-based parts; the stimulus specification is saved in a file and adheres to PSpice netlist syntax.

To determine the part name for an equivalent current source

1. In the table of voltage source parts, replace the first V in the part name with I.

For example, the current source equivalent to VDC is IDC, to VAC is IAC, to VEXP is IEXP, and so on.

## Using VSTIM and ISTIM

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

## 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 parts with AC and DC properties
- VSRC or ISRC parts

## Using time-based stimulus parts with AC and DC properties

The time-based stimulus parts 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



## PSpice Help

### Setting up your design for simulation

---

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 properties, each of these parts also has a DC and AC property. When you use one of these parts, you must define all of the transient properties. However, it is 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 property...	Has this syntax...
DC	DC_value[units]
AC	magnitude_value[units] [phase_value]

For the meaning of transient source properties, refer to the I/V (independent current and voltage source) device type syntax in the Analog Devices chapter in the online PSpice Reference Guide.

### Using VSRC or ISRC parts

The VSRC and ISRC parts have one property 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 property...	Has this syntax...
DC	DC_value[units]
AC	magnitude_value[units] [phase_value]
TRAN	time-based_type (parameters)

where time-based\_type is EXP, PULSE, PWL, SFFM, or SIN, and the parameters depend on the time-based\_type.

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 PSpice Reference Guide.

**Note:** If you are running only a transient analysis, it is recommended that you use a VSTIM or ISTIM part if you have the standard package, or one of the other time-based source parts that has properties specific for a waveform shape.

## Digital stimuli

If you want this kind of input...Use this part....

For transient  
analyses

signal or bus (any width)	DIGSTIMn
clock signal	DIGCLOCK
1-bit signal	STIM1
4-bit bus	STIM4
8-bit bus	STIM8
16-bit bus	STIM16
file-based signal or bus (any width)	FILESTIMn

## Things to watch for

This section includes troubleshooting tips for some of the most common reasons your circuit design may not netlist or simulate.

Common reasons for simulation problems include:

Unmodeled parts

Unconfigured model, stimulus, or include files

Unmodeled pins

Missing ground

Missing DC path to ground

Using the FLOAT property

**Note:** You must use the '0' (zero) ground part in designs intended to be simulated by PSpice. If you have used other ground parts, you can rename them to '0' so that they will be accepted by PSpice.

## Unmodeled parts

If you see messages like this in the Capture Session Log:

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

then you may have done one of the following things:

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

### ***Do this if the part in question is from the PSpice libraries***

Replace the part with an equivalent part from an analog library or digital library with modeled parts.

- Make sure that you can simulate the part by checking the following:
- That it has a PSPICETEMPLATE property and that its value is non-blank.
- That it has Implementation `type=PSpice Model` and an Implementation property with a non-blank value.

**Note:** 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 PSPICETEMPLATE property if you are unsure or still getting warnings when you try to simulate your circuit.

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

- Are there blank (or inappropriate) values for the part's MODEL and PSPICETEMPLATE properties?

If so, load this part into the part editor and set these properties appropriately. One way to approach this is to edit the part that appears in your design.

***To edit the properties for the part in question***

1. In the schematic page editor, select the part.
2. From the Edit menu, choose part.

The part editor window appears with the part already loaded.

3. From the Edit menu, choose Properties and proceed to change the property values.

**Unconfigured model, stimulus, or include files**

If you see messages like these in the Capture Session Log,

(design\_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 Simulation Settings dialog box, click the Configuration Files tab and click Include, Library and Stimulus in the Category field.

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 Configuration Files tab, click Library in the Category field.

**Unmodeled pins**

If you see messages like these in the Capture Session Log,

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 PSpice libraries include parts that are suitable for both simulation and board layout. These parts may have a mix of modeled pins and unmodeled pins. 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.

## 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 ground parts named 0 (zero) connected appropriately in your design?

If not, place and connect one (or more, as needed) in your design. You can use the 0 (zero) ground part in SOURCE.OLB or any other ground part as long as you change its name to 0.

**Note:** You must use the '0' (zero) ground part in designs intended to be simulated by PSpice. If you have used other ground parts, you can rename them to '0' so that they will be accepted by PSpice.

## 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,  $1G\Omega$ ) resistor either:

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

## Using the FLOAT property

When preparing a circuit for simulation with PSpice, it's important to be sure that all pins for all parts are connected properly. If a pin is intentionally meant to remain unconnected, you need to use the PSpice pin property FLOAT, rather than a No Connect symbol. Otherwise, the circuit may not netlist correctly for PSpice.

## PSPice Help

### Setting up your design for simulation

---

The pin property FLOAT may have one of the following three values:

Value	Description
Error	The pin will not netlist. An error message will be returned when the PSPice simulation netlist is generated. Use Error when you want to be reminded that this pin is a "no connect" and should be treated in a special way. Error is the default value.
RtoGND	The pin is connected to a virtual resistor, whose opposite pin is tied to GND. The resistor has a value of 1/GMIN. This value allows the simulation netlist to be created and allows PSPice to perform the analysis. The virtual resistor will not be processed as part of a layout netlist or appear in a BOM.
UniqueNet	The pin, when left unconnected, is attached to a unique node when the PSPice simulation netlist is generated. Use UniqueNet when you want the pin to remain unconnected but correspond to the Probe data associated with its part.

The FLOAT property can either be defined in the Part Editor when creating a new part, or you can edit a pin on an existing part using the Property Editor spreadsheet.

To define the FLOAT property using the Property Editor spreadsheet

1. In Capture, double-click on the pin to open the Property Editor spreadsheet.
2. Click on the Pins tab.
3. Click New and type FLOAT (upper case) in the Property Name text box, then click OK.
4. Click in the cell under the FLOAT column for the pin, and then type the property value you want to use.
5. Click Apply to have the changes take effect.

### **Analog libraries with modeled parts**

1_SHOT	EPWRBJT	MOTOR_RF
ABM	FILTSUB	NAT_SEMI
ADV_LIN	FWBELL	OPAMP
AMP	HARRIS	OPTO
ANALOG	IGBT	PHIL_BJT
ANA_SWIT	JBIPOLAR	PHIL_FET
ANLG_DEV	JDIODE	PHIL_RF
ANL_MISC	JFET	POLYFET
APEX	JJFET	PWRBJT
BIPOLAR	JOPAMP	PWRMOS
BREAKOUT	JPWRBJT	SIEMENS
BUFFER	JPWRMOS	SWIT_RAV
BURR_BRN	LIN_TECH	SWIT_REG
CD4000	MAGNETIC	TEX_INST
COMLINR	MAXIM	THYRISTR
DIODE	MIX_MISC**	TLINE
EBIPOLAR	MOTORAMP	XTAL
EDIODE	MOTORMOS	ZETEX
ELANTEC	MOTORSEN	

\*\* Contains mixed-signal parts.

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



**Digital libraries with modeled parts**

7400	74H	DIG_ECL
74AC	74HC	DIG_GAL
74ACT	74HCT	DIG_MISC
74ALS	74L	DIG_PAL
74AS	74LS	DIG_PRIM
74F	74S	

## **PSpice Help**

### Setting up your design for simulation

---

---

# Performing Circuit Analysis

---

## Analyzing waveforms with PSpice

PSpice offers integrated waveform analysis functionality for viewing simulation results. Use waveform analysis for circuit performance analysis and data comparison from multiple files.

### What is waveform analysis?

After completing the simulation, PSpice plots the waveform results so you can visualize the circuit's behavior and determine the validity of your design.

Waveform analysis not only displays simple voltages and currents, but also complex arithmetic expressions involving voltages and currents and the Fourier Transform of these expressions. You can also use waveform analysis for Performance Analysis; see [Using Performance Analysis](#) for more information.

When a mixed analog/digital simulation runs successfully, PSpice displays analog and digital waveforms simultaneously with a common time base. You can add text labels and other annotation symbols to plots for clarification.

Taken together, simulation and waveform analysis is an iterative process. After analyzing simulation results, you can refine your design and simulation settings and then perform a new simulation and waveform analysis.

PSpice saves two kinds of waveform data file formats: ASCII and binary.

### Performing post-simulation analysis of the results

This means you can plot additional information derived from the waveforms. What you can plot depends on the type of analyses you run. Bode plots, phase margin, derivatives for small-signal characteristics, waveform families, and histograms are only a few of the possibilities. You can also plot other waveform characteristics such as rise time versus temperature, or percent overshoot versus component value.

## **Pinpointing design errors in digital circuits**

When PSpice detects setup and hold violations, race conditions, or timing hazards, a detailed message appears along with corresponding waveforms. PSpice also helps you locate the problem in your design.

## **What are the features of PSpice simulation profiles?**

PSpice has the following features in simulation profiles:

### **Increased reusability of simulation profiles**

Previous versions of PSpice allowed you to import settings only from a simulation profile that exists in the same project. You can now create a new simulation profile by importing settings from a simulation profile that exists in another project also. This increases the reusability of simulation profiles. For more information, see [Creating a new simulation profile](#).

### **Support for profile level configuration**

Previous versions of PSpice allowed you to configure model libraries, stimulus files and include files only at the global level (applicable to all designs) or at the design level (applicable only to the design). You can now configure model libraries, stimulus files and include files at the simulation profile level also.

The ability to configure model libraries, stimulus files and include files at the profile level has the following benefits:

- You can simulate the same circuit with different model libraries for each type of analysis and compare results. For example, you may want a part to be modeled using the DECODER model in the FRQCHK.LIB model library for AC analysis and using the DECODER model in the FRQCHK\_1.lib model library for transient analysis. You can create two simulation profiles—one for AC and one for transient analysis, configure the FRQCHK.LIB model library at the profile level in the simulation profile for AC analysis, and configure the FRQCHK\_1.LIB model library at the profile level in the simulation profile for transient analysis. Previous versions of PSpice required you to configure the model libraries at the design level and change the search order of the model libraries in a simulation profile before running the simulation, if you wanted to simulate the same circuit with different model libraries for each type of analysis.
- You can simulate the same circuit with different stimuli to test different characteristics like small signal response and large signal response, for each type of analysis. For example, you may want to simulate a circuit using the RF\_AMP.STL stimulus file for AC analysis

and the RF\_AMP1.STL stimulus file for transient analysis. You can create two simulation profiles—one for AC and one for transient analysis, configure the RF\_AMP.STL stimulus file at the profile level in the simulation profile for AC analysis, and configure the RF\_AMP1.STL stimulus file at the profile level in the simulation profile for transient analysis. Previous versions of PSpice required you to configure the stimulus files at the design level and change the search order of the stimulus files in a simulation profile before running the simulation, if you wanted to simulate the same circuit with different stimuli for each type of analysis.

- You can simulate the same circuit using different include files for each type of analysis. For example, you may want to simulate a circuit using the RF\_AMP.INC include file for AC analysis and the RF\_AMP1.INC include file for transient analysis. You can create two simulation profiles—one for AC and one for transient analysis, configure the RF\_AMP.INC include file at the profile level in the simulation profile for AC analysis, and configure the RF\_AMP1.INC include file at the profile level in the simulation profile for transient analysis. Previous versions of PSpice required you to configure the include files at the design level and change the search order of the include files in a simulation profile before running the simulation, if you wanted to simulate the same circuit with different include files for each type of analysis.

## Creating a new simulation profile

A simulation profile (\*.SIM) saves your simulation settings for an analysis type so you can reuse them easily.

You can create a new simulation profile from scratch or import the settings from an existing simulation profile. Importing settings from existing simulation profiles allows you to reuse the settings from other simulation profiles.

You can use PSpice, to create a new simulation profile by importing settings from a simulation profile that exists in the same project or in another project. Previous versions of PSpice allowed you to import settings only from a simulation profile that exists in the same project.

To create a new simulation profile

1. From the File menu in PSpice, point to New and choose Simulation Profile to display the New Simulation dialog box.
2. In the Profile Name text box, type a name for the profile (such as the name of the analysis type for the new profile).
3. In the Inherit From an Existing Profile text box, enter the name of another profile to import its settings into the new profile. You must enter an existing profile name in this text box or click browse to select an existing profile.

4. Click Create to create the profile and display the Simulation Settings dialog box.

See the following topics for information on the settings in the tabs in the Simulation Settings dialog box:

General tab	<a href="#"><u>General simulation settings for simulation profiles</u></a>
Analysis	<a href="#"><u>Analysis settings for simulation profiles</u></a>
Configuration Files	<a href="#"><u>Library settings for simulation profiles</u></a> <a href="#"><u>Stimulus settings for simulation profiles</u></a> <a href="#"><u>Include files settings for simulation profiles</u></a>
Options tab	<a href="#"><u>Options for simulation profiles</u></a>
Data Collection tab	<a href="#"><u>Data collection options for simulation profiles</u></a>
Probe window	<a href="#"><u>Probe windows settings for simulation profiles</u></a>

## Using a simulation profile

You can simulate the circuit using any of the simulation profiles you have created in the project. To use a simulation profile, open it in PSpice or make it active in the design entry program such as Capture or Design Entry HDL.


To open a simulation profile in PSpice

1. From the File menu, choose Open Simulation.
2. Select the <profile\_name>.SIM file and click Open.

To make a simulation profile active in Capture

Do one of the following:

- From the PSpice toolbar in Capture, select the simulation profile you want to use. See PSpice toolbar in Capture for more information.
- Select the simulation profile in the Capture Project Manager, and choose Make Active from the PSpice menu in Capture.

The  icon indicates an active simulation profile.

Each simulation profile is associated with a schematic. When you select a simulation profile in Capture, the folder for the associated schematic becomes the root schematic folder. When

you run PSpice, only the pages in the root schematic folder are netlisted. If the schematic pages in the root schematic folder refer to pages in another schematic folder (as in the case of hierarchical designs), the pages in that schematic folder also get netlisted.

## **Editing a simulation profile**

You can edit an existing simulation profile to change the simulation settings. You can edit the current simulation profile in PSpice. In the design entry program, you can edit the current simulation profile or select the simulation profile you want to edit.

### **To edit a simulation profile in PSpice**

1. From the Simulation menu, choose Edit Profile to display the Simulation Settings dialog box.
2. Make the required changes in the Simulation Settings dialog box.
3. Click OK to save the changes.

### **To edit the current simulation profile in design entry program**

1. From the PSpice menu, choose Edit Simulation Profile to display the Simulation Settings dialog box.
2. Make the required changes in the Simulation Settings dialog box.
3. Click OK to save the changes.

### **To select a simulation profile you want to edit in design entry program**

1. From the PSpice toolbar in the design entry program, select the simulation profile you want to edit.
2. From the PSpice menu in the design entry program, choose Edit Simulation Profile to display the Simulation Settings dialog box.
3. Make the required changes in the Simulation Settings dialog box.
4. Click OK to save the changes.

See the following topics for information on editing the settings in the tabs in the Simulation Settings dialog box:

General tab	<a href="#"><u>General simulation settings for simulation profiles</u></a>
Analysis	<a href="#"><u>Analysis settings for simulation profiles</u></a>
Configuration Files	<a href="#"><u>Library settings for simulation profiles</u></a> <a href="#"><u>Stimulus settings for simulation profiles</u></a> <a href="#"><u>Include files settings for simulation profiles</u></a>
Options tab	<a href="#"><u>Options for simulation profiles</u></a>
Data Collection tab	<a href="#"><u>Data collection options for simulation profiles</u></a>
Probe window	<a href="#"><u>Probe windows settings for simulation profiles</u></a>

## Deleting a simulation profile

You can delete a simulation profile only from the Capture Project Manager.

### To delete a simulation profile

1. Select the simulation profile in the Capture Project Manager and press Delete

**Note:** When you delete a simulation profile, the profile is deleted from the project but the files related to the simulation profile are not deleted from the project directory. You can manually delete the files from the project directory. For example, if you delete a simulation profile named TRAN, you must delete the TRAN folder located at

`<project_directory>\<design_name>-  
PSpiceFiles\<root_schematic_name>\.`

**Note:** If you delete a simulation profile but do not delete the files related to that profile, and later create a profile with the same name for the schematic, Capture displays an error message that a profile with the same name exists. You can overwrite the old profile or create another profile with a different name.

## Viewing the simulation queue

Use the Simulation Queue dialog box to set the order of a batch simulation and to view the progress of a batch simulation.





### To set up a batch simulation

1. From the File menu, choose Open Simulation.
2. Select multiple .SIM or .CIR files by doing one of the following:
  - ☐ Press CTRL while selecting to select individual files.
  - ☐ Press SHIFT while selecting to select a range of files.

**Note:** If the files you want to select are located in different directories, click Add in the Simulation Queue dialog box to find them in the various directories. The Simulation Queue dialog box will appear after you click Open (see below).

### To set the batch simulation order

1. After selecting the files for a batch simulation, click Open.  
The Simulation Queue dialog box appears. The file at the top of the Pending Simulations list is set to be simulated first.
2. To change the order, do one of the following:
  - ☐ Select a filename in the list and click either  or  to move it up or down.
  - ☐ Click the Reset button to reset the order.
  - ☐ Click Add to add more files to the simulation.
3. Click the Settings button to display the Simulation Settings dialog box, where you can view or change the simulation profile settings.
4. Click the Start button to start the simulations.

## General simulation settings for simulation profiles

Use the General tab of the Simulation Settings dialog box to specify the simulation input and output files and save a description of the simulation profile.

### Profile name

The Profile name text box displays the name of the current profile. You can change the name here.

## **Input settings**

Choose Schematic to have the simulation input come from a Capture design (\*.DSN file), or choose Circuit File (\*.CIR) to have the simulation input come from a circuit file. For either input type, enter the name of the file in the text box.

For a design-based simulation, also select the name of the top-level schematic page. Use the Analysis tab of this dialog box to enter simulation settings.

For a circuit-file-based design, the Analysis tab settings are unavailable. You can change the simulation settings by editing the commands in the .CIR file. See the online PSpice Reference Guide for more information about circuit file commands.

## **Output settings**

In the Output Filename text box, enter a file name (\*.OUT) for the simulation output to be saved to.

In the Waveform Data Filename text box, enter a filename (\*.DAT) for the simulation waveform data to be saved to.

**Note:** Use this text box to describe the design or simulation profile.

## **Analysis settings for simulation profiles**

Use the Analysis tab of the Simulation Settings dialog box to define the basic analysis type and set up additional advanced analyses and simulation options.

## **Analysis types**

From the Analysis Type list, select the basic analysis type you want to use. See Setting up analyses for more information about these.

## **Analysis options**

Under Options, select an advanced analysis you want to use. See Setting up analyses for more information about these.

## Enable PSpice AA support for legacy

Select the *Enable PSpice AA support for legacy* option to ensure that the legacy PSpice models are simulated in PSpice Advanced Analysis.

## Include files settings for simulation profiles

Use the Configuration Files tab of the Simulation Settings dialog box to configure and edit include files for simulation.

1. Click *Include in the Category field in the Configuration Files* tab, then configure or edit the include files.

Click this button...

To do this...

Browse

Browse directories for files to include—the selected file name appears in the Filename text box

Add as Global

Add the file listed in the Filename text box to be used for all designs

Add to Design

Add the file listed in the Filename text box to be used for the current design only

Add to Profile

Add the file listed in the Filename text box to be used for the current profile only

A file added at the design level cannot be added at the profile level also.

Note: Profile specific configuration is available only if the project was created using Capture from release 10.0 and beyond. This feature is also available for the project created using older versions of Capture but converted to the Capture 10.0 format.

Edit

Open the selected file for editing in PSpice.

Change

Change the selected file with the file specified in the Filename text box.



Remove the selected file from the Variant Files list—this does not delete the file from your computer



and




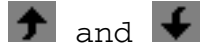
Move the selected file up or down in the Variant Files list.

## Library settings for simulation profiles

Use the Configuration Files tab of the Simulation Settings dialog box configure global, design and profile level model libraries.

1. Click Library in the Category field in the Configuration Files tab, and then configure the model libraries.

Click this button...	To do this...
Browse	Browse directories for libraries to make available to the profile—the selected file name appears in the Filename text box
Add as Global	Add the file listed in the Filename text box to be available to all designs
Add to Design	Add the file listed in the Filename text box to be available to the current design only Note: When you edit a model from Capture and save the changes, Model Editor automatically configures the model library at the design level.
Add to Profile	Add the file listed in the Filename text box to be available to the current profile only A file added at the design level cannot be added at the profile level also.  <b>Note:</b> Profile specific configuration is available only if the project was created using Capture from release 10.0 and beyond. This feature is also available for the project created using older versions of Capture but converted to the Capture 10.0 format.
Edit	Open the selected file for editing in Model Editor
Change	Change the selected file with the file specified in the Filename text box
	Remove the selected file from the Variant Files list—this does not delete the file from your computer



Move the selected file up or down in the Variant Files list.

PSpice locates models by searching the model libraries in the order in which they are listed in the Variant Files list. PSpice locates models by first searching the model libraries configured at the profile level, then the model libraries configured at the design level and finally the model libraries configured at the global level.

**Note:** You can change the search order only within the files configured at the same level. For example, you can change the search order only within the files configured at the profile level.

### Library search path

The Library Path text box displays the path that PSpice uses to search for models configured at the global level. You can edit or replace this path by typing a path in this text box, following these rules:

Use a semicolon (;) to separate two path names.

Do not follow the last path name with a semicolon.

#### **Example:**

To search first *C:\OrCAD\Orcad\_16.5\PSpice \Library*, then *C:\MYLIBS*, type the following:

```
"C:\OrCAD\Orcad_16.5\PSpice \Library";"c:\mylibs"
```




### Stimulus settings for simulation profiles

Use the Configuration Files tab of the Simulation Settings dialog box to configure global, design and profile level stimulus files.

Click Stimulus in the Category field in the Configuration Files tab, and then configure the stimulus files.

Click this button...    To do this...

Browse	Browse directories for libraries to make available to the profile- the selected file name appears in the Filename text box
--------	---

Add as Global	Add the file listed in the Filename text box to be available to all designs
Add to Design	Add the file listed in the Filename text box to be available to the current design only
Add to Profile	<p>Add the file listed in the Filename text box to be available to the current profile only.</p> <p>A file added at the design level cannot be added at the profile level also.</p> <p><b>Note:</b> When you edit a stimulus from Capture and save the changes, Stimulus Editor automatically configures the stimulus file at the profile level.</p> <p><b>Note:</b> Profile specific configuration is available only if the project was created using Capture from release 10.0 and beyond. This feature is also available for the project created using older versions of Capture but converted to the Capture 10.0 format.</p>
Edit	Open the selected file for editing in Stimulus Editor
Change	Change the selected file with the file specified in the Filename text box
	Remove the selected file from the Variant Files list—this does not delete the file from your computer
 and 	<p>Move the selected file up or down in the Variant Files list.</p> <p>PSpice locates the stimuli by searching the stimulus files in the order in which they are listed in the Variant Files list. PSpice locates the stimuli by first searching the stimulus files configured at the profile level, then the stimulus files configured at the design level and finally the stimulus files configured at the global level.</p> <p><b>Note:</b> You can change the search order only within the files configured at the same level. For example, you can change the search order only within the files configured at the profile level.</p>

## Options for simulation profiles

Use the Options tab of the Simulation Settings dialog box to fine-tune how PSpice performs calculations for analog and gate-level digital simulation, as well as what information to save to the simulation output file (\*.OUT).

## Option categories

From the Category list, select Analog Simulation, Gate-level Simulation, or Output file to display the settings for each option category. Click the Reset button to reset all the options to their default values.

## Analog simulation options

Use the Analog Simulation settings to fine-tune analog simulation accuracy, set iteration limits, set operating temperature, and specify MOSFET parameters.

The option names shown to the right of each text box correspond to the option names used in the PSpice .OPTIONS command. For more information about this command, refer to the online PSpice Reference Guide.

Click this button...    To do this...

AutoConverge	Suggest relaxed limits for various options that PSpice can modify during a simulation to achieve convergence.
MOSFET options	Enter values for the default drain area, default source area, default length, and default width.
Advanced options	Enter values for the total transient iteration limit, relative magnitude for matrix pivot, and absolute magnitude for matrix pivot.

The following tables defines all the options in the tab for the Analog Simulation category:

---

Flag option	Meaning
ADVCONV	Enables all convergence algorithms, such as Pseudo Tran, STEPGMIN, and step sources. ON by default.
DMFACTOR	Sets the relative factor for minimum delta. The value specifies the relative value by which the minimum time step size is changed. The value should be less than or equal to 1 and a factor of 10, such as .1, .001, or .0001.
GMINSRC	Enables step GMIN inside source-stepping
NOGMINI	Specifies not to add GMIN across current sources.
PSEUDOTRAN	Uses Pseudo-Transient Method.

## PSpice Help

### Performing Circuit Analysis

Flag option,	Meaning, <i>continued</i>
STPEGMIN	Enables GMIN stepping. This causes a GMIN stepping algorithm to be applied to circuits that fail to converge. GMIN stepping is applied first, and if that fails, the simulator falls back to supply stepping.
BRKDEPSRC	Sets automatic break-points for behavioral sources

Options	Description	Units	Default
ABSTOL <sup>1</sup>	best accuracy of currents	amp	1.0 pA
CHGTOL	best accuracy of charges	coulomb	0.01 pC
GMIN <sup>1</sup>	minimum conductance used for any branch	ohm <sup>-1</sup>	1.0E-12
GMINSTEPS	the GMIN stepping size in integer (any positive value). Set to 0 for engine default.		Same as ITL1
ITL1	DC and bias point blind repeating limit (the first evaluation of the operating point of the system)		150.0
ITL2	DC and bias point educated guess repeating limit (DC transfer curve iteration limit)		20
ITL4 <sup>1</sup>	the limit at any repeating point in transient analysis		10
ITL5 <sup>2</sup>	total repeating limit for all points for transient analysis (ITL5=0 means ITL5=infinity)		0.0
ITL6	the number of steps of the source stepping algorithm. Can have any positive integer value. Set to 0 for engine default.		Same as ITL1
LIMIT	the absolute voltage limit. The default, 0, specifies that there is no limit on data values. You can modify it to a large value, such as 1e12, to eliminate overflow errors, especially when using exponential sources.		0



**PSpice Help**  
Performing Circuit Analysis

Options	Description	Units	Default
method	integration method (values can be either TRAPEZOIDAL or GEAR)		
NUMDGT	number of digits output in print tables (maximum of 8 useful digits)		4.0
PIVREL <sup>2</sup>	relative magnitude required for pivot in matrix solution		1.0E-3
PIVTOL <sup>2</sup>	absolute magnitude required for pivot in matrix solution		1.0E-13
PTRANSTEP	number of steps for a pseudo transient analysis to find the operating point. Can be any positive integer value. Set to 0 for engine default.		Same as ITL1
RELTOL <sup>1</sup>	relative accuracy of V and I		0.001
SOLVER <sup>2</sup>	performance package solution algorithm (Solver = 0 selects the original solution algorithm; Solver = 1 selects the advanced solution algorithm)		1
THREADS	maximum number of threads. Set to 0 for engine default. THREADS=1 implies single-thread.		Number of cores/2
TRTOL	tolerance for integration error calculated using transient analysis. It is a relative tolerance where a higher TRTOL value results in bigger time steps and reduced accuracy. The TRTOL value should NOT be greater than 1/RELTOL.		7
VNTOL <sup>1</sup>	best accuracy of voltages	volt	1.0 uV
WCDEVIATION	worst case deviation. It can have double values between 0 and 1.		Same as RELTOL

1. These options can have an expression that uses the SCHEDULE function, which is a function of time.

2. PSpice now contains two solution algorithms for simulation. Solver 1 increases simulation speed over Solver 0, particularly for larger circuits with substantial runtimes. Solver 1 has slightly better convergence characteristics than Solver 0. Having both algorithms available improves convergence, since there are two different algorithms that can perform the simulation.

## Gate-level simulation options

Use the Gate-level Simulation settings to set timing, I/O levels for interfaces, drive strength, and error message limits.

Click this button... To do this...

Advanced options      Enter values for the minimum output drive resistance, maximum output drive resistance, overdrive ratio, default delay calculation, and error message limits.

The option names shown to the right of each text box correspond to the option names used in the PSpice .OPTIONS command. For more information about this command, refer to the online PSpice Reference Guide.

## Output file options

Use the Output File settings to select the types of information PSpice saves to the simulation output file.

The option names shown to the right of each text box correspond to the option names used in the PSpice .OPTIONS command. For more information about this command, refer to the online PSpice Reference Guide.

The following tables defines all the options in the tab for the Output file category:

---

Flag option	Meaning
ACCT	Summary and accounting information is printed at the end of all the analyses (refer to your <i>PSpice User's Guide</i> for further information on ACCT).
EXPAND	Lists devices created by subcircuit expansion and lists contents of the bias point file.
LIBRARY	Lists lines used from library files.
LIST	Lists a summary of the circuit elements (devices).

## PSpice Help

### Performing Circuit Analysis

---

---

Flag option,	Meaning, <i>continued</i>
NOBIAS	Suppresses the printing of the bias point node voltages.
NODE	Lists a summary of the connections (node table).
NOECHO	Suppresses a listing of the input file(s).
NOMOD	Suppresses listing of model parameters and temperature updated values.
NOOUTMSG	Suppresses simulation error messages in output file.
NOPAGE	Suppresses paging and the banner for each major section of output.
OPTS	Lists values for all options.
NUMDG	Number of digits in printed values. This is 4 by default.

---

### Data collection options for simulation profiles

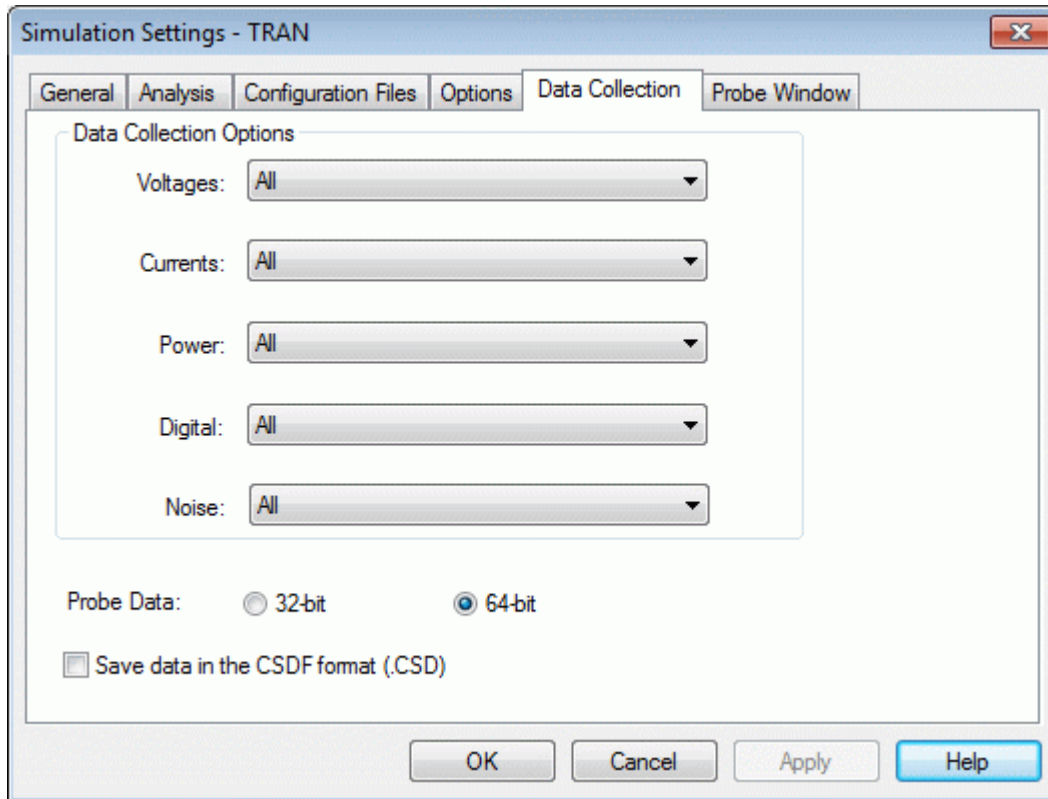
Use the Data Collection tab of the Simulation Settings dialog box to restrict the captured simulation data. This is especially useful for large circuit designs that produce more data than you need for waveform analysis.

You can also set the probe data accuracy from this tab. By default, probe data has 64-bit accuracy but you can choose a lower 32-bit accuracy.

## PSpice Help

### Performing Circuit Analysis

---



You can choose to apply the options in the table below to apply to Voltages, Currents, Power, Digital or Noise data. For more information on setting these options, click [Setting data collection options](#).

### Data Collection Options

Option	Description
All	All data will be collected and stored. (This is the default setting.)
All but Internal Subcircuits	All data will be collected and stored except for internal subcircuits of hierarchical designs (top level data only).
At Markers Only	Data will only be collected and stored where markers are placed.
None	No data will be collected.

### Waveform data file format option

Select the *Save data in the CSDF format (.CSD)* check box to save the waveform data file in the ASCII Common Simulation Data Format instead of in the default binary format.

### Probe Data accuracy options

By default, the probe data has 64-bit accuracy. You can set the accuracy by selecting a different radio button for *Probe Data*. A 64-bit accuracy setting ensures that the output does not have ramps. For example, when a small amplitude voltage is superimposed on a large amplitude voltage, the resolution might be lost, resulting in ramps.

### Probe windows settings for simulation profiles

Use the Probe Windows tab of the Simulation Settings dialog box to set up how Probe windows are displayed for a simulation profile.

For more Probe window options, see Setting Probe window options.

### Probe window display options

Use this option...

To do this...

Display Probe window  
when profile is opened

Display the Probe windows that were displayed the last time the  
profile was opened.

Display Probe window:

Display the Probe windows when:

- during simulation
- after simulation has  
completed.

- the simulation is running, and update the waveforms as the  
simulation progresses.
- the simulation is finished.

Show

Show the traces:

- All markers on open  
schematics
- Last plot

- For all the markers that are placed on currently open  
designs in the design entry program.
- That were used the last time the profile was opened.

---

# Traces

---

## Adding traces

You can add one or more analog or digital traces to the selected plot in the current Probe window. To view information about the trace, including which section and/or file its data came from, right-click the trace in the Probe window and choose Information.



*Tip*

You can add traces that use data from individual data files loaded into PSpice. Click here [To use a function or arithmetic operator](#) for more information.

## To add one or more traces to a plot

1. On the toolbar, click the *Add Trace* button to display the *Add Traces* dialog box:
2. Select the simulation output variable that you want to display by clicking one of the variables in the list. Click here [Narrowing the list of output variables](#) for information on narrowing this list.
3. Optional: Select operators, functions, or macros from the Functions or Macros list box to refine the data to be displayed. Depending on what you use, you may need to select more than one output variable.
  - ☐ Click here [Defining analog trace expressions](#) for more information on using operators, functions, and macros for analog traces.
  - ☐ Click here [Defining digital trace expressions](#) for more information on using operators, functions, and macros for digital traces.
4. Click OK.

## To modify a trace that is already displayed

1. Double-click the trace name in the plot legend. In the Modify Traces dialog box, you can perform the same functions as in the Add Traces dialog box, but apply them to the selected trace.

## Viewing trace information

You can view the trace name, the path for the data file from which the trace was generated (when you have more than one waveform data file loaded), information about the simulation that produced the waveform data file, and the number of data points used.

To view trace information

1. Do one of the following:

- ☐ Right-click a trace and choose Information.
- ☐ Double-click the trace symbol in the plot legend.

The Section Information dialog box appears, displaying information for the trace.

## Editing trace display properties

When you create a trace and it is assigned a color by your scheme setting, the color stays even if you delete a trace; the colors are not reassigned. The same is true if you change the color of an individual trace. Color schemes are specified in the Probe Options dialog box (from the Tools menu, choose Options).

Trace colors are reassigned when you apply a new scheme.

To edit the trace properties

1. Select and right-click a trace and choose Properties to display the Trace Properties dialog box. (You can also SHIFT+click to select multiple traces or use the Select All command from the Edit menu to select all the traces before right-clicking to change them all as a group.)
2. Set any of the following options:
  - ☐ From the Color list, select a color to use for the trace.
  - ☐ From the Pattern list, select a pattern to use for the trace.
  - ☐ From the Width list, select a width to use for the trace.
  - ☐ From the Symbol list, select a symbol to associate with the trace in the plot legend. Select Show Symbol to display the symbol on the trace itself.
3. Click OK to apply the new settings and close the Trace Properties dialog box.
4. If you do not see the changes immediately, from the View menu, choose Redraw to redraw the display.



## Setting grid display properties

You can change the grid properties separately for the major and minor grids on the x- and y-axes.

To edit the grid properties

1. Right-click a gridline, then choose Properties to display a Grid Properties dialog box.
2. Set any of the following options:
  - ☐ From the Color list, select a color to use for the grid.
  - ☐ From the Pattern list, select a pattern to use for the grid.
  - ☐ From the Width list, select a width to use for the grid.
  - ☐ Select the check box to apply these settings to the grid on the other axis.
3. Click OK to apply the new settings and close the Grid Properties dialog box.

## Setting plot edge properties

To edit the plot edge properties

1. Right-click a plot edge, then choose Properties to display a Plot Edge Properties dialog box.
2. Set any of the following options:
  - ☐ From the Color list, select a color to use for the grid.
  - ☐ From the Pattern list, select a pattern to use for the grid.
  - ☐ From the Width list, select a width to use for the grid.
3. Click OK to apply the new settings and close the Plot Edge Properties dialog box.

## Defining analog trace expressions

When defining analog trace expressions, you can include any combination of analog simulation output variables, arithmetic operators, functions, macros, and sweep variables.

For AC analysis, PSpice uses complex arithmetic to evaluate expressions and displays the magnitude of complex results. If the result is real (for example,  $\text{IMG}(V(4)+V(5))$ ), then it can be negative. If the result is complex, (for example,  $V(4)+(5)$ ), then the magnitude is displayed, which is always positive.

For procedures, see the following topics:

- ☐ [To use a function or arithmetic operator](#)
- ☐ [To use a sweep variable](#)



#### *Tip*

You can add traces that use data from individual data files that you loaded into PSpice. Click here [To use a function or arithmetic operator](#) for an example.



#### *Tip*

Device noise is available in addition to total input and output noise. These are generated only if you run a noise analysis. Click here [Device noise variables](#) to display the device noise output variables.

**Note:** Click here [How noise units are reported](#) for information on how noise units are reported.

## Analog Operators

Valid analog arithmetic operators:

( )	grouping
* /	multiplication/division
+ -	addition/subtraction
@	at a specific section and/or data file
#	refers to already added traces in the Probe window.

To refer to a trace use the syntax #*n*, where *n* is the number of the trace. For example, if *V(in)* is the first trace and *V(out)* is the second trace in the Probe window, *V(in)/V(out)* can be added just by writing #1/#2 in the *Add Trace* window.

## Analog Functions

Valid analog arithmetic functions:

ABS ( <i>x</i> )	<i>x</i>
ARCTAN ( <i>x</i> )	arc tangent of <i>x</i> with results in radians

## PSpice Help

### Traces

---

ATAN (x)	arc tangent of x with results in radians
AVG (x)	running average of x over the range of the X axis variable
AVGX (x, d)	running average of x from X_axis_value(x)-d to X_axis_value(x)
COS (x)	cos(x) with x in radians
D (x)	derivative of x with respect to the X axis variable <b>Note:</b> dV(node) is equivalent to d(V(node))
DB (x)	magnitude in decibels of x
ENVMAX (x, d)	envelope of x. Peaks selected have a minimum number of d consecutive datapoints.
ENVMIN (x, d)	envelope of x. Valley lows selected have a minimum number of d consecutive datapoints.
EXP (x)	the natural exponential function of x
G (x)	group delay of x with results in seconds
IMG (x)	imaginary part of x
LOG (x)	ln(x) with log base e
LOG10 (x)	log(x) with log base 10
M (x)	magnitude of x
MAX (x)	maximum value of x
MIN (x)	minimum value of x
P (x)	phase of x with results in degrees
PWR (x, y)	x to the power of y
R (x)	real part of x
RMS (x)	running RMS average of x over the range of the X axis variable
s (x)	integral of x over the range of the X axis variable <b>Note:</b> sIC(node) is equivalent to s(IC(node))
SGN (x)	+1 (if x>0), 0 (if x=0), -1 (if x<0)
SIN (x)	sin(x) with x in radians
SQRT (x)	the square root of x
TAN (x)	tan(x) with x in radians

## To use a function or arithmetic operator

In the Add Trace dialog box:

1. From the Functions or Macros list, select Analog Operators and Functions.
2. In the corresponding list, click the operator symbol or function name you want to use.
3. If the selection is a function, fill in the arguments list by doing the following:
  - a. In the Simulation Output Variables list, click the name of an output variable.
  - b. Repeat for as many arguments as are needed for the function call.

## Examples

- `V(Out1)@1`  
displays the `V(Out1)` data contained in the first section of available data.
- `V(Out2)@f2`  
displays the `V(Out2)` data contained in the second file of a set of loaded data files.
- `V(Out2)@"path_name"`  
displays the `V(Out2)` data contained in the specified .dat file, which must already be loaded.
- `V(Out1)@1@f2`  
displays the `V(Out1)` data in the first section in the second file of a set of loaded data files.

## Using analog output variables

To add a trace

1. From the Trace menu, choose Add Trace to display the Add Traces dialog box.

Enter names in the Trace Expression text box using the following notations.

□ `;<display_name>`

is the (optional) name you want to use to represent this trace expression on the plot.

□ `V(<node_name>)` or `V(<node_name_1>, <node_name_2>)`

For example, `V(3)` or `V(1,3)`.

## PSpice Help

### Traces

---

- ❑  $Vx(<device\_name>)$  and  $Ix(<device\_name>)$ , but not  $V(<device\_name>)$  or  $Vxy(<device\_name>)$ .

For example,  $VC(Q5)$ ,  $IB(Q2)$ , and  $VG(M2)$  are valid, but not  $V(R3)$  or  $VCE(Q13)$ .

- ❑  $I(device\_name)$  for current value through a device.
- ❑  $VG(x)$  or  $IG(x)$  for group delay for voltage and current values, respectively.
- ❑  $N<noise\_type>(<device\_name>)$  for the contribution from  $<noise\_type>$  of  $<device\_name>$  to the total output noise.

For example,  $NFID(M1)$  represents the flicker noise at MOSFET M1.

For noise values, use the variables as shown below:

#### **Total Noise Variables**

Use this variable...	For this noise value...
----------------------	-------------------------

$V(ONoise)$	output voltage
-------------	----------------

$V(INoise)$	input voltage
-------------	---------------

$I(INoise)$	input current
-------------	---------------

**Note:** Device noise is available in addition to total input and output noise. These are generated only if you run a noise analysis. Click here [Device noise variables](#) to display the device noise variables. Click here [How noise units are reported](#) for information on how noise units are reported.

#### **Device noise variables**

Click here [About noise units](#) for more information on how noise units are reported.

For this device...	Use these variables...
$B(GaAsFET)$	$NFID$ (Idrain flicker noise) $NRD$ (RD thermal noise) $NRG$ (RG thermal noise) $NRS$ (RS thermal noise) $NSID$ (Idrain shot noise) $NTOT$ (total noise)

## PSpice Help

### Traces

---

D (Diode)	NFID (Idrain flicker noise)
	NRS (RS thermal noise)
	NSID (Idrain shot noise)
	NTOT (total noise)
J (JFET)	NFID (Idrain flicker noise)
	NRD (RD thermal noise)
	NRG (RG thermal noise)
	NRS (RS thermal noise)
	NSID (Idrain shot noise)
M (MOSFET)	NTOT (total noise)
	NFID (Idrain flicker noise)
	NRB (RB thermal noise)
	NRD (RD thermal noise)
	NRG (RG thermal noise)
	NRS (RS thermal noise)
N (Digital Input)	NSID (Idrain shot noise)
	NTOT (total noise)
	NRHI (resistance noise between the digital device output and its PWR pin)
	NRLO (resistance noise between the digital device output and its GND pin)
O (Digital Output)	NTOT (total noise)
	NTOT (total noise)
Q (BJT)	NFIB (base current flicker noise)
	NRB (RB thermal noise)
	NRC (RC thermal noise)
	NRE (RE thermal noise)
	NSIB (base current shot noise)
	NSIC (collector current shot noise)
	NTOT (total noise)

R (Resistor)	NTOT (total noise)
S (Vswitch)	NTOT (total noise)
W (Iswitch)	NTOT (total noise)

### How noise units are reported

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 forms V(ONoise) or V(INoise)	$(volts)/(\sqrt{Hz})$
---	-----------------------

### Defining digital trace expressions

When defining digital trace expressions, you can include any combination of digital signals, buses, signal constants, bus constants, digital operators, and macros.

The following rules apply:

- An arithmetic or logical operation between two bus operands results in a bus value that is wide enough to contain the result.
- An arithmetic or logical operation between a bus operand and a signal operand results in a bus value.

For procedures, see the following topics:

- ☐ [To add a digital signal](#)
- ☐ [To add a bus](#)
- ☐ [To add a digital signal or bus constant](#)
- ☐ [To use a digital operator](#)
- ☐ [To add a digital signal or bus constant](#)
- ☐ [To use a digital operator](#)

## To add a digital signal

In the Add Trace dialog box:

1. Do one of the following:

- ☐ In the Simulation Output Variables list, click the signal you want to display.

or

- ☐ In the Trace text box, create a digital expression by either typing the expression, or by selecting digital signals from the Simulation Output Variables list and digital operators from the Digital Operators and Functions list.

2. If you want to name the signal with a name that is different from the node name:

- ☐ Click in the Trace text box after the last character in the signal name.
- ☐ Type ; .
- ☐ Type the name.

## Syntax

To specify the digital node name or expression to use in adding a digital signal, use the syntax:

`<digital_node_name>;<display_name>`

or:

`<digital_expression>;<display_name>`

where:

`<digital_node_name>` is the digital signal from the Simulation Output Variables list.

`<digital_expression>` is the expression using digital signals and operators understood by PSpice.

`<display_name>` is the (optional) name you want to use to represent this signal on the plot.

## Example

U2:Y;OUT1

specifies a digital trace using the node U2:Y, named OUT1 on the plot.



## To add a bus

In the Add Trace dialog box:

1. From the Functions and Macros list, select Digital Operators and Functions.
2. Click the { } entry.
3. In the Simulation Output Variables list, click the digital signals in high order to low order sequence.
4. If you want to name the bus with a name that is different from the default:
  - a. Click in the Trace text box after the last character in the signal name.
  - b. Type ; .
  - c. Type the name.
5. If you want to set the radix to something other than the default:
  - d. Click in the Trace text box after the last character in the bus definition.
  - e. Type ; .
  - f. Type the radix value.

## Syntax

Specify the contained signals and name of the bus using the syntax:

```
{digital signals list};display name;radix
```

or:

```
{bus_prefix[msb:lsb]};display name;radix
```

where:

<code>{digital signals list}</code>	is a comma- or space-separated list of up to 32 digital simulation output variables sequenced from high order to low order.
-------------------------------------	---

<code>{bus_prefix[msb:lsb]}</code>	is an alternate way to specify up to 32 signals in the bus.
------------------------------------	---

<code>Display name</code>	(optional) is the name you want to use to represent this bus on the plot.
---------------------------	---

`radix` (optional) is the numbering system in which you want to display the bus values.

### **Examples**

`{Q2 Q1 Q0};A;O`

specifies a 3 bit bus whose high order bit is the digital value at node Q2. On a plot, PSpice names the bus signal A, and values appear in octal notation.

`{a3 a2 a1 a0};;d`

specifies a 4 bit bus. On a plot, values appear in decimal notation. Since no display name is specified, PSpice names the bus signal with the signal list.

**Note:** `{a[3:0]}` is equivalent to `{a3 a2 a1 a0}`.

### ***radix***

Valid radix values:

H or X	hexadecimal (default)
D	decimal
O	octal
B	binary

### **To add a digital signal or bus constant**

For a signal constant

1. If needed, click in the Trace text box at the location where you want the constant.
2. In the Digital Operators and Functions list box, click on the signal constant value.

For a bus constant

1. Click in the Trace text box at the location where you want the constant.
2. Type the bus expression using the syntax:

`r'ddd`

where

r	is the lower-case bus constant radix
ddd	is the string of digits reflecting the constant value in the specified radix

***Examples***

x'3FFFF	hexadecimal
h'5a	hexidecimal
d'79	decimal
o'177400	octal
b'100110	binary

***signal constants***

`0	low
`1	high
`F	falling
`R	rising
`X	unknown
`Z	high impedance

***bus constant radix***

Valid bus constant radix values are all lower case as follows:

h or x	hexadecimal
d	decimal
o	octal
b	binary

**To use a digital operator**

In the Add Trace dialog box:

1. From the Functions and Macros list, select Digital Operators and Functions.
2. Click one of the digital operators in the corresponding list.
3. If the operator is the grouping operator:
  - a. Click the name of a node in the Simulation Output Variables list.
  - b. Repeat step a for each node in the group.

### ***Digital operators***

Valid digital arithmetic and boolean operators, listed in order of precedence:

( )	grouping
~	logical complement
* /	multiplication/division (bus values only)
+ -	addition/subtraction (bus values only)
&	and
^	exclusive or
	or

### **To use a sweep variable**

In the Add Trace dialog box, either select the sweep variable from the Simulation Output Variables list or type in the name as follows:

- ☐ For a DC sweep, use the name of the voltage or current source that was swept in the analysis.
- ☐ For an AC sweep, use Frequency.
- ☐ For a transient analysis, use Time.

### **Narrowing the list of output variables**

To restrict the simulation output variables

1. In the Simulation Output Variables text box, type a wildcard string that approximates the output variables you want to see, then press Enter.

Valid wildcards:

- |   |                                 |
|---|---------------------------------|
| * | matches zero or more characters |
| ? | matches exactly one character   |

**Note:** You can also restrict the list to include only the variable types of interest by selecting the check boxes to the right of the Simulation Output Variables list.

### ***Examples***

C\*

matches any output variable starting with C.

C?

matches any two-character output variable starting with C.

### **Deleting traces**

You can remove one or more analog or digital traces from a given plot in a Probe window.

To delete one or more traces from a plot

1. Do one of the following to select the first trace that you want to delete:
  - ☐ For an analog trace, select the trace name in the legend below the X axis.
  - ☐ For a digital signal, select the trace name to the left of the Y axis.
2. Shift-click on other trace names to select more traces for removal.
3. On the toolbar, click the Cut button to remove the trace or traces.

To delete all traces in the current plot

1. From the Trace menu, choose Delete All Traces.

### **Setting the digital plot size**

You can set the size of the digital plot to display on the screen to make more (or less) room for traces or trace names.

To set the digital plot size using the mouse

1. Display at least one digital trace and one analog trace in the Probe window for which you want to set the digital size.
2. To change the bottom position of the digital Probe window, do the following:
  - a. Place the cursor between the analog and digital parts of the plot.
  - b. Click the plot separator that you want to change.
  - c. Drag the plot separator until you have the digital size you want.
3. To change the left side of the digital Probe window, do the following;
  - a. Place the cursor at the left edge of the digital Probe window you want to resize.
  - b. Click the left edge that you want to move.
  - c. Drag the left edge of the digital Probe window to adjust the space available for displaying digital trace names.

To set the digital plot size using menu options

1. Display at least one digital trace in the plot for which you want to set the digital size.
2. From the Plot menu, choose Digital Size.
3. In the Digital Size dialog box, select the following:
  - a. Percentage of Plot to be Digital
  - b. Length of Digital Trace Name
4. Click OK.

## Using cursors

You can display the cursors on the plot, using two cursors per Probe window at a time. Each cursor displays the exact value of a single point on a curve. You can also change the number of digits displayed in the cursor box.

**Note:** If all the selected sections have one data point at the same X value, then the x-axis has only one tick mark and one value.

For more information, see the following topics:

- ☐ [To display both cursors](#)
- ☐ [To move the cursors using the mouse](#)

- ☐ To apply the cursors to a different trace
- ☐ To change the number of digits displayed
- ☐ To move the cursors using the keyboard

### **To display both cursors**

Do one of the following:

- On the toolbar, click the Cursor button.
- From the Trace menu, choose Cursor, then choose Display.

The cursor box appears on the screen, showing the current position of the cursor on the x- and y-axes. Press and hold either the left or right mouse buttons to alternate moving one or the other cursor. As you move the cursors, the values in the cursor box change. Move the cursor box by dragging the box to another location.

### **To move the cursors using the mouse**

Do one of the following:

- Click and drag to move cursor 1.
- Right-click and drag to move cursor 2.

### **To apply the cursors to a different trace**

If you want to apply the cursors to a different trace, click the trace symbol in the plot legend for the trace you want to change to.

If you are displaying a large number of traces, and the symbol is not shown for the trace you want to apply the cursors to, you can apply the cursors to that trace in one of the following ways:

1. Click in the cursor box (below the title bar) to freeze the cursor locations on the current trace.
2. After freezing the cursor locations, right-click on the new trace you want to apply the cursors to.

or -

1. From the Trace menu, choose Cursor, then choose Freeze to freeze the cursor locations on the current trace.
2. After freezing the cursor locations, right-click on the new trace you want to apply the cursors to.

or -

1. Press and hold the SPACEBAR while right-clicking on the new trace you want to apply the cursors to.

### **To change the number of digits displayed**

1. From the Tools menu, choose Options.
2. In the Number of Cursor Digits text box, type a number between 2 and 18.
3. Click OK.

### **To move the cursors using the keyboard**

Use the following key combinations to move the cursors.

To move this...	Press this...
Cursor 1 to the right or left	Right or left arrow
Cursor 2 to the right or left	Shift+right arrow or Shift+left arrow
Cursor 1 to the previous or the next trace	Ctrl+right or Ctrl+left arrow keys
Cursor 2 to the previous or next trace	Shift+Ctrl+right arrow or Shift+Ctrl+left arrow
Cursor 1 to the beginning or the end of the trace	Home or End



Cursor 2                      Shift+Home or  
to the beginning or      Shift+End  
the end of the  
trace

## Moving cursors along a trace

You can move cursors to view the coordinates of any point on a trace.

Move a cursor by moving the mouse to the desired location and clicking the appropriate mouse button. The left mouse button controls the first cursor, and the right mouse button controls the second cursor. The move is made by the last cursor that was moved. For example, if you right-click the plot to move cursor 2 and then click the max button on the toolbar, cursor 2 will move to the maximum point on the trace it is positioned on.

There are also default keyboard shortcuts for moving cursors.

The trace each cursor follows is determined by the selected trace symbol. To change the trace a cursor is on, click a different trace symbol in the legend.



***Cursor movements are not recorded during command logging.***

For more information, see the following topics:

- ☐ [To turn cursors on or off](#)
- ☐ [To move the cursor to the next peak](#)
- ☐ [To move the cursor to the next trough](#)
- ☐ [To move the cursor to the next slope](#)
- ☐ [To move the cursor to a minimum or maximum point](#)
- ☐ [To move the cursor to the next data point](#)
- ☐ [To move the cursor to the next transition](#)
- ☐ [To move the cursor to the previous transition](#)
- ☐ [To mark every data point with a symbol](#)

**To turn cursors on or off**

1. On the toolbar, click the Cursor button.

**To move the cursor to the next peak**

1. On the toolbar, click the Peak button.

**Note:** A first peak is a place where the points on each side of the peak have a lower y value.

**To move the cursor to the next trough**

1. On the toolbar, click the Trough button.

**Note:** A trough is the opposite of a peak. It is the low point in a curve of data.

**To move the cursor to the next slope**

1. On the toolbar, click the Slope button.

**Note:** The slope is the change in Y over the change in X. The slope can be either positive or negative.

**To move the cursor to a minimum or maximum point**

- To see the minimum value, on the toolbar, click the Min button..
- To see the maximum value, on the toolbar, click the Max button..

**To move the cursor to the next data point**

1. On the toolbar, click the Point button.

**To move the cursor to the next transition**

1. On the toolbar, click the Next Transition button.

**To move the cursor to the previous transition**

1. On the toolbar, click the Previous Transition button.

### **To mark every data point with a symbol**

1. On the toolbar, click the Mark Data Points button.
2. In the Probe Options dialog box, select the Mark Points check box.
3. Click OK.

### **Changing views**

You can change the way PSpice displays the traces on a plot. You can zoom in or out, change the center point, display only a selected area, see the previous view, and fit the display to the plot. You can also turn the display of the Toolbar and Status Bar on or off.

- ☐ [To zoom in or out](#)
- ☐ [To change the center point](#)
- ☐ [To display the selected area](#)
- ☐ [To redraw the screen](#)
- ☐ [To see the previous view](#)
- ☐ [To fit the view](#)
- ☐ [To display the Toolbar](#)
- ☐ [To display the status bar](#)

**Note:** The new items were formerly part of the Probe Options dialog box.

### **To zoom in or out**

1. On the toolbar, click View In or View Out.
  - ☐ View In zooms in by a factor of 2 around the point you specify.
  - ☐ View Out zooms out by a factor of 2 around the point you specify.

### **To change the center point**

1. From the View menu, choose Pan New Center.
2. Click the new center.

The screen redraws with the new center, maintaining the previous scale.

### **To display the selected area**

1. On the toolbar, click the Area button.
2. Click the mouse in the display.
3. Drag a box around the area you want to view.

The area is displayed.

### **To redraw the screen**

- From the View menu, choose Redraw.

The screen is immediately redrawn.

### **To see the previous view**

- From the View menu, choose Previous.

The screen is redrawn to the previous view, whether it was the last scroll position or the last screen setting.

To go back to more views, repeat the command.

### **To fit the view**

1. On the toolbar, click the Fit button.

The selected plot changes scale so that all data fits in the plot view on the screen.

### **To display the Toolbar**

- From the View menu, choose Toolbar.

A checkmark next to the menu command indicates that the Toolbar is displayed.

### **To display the status bar**

- From the View menu, choose Status Bar.

A checkmark next to the menu command indicates that the Status Bar is displayed.

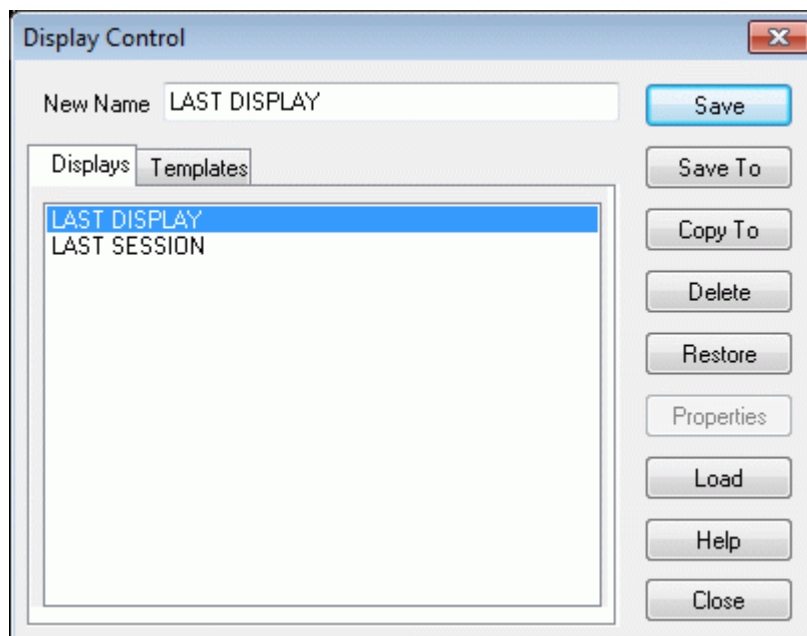
### To save or load a display

1. From the Window menu, choose Display Control.
2. Do one of the following:
  - ☐ To save the current display, type a name in the New Name box. Click Save.
  - ☐ To save to a different location, click the Save To button. Type a new name. Click OK.
  - ☐ To load a listed display, click the name and click the Restore button.
  - ☐ To load a display not listed, click the Load button, then select the name of the .PRB file to load. Click OK. Click the name of the display to load, then click the Restore button.
3. Click Close.

### To use a saved display

1. From the Window menu, choose Display Control.

The Display Control dialog box appears.



2. Click on the Displays tab.
3. Do one of the following:

- ☐ To use a display listed here, click the name.
- ☐ To use a display from another .PRB, click Load. Select the file. Click OK. Click the name of the display.

4. Click Restore.

**Note:** You can use a saved display to display traces as long as the current data file has variables with the same names as the variables in the display file.

### To load displays from another .PRB file

1. From the Window menu, choose Display Control.

The Display Control dialog box appears.

2. Click on the Displays tab.

3. Click Load.

4. Select the file.

5. Click OK.

**Note:** You can use displays saved in another .PRB file.

## Creating a Fourier Transform

Fourier Transforms (FFT) can be applied to all the analog traces in a Probe window.

You can use Fourier Transforms to examine the spectrum of the output of non-linear circuits. The resolution of the transformed display is determined by the extent of the original x-axis. The extent of the transformed x-axis is determined by the number of original data points.

If you want the Fourier transform display in PSpice to show more resolution, run the transient analysis for a longer time interval. Run the circuit for many cycles if necessary.

When Fourier is used to end the Fourier transform mode, all traces are drawn normally and the x-axis variable goes back to its original domain.

**Note:** You can take the Fourier transform of an expression of nodes (e.g., V(4) ? V(5)), but you cannot display an expression of Fourier transforms (e.g., FFT(V(4)) ? FFT(V(5))).

To view a Fourier Transform

1. Select a plot to view.

2. On the toolbar, click the Fourier Transform button.

Fourier transforms of all traces are displayed.

To end the Fourier Transform

1. On the toolbar, click the Fourier Transform button.



***Measurement definitions are not supported in Fourier transform mode.***

## Cautions when using FFTs

To correctly evaluate the harmonic components of a waveform, you must apply the Fourier Transform to a waveform with an integral number of periods. The FFT of a waveform with a partial period (e.g., 2.9 periods instead of 3) will generate false harmonic information.

To get an exact number of periods

1. Restrict the data as necessary.

In general, using several periods will give better results.

## Changing axis settings

You can specify the way PSpice displays the x-axis or y-axis, or the x or y grids. The Axis Settings dialog box provides tabbed dialog boxes for defining the display characteristics for each of these options:

- To set the x-axis
- To set the y-axis
- To set the x-grid
- To set the y-grid

### To set the x-axis

1. Do one of the following to display the Axis Settings dialog box:
  - ☐ From the Plot menu, choose Axis Settings.
  - ☐ Double-click in the area below the plot where the x-axis values are listed.

2. Click the X Axis tab, and type or select the following:

- ☐ Data Range
- ☐ Scale
- ☐ Use Data
- ☐ Processing Options
- ☐ Axis Title
- ☐ Axis Variable

3. Click OK to apply the changes.

### **To set the y-axis**

1. Do one of the following to display the Axis Settings dialog box:

- ☐ From the Plot menu, choose Axis Settings.
- ☐ Double-click in the area below the plot where the x-axis values are listed.

2. Click the Y Axis tab, and type or select the following:

- ☐ Data Range
- ☐ Scale
- ☐ Y Axis Number: select an identification number from the list.
- ☐ Axis Position
- ☐ Axis Title: enter a title for the y-axis.

3. Click OK to apply the changes.

### **To set the x-grid**

1. Do one of the following to display the Axis Settings dialog box:

- ☐ From the Plot menu, choose Axis Settings.
- ☐ Double-click in the area below the plot where the x-axis values are listed.

2. Click the X Grid tab, and type or select the following:

- ☐ Automatic: select this to calculate the grid spacing automatically.



- ☐ Major
- ☐ Minor

3. Click OK to apply the changes.

### **To set the y-grid**

1. Do one of the following to display the Axis Settings dialog box:
  - ☐ From the Plot menu, choose Axis Settings.
  - ☐ Double-click in the area below the plot where the x-axis values are listed.
2. Click the Y Grid tab, and type or select the following:
  - ☐ Automatic: select this to calculate the grid spacing automatically.
  - ☐ Y Axis Number: select this to identify which y-axis the settings should be applied to.
  - ☐ Major
  - ☐ Minor
3. Click OK to apply the changes.

### **Using the ‘Save as Default’ and ‘Reset Defaults’ buttons**

The Save as Default button provides a means of setting the default preferences that are used when traces are added or when new axes are created. This will not affect the settings of previously existing axes or traces.

The Reset Defaults button will restore the default settings to the original settings that are used when the program is first run.

### **To set the x-axis data range**

1. Double-click the x-axis to display the Axis Settings dialog box.
2. Click the X Axis tab.
3. In the Data Range frame, choose either Auto Range or User Defined.  
If you choose User Defined, specify the range in the text boxes.
4. In the Scale frame, do the following:

- a. Type a beginning value and end value for the range.
  - b. Choose Linear or Log scaling.
5. In the Use Data frame, choose either Full or Restricted.  
If you choose Restricted, specify the range in the text boxes.
6. In the Processing Options frame, choose Fourier, Performance Analysis., or neither.
7. Click Axis Variable to select the variable for the x-axis.
8. Click the variable or trace you want as the variable for the x-axis. To see other available variables or traces, click the choices. Click here [Defining analog trace expressions](#) for more information on defining traces.
  - a. Click OK to close this dialog box.
9. Click OK.

#### **To set the y-axis data range**

1. Do one of the following:
  - ☐ From the Plot menu, choose Y Axis Settings.
  - ☐ Double-click the y-axis to display the Axis Settings dialog box.
2. Click Y Axis tab.
3. In the Data Range frame, choose either Auto Range or User Defined.  
If you choose User Defined, specify the range in the text boxes.
4. In the Scale frame, do the following:
  - a. Type a beginning value and end value for the range.
  - b. Choose Linear or logarithmic scaling.
5. In the Y Axis Number box, select an identification number for the y-axis.
6. In the Axis Title box, type a title for the y-axis.
7. Click OK.

## Adding a new Y axis

You can add a new y-axis to the active Probe window. You can add up to 3 y-axes. The added y-axis becomes the selected y-axis. All subsequent traces added are added to the selected y-axis.

To add a Y axis

1. Click the plot that you want to add a new y-axis to.
2. From the Plot menu, choose Add Y Axis.

## Deleting a Y axis

You can delete a y-axis you no longer want.



***Any traces on the axis are deleted when the axis is deleted.***

To delete a y-axis

1. Click the y-axis that you want to delete.
2. From the Plot menu, choose Delete Y Axis.

Adding a new Y axis

Changing axis settings

## Using multiple plots

You can display multiple plots at the same time in one Probe window. When several plots are in the same Probe window, you can select one, delete one, or work with them as synchronized or unsynchronized. You can also copy and paste traces among plots.

Plots from the same set of waveform data are automatically synchronized. You can use unsynchronized plots to independently apply different scales, Fourier or Performance Analysis, or evaluate measurement expressions.

Unsynchronizing plots releases the selected plot to have its own x-axis. Plots that share x-axes are always displayed together, one above the other.

For more information, see the following topics:

- ☐ [To add a new plot](#)
- ☐ [To select the current plot](#)
- ☐ [To delete a plot](#)
- ☐ [To unsynchronize plots](#)

### **To add a new plot**

1. Click in the Probe window to which you want to add the plot.
2. From the Plot menu, choose Add Plot to Window.

The new plot appears above the selected plot in the Probe window.

Now you can add traces.

### **To select the current plot**

Click the plot you want.

### **To delete a plot**

1. Click the Plot you want to delete.
2. From the Plot menu, choose Delete Plot.

### **To unsynchronize plots**

If the selected plot is the middle plot of three plots sharing an X axis, then the middle plot is moved to the top position.

**Note:** After you have unsynchronized a plot, you cannot resynchronize it. You must delete the plot and add a new plot.

To unsynchronize a plot

1. Click the plot you want to become unsynchronized.
2. From the Plot menu, choose Unsynchronize X Axis.

## Using Probe windows

When you open a waveform data file, a new Probe window appears.

You can have more than one Probe window at a time. Each Probe window can contain one or more plots. Each plot can contain both analog and digital traces. You can copy and paste traces among plots.

You can create, select, arrange, and delete Probe windows by using the Plot and Windows menus. The title for the Probe window is a list of the waveform data files open for that window.

Probe windows feature automatic grid spacing. As you resize a Probe window, the major grid spacing changes, and the grid numbering appears as the numerals fit on the screen.

You can save the contents of a Probe window by using Display Control. Display control saves the plot configuration, including number of plots, traces and labels in each plot, x- and y-axis settings, and x-axis variable.

You can print any or all of the plots in a Probe window.

## Toggling between display modes

You have the choice of using two different display modes.

The standard (default) display mode in PSpice includes the main Probe window, plus the output window and the simulation status window. This provides all possible information about the simulation run and contains all of the toolbars and settings.

The alternate display mode shows only the Probe window with any waveforms that have been plotted. This mode gives you just the plots you are interested in seeing without the additional simulation data normally provided by PSpice .

The toolbar and window settings are saved for each mode. Any changes you make in the settings will become the new default the next time you choose that display mode.

The alternate display mode can be very handy when you want to see the waveforms superimposed on the schematic diagram for easy debugging and testing of the circuit. You can customize the alternate display mode to view various toolbars or other PSpice windows, according to your own preferences.

**Note:** By default, the alternate display mode is set to be visible at all times (see Keeping the Probe window visible at all times).

To toggle between the standard and the alternate display modes:

1. From the View menu, choose Alternate Display or click the Alternate Display toolbar button.

## **Keeping the Probe window visible at all times**

Like any other application running under Windows, the PSpice window will remain in the forefront of the desktop only as long as it is the active window. In order to keep the PSpice window visible at all times, you can use the push pin feature.

By keeping the Probe window on top of other active windows, you can easily view the schematic page at the same time you see the corresponding waveform for that circuit. This allows you to cross-probe quickly and easily without having to activate the Probe window each time.

**Note:** The push pin button is a toggle; clicking on it when it is enabled will disable the “on top” function.

To make the Probe window visible at all times:

1. Click the push pin button in the toolbar or, from the View menu, choose Always on Top.

## **To print plots**

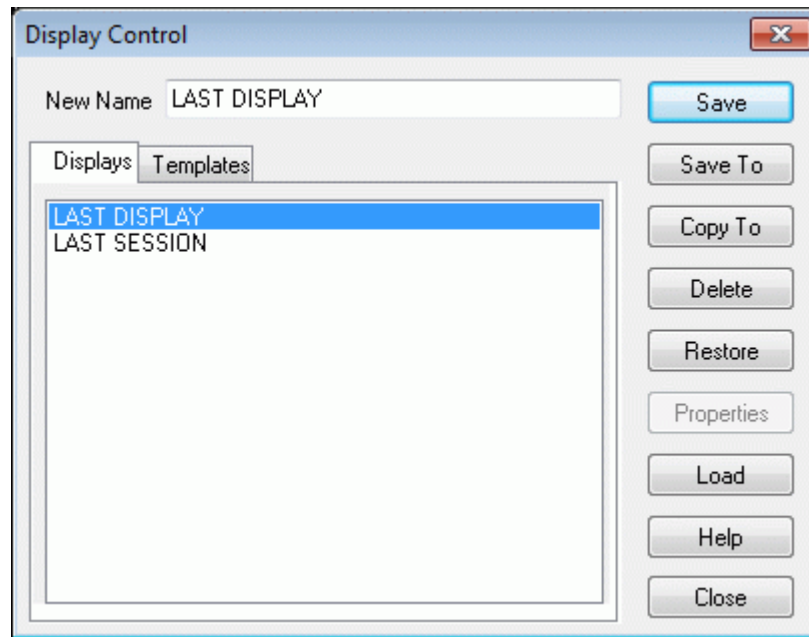
Do one of the following:

1. To print one copy of the current Probe window using the default print settings, click the Printer button on the toolbar.
2. Select the plots you want to print.
3. If needed, select a printer and printer information.
4. Click OK.

## **Using Display Control**

You can create displays to save the contents of a Probe window. You can view a display again at a later time with a different simulation so long as the new simulation has identically named variables.

Once the display is saved, you can copy it, edit it, and delete it.



For more information, see the following topics:

- ☐ [To save a display](#)
- ☐ [To copy a display](#)
- ☐ [To delete a display](#)
- ☐ [To use a saved display](#)
- ☐ [To load displays from another .PRB file](#)

#### **To save a display**

1. Set up the plots, traces, labels, and axes in the Probe window you want to save.
2. From the Window menu, choose Display Control.

The Display Control dialog box appears.

3. Click on the Displays tab.
4. In the New Name text box, type a name for the display.
5. Do one of the following:
  - ☐ To save the display in the current .PRB file, click Save.

- ☐ To save the display in another .PRB file, click Save To. Specify the name and location of the file. Click OK.

6. Click Close.

### **To copy a display**

1. From the Window menu, choose Display Control.

The Display Control dialog box appears.

2. Click on the Displays tab.
3. Click the name of the display to copy.
4. Click Copy To.
5. Specify the name and location of the copied display.
6. Click OK.
7. Click Close.

### **To delete a display**

1. From the Window menu, choose Display Control.

The Display Control dialog box appears.

2. Click on the Displays tab.
3. Do one of the following:

- ☐ To delete a display from the current .PRB file, click the name, then click Delete.
- ☐ To delete a display from a global or remote .PRB file, click Delete From, then select the .PRB file.

4. Click Close.

## **Using plot window templates**

PSpice provides plot window templates that allow you to create and reuse custom displays in Probe using defined arguments. A plot window template is a plot window consisting of one or more arguments used to represent node voltage, pin current, power or digital names within a display. An argument provides the means to replace a fixed node voltage or pin current name with a node voltage or pin current name you choose.



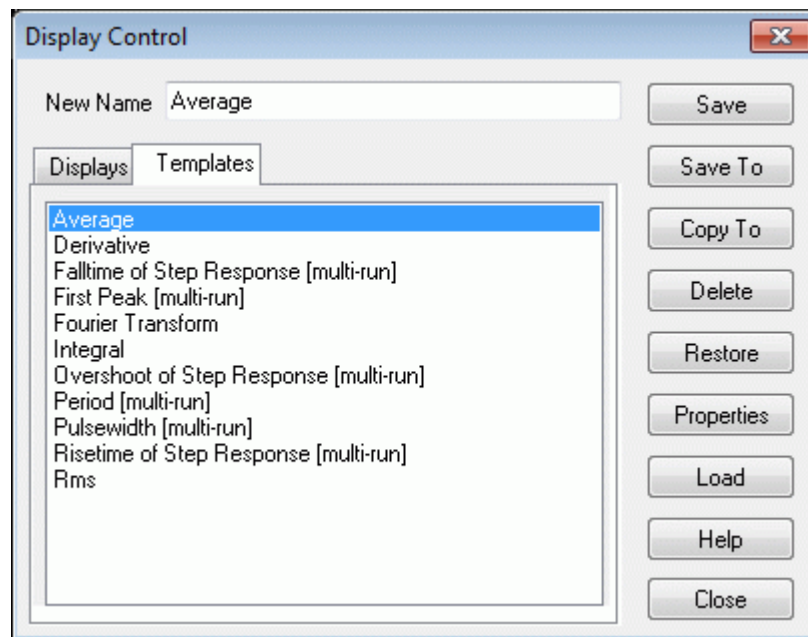
## PSpice Help

### Traces

---

You can create unique plot window templates for a particular design or general templates that can be applied to various designs. A set of some of the more commonly used templates are predefined and included with PSpice .

To work with plot window templates, from the Window menu, choose Display Control, and click the Templates tab. Here you can customize plot window templates in various ways. See the Related Topics below for more detailed information.



## Related Topics

For information about...	<a href="#">Click this topic...</a>
Creating a new template...	<a href="#">Creating a plot window template</a>
Modifying an existing template...	<a href="#">Modifying a plot window template</a>
Deleting a template...	<a href="#">Deleting a plot window template</a>
Copying a template...	<a href="#">Copying a plot window template</a>
Restoring a template...	<a href="#">Restoring a plot window template</a>
Viewing the properties of a template...	<a href="#">Viewing the properties of a plot window template</a>
Loading a template...	<a href="#">Loading a plot window template</a>
Placing plot window template markers...	<a href="#">Placing plot window template markers</a>

## Creating a plot window template

In order to create and save a new plot window template, you must first set up the active plot window in Probe with the configuration you want. The active plot window will be the basis for the template properties you save.

**Note:** Only those templates which apply to the active simulation are listed in the Template dialog box. For example, an AC simulation will show frequency domain templates such as Bode plot, while a transient analysis will show time domain templates such as risetime or pulsewidth. In addition, some predefined templates require multirun analyses (Monte Carlo analysis, time sweep, or parametric sweep).

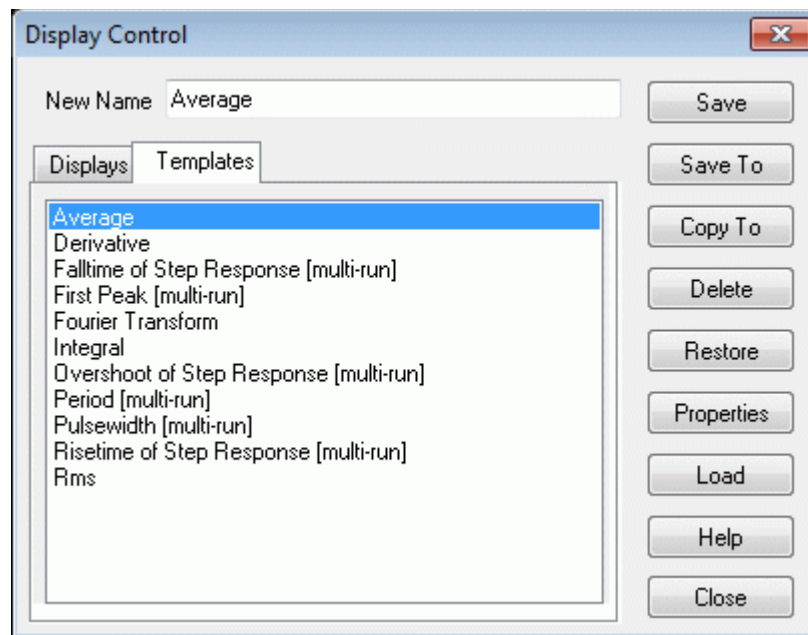
To create a new plot window template

1. In PSPice , from the Window menu, choose Display Control.
2. Click the Templates tab.

## PSpice Help

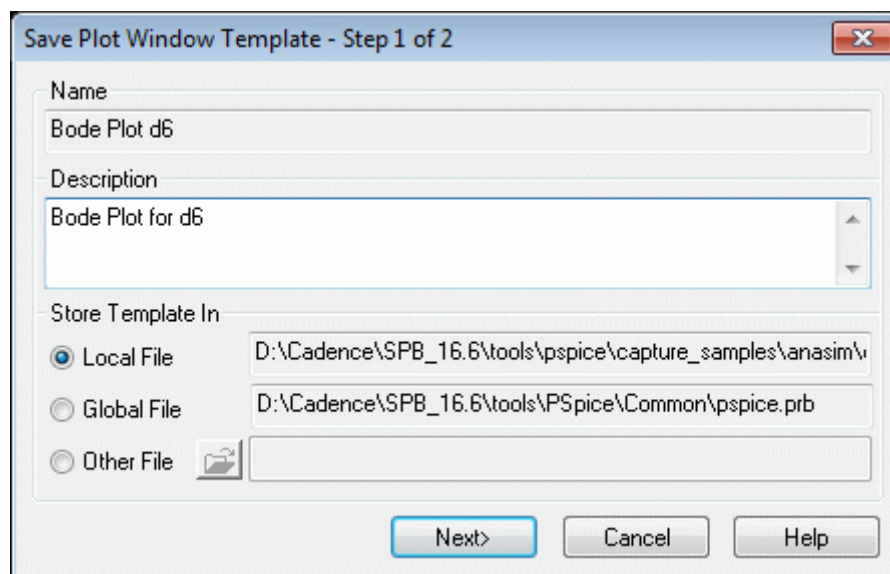
### Traces

---



3. In the New Name text box, enter the name for the new template you want to create.
4. Click Save or Save To.

The Save Plot Window Template – Step 1 of 2 dialog box appears.



5. In the Description text box, type in a description for the template, if you would like one. (This is optional.)

## PSpice Help

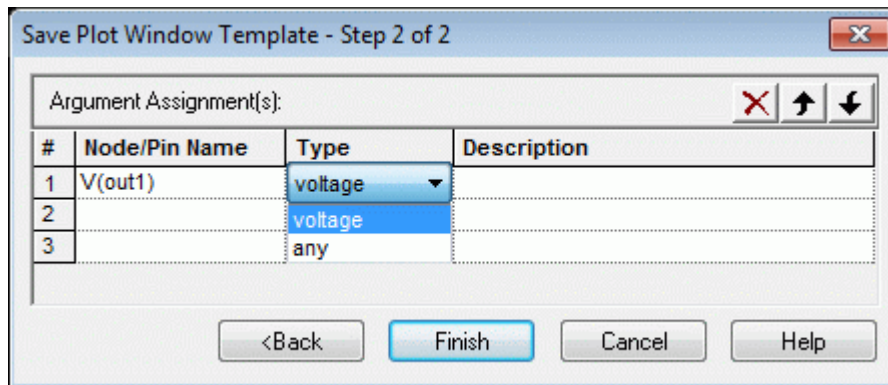
### Traces

6. If you clicked Save To instead of Save, choose the .PRB file you wish to save the template to by selecting the appropriate radio button under the Store Template In frame. (The default is the local .PRB file. For the Save function, the Local File is the only option.)

- ☐ Local File – the .PRB file for the current simulation in PSpice .
- ☐ Global File – the .PRB file to be used globally for all Probe displays.
- ☐ Other File – another .PRB file stored elsewhere on your hard disk or network drive. Use the Browse button to locate the file on a particular drive.

7. Click Next.

The Save Plot Window Template – Step 2 of 2 dialog box appears. The number of Node/ Pin Name arguments that are listed here is determined by the current display.



8. Define the association of each argument by selecting the node or pin name from the drop-down list under the column Node/Pin Name.

This drop-down list shows all of the available node voltage, pin current, power or digital names. If the drop-down list does not appear, click in the text box to activate the drop-down button.

9. For each argument, set the Type of argument to be used by selecting the argument name from the drop-down list under the column Type.

This drop-down list shows all of the available argument types (any, current, power, voltage). If the drop-down list does not appear, click in the text box to activate the drop-down button.

10. For each argument, under the Description column, type in a description, if you would like one. (This is optional.)

The description you enter here will be displayed in the status line of Capture when placing a marker associated with the argument.

11. If desired, change the order of the arguments by using the Arrow buttons to move an argument up or down in the listing. Or, you can delete an argument by selecting it and clicking the Delete button.
12. Click Finish.

**Note:** At least one argument is required to create a plot window template. The maximum number of arguments allowed is the number of unique node voltage, pin current, power or digital names in the active display.

## Modifying a plot window template

Modifying a plot window template is essentially the same as creating a new template. In order to modify a plot window template, that particular template must be the active plot window in Probe. If the active display is not the template you want to modify, use the Restore button to make a different template the active display in Probe (see Restoring a plot window template).

To modify a plot window template

1. From the Window menu, choose Display Control.
2. Click the Templates tab.
3. Select the template you want to modify by clicking on its name in the list of loaded templates. If the template you are looking for is not in the list, use the Restore button to make it the active display.
4. Click Save to display the Save Plot Window Template – Step 1 of 2 dialog box.
5. Make the desired changes, then click Next to display the Save Plot Window Template – Step 2 of 2 dialog box.
6. Make the desired changes, then click Finish.

The modifications will be saved and the display will be updated automatically.

**Note:** If an argument assignment no longer applies because the node voltage, pin current, power or digital names are mapped to an argument that has changed, then information regarding that argument will not be available in the Step 2 of 2 dialog box.

## Deleting a plot window template

You can remove a plot window template from the list of loaded templates. By deleting a plot window template, you remove it from the list of templates you can access and erase it from the .PRB file.

To delete a plot window template

1. From the Window menu, choose Display Control.
2. Click the Templates tab.
3. Click on the name of the plot window template you want to delete.
4. Click Delete.

## **Copying a plot window template**

You can copy a plot window template into another .PRB file to make it available for use later with that file.

To copy a plot window template

1. From the Window menu, choose Display Control.
2. Click the Templates tab.
3. Click on the name of the plot window template you want to copy.
4. Click Copy To.

The Probe File for Save Template dialog box appears.

5. Choose the .PRB file you wish to save the template to by selecting the appropriate radio button under the Store Template In frame. (The default is the local .PRB file.)
  - ☐ Local File – the .PRB file for the current simulation in PSPice .
  - ☐ Global File – the .PRB file to be used globally for all Probe displays.
  - ☐ Other File – another .PRB file stored elsewhere on your hard disk or network drive. Use the Browse button to locate the file on a particular drive.
6. Click OK.

## **Restoring a plot window template**

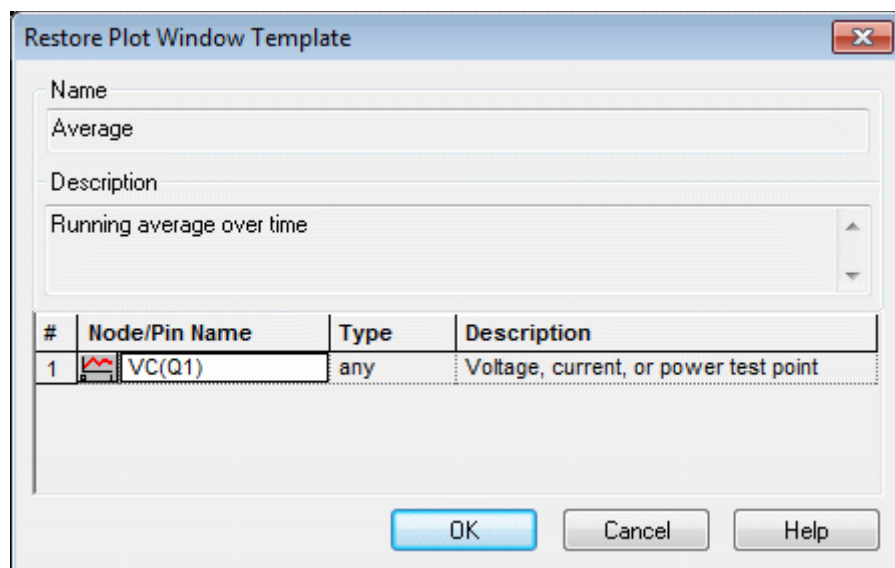
In order to make a plot window template the active display in Probe, you must restore it. This process recalls a previously defined plot window template and sets up a new plot window in Probe using the arguments associated with that template. In order for the arguments in the template to apply, you must replace the node voltage names or pin current names for each argument contained in the restored template.

**Note:** You can only restore plot window templates that are already loaded. If you want to restore a plot window template that does not appear in the list, you must first load it. To load a template, see Loading a plot window template.

To restore a plot window template

1. From the Window menu, choose Display Control.
2. Click the Templates tab.
3. Choose the plot window template you want to restore by clicking on its name in the list of loaded templates.
4. Click Restore.

The Restore Plot Window Template dialog box appears.



5. Reassign the node voltage names or pin current names for each argument in the list.
6. Click OK.

A new Probe window will be created and the restored plot window template will be displayed.

**Note:** You may also restore a plot window template by choosing the Add Trace command from the Trace menu, and then selecting Plot Window Templates from the drop-down list in the Functions or Macros frame.

## Viewing the properties of a plot window template

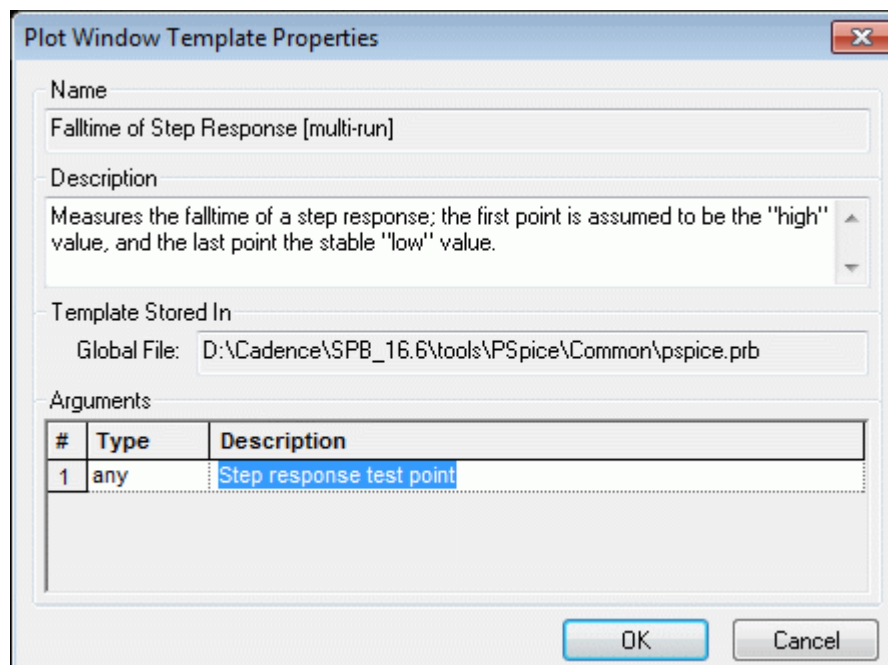
You can view the properties of a plot window template and change the description fields for the template or arguments it contains.

**Note:** Only those templates which apply to the active simulation are listed in the Template dialog box. For example, an AC simulation will show frequency domain templates such as Bode plot, while a transient analysis will show time domain templates such as risetime or pulsewidth. In addition, some predefined templates require multirun analyses (Monte Carlo analysis, time sweep, or parametric sweep).

To view the properties of a plot window template

1. From the Window menu, choose Display Control.
2. Click the Templates tab.
3. Click on the name of the plot window template you want to view.
4. Click Properties.

The Plot Window Template Properties dialog box appears.



5. Change the Description field for the template, or change the description for any of the Arguments, as desired.
6. Click Finish to exit and save any changes.



**Note:** When viewing the properties of a template, you can only edit the description fields. No other changes are allowed. If you want to modify the arguments or assignments, see [Modifying a plot window template](#).

## Loading a plot window template

You can load a plot window template from another .PRB file, and add it to the list of available templates. When you load a template, you do not make it the active display in Probe. You are only adding it to the list of available templates. (To restore the display of a newly loaded template, see [Restoring a plot window template](#).)

If a duplicate template is loaded, then the one you are loading will replace the current one in the list. If you close the data file and reopen it, any plot window templates that you loaded earlier will have to be loaded again to make them available. (Loaded templates are not saved with the data file.)

To load a plot window template

1. From the Window menu, choose Display Control.
2. Click the Templates tab.
3. Click Load.

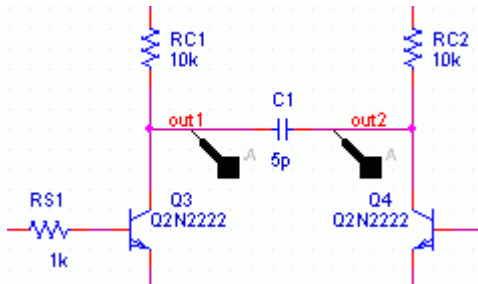
The Load Displays dialog box appears.

4. Locate the .PRB file that contains the plot window template you want to load.
5. Select the file and then choose Open.

The loaded templates will be listed in the Display Control dialog box.

## Placing plot window template markers

You can place a marker in Capture that represents a plot window template. The marker will restore the associated template when you run the simulation in PSpice . Markers for plot window templates are distinguished from other markers (for voltage, current, or power) by being square rather than round in shape.



A simulation profile must be active in order to place a marker for a plot window template. The analysis type defined in the profile will determine what type of template will be loaded (either for AC, DC or transient analysis). Plot window templates are defined for one analysis type only. For example, an AC simulation will show frequency domain templates such as Bode plot, while a transient analysis will show time domain templates such as risetime or pulsewidth. In addition, some predefined templates require multirun analyses (Monte Carlo analysis, time sweep, or parametric sweep).

When placing a plot window template marker, the argument description for the template being placed will appear in the status bar of Capture. Markers will continue to be placed until all arguments for the template have been satisfied. If an active simulation exists, then the template markers will turn black; otherwise, they will remain gray.

If an argument type is set to "Any" rather than a specific type, the marker type will depend on the marker placement location. If a marker is placed on a pin, then it will be assumed to be a current marker. If a marker is placed on a node, it will be assumed to be a voltage marker. If a marker is placed on a device, it will be assumed to be a power marker.

To place a plot window template marker

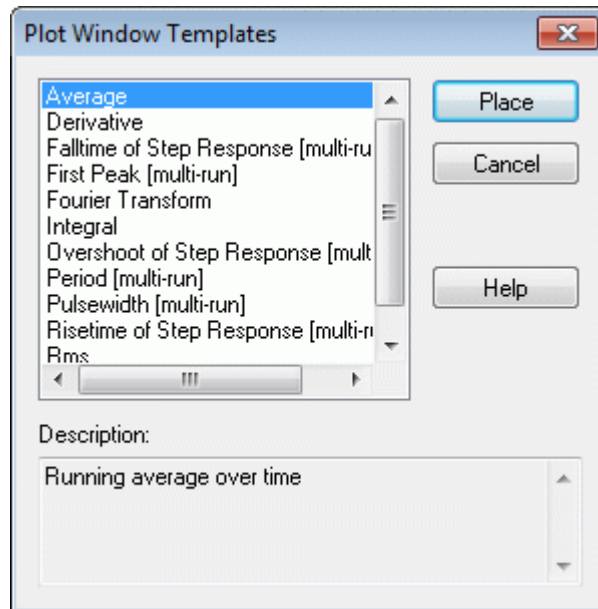
1. In Capture, from the PSpice menu, choose Markers, then select Plot Window Templates.

The Plot Window Templates dialog box appears.

## PSpice Help

### Traces

---



2. Click on the template you want to associate with the marker you will place.
3. Click Place.

A plot window template marker will appear and be attached to the cursor.

4. Place the marker at a particular location on the schematic page.
5. Continue to place markers at the appropriate locations until all the arguments for the template have been satisfied.

**Note:** PSpice does not have to be running in order for you to place a marker for a plot window template. The list of loaded templates comes from either the default PSpice .PRB file or from the .PRB file for the active profile, if that exists.

## Related Topics

For information about...	<a href="#">Click this topic...</a>
Plot window templates in PSPice ...	<a href="#">Using plot window templates</a>
Working with current, voltage or power markers...	<a href="#">Using markers</a>
Defining simulation profiles...	<a href="#">Creating a new simulation profile</a>

## Labeling plots

You can place labels to annotate an analog plot. Labels can be placed anywhere on the trace window, including outside the current Probe window where they are not visible. Unless you specifically set the plot region, PSPice rescales the plot so that all labels are visible.

You can place the following labels. Click one to display a procedure of how to use it.

text	<a href="#">To place text</a>
Line	<a href="#">To draw a line</a>
Box	<a href="#">To draw a box</a>
circle	<a href="#">To draw a circle</a>
poly-line	<a href="#">To draw a poly-line</a>
Arrow	<a href="#">To draw an arrow</a>
Ellipse	<a href="#">To draw an ellipse</a>
mark	<a href="#">To mark the cursor location</a>

## To move a label

1. Click to select a label. Shift+click to select several labels.
2. Drag the labels to a new location, holding the mouse button down on the edge of one of the labels.

3. Release the mouse button to place the labels.

#### **To delete a label**

1. Click to select a label. Shift+click to select several labels.
2. From the Edit menu, choose Delete.

#### **To place text**

1. On the toolbar, click the Text button.
2. In the Text Label dialog box, type a label in the text box. You can use up to 124 characters, including spaces.
3. Click OK.
4. Move the cursor to where you want to place the text.
5. Click to place the text.

#### **To draw a line**

1. From the Plot menu, point to Label, then choose Line.
2. Click the start point for the line.
3. Move the pointer to the end point for the line.
4. Click to set the end point and draw the line.

#### **To draw a poly-line**

1. From the Plot menu, point to Label, then choose Poly-line.
2. Click the start point for the line.
3. Move the pointer to the end point for the first segment.
4. Click to set the end point and draw the first segment.
5. Repeat steps 3 and 4 for the other segments.
6. Right-click to complete the poly-line label.

**Note:** When you create single poly-lines, PSpice changes all the line segments to single poly-line labels.

**To draw an arrow**

1. From the Plot menu, point to Label, then choose Arrow.
2. Click the start point for the arrow. The arrowhead appears on the other end.
3. Move the pointer to the end point for the line.
4. Click to set the end point and draw the arrow.

**To draw a box**

1. From the Plot menu, point to Label, then choose Box.
2. Click to set the first corner of the box.
3. Move the pointer to the other corner of the box.
4. Click to set the corner and draw the box.

**To draw a circle**

1. From the Plot menu, point to Label, then choose Circle.
2. Click to set the center of the circle.
3. Move the pointer to the outside point of the circle.
4. Click to set the radius and draw the circle.

**To draw an ellipse**

1. From the Plot menu, point to Label, then choose Ellipse.
2. Type the inclination angle and click OK.
3. Click to place the center of the ellipse.
4. Move the pointer to size and shape the ellipse.
5. Click to draw the ellipse.

**To mark the cursor location**

The Mark command places a cursor mark at the position of the most recently moved cursor.

A cursor mark consists of a text label with the coordinates of the cursor placed above and to the right of the cursor and a line label with one end anchored to the trace at the cursor and the other end placed just below the text.

If the line label is moved, the end anchored to the trace does not move, and the line stretches and rotates about the anchor point.

## Editing labels

There are several ways to edit items in PSpice :

- You can copy the current Probe window to the Windows clipboard to be pasted in another Windows application.
- Other items can be cut and pasted.
- Objects like a single trace expression, a single text label, or a single ellipse label can be edited; edit traces by changing the expression that describes the trace.
- The text of text labels or inclination angle of ellipse labels can also be edited. For example, you can change the title of the currently active Probe window.

For more information, see the following topics:

- ☐ [Copying Probe data to other applications](#)
- ☐ [To copy and paste an item](#)
- ☐ [To modify a label or an ellipse angle](#)

### To copy a Probe window to the clipboard

1. Click the tab of the Probe window you want to copy.
2. From the Window menu, choose Copy to Clipboard. The status line and the menu bar are not copied.
3. Paste the bitmap into a graphics program like Microsoft Paint.
4. Edit the bitmap as needed.
5. Do one of the following:
  - ☐ Save the bitmap to a file. Use this option if you are going to need the bitmap in several applications.
  - ☐ Copy the bitmap again and paste it in another application.

### **To copy and paste an item**

1. Click an item.
2. On the toolbar, click the Copy button.
3. On the toolbar, click the Paste button.
4. Click to place the item.

### **To modify a trace**

1. Do one of the following:
  - ☐ Click the trace name. From the Edit menu, choose Modify Object.
  - ☐ Double-click the trace name.
2. Do any of the following:
  - ☐ Select a new expression from the list.
  - ☐ Type in a new expression.
  - ☐ Modify the existing expression.
3. Click OK.

### **To modify a label or an ellipse angle**

1. Click a label or an ellipse.
2. From the Edit menu, choose Modify Object.
3. Make any changes in the dialog box.
4. Click OK.

## **Copying Probe data to other applications**

### **Copying Probe window to the Windows Clipboard and word processing applications**

To copy a Probe window to the Windows Clipboard and word processing applications like Microsoft Word, do the following:

1. Click the tab of the Probe window you want to copy.



2. From the Window menu, choose Copy to Clipboard. The Copy to Clipboard dialog box appears.
3. Select the Make window and plot backgrounds transparent check box if you want to copy the probe window and plot background with a transparent background.
4. Select the appropriate check box for the foreground color.
5. Click OK.

The Probe window is copied to the Windows Clipboard. The status line and the menu bar are not copied.

6. Paste the bitmap into a graphics program like Microsoft Paint.
7. Edit the bitmap as needed.
8. Do one of the following:
  - ☐ Save the bitmap to a file. Use this option if you are going to need the bitmap in several applications.
  - ☐ Copy the bitmap again and paste it in another application.

### **Copying Probe data to spreadsheet applications and math programs**

You can copy the X and Y-axis data for traces on the Probe window to text editors, spreadsheet applications and math programs. You can then manipulate the data for your own purposes. For example, you can define custom measurement functions in Microsoft Excel and analyze the Probe data using those functions. You can also use the probe data in spreadsheet applications to create charts or graphs for presentation purposes.

1. Click the tab of the Probe window from which you want to copy the data.
2. Click the trace name in the plot legend. To select more than one trace name, use SHIFT+click or CTRL+click.
3. From the Edit menu choose Copy.
4. Do one of the following:
  - ☐ Paste the data into a spreadsheet or math program.
  - ☐ Paste the data into a text editor and save it to an ASCII text file. The data is stored in a tab-delimited format.

Use this option if you want to import the data into other applications.

## Loading large data file

To load a large data file, you can select one of the following methods:

- Displaying fewer data points
- Displaying partial trace

### Displaying fewer data points

Instead of using all the data points in the .dat file, only a few points are used to construct the complete trace. The number of data points used to construct the complete trace depends on the number of data points per trace defined in PSpice . By default, this limit is set to 1 million points, but if required, users can increase this limit. See Setting large data file options

### Displaying partial trace

The complete trace is divided into multiple smaller parts. Only a part of the trace is loaded and displayed. Number of partial traces created depends on total number of data points used to define the trace and also on the number of data points per trace allowed in PSpice . By default, the number of data points per trace is set to 1 million points, but if required, users can increase this limit. See Setting options for large data files.

## Importing traces

PSpice now allows you to import the traces stored in tabular format in a text (.txt) or comma-separated (.csv) file. Using the import feature you can import waveforms generated by measuring instruments such as digital oscilloscope to PSpice . Import feature can also be used to import waveforms that can be appended to existing data file for comparing two or more traces.

**Note:** To be able to append a trace to an existing data file, the X-axis name and range should match.

To import a trace into PSpice , saved in a text (.txt) or a comma-separated (.csv) file complete the following steps.

1. From the File menu, choose Import.
2. In the Import File dialog box, select the text file to be imported in PSpice .

The Import Traces dialog box appears. All the nodes listed in the source file are listed in the X-axis drop-down list and the Available Nodes list.

3. In the Import Traces dialog box, specify the name and the location of the .DAT file in which the imported trace is to be stored.
4. From the X-Axis drop-down list box, select the node name to be plotted on the X-Axis.
5. Specify a name for the X-axis. Select one of the following options for naming the X-axis.
  - a. Time
  - b. Frequency
  - c. Sweep Variable: when you select Sweep Variable, you need to specify the variable name in the enabled textbox.
6. From the Available Traces list box, select the traces that are to be imported in PSpice .
7. Click Add.

The selected traces appear in the Import Trace list box. To import all the traces available in the source file, click the Add All.

8. Select OK to import the selected trace(s).

## Import Traces

This dialog box appears when you try to import a traces saved in the text format or in a .csv file.

DAT File Path	Specify the name and location of the data file in which the imported trace will be saved.
X-Axis	Select the trace that is to be plotted along the X-Axis.  The X-Axis drop-down list box lists all the traces available in the .txt or the .csv file.
Specify a name for the X-Axis using one of the following options.	
Time	Select this, if you want to rename the X-axis as Time.  This might be helpful when you want to append the imported trace to some other data file for comparing results.
Frequency	Select this, if you want to rename the X-axis as Frequency.
Sweep Variables	Select this, if you want to specify a user-defined name for the X-axis. Specify the name in the enabled text box.

## PSpice Help

### Traces

---

Available Traces	Lists all the traces available in the .txt or .csv file.
Imported Traces	Lists all the traces that will be imported in the .dat file.
Add	Click this to add the selected trace to the Import Trace list.
Add All	Click this to add all the available traces into the Import Traces list.  <b>Note:</b> The trace selected as X-Axis, will not appear in the Import Traces list box.
Remove	Click this to remove the selected traces from the Import Trace list.
Remove All	Click this to empty the Import Traces list

---

# Using performance analysis and measurements

---

## Using Performance Analysis

Performance Analysis allows you to add traces that show how a derived value changes between simulation runs. The derived value is calculated for each run based on the measurement expression you specify when you add each trace.

To prepare for using Performance Analysis

1. Have multiple sections of data in the Probe data file. You can also append data files together in PSpice , which will treat them as separate sections. Enabling a parametric analysis, a temperature analysis with multiple temperatures, or a Monte Carlo analysis causes the simulator to do multiple simulations of the circuit. Each simulation creates one section of data.
2. Create measurement definitions you want to use and bring them into PSpice . Do one of the following:
  - a. Use the pre-defined measurement definitions in the file PSpice .PRB.
  - b. Create measurement definitions in a .PRB file, using an ASCII text editor like Notepad. Save the file with a .PRB extension. Load the file into PSpice as a global file.
  - c. Create new or edit existing measurement definitions while running Probe: from the Trace menu, select Measurements, then select New or Edit.

## Using Measurement Expressions

Measurement expressions evaluate the characteristics of a waveform. A measurement expression is made by choosing the waveform and the waveform calculation you want to evaluate.

The waveform calculation is defined by a measurement definition such as rise time, bandpass bandwidth, minimum value, and maximum value.

For example, if you want to measure the risetime of your circuit output voltage, use the following expression:

```
Risetime_NoOvershoot(v(out))
```

### Measurement strategy

- Start with a circuit created in Capture and a working PSpice simulation.
- Decide what you want to measure.
- Select the measurement definition that matches the waveform characteristics you want to measure.
- Insert the output variable (whose waveform you want to measure) into the measurement definition, to form a measurement expression.
- Test the measurement expression.

## Composing Measurement Expressions

These steps show you how to create a measurement expression in PSpice . Measurement expressions created in PSpice can be imported into Advanced Analysis.

1. Work in the Simulation Results view in PSpice . In the side toolbar, click on  .

2. From the Trace menu in PSpice , select Measurements.

The Measurements dialog box appears.

3. Select the measurement definition you want to evaluate.

4. Click Eval (evaluate).

The Arguments for Measurement Evaluation dialog box appears.

5. Click the Name of trace to search button.

The Traces for Measurement Arguments dialog box appears.

**Note:** You will only be using the Simulation Output Variables list on the left side. Ignore the Functions or Macros list.

6. Uncheck the output types you don't need (if you want to simplify the list).

7. Click on the output variable you want to evaluate.

The output variable appears in the Trace Expression field.

8. Click OK.

The Arguments for Measurement Evaluation dialog box reappears with the output variable you chose in the Name of trace to search field.

9. Click OK.

Your new measurement expression is evaluated and displayed graphically in the PSpice window.

10. Click OK in the Display Measurement Evaluation pop-up box to continue working in PSpice .

Your new measurement expression is saved, but it no longer displays in the window. The only way to get another graphical display is to redo these steps.

You can see the numerical evaluation of your measurement expression by following the steps in the topic [Viewing Measurement Results](#).

See the topic [Measurement Expression Example](#) for an example.

## Measurement Expression Example

These steps show you an example of creating a measurement expression in PSpice .

Work in the Simulation Results view in PSpice . In the side toolbar, click on .

1. Choose Trace - Measurements.

The Measurements dialog box appears.

2. Select the measurement definition you want to evaluate.

3. Click Eval (evaluate).

The Arguments for Measurement Evaluation dialog box appears.

4. Click the Name of trace to search button.

The Traces for Measurement Arguments dialog box appears.

**Note:** You will only be using the Simulation Output Variables list on the left side. Ignore the Functions or Macros list.

5. Uncheck the output types you don't need (if you want to simplify the list).
6. Click on the output variable you want to evaluate.

The output variable appears in the Trace Expression field.

7. Click OK.

The Arguments for Measurement Evaluation dialog box reappears with the output variable you chose in the Name of trace to search field.

8. Click OK.

Your new measurement expression is evaluated and displayed graphically in the PSpice window.

9. Click OK in the Display Measurement Evaluation pop-up box to continue working in PSpice .

Your new measurement expression is saved, but it no longer displays in the window. The only way to get another graphical display is to redo these steps.

You can see the numerical evaluation of your measurement expression by following the steps in the topic [Viewing Measurement Results](#).

## Viewing Measurement Results

To view the results of measurement expressions you have previously composed:

1. From the View menu in PSpice , select Measurement Results.

The Measurement Results table displays below the plot window.

2. Click the box in the Evaluate column.

The PSpice calculation for your measurement expression appears in the Value cloumn.

See the topic [Measurement Results Example](#) for an example.

## Evaluating a measurement

After simulating a circuit, you can add new measurements in the Measurement Results window, and evaluate them. To add a measurement for evaluation:

1. From the *Trace* menu in PSpice, choose *Evaluate Measurement*.

The *Evaluate Measurement* dialog box displays, with *Measurements* selected in the *Functions or Macros* drop-down list box.

**Note:** Alternatively, the Evaluate Measurement dialog box can be invoked using the



*Evaluate Measurement* icon from the toolbar.

1. Select the measurement that you want to evaluate by clicking one of the entries in the *Measurements* list.
2. From the *Simulation Output Variables* list, click the variable to be passed as a parameter to the selected measurement, and click *OK*.

The new measurement gets added in the *Measurement Results* table displayed below the plot window.

The PSpice calculation for your measurement expression appears in the *Value* column.

**Note:** To view information about measurements provided by PSpice, see [Measurement Definitions Included with PSpice](#).

To view information on how to create new measurements, see [Composing Measurement Expressions](#).

## Measurement Results Example

1. Choose View - Measurement Results.

The Measurement Results table displays below the plot window.

2. Click the box in the Evaluate column.

A checkmark appears in the Evaluate column check box and the PSpice calculation for your measurement expression appears in the Value column.

## Measurement Definitions Included with PSpice

Definition	Finds the. . .
Bandwidth	Bandwidth of a waveform (you choose dB level)
Bandwidth_Bandpass_3dB	Bandwidth (3dB level) of a waveform
Bandwidth_Bandpass_3dB_XRange	Bandwidth (3dB level) of a waveform over a specified X-range
CenterFrequency	Center frequency (dB level) of a waveform
CenterFrequency_XRange	Center frequency (dB level) of a waveform over a specified X-range
ConversionGain	Ratio of the maximum value of the first waveform to the maximum value of the second waveform
ConversionGain_XRange	Ratio of the maximum value of the first waveform to the maximum value of the second waveform over a specified X-range
Cutoff_Highpass_3dB	High pass bandwidth (for the given dB level)
Cutoff_Highpass_3dB_XRange	High pass bandwidth (for the given dB level)
Cutoff_Lowpass_3dB	Low pass bandwidth (for the given dB level)
Cutoff_Lowpass_3dB_XRange	Low pass bandwidth (for the given dB level) over a specified range
DutyCycle	Duty cycle of the first pulse/period
DutyCycle_XRange	Duty cycle of the first pulse/period over a range
Falltime_NoOvershoot	Falltime with no overshoot.
Falltime_StepResponse	Falltime of a negative-going step response curve
Falltime_StepResponse_XRange	Falltime of a negative-going step response curve over a specified range
GainMargin	Gain (dB level) at the first 180-degree out-of-phase mark
Max	Maximum value of the waveform
Max_XRange	Maximum value of the waveform within the specified range of X

## PSPice Help

### Using performance analysis and measurements

---

Min	Minimum value of the waveform
Min_XRange	Minimum value of the waveform within the specified range of X
NthPeak	Value of a waveform at its nth peak
Overshoot	Overshoot of a step response curve
Overshoot_XRange	Overshoot of a step response curve over a specified range
Peak	Value of a waveform at its nth peak
Period	Period of a time domain signal
Period_XRange	Period of a time domain signal over a specified range
PhaseMargin	Phase margin
PowerDissipation_mW	Total power dissipation in milli-watts during the final period of time (can be used to calculate total power dissipation, if the first waveform is the integral of V(load))
Pulsewidth	Width of the first pulse
Pulsewidth_XRange	Width of the first pulse at a specified range
Q_Bandpass	Calculates Q (center frequency / bandwidth) of a bandpass response at the specified dB point
Q_Bandpass_XRange	Calculates Q (center frequency / bandwidth) of a bandpass response at the specified dB point and the specified range
Risetime_NoOvershoot	Risetime of a step response curve with no overshoot
Risetime_StepResponse	Risetime of a step response curve
Risetime_StepResponse_XRange	Risetime of a step response curve at a specified range
SettlingTime	Time from <begin_x> to the time it takes a step response to settle within a specified band
SettlingTime_XRange	Time from <begin_x> to the time it takes a step response to settle within a specified band and within a specified range
SlewRate_Fall	Slew rate of a negative-going step response curve
SlewRate_Fall_XRange	Slew rate of a negative-going step response curve over an X-range
SlewRate_Rise	Slew rate of a positive-going step response curve

## PSpice Help

### Using performance analysis and measurements

---

SlewRate_Rise_XRange	Slew rate of a positive-going step response curve over an X-range
Swing_XRange	Difference between the maximum and minimum values of the waveform within the specified range
XatNthY	Value of X corresponding to the nth occurrence of the given Y_value, for the specified waveform
XatNthY_NegativeSlope	Value of X corresponding to the nth negative slope crossing of the given Y_value, for the specified waveform
XatNthY_PercentYRange	Value of X corresponding to the nth occurrence of the waveform crossing the given percentage of its full Y-axis range; specifically, nth occurrence of $Y = Y_{min} + (Y_{max} - Y_{min}) * Y_{pct} / 100$
XatNthY_PositiveSlope	Value of X corresponding to the nth positive slope crossing of the given Y_value, for the specified waveform
YatFirstX	Value of the waveform at the beginning of the X_value range
YatLastX	Value of the waveform at the end of the X_value range
YatX	Value of the waveform at the given X_value
YatX_PercentXRange	Value of the waveform at the given percentage of the X-axis range
ZeroCross	X-value where the Y-value first crosses zero
ZeroCross_XRange	X-value where the Y-value first crosses zero at the specified range

## Creating Custom Measurement Definitions

Measurement definitions establish rules to locate interesting points and compute values for a waveform. In order to do this, a measurement definition needs:

- A measurement definition name
- A marked point expression

These are the calculations that computer the final point on the waveform.

- One or more search commands

These commands specify how to search for the interesting points.

## Strategy

1. Decide what you want to measure.
2. Examine the waveforms you have and choose which points on the waveform are needed to calculate the measured value.
3. Compose the search commands to find and mark the desired points.
4. Use the marked points in the Marked Point Expressions to calculate the final value for the waveforms.
5. Test the search commands and measurements.

**Note:** An easy way to create a new definition:

Choose Trace- Measurements to open the Measurements dialog box, then:

- Select the definition most similar to your needs
- Click Copy and follow the prompts to rename and edit.

## Composing a New Measurement Definition

These steps show you how to create a measurement definition in PSpice . Measurement definitions can be directly evaluated or used as part of measurement expressions.

Work in the Simulation Results view in PSpice . In the side toolbar, click on .

1. From the PSpice Trace menu, choose Measurements.

The Measurements dialog box appears.

2. Click New to start with the basic template definition, or click Copy to start your new measurement definition using an existing definition.

The New Measurement dialog box or the Copy Measurement dialog box appears.

3. Type a name for the new measurement in the New Measurement name field. All measurement definitions must have unique names.

4. Make sure local file is selected.

This stores the new measurement in a .PRB file local to the design.

5. Click OK.

The Edit New Measurement dialog box appears. If you have copied a measurement definition, you will need to click Edit on the Measurements dialog box.

6. Type in the marked point expression.
7. Type in any comments you want.
8. Type in the search command function.

Your new measurement definition is now listed in the Measurements dialog box.

See the topic Measurement Definition Example for an example.

## Managing Measurements

You can use this dialog to evaluate measurement expressions or to create, edit, or delete existing measurements definitions.

- New allows you to compose a new measurement definition.
- Copy allows you to copy an existing measurement definition. This is a good way to start when you want to create a new measurement definition.
- View allows you to see the marked point expression, search function, and comments of the selected measurement definition.
- Edit allows you to modify the selected measurement definition.
- Delete allows you to remove the selected measurement definition. Once it has been deleted, you cannot undelete it.
- Eval allows you to evaluate a measurement expression using the selected measurement definition.
- Load allows you to load a different .PRB file with different measurement definitions. The load command on this Measurements dialog box will only load the measurement definitions from the .PRB file. Macros and displays will not be included.

## Copying a Measurement Definition

Copying a measurement definition allows you to modify an existing measurement definition without losing the original definition.

Work in the Simulation Results view in PSpice . In the side toolbar, click on



1. Choose Trace - Measurements.

The Measurements dialog box appears.

2. Select a measurement from the list and click Copy.

The Copy Measurement dialog box appears.

3. Type a name for the new measurement definition in the New Measurement name field. All measurement definitions must have unique names.

4. Make sure local file is selected.

This stores the new measurement in a .PRB file local to the design.

5. Click OK.

To edit your new measurement definition, click Edit on the Measurements dialog box. See [Editing a Measurement Definition](#) for more information.

## Editing a Measurement Definition

Editing a measurement definition allows you to change the result it will evaluate.

Work in the Simulation Results view in PSpice . In the side toolbar, click on .

1. Choose Trace - Measurements.

The Measurements dialog box appears.

2. Select a measurement from the list and click Edit.

The Edit Measurement dialog box appears.

3. Edit the marked point expression as needed.

4. Edit the comments as needed.

5. Edit the search command function as needed.

## Measurement Definition Example

1. From the PSpice Trace menu, choose Measurements.

The Measurements dialog box appears.

2. Click New.

The New Measurement dialog box appears.

3. Type a name for the new measurement in the New Measurement name field.

4. Make sure local file is selected.

This stores the new measurement in a .prb file local to the design.

5. Click OK.

The Edit New Measurement dialog box appears.

6. Type in the marked point expression.

```
Point707(1) = y1
```

7. Type in any comments you want.

```
*  
#Desc# Find the .707 value of the trace.  
*  
#Arg1# Name of trace to search  
*
```

8. Type in the search function.

```
{  
1|Search forward level(70.7%, p) !1;  
}
```

**Note:** The search function is enclosed within curly braces.

Always place a semi-colon at the end of the last search function.

You now have your edited measurement definition.

9. Click OK to save your edits.

Your new measurement definition is now listed in the Measurements dialog box.

## Measurement Definition Syntax

Measurement definitions have the following structure and syntax:

```
Measurement_name (1, [2, ..., n][, subarg1, subarg2, ..., subargm]) =  
    Marked_point_expression  
  
{  
1| search_commands_and_marked_points_for_expression_1;
```



## PSpice Help

### Using performance analysis and measurements

---

```
2| search_commands_and_marked_points_for_expression_2;  
  
n| search_commands_and_marked_points_for_expression_n;  
}
```

For examples of syntax, check out the existing measurement definitions in PSpice .

1. From the Trace menu in PSpice , choose Measurements.

The Measurement dialog box appears.

2. Highlight an existing measurement definition, for example Risetime\_NoOvershoot, and select View to examine the syntax.

The View Measurement dialog box appears.

The name of the measurement is Risetime\_NoOvershoot. Risetime\_NoOvershoot takes 1 argument, a trace name (as seen from the comments).

The first search function searches forward (positive x direction) from the point on the trace where the waveform crosses the 10% point in a positive direction. That point's X and Y coordinates will be marked and saved as point 1.

The second search function searches forward in the positive direction for the point on the trace where the waveform crosses the 90% mark. That point's X and Y coordinates will be marked and saved as point 2.

The marked point expression is  $x_2 - x_1$ . This means the measurement calculates the X value of point 2 minus the X value of point 1 and returns that number.

## Measurement Name Syntax

Names can contain any alphanumeric character (A-Z, 0-9) or an underscore `_`, up to 50 characters in length. The first character should be an upper or lower case letter.

Examples of valid function names: Bandwidth, CenterFreq, delay\_time, DBLevel1.

## Marked Point Expression Syntax

A marked point expression calculates a single value, which is the value of the measurement, based on the X and Y coordinates of one or more marked points on a curve. The marked points are found by the search command.

- All the arithmetic operators (+, -, \*, /, () ) and all the functions that apply to a single point (for example, ABS(), SGN(), SIN(), SQRT() ) can be used in marked point expressions.
- The result of the expression is one number (a real value).
- Marked point expression differ from a regular expression in the following ways:
- Marked point coordinate values (for example, x1, y3), are used instead of simulation output variables (v(4), ic(Q1) ).
- Multiple-point functions such as d(), s(), AVG(), RMS(), MIN(), and MAX() cannot be used.
- Complex functions such as M(), P(), R(), IMG(), and G() cannot be used.
- One additional function called MPAVG can also be used. It is used to find the average Y value between 2 marked points. The format is:

```
MPAVG(p1, p2, [<.fraction>])
```

where p1 and p2 are marked X points and fraction (expressed in decimal form) specifies the range. The range specified by [<.fraction>] is centered on the midpoint of the total range. The default value is 1.

An example:

The marked point expression

```
MPAVG(x1, x5, .2)
```

Will find the halfway point between x1 and x5 and will calculate the average Y value based on the 20 percent of the range that is centered on the halfway point.

## Comments Syntax

A comment line always starts with an asterisk. Special comment lines include the following examples:

```
*#Desc#*   The measurement description.
```

```
*#Arg1#*   The first argument used in the measurement definition.
```

These comment lines will be used in dialog boxes, such as the Arguments for Measurement Evaluation box.

## Search Command Syntax

Search commands can be used to find a particular point along a trace based on characteristics of that trace. Search commands can be used for two purposes:

- To position the cursor at a specific place along a trace.
- As part of a measurement definition<definition>, where the point found by the search can be used as a marked point.

```
Search      [direction]      [/start_point/]      [#consecutive_points#]
            [(range_x[,range_y])]
[for]                [repeat:]                <condition>
```

Brackets indicate optional arguments.

You can use uppercase or lowercase characters, because searches are case independent.

[direction]

The direction of the search. The search begins at the origin of the curve.

forward	Search forward on the trace
backward	Search backwards on the trace

[Forward] searches in the normal X expression direction, which may appear as backwards on the plot if the X axis has been reversed with a user-defined range.

Forward is the default direction.

[/start\_point/]

The starting point to begin a search. The current point is the default.

Use this	To start the search at this...
^	The first point in the search range
Begin	The first point in the search range
\$	The last point in the search range
End	The last point in the search range
xn	A marked point number or an expression of marked points.

For example:

x1

$(x1 - (x2 - x1) / 2)$

[#consecutive\_points#]

Defines the number of consecutive points required for a condition to be met. Usage varies for individual conditions; the default is 1.

A peak is a data point with one neighboring data point on both sides that has a lower Y value than the data point.

If [#consecutive\_point#] is 2 and <condition> is PEak, then the peak searched for is a data point with two neighboring data points on both sides with lower Y values than the marked data point.

[(range\_x[,range\_y])]

Specifies the range of values to confine the search.

The range can be specified as floating-point values, as a percent of the full range, as marked points, or as an expression of marked points. The default range is all points available.

Some examples:

This range...	Means this...
(1n,200n)	X range limited from 1e-9 to 200e-9, Y range defaults to full range
(1.5,20e-9,0,1m)	Both X and Y ranges are limited
(5m,1,10%,90%)	Both X and Y ranges are limited
(0%,100%,1,3)	Full X range, limited Y range
(,,1,3)	Full X range, limited Y range
(,30n)	X range limited only on upper end

[for] [repeat:] <condition>

Specifies which occurrence of <condition> to find.

If repeat is greater than the number of found instances of <condition>, then the last <condition> found is used.

For example, the argument:

`2:LEvel`

would find the second level crossing.

The `<condition>` must be exactly one of the following:

- `LEvel(value[,posneg])`
- `SLope[(posneg)]`
- `PEak`
- `TRough`
- `MAx`
- `MIn`
- `POint`
- `XValue(value)`

Each `<condition>` requires just the first 2 characters of the word. For example, you can shorten `LEvel` to `LE`.

If a `<condition>` is not found, then either the cursor is not moved or the measurement definition is not evaluated.

`LEvel(value[,posneg])`

`<value>` can take any of the following forms:

Value Form	Example
A floating number	1e5 100n 1
A percentage of full range	50%
A marked point Or an expression of marked points	X1 Y1 (x1-x2)/2
A value relative to startvalue	.-3 (startvalue -3) .+3 (startvalue +3)

## PSpice Help

Using performance analysis and measurements

## **PEak**

Finds the nearest peak. At least [#consecutive\_points#] points on each side of the peak must have Y values less than the peak Y value.

## **TRough**

Finds nearest negative peak. At least [#consecutive\_points#] points on each side of the trough must have Y values greater than the trough Y value.

## **MAx**

Finds the greatest Y value for all points in the specified X range. If more than one maximum exists (same Y values), then the nearest one is found.

MAx is not affected by [direction], [#consecutive\_points#], or [repeat:].

## **MIn**

Finds the minimum Y value for all points in the specified X range.

MIn is not affected by [direction], [#consecutive\_points#], or [repeat:].

## **POint**

Finds the next data point in the given direction.

## **XValue(value)**

Finds the first point on the curve that has the specified X axis value.

The <(value)> is a floating-point value or percent of full range

XValue is not affected by [direction], [#consecutive\_points#], [(range\_x [,range\_y])], or [repeat:].

<(value)> can take any of the following forms:

Value Form	Example
A floating number	1e5 100n 1

## PSpice Help

### Using performance analysis and measurements

---

A percentage of full range	50%
A marked point Or an expression of marked points	X1 Y1 (x1-x2)/2
A value relative to startvalue	.-3 (startvalue -3) .+3 (startvalue +3)
A db value relative to startvalue	.-3db (3db below startvalue) .+3db (3db above startvalue)
A value relative to max or min	Max-3 (maxrng -3) Min+3 (minrng +3)
A db value relative to max or min	Max-3db (3db below maxrng) Min+3db (3db above maxrng)

### ForceDBArg1

Converts a non-DB trace to a dB trace. Used in measurement functions such as Bandwidth\_Bandpass\_3dB and Cutoff\_Highpass\_3dB.

## Limiting a Measurement Expression to a Specific Range of Data

You can restrict the data that measurement expressions apply to.

To limit the data

1. From the Plot menu, select Axis Settings.
2. Click the X Axis tab
3. Select User Defined in the Data Range frame and enter the desired start and finish values.
4. Click OK.

All measurement expressions used are applied to the data range you specified.



## **Control Elements in Dialog Boxes**

### **Help**

Click to display Help.

### **Close**

Click to close the measurement dialog box.

### **Measurement Function Expression**

Type the measurement function argument.

### **Name of Trace to Search Frame**

Type any other measurement arguments needed to complete the expression.

### **Box**

Type your changes in this box.

### **This Measurement Function is Saved in the File**

The location of the measurement function on your system.

### **New Measurement Function Name**

Type the new name of the copied measurement function.

### **File to Keep Measurement Function in**

Click to select the location of the copied measurement function.

### **List**

The available sections are listed. Click to select.

## **All**

Click to select all of the listed sections.

## **None**

Click to select none of the listed sections.

## **expression arguments (1, 2, ..., n )**

These are placeholders for the Probe trace expressions that search commands are applied to. The number that identifies the expression argument (e.g., 1) corresponds to the search command labeled with the same number below.

When a measurement definition is used in the Add Trace dialog box, expression arguments are replaced with regular Probe trace expressions. For example, for the measurement definition

### **GainMargin (1,2)**

You need to specify two trace expressions in the Trace Expression text box, e.g.,

```
GainMargin (Vp(Out), VdB(Out)) .
```

## **substitution arguments (subarg1, subarg2, ..., subargm)**

These are optional arguments that follow the expression arguments.

In a measurement definition, these arguments are text strings, which can have any alphanumeric character or underscore (\_) but cannot start with a number.

For example, in the measurement definition

```
bandwidth (1, db_level) = x2 -x1
```

`db_level` is the substitution argument. This measurement definition finds the `db_level` bandwidth of a signal.

When a measurement definition is used in the Add Trace dialog box, the values of these arguments are specified by replacing them with numbers, which are substituted into any of the measurement definition's search commands that contain the subargument text.

#### Example

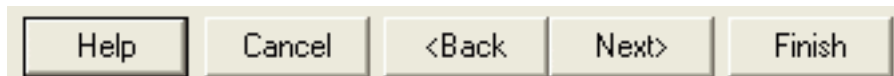
```
bandwidth (VdB(out),3)
```

## Introducing the Performance Analysis Wizard

The Performance Analysis wizard helps you complete the information necessary to generate a Performance Analysis trace. Just type or select the information as you go.

To start the Performance Analysis wizard, choose *Performance Analysis* from the *Trace* menu.

While you are in the wizard, a button bar is displayed at the bottom of every screen.



The following table describes the buttons.

Cancel	Cancel the Wizard and return to PSpice .
Back	Go back one Wizard screen.
Next	Go forward one Wizard screen.
Finish	Leave the Wizard and see the Performance Analysis.
Help	Click to go to the Wizard help system.

The Performance Analysis wizard has four steps:

1. Introducing the Performance Analysis wizard  
Contains information about the wizard process
2. Selecting a measurement  
Helps you select from the available, defined, measurements.

### 3. Selecting measurement arguments

Allows you to specify the arguments for the measurement.

### 4. Testing the measurement

Shows you the results of your measurement with the current arguments. You can redefine the arguments if the results are not what you expected.

When you are done with the Performance Analysis wizard, your trace is shown in Performance Analysis mode.

The Performance Analysis wizard creates simple Performance Analysis traces. If you need a Performance Analysis trace that is an expression of several measurements, you can add that trace manually.

## Selecting a Measurement

The second step of the wizard helps you select a measurement definition.

To create a Performance Analysis trace, select the measurement definition you want. This measurement definition determines the characteristic that is extracted from each PSpice run and is plotted versus the variable that changes between simulation runs.

To find more information about each measurement definition listed, click a name in the list on the left side of the dialog box. If included in the .PRB file, an explanation about the selected measurement definition appears on the right side of the dialog box.

If the measurement definition you want is not listed, you can edit an existing measurement definition or create a new one to meet your needs.

To edit or create a new measurement definition, click the *Measurements* button.

## Selecting Measurement Arguments

The third step of the wizard helps you fill in the measurement expression arguments.

Measurement expressions can have 2 types of arguments:

#### ■ Expression arguments

Every measurement expression must have at least one expression argument, but can have more.

■ Substitution arguments

A measurement expression can have 0 or more substitution arguments.

To specify the Measurement Expression argument

1. Do one of the following:

- ☐ Click the toolbar button{bmc BM18.SHG}. To change the listed traces, check or uncheck the output types. To select a trace, double-click a name in the Simulation Output Variables list. The name is copied to the appropriate argument box in the wizard.

or

- ☐ In the Name of Trace to Search box, type the name of the trace you want to evaluate.
2. Depending on the measurement definition selected in the previous step, type the rest of the information in the boxes, or click the toolbar buttons to select the information.
3. When you are done, click *Next*.

## Testing the Measurement

The final step of the wizard views the plotted measurement. If the value of this measurement is not correct, or if the marked points are not on the right places on the trace, do one of the following:

- ☐ Return to Step 2 and select or create a more appropriate measurement definition for your data

or

- ☐ Return to Step 3 and change the measurement expression arguments. You might have chosen the wrong expression argument or arguments.



---

# Setting Options

---

## Setting Probe window options

Use the Probe Options dialog box to specify how PSpice displays Probe window items such as scroll bars, symbols, and trace colors. You can also specify the auto-update interval, number of histogram divisions, and how large data files should be loaded onto PSpice .

For more Probe window options, see Probe windows settings for simulation profiles.

To set Probe window options

1. From the Tools menu, select Options to display the General tab of the Probe Options dialog box.
2. In the Use Symbols frame, choose one of the following options:
  - ☐ Attributes
  - ☐ Never
  - ☐ Always
3. In the Trace Color Scheme frame, choose one of the following options:
  - ☐ Normal
  - ☐ Match Axis
  - ☐ Sequential Per Axis
  - ☐ Unique By File
4. In the Use ScrollBars frame, choose one of the following options:
  - ☐ Auto
  - ☐ Never
  - ☐ Always
5. In the Auto-Update Interval frame, choose one of the following options:
  - ☐ Auto

- ☐ Every sec
  - ☐ Every %
6. In the Number of Histogram Divisions text box, enter the number of histogram divisions used when PSpice displays Performance Analysis of a Monte Carlo run.
  7. In the Number of Cursor Digits text box, enter the number of digits PSpice displays when the cursor position is shown.
  8. Select any or all of the following options:
    - ☐ Mark Data Points
    - ☐ Display Evaluation
    - ☐ Display Statistics
    - ☐ Highlight Error States
  9. Click OK to close the Probe Options dialog box.

## Setting large data file options

1. From the Tools menu, select Options to display the Probe Options dialog box.
2. Select the Large Data tab.
3. Select one of the Options to be used as the default setting for loading a large data file.
  - a. Select Use fewer data points to display a complete trace, when you want that full trace, which may not be accurate, to be loaded and displayed on the Probe window. In this case the trace is not accurate because it is created using a subset of data points picked randomly from the data file.
  - b. Select Use all data points to display trace in parts, when you want the trace to be accurate but only a small part of the trace is displayed at one time. In this case, the complete trace is divided into multiple parts and each part can be viewed separately.
  - c. Select Always ask, when you want the Large Data File dialog box to be displayed every time a large data file is opened in PSpice . By default, this option is selected.
  - d. Select Ignore this Warning, when you do not want to be notified whether the data file being loaded is a large data file or not. Therefore, the Large Data File dialog box never appears. In this case, either the Out of memory error appears, or if the system memory is sufficient the data file will be loaded successfully.



4. In the Data points in one part text box, specify the threshold limit for the number of data points in a large data file. By default this value is set to 1 million (1000000).

For example, if you want that any data file that has more than 2 million data points per trace should be categorized at a large data file then enter 2000000 in the Data points in one part text box.

5. Click OK to save the settings and to close the Probe Setting dialog box.

## Selecting a Printer

You can specify a different printer and settings for PSpice to use to print your plots. (If you do not change the settings, PSpice uses the default Windows printer settings.)

To select a printer

1. From the File menu, select Printer Setup.
2. In the Printer frame, select the printer you want to use. Click Properties to change its settings.
3. In the Paper frame, select the type of paper you want to use.
4. In the Orientation frame, choose which page orientation you want to use.
5. When you have finished selecting the printer, click OK.

## Using Print Preview

You can use Print Preview to view pages before you print them.

To use print preview

1. From the File menu, select Print Preview to preview all plots, using the current settings. Each plot is displayed as a separate page, unless you have set it differently in Page Setup.
2. In the Preview screen, click Print to print the page, or click Close to return to the Probe window.

## Setting up the Page

You can set up the page in PSpice to establish the way the printed plots appear when you print.

To set up the page

1. From the File menu, select Page Setup.
2. In the Page Setup dialog box, enter the following options:
  - ☐ In the Margins frame, type the margin spacing.
  - ☐ In the Plots Per Page frame, enter the number of plots to be printed on each page, from 1 to 9 per page.
  - ☐ In the Orientation frame, choose Landscape or Portrait.
  - ☐ Select the Draw Border check box to put a border on the page.
  - ☐ Select the Draw Plot Title box to put the plot title on the page.
  - ☐ In the Cursor Information area, choose where you want the cursor information to be displayed on the page.
3. To add a header or a footer, click the Header button or the Footer button.
  - ☐ The Left Side, Center, and Right Side boxes display the header or footer content that is left-aligned, centered, and right-aligned, respectively. Type information into the appropriate box. You can use codes for certain kinds of information.
  - ☐ Press Ctrl+Enter to start a new line.
  - ☐ Click OK to exit this dialog box and save the new settings.
4. Click OK to close the Page Setup dialog box and save the new settings.

## Printing in PSpice

To print the current plot

1. On the toolbar, click the Print button to print one copy of the current Probe window, using the current settings.

To print specific plots

1. From the File menu, select Print.
2. Set up the following options:
  - ☐ Under Printer, select which printer to use. Click Properties to set the printer's properties.
  - ☐ In the Copies box, enter how many copies to print.

- ☐ Click Page Setup to change the page settings.
  - ☐ Under Plots to Print, select which plots to print. Each plot is printed as a separate page, unless you have set it differently in Page Setup.
  - ☐ Under Automatic Grid Spacing, choose either Based on Print Area or Same as Display.
3. Click OK to print using these settings, or click Cancel to exit this dialog box without printing anything.

## Setting the Width of Printed Plot Lines

You can set the width of the printed plot lines by using an ASCII text editor, such as Notepad, to edit the PSpice .INI file.

To set the plot line width

1. Using a text editor, open the file PSpice .INI.
2. In the [Probe] section of the file, add or modify the following entry. The format is:

```
PRINTERLINEWIDTH = <value>
```

If the above entry is not present in the .INI file, the default line width is 0.

The line width of a printed plot is scaled automatically, so you do not need to set custom values for the PRINTERLINEWIDTH command in the PSpice .INI file. This makes the printout more legible for most printer configurations using the default value in the .INI file.

You can select all the traces in a plot to change their properties collectively. To do this, use the Select All command from the Edit menu, then edit the properties as needed. The new trace properties will then be reflected on the screen, in Print Preview, and on the printout.

## Changing the Screen Colors

You can change the colors PSpice uses to display plots on the screen by using an ASCII text editor, like Notepad, to edit the PSpice .INI file.

To edit screen colors

1. Using a text editor, open the file PSpice .INI.
2. In the [Probe Display Colors] or the [Probe Pointer Colors] section of the file, add or modify a color entry. The format is:

`<item name>=<color>`

For example, `background=red` draws the background in red.

Valid item names are:

<code>background</code>	The colors for the window background
<code>foreground</code>	The default color for items not explicitly specified
<code>trace 1 through trace 12</code>	Colors the traces with specified color when they appear on the Probe window. You can specify up to 12 colors.

**Valid color names**

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

Specify varying degrees of color by using the R, G, B value of the color. The range is from 0-255. For example, `<attribute>=255 0 0` is the same as `<attribute>=red`.

3. Set the trace colors using the following form:

`trace_x=colorname`

where `trace_x` is *trace\_1 through trace\_12* and `colorname` is a valid color name.

4. If needed, set `numtrace colors` to N, where N is between 1 and 12. The numbers 1 through 12 represent the number of trace colors displayed on the screen or printed before the color order repeats.
5. Save the file as ASCII text and close it when you have finished. The settings take effect the next time you start PSpice .

## Header and footer codes

These codes can be inserted into any of the boxes as an abbreviated method to include certain information.

This code ...	Does this...
---------------	--------------

&D	specifies the current date in the header and/or footer
&T	specifies the current time in the header and/or footer
&N	specifies where page numbering appears in print job
&A	prints simulation date from current data file
&M	prints simulation time from current data file
&I	prints simulation title from current data file
&E	prints simulation temperature from data file
&P	prints specified parameter or other value changing between data file sections

## Customizing toolbars

Toolbar settings can be saved in settings schemes.

To change the display of toolbars

To create a new toolbar

To add buttons to toolbars

To remove buttons from toolbars

To reset toolbars to their default settings

### To change the display of toolbars

1. From the Tools menu, choose Customize.
2. Click the Toolbars tab.
3. In the Toolbars list, select the toolbars you want to be displayed.
4. Select any of the following options:
  - ☐ Select Show Tooltips to enable ToolTips.
  - ☐ Select Cool Look to make the toolbar buttons appear flat.
  - ☐ Select Large Buttons to display the toolbar buttons at a larger size.

5. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

### **To create a new toolbar**

1. From the Tools menu, choose Customize.
2. Click the Toolbars tab.
3. Click New.
4. In the text box that appears, type a name for the toolbar, then click OK.

The name of the new toolbar appears in the Toolbars list.

5. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.



#### *Tip*

You can also create a new toolbar by dragging a button from the Customize dialog box to any open area on the workspace.

### **To add buttons to toolbars**

1. From the Tools menu, choose Customize.
2. Click the Commands tab.
3. In the Categories list, click a category to display related toolbar buttons in the Buttons frame.
4. Under Buttons, click a button to display a description of its function in the Description frame.
5. To add the selected button to a toolbar, drag it from the Customize dialog box to any toolbar displayed in the program window.
6. Click OK to close the dialog box.

**Note:** Changes are applied even if you don't click Apply.

### **To remove buttons from toolbars**

1. Drag the button you do not want from the toolbar to the Customize dialog box.

### **To reset toolbars to their default settings**

1. From the Tools menu, choose Customize.
2. Click the Toolbars tab.
3. Do one of the following:
  - ☐ Under Scheme, from the list, select Default.
  - ☐ Click Reset.
4. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

**Note:** Changes are applied only to the currently selected toolbar.

### **Customizing Commands**

To customize commands:

1. Choose Tools - Customize
2. Click the Commands tab
3. Select a Menu name from the Categories list

The buttons for the selected Menu item are displayed in the Buttons box. Select a button to view its description in the Description box.

4. Drag a button and place it in the toolbar

### **Customizing keyboard shortcuts**

Keyboard shortcuts can be saved in settings schemes.

To create a keyboard shortcut

To remove a keyboard shortcut

To reset all keyboard shortcuts to their default settings

#### **To create a keyboard shortcut**

1. From the Tools menu, choose Customize.

2. Click the Keyboard tab.
3. Under Select a Command, select a command to display its description in the Description frame.  
  
Any shortcuts already assigned to the selected command are displayed in the Assigned Shortcuts frame.
4. Click Create Shortcut.
5. In the dialog box that appears, enter the shortcut that you want to assign to the command, then click OK.
6. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

#### **To remove a keyboard shortcut**

1. From the Tools menu, choose Customize.
2. Click the Keyboard tab.
3. Under Select a Command, select the command whose shortcut you want to remove.  
  
A description of the command is displayed in the Description frame. Any shortcuts already assigned to the selected command are displayed in the Assigned Shortcuts frame.
4. Under Assigned Shortcuts, select the shortcut you want to remove, then click Remove.
5. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

#### **To reset all keyboard shortcuts to their default settings**

1. From the Tools menu, choose Customize.
2. Click the Keyboard tab.
3. Do one of the following:
  - ☐ Under Scheme, from the list, select Default.
  - ☐ Click Reset.



4. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

## **Settings schemes**

Toolbar and keyboard shortcut settings can be saved in schemes. This is useful for people sharing a computer; they can each save their settings to suit their own work habits.

To apply a scheme

To create a scheme

To delete a scheme

### **To apply a scheme**

1. From the Tools menu, choose Customize.
2. Under Scheme, from the list, select the name of the scheme you want to use.
3. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

### **To create a scheme**

1. From the Tools menu, choose Customize.
2. Set up your toolbar and keyboard shortcut preferences, while clicking Apply to keep the dialog box open.
3. Under Scheme, click Save As.
4. In the text box that appears, type a name for your scheme, then click OK.
5. Click OK to close the dialog box.

### **To delete a scheme**

1. From the Tools menu, choose Customize.
2. Click any of the tabs.
3. Under Scheme, select the name of the scheme you want to delete, then click Delete.
4. When asked whether you are sure you want to delete the scheme, click Yes.

5. Click OK to close the dialog box.

---

# Reference Information

---

## Using files

Opening, closing, and appending files in PSpice is similar to using files in other Windows programs.

If you open a section that is currently being created by the simulation, added traces are automatically updated.

Double-click the symbol next to a trace name in the plot window legend to view information about the trace, including which file its data originated from.

## To open a file

1. On the toolbar, click the Open File button.
2. Select the name of the file from the list. To use a file from a different directory or drive, select it from the list.
3. Click OK.
4. If the file has multiple sections with different analysis types, the Analysis Type dialog box appears.
  - ☐ Click the appropriate Analysis type. Select from AC sweep, DC sweep, or Transient.
5. If the file has multiple sections of data of the selected analysis type, the Available Sections dialog box appears. Do one of the following:
  - ☐ Click the appropriate sections.
  - ☐ To use all the listed sections, click All.
6. Click OK.
7. If any simulation-generated errors are found, a message appears. Click OK to view the error messages, or click Cancel to ignore the errors.

## To close a file

1. From the File menu, select Close.

All data files used by the currently selected plot window are closed. Any other plot windows that use these files are also closed.

### **To append a file**

When you append a file, PSpice treats all the appended files as one set of waveform data.

1. On the toolbar, click the Append File button.
2. Select the name of the file to append from the list. To select a file from a different directory or drive, click the name on the list.
3. Click OK.
4. If the file has multiple sections of data of the selected analysis type, the Available Sections dialog box appears. Do one of the following:
  - ☐ Click the appropriate sections.
  - ☐ Click the All button to use all sections.
5. Click OK.
6. If any simulation-generated errors are found, a message appears. Click OK to view the error messages, or click Cancel to ignore the errors.

### **PSpice default keyboard shortcuts**

PSpice has a set of keyboard shortcuts for frequently used commands.

Ctrl+X	cut the selected item
Ctrl+C	copy the selected item
Ctrl+V	paste the cut or copied item
Delete	delete the selected item
Ctrl+Y	add a y-axis
Shift+Ctrl+Y	delete a y-axis
Insert	add traces
Ctrl+Delete	delete all traces in the selected plot
Ctrl+U	restore the last deleted traces
Shift+Ctrl+C	turn the data cursor on or off

## PSpice Help

### Reference Information

---

Shift+Ctrl+F	freeze the cursor
Shift+Ctrl+S	search
Shift+Ctrl+X	move to the next maximum value
Shift+Ctrl+M	move to the next minimum value
Shift+Ctrl+R	move to the previous transition
Shift+Ctrl+N	move to the next transition
Shift+Ctrl+P	move to the next peak
Shift+Ctrl+T	move to the next trough
Shift+Ctrl+L	move to the next slope
Shift+Ctrl+I	move to the next point
Ctrl+A	zoom into the selected area
Ctrl+I	zoom in around a specified point
Ctrl+L	redraw the screen
Ctrl+O	open a file
Ctrl+N	create a new text file
F12	restore the last Probe window session
Ctrl+P	display the Print dialog box
F1	view online Help
Alt+F4	exit PSpice

Customizing keyboard shortcuts <link>

### Invalid node names

Certain characters are not valid for use in net names or node names because they are either reserved for special use or are not recognized, by PSpice . A list of invalid characters, which should not be used when naming wires, is given below.

@	'at' symbol
%	percent sign

## PSpice Help

### Reference Information

---

&	ampersand
*	asterisk
+	plus sign (only as initial character)
<space>	Space
?	Question mark
~	Tilde
#	Hash
^	Power of
"	"Double quotes
\\	Back slashes
!	Exclamation mark
(	Opening braces
)	Closing braces
'	'Single quotes
<	Lesser than
>	Greater than
=	Equals
[	Opening square braces
]	Closing square braces

Besides the characters listed above, you should not use the reserved word CN,- for naming parts or nets. CN is a reserved word used for canonical names.

## Limits in PSpice and Probe

The following limits apply to PSpice and Probe:

Feature	Maximum limit
analog display in Probe	400 traces, or 2 times the number of sections, whichever is greater
digital display in Probe	400 traces, or 2 times the number of sections, whichever is greater

## PSpice Help

### Reference Information

---

number of analog nodes that can be stored in a.DAT file       $2^{31}$  (~2,147K)

number of digital nodes that can be stored in a.DAT file       $2^{15}$  (~32K)

Monte Carlo analysis      10,000 runs





---

# Files and Commands

---

## Using .PRB files

.PRB files are ASCII text files where displays, measurement functions, and macros are stored. Each of these are listed in a section in the .PRB file. The section begins with a section header:

```
[Displays]
```

```
[Measurement Function]
```

```
[Macros]
```

The sections can be in any order or not appear at all.

Comment lines begin with an asterisk \*. Blank lines are ignored.



*Tip*

PSpice.PRIB is shipped with your OrCAD applications. This file contains examples of measurement functions and can be used as templates for your own measurement function definitions.

## Loading .PRB files

Certain .PRB files are automatically loaded by PSpice :

- The global .PRB file is loaded when a data file is opened. The global .PRB file is specified in the [PROBE] section of the pspice.ini file by the line:

```
PRBFILE=filename
```

- The local .PRB file is loaded after the global .PRB file. The local .PRB file has the same name as the data file that is being opened. For example, if the data file is named PROJECT.DAT, the local file is named PROJECT.PRIB.

You can load a specific .PRB file after the global and local files are loaded by using the command line option -p. For example:

PSpice .EXE -P MYFILE.PRB

You can also use .PRB files by clicking the Load button in the Display Control, Measurement Functions, or Macro dialog boxes. Only the section is loaded from the .PRB file. For example, if you click Load in the Measurement Function dialog box, only the measurement functions in the specified .PRB file are loaded.



***When you load a .PRB file that contains any displays, measurement functions, or macros with the same names as ones already loaded, the new ones replace the previous ones.***

## PRB file

A .PRB file is an ASCII file which contains three sections: one for measurement functions, one for macros, and one for display definitions. Each section begins with a header (e.g., [MEASUREMENT FUNCTIONS]).

Measurement functions are stored in a .PRB file. A single .PRB file can contain many measurement function definitions.

### Text in a .PRB file

A .PRB file can contain any number of measurement function, macro, and display definitions.

Text in a .PRB file uses the following format:

- Section headers are in brackets—for example, [MACROS].
- Comment lines begin with an asterisk \*.
- Blank lines are ignored, so you can add blank lines to improve the readability of the file.
- Lines of text can wrap to the next line. The maximum text line is 255 characters.

## Moving data to other applications

You can copy trace data to other applications in two ways: as data or as a bitmap.

Copying bitmaps to another application

Copying data to another application

## Copying trace data to another application

You can copy and save trace data to other applications.

To copy trace data to another application

1. Click the trace name in the plot legend to select the trace. (The name turns red when selected.)
2. From the Edit menu, choose Copy.
3. Open a file in another application, such as Microsoft Excel, Microsoft Word, or Notepad.
4. From the Edit menu of the other application, choose Paste to paste the data in the file as numbers and text.
5. Save the file.

## Logging commands

You can use command logging to create a file of commands that you can use at a later date. This file is played back to repeat a series of commands.

To create a command file

1. From the File menu, choose Log Commands.
2. Type the name of a new file or select an existing file. If you use an existing file, it is overwritten with the new command file.
3. Click OK.

All functional actions you make on the screen are logged to the specified file. The resulting file contains actions like opening files, setting axes, and adding traces.

4. To stop logging commands, from the File menu, choose Log Commands again.



***Cursor movements are not recorded during command logging.***

To run a command file

1. From the File menu, choose Run Commands.
2. Select the file from the list. If the file is in a different directory or drive, select the correct file or directory from the lists.

3. Click OK. to play the file immediately on your screen.

## Creating and changing macros

Macros can contain constraints, functions, and/or expressions of any or all of these. Macros are used when specifying a trace to add. Macros can refer to other macros, but recursive definitions are not allowed.

You can define, load, save, and delete macro definitions. Macros are stored in a .PRB file.

For more information, see the following topics:

- ☐ [To create a new macro](#)
- ☐ [To modify an existing macro](#)
- ☐ [To load a macro](#)
- ☐ [To delete a macro](#)
- ☐ [To delete a macro from another file](#)
- ☐ [Macro syntax and examples](#)

### To add a macro

In the Add Trace dialog box:

1. From the Functions or Macros list, select Macro.
2. From the corresponding list, select the name of the macro.
3. Fill in the arguments list, using the macro syntax format:
  - a. In the Node Names list, click the name of a digital node.
  - b. Repeat step a for all arguments needed for the macro call.

### To create a new macro

1. From the Trace menu, choose Macros.
2. In the Definition text box, type a unique name and a definition for the macro, observing the macro rules.
3. Click the Save button to save the macro.



***If the macro name already exists, clicking the Save button overwrites the existing macro of the same name.***

### **To modify an existing macro**

1. From the Trace menu, choose Macros.
2. From the list, select a macro.
3. In the Definition text box, edit the macro definition, observing the macro rules.
4. To save the modified macro, do one of the following:
  - ☐ To save with the same name, click the Save button.
  - ☐ To save to a specified PRB file, click the Save To button.

### **To load a macro**

1. From the Trace menu, choose Macros.
2. Click the Load button.
3. Select the file you want to load and click Open.

### **To delete a macro**

1. From the Trace menu, choose Macros.
2. Select a macro from the list.
3. Do one of the following:
  - ☐ Click the Delete button. The macro is immediately removed from the list.
  - ☐ If the selected macro is being used by a current trace, you are warned that it is in use. To delete the macro anyway, click OK



***If you delete a macro that is being used by a trace, the trace is deleted too.***

### To delete a macro from another file

1. From the Trace menu, choose Macros.
2. Click the Delete From button.
3. Select the remote or global .PRB file that has the macro you want to delete.
4. Click Open to delete the macro.



***If you delete a macro that is being used by a trace, the trace is deleted too.***

### Macro syntax and examples

A macro can have a maximum of 80 characters, including macro name and arguments. Macros can refer to other macros, but recursive definitions are not allowed. Arguments are enclosed in parentheses ( ) without spaces. A line beginning with an asterisk \* is a comment line. In-line comments are marked with a semicolon after the macro definition. Blank lines are ignored.

The macro format is:

```
<macro name>[(arg[,arg]*)] = <definition>
```

#### Examples

- ❑ `ADD(A,B) = A+B`
- ❑ `10X(A) = 10*A; here is an in-line comment`
- ❑ `100X(A) = 10*10X(A)`
- ❑ `PI = 3.14159`

### CSDF

When you (?) the Save Data in the CSDF Format option in the data collection options for simulation profiles, the simulation results are generated in generic ASCII text so that any computer link can handle the file transfer and so that any computer platform can process the data. The default file name extension for CSDF files is .CSD.

## Binary

Binary files are compact and provide the quickest access to the waveform data.

However, binary files can present problems if you want to manipulate the data, or when you want to transfer the data file to a different type of computer. There are several problems that you can encounter. Examples include:

- Binary formats for real numbers differ between platforms
- Link between computers cannot handle binary data adequately
- Binary data format is difficult to access or process

## Specifying default command line options

To start PSpice with a set of customized options that will be used each time a simulation is run, you can modify the initialization file (pspice.INI).

1. Open the file pspice.ini (located in the Windows directory) in any ASCII text editor, such as Notepad.
2. Under the heading [PSpice ] add the following command line with the desired switches:  

```
PSPICECMDLINE=<option1> <option2>
```
3. Save the file.

For more detailed information on defining simulation command line options in PSpice and a listing of the valid options, see the online PSpice Reference Guide.

## Configuring the pspice.INI file

The pspice.ini file is an ASCII text file that contains the initialization settings for PSpice and other applications that run with PSpice . It controls how PSpice application programs are started, and how their environment and initialization settings are defined. The pspice.ini file is created when you perform the installation process.

Under normal installations to your local hard drive, the pspice.ini file is located in the directory where the PSpice executable (pspice.exe) is installed (typically, in *<installation>\tools\pspice*). The file is divided into sections with title names enclosed in brackets.

Each section contains settings which follow the format:

<keyword>=<value>

where the <keyword> is the name of the setting and <value> defines the value of that setting.

For example, the first section is [PSPICE]. The setting:

```
LIBPATH="C:\Cadence\SPB_16.5\tools\pspice\library"
```

indicates that the LIBPATH setting will point to the directory C:\Cadence\SPB\_16.5\tools\pspice\library, where the PSpice model libraries are typically installed.

## Changing the settings

When the pspice.ini file is created (during installation), the installation assigns default values to the settings. Most of the settings can be changed through dialog boxes within PSpice or Probe. However, some settings may only be modified with a text editor.

**Note:** If you use a text editor to modify the settings in pspice.ini, be sure to close any PSpice applications that may be running. The changes you make to pspice.ini should be saved to the file. They will take effect the next time you start the programs.

You can change the default settings using any standard text editor, such as Notepad.

To change items that are configurable through the text editor, you should note that:

- The <keyword> is followed by an equals sign (=) with no spaces in-between.
- The <value> can be an integer, a string, or a quoted string, depending on the setting; start typing the <value> immediately after the equals sign.
- For settings that turn a feature on or off, the <value> should be either ON or OFF.
- You can include comments within the file, but you must begin each comment line with a semicolon (;).

## Section descriptions

The pspice.ini file consists of several sections that define the configuration settings for the various PSpice applications. The descriptions listed below provide information on the elements within each of these sections in pspice.ini. The descriptions provide details on the permissible <keyword> and <value> settings.

**Note:** Several of the configuration settings described here will not appear in pspice.ini unless the default values have been changed.



For specific information about a particular section, click the buttons next to the section titles listed below. (The sections are listed here in alphabetical order for easy reference. They are typically arranged in a different order in the actual pspice.ini file.)

[MODEL EDITOR section](#)

[MSGVIEW section](#)

[OPTIMIZER section](#)

[PACKAGE TYPES section](#)

[PART LIBS section](#)

[PROBE section](#)

[PROBE DISPLAY COLORS section](#)

[PROBE PRINTER COLORS section](#)

[PSPICE section](#)

[PSpice NETLIST section](#)

[SCHEMATICS section](#)

[SCHEMATICS BORDER section](#)

[SCHEMATICS INTERFACES section](#)

[SCHEMATICS LAYERS section](#)

[SCHEMATICS MRP LIST section](#)

[STIMULUS EDITOR DISPLAY COLORS section](#)

[STIMULUS EDITOR PRINTER COLORS section](#)

[SUBCKT SETTING section](#)

### ***MODEL EDITOR section***

The [MODEL EDITOR] section defines the configuration settings used by PSpice Model Editor. The following table describes the different settings.

Keyword	Description
---------	-------------

## PSpice Help

### Files and Commands

---

ALWAYSCREATEPART	Controls whether a part is created. Valid settings are On or Off. The default setting is ON.
ALWAYSCREATE SYMBOL	Controls whether a symbol is created. Valid settings are On or Off. The default setting is OFF.
AUTOGRAPHREFRESH	Controls whether the graph should be refreshed automatically with each change. Valid settings are On or Off. The default setting is ON.
AUTOSPLITTERRESIZING	Controls whether the splitter control should be resized automatically with each change. Valid settings are On or Off. The default setting is ON.
BASESCHSYMPATH	specifies the directory in which new PSpice Schematics symbol definitions are saved. The syntax for a typical installation is:  BASESCHSYMPATH=c:\Cadence\SPB_16.5\tools\PSpice\Library\modeled.slb.
BASESYMLIBPATH	Specifies the directory in which new PSpice model libraries are saved. The syntax for a typical installation is:  BASESYMLIBPATH=c:\Cadence\SPB_16.5\tools\PSpice\..\Capture\Library\PSpice\ModelEd.etc.
CURSORBOTTOM	Specifies the bottom position of the cursor box. The default is 0.
CURSORNDIGITS	Specifies the number of digits to be displayed for the cursor position. The default is 5.
CURSORRIGHT	Specifies the right position of the cursor box. The default is 0.
DISPLAYTOOLBAR	Controls the display of toolbars. Valid settings are On or Off. The default setting is ON.
ERRORMSGDLGLEFT	Specifies the left position of the Error Message box. The default is -1.
ERRORMSGDLGTOP	Specifies the top position of the Error Message box. The default is -1.
SCHSYMCREATIONTYPE	Controls whether the PSpice Schematics symbol is created with the same type of model as the previous symbol. Valid settings are On or Off. The default setting is OFF.
SCHSYMSAMEPATH	Controls whether the PSpice Schematics symbol is stored in the same directory path as the model definition. Valid settings are On or Off. The default setting is ON.

## PSpice Help

### Files and Commands

---

STATUSLINEON	Turns the status line on or off. Valid settings are On or Off. The default setting is ON.
SYMBOLPATHASMODEL	Controls whether the Capture symbol is stored in the same directory path as the model definition. Valid settings are On or Off. The default setting is ON.
TRANSLATOR	<p>Specifies the directory where the Schematics-to-Capture Translator executable is located. The syntax for a typical installation is:</p> <pre>TRANSLATOR=c:\Cadence\SPB_16.5\tools\Capture\s ch2cap.exe</pre>
USERSCHSYMPATH	<p>Specifies the directory in which new user-defined PSpice Schematics symbols are saved. This is normally left undefined until a new symbol is created. The syntax for a typical installation is:</p> <pre>USERSCHSYMPATH= (none)</pre>
USERSYMLIBPATH	<p>Specifies the directory in which new user-defined simulation models are saved. This is normally left undefined until a new model is created. The syntax for a typical installation is:</p> <pre>USERSYMLIBPATH= (none)</pre>

### **MSGVIEW section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [MSGVIEW] section defines the configuration settings used by Message Viewer. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
COL0WIDTH	Specifies the width of column 0. The default is -1.
COL1WIDTH	Specifies the width of column 1. The default is -1.
COL2WIDTH	Specifies the width of column 2. The default is -1.
COL3WIDTH	Specifies the width of column 3. The default is -1.

FONTITALIC	Controls the use of the italic font. Valid settings are 0 (Off) or 1 (On). The default setting is 0.
FONTNAME	Controls the font name. The default is MS Sans Serif.
FONTSIZE	Controls the font size. The default is 13.
FONTWEIGHT	Controls the font weight. The default is 400.
FONTWIDTH	Controls the font width. The default is 0.
HEIGHT	Specifies the height of the Message Viewer window. The default is 350.
LEFT	Specifies the left position of the Message Viewer window. The default is 50.
SENDERCLR	Controls the display of sender information in the Message Viewer window. Valid settings are 0 (Off) or 1 (On). The default setting is 0.
STATUSBAR	Controls the display of the status bar. Valid settings are 0 (Off) or 1 (On). The default setting is 1.
TIMESTAMPS	Controls the inclusion of time stamps (date & time) in the Message Viewer file. Valid settings are 0 (Off) or 1 (On). The default setting is 1.
TOOLBAR	Controls the display of toolbars. Valid settings are 0 (Off) or 1 (On). The default setting is 1.
TOP	Specifies the top position of the Message Viewer window. The default is 50.
WIDTH	Specifies the width of the Message Viewer window. The default is 450.

### **OPTIMIZER section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [OPTIMIZER] section defines the configuration settings used by PSpice Optimizer. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
---------	-------------

LASTFILEn                      Specifies up to four of the most recently used files. This should not be changed by the user.

### ***PACKAGE TYPES section***



***If you did not choose to install PSpice Schematics, this section will not appear in the `pspice.ini` file.***

The [PACKAGE TYPES] section lists the packages that are configured for use by PSpice Schematics. The individual entries in the listing can be edited in the form:

`<package name>=<package type>`

where `<package name>` is the name of the physical package (DIP14) and `<package type>` is the type of package (DIP).

### ***PART LIBS section***



***If you did not choose to install PSpice Schematics, this section will not appear in the `pspice.ini` file.***

The [PART LIBS] section lists the part libraries that are configured for use by PSpice Schematics. The individual entries in the listing can be edited in the form:

`LIB1=<library name> [extension]`

where `<library name>` is the name of the library file (ANALOG) and `[extension]` is the extension for the library file (.SLB, .PLB, etc). You may list multiple extensions. The .SLB extension refers to the symbol libraries used by PSpice Schematics. The .PLB extension refers to the package libraries used by PSpice Schematics for generating valid PCB layout netlists.

### ***PROBE section***

The [PROBE] section defines the configuration settings used by Probe. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
AUTOUPDATEINTERVAL	Controls the update interval for Probe waveforms. Valid settings are Auto, Seconds, or Percent. The default setting is AUTO.
AUTOUPDATEPERCENT	Specifies the auto update interval for Probe waveforms in percent, if AUTOUPDATEINTERVAL is set to Percent. The default is 10.
AUTOUPDATESECONDS	Specifies the auto update interval for Probe waveforms in seconds, if AUTOUPDATEINTERVAL is set to Seconds. The default is 10.
CURSORBOTTOM	Specifies the bottom position of the cursor box. The default is 0.
CURSORDIGITS	Specifies the number of digits to be displayed for the cursor position. The default is 5.
CURSORRIGHT	Specifies the right position of the cursor box. The default is 0.
DGTLNAMELEFTJUSTIFY	Controls whether digital names are left or right justified. Valid settings are On or Off. The default is OFF, meaning right justified.
DISPLAYEVALON	Turns on the display of traces and marked points used by Display Evaluation in the currently selected plot, on the currently selected Y axis. Valid settings are On or Off. The default is OFF.
ERRORMSGDLGLEFT	Specifies the left position of the Error Message box. The default is -1.
ERRORMSGDLGTOP	Specifies the top position of the Error Message box. The default is -1.
HISTNDIVISIONS	Specifies the number of divisions for histograms. The default is 10 divisions.
HISTSHOWSTATSON	Specifies that the histogram statistics will be displayed in the Probe window. Valid settings are On or Off. The default is ON.
MARKDATAPOINTS	Specifies whether data points will be marked on Probe traces. Valid settings are On or Off. The default setting is OFF.

## PSpice Help

### Files and Commands

---

PRBFILE	Specifies the name of a global .PRB file which contains Probe macro, display, measurement function, and plot template definitions. The syntax for a typical installation is:  PRBFILE=c:\Cadence\SPB_16.5\tools\PSpice\Common\pspice.prb
PRINTERLINEWIDTH	Specifies the width of lines drawn during printing (in pixels). The default is 1.
SCROLLBARS	Controls display of scroll bars. Valid settings are Always, Never, or Auto. The default setting is AUTO.
PRINTSAMEASDISPLAY	Controls whether the Probe printout matches the display. Valid settings are 0 (Off) or 1 (On). The default setting is 0.
TRACECOLORSCHEME	Specifies the trace coloring scheme. Valid settings are Normal, Match, or Sequential. The default setting is NORMAL.
TRACESYMBOLS	Specifies how trace symbols will appear on traces in the Probe plot window. Valid settings are Always, Never, or Auto. The default setting is AUTO.

### ***PROBE DISPLAY COLORS section***

The [PROBE DISPLAY COLORS] section defines the color settings used by Probe. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

To change the settings in the [PROBE DISPLAY COLORS] section, use the format:

```
<item name>=<color>
```

where <item name> specifies the Probe item and <color> specifies the color.

For example, the entry FOREGROUND=DARKGREEN results in graph axes being drawn in dark green, Or, BACKGROUND=CYAN results in the screen background changing to the color sycan instead of the default color black.

Alternatively, you may specify varying degrees of color by using the RGB (red, green, blue) value of the color you desire. For example, TRACE\_2=255 0 0 is the same as TRACE\_2=RED.

The available colors are: black, blue, brown, brightwhite, cyan, darkblue, darksyan, darkgray, darkgreen, darkmagenta, darkred, green lightgray, magenta, red, and yellow.

## PSpice Help

### Files and Commands

---

In some cases you may want to limit the number of colors used for drawing. For a super VGA display, the default maximum number of colors (NUMTRACECOLORS) is twelve.

Keyword	Description
BACKGROUND	Specifies the background color. The default setting is BLACK.
FOREGROUND	Specifies the foreground color. The default setting is WHITE.
NUMTRACECOLORS	Specifies the maximum number of trace colors. The default setting is 0 (one color). For color printers, set this to 12. After reaching the maximum number, Probe begins displaying the next trace color by repeating the first trace color, then the second, etc.
TRACE_1	Specifies the color of the first trace. The default setting is BRIGHTGREEN.
TRACE_2	Specifies the color of the second trace. The default setting is BRIGHTRED.
TRACE_3	Specifies the color of the third trace. The default setting is BRIGHTBLUE.
TRACE_4	Specifies the color of the fourth trace. The default setting is BRIGHTYELLOW.
TRACE_5	Specifies the color of the fifth trace. The default setting is BRIGHTMAGENTA.
TRACE_6	Specifies the color of the sixth trace. The default setting is BRIGHTCYAN.
TRACE_7	Specifies the color of the seventh trace. The default setting is MUSTARD.
TRACE_8	Specifies the color of the eighth trace. The default setting is PINK.
TRACE_9	Specifies the color of the ninth trace. The default setting is LIGHTGREEN.
TRACE_10	Specifies the color of the tenth trace. The default setting is DARKPINK.
TRACE_11	Specifies the color of the eleventh trace. The default setting is LIGHTBLUE.
TRACE_12	Specifies the color of the twelfth trace. The default setting is PURPLE.



### ***PROBE PRINTER COLORS section***

The [PROBE PRINTER COLORS] section defines the color settings used for printing from Probe. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

To change the settings in the [PROBE PRINTER COLORS] section, use the format:

<item name>=<color>

where <item name> specifies the Probe item and <color> specifies the color.

For example, the entry FOREGROUND=DARKGREEN results in graph axes being printed in dark green. The default maximum number of colors is twelve.

The available colors are: black, blue, brown, brightwhite, cyan, darkblue, darkcyan, darkgray, darkgreen, darkmagenta, darkred, green lightgray, magenta, red, and yellow.

If you are using a four pen plotter, you will get the best results if you set the maximum number of colors to three. The first three trace colors will be used for the colors of digital traces.

Keyword	Description
BACKGROUND	Specifies the background color. The default setting is BRIGHTWHITE.
FOREGROUND	Specifies the foreground color. The default setting is BLACK.
NUMTRACECOLORS	Specifies the maximum number of trace colors. The default setting is 12. After reaching the maximum number, Probe begins printing the next trace color by repeating the first trace color, then the second, etc.
TRACE_1	Specifies the color of the first trace. The default setting is GREEN.
TRACE_2	Specifies the color of the second trace. The default setting is RED.
TRACE_3	Specifies the color of the third trace. The default setting is BLUE.
TRACE_4	Specifies the color of the fourth trace. The default setting is YELLOW.
TRACE_5	Specifies the color of the fifth trace. The default setting is MAGENTA.

TRACE_6	Specifies the color of the sixth trace. The default setting is CYAN.
TRACE_7	Specifies the color of the seventh trace. The default setting is MUSTARD.
TRACE_8	Specifies the color of the eighth trace. The default setting is PINK.
TRACE_9	Specifies the color of the ninth trace. The default setting is LIGHTGREEN.
TRACE_10	Specifies the color of the tenth trace. The default setting is DARKPINK.
TRACE_11	Specifies the color of the eleventh trace. The default setting is LIGHTBLUE.
TRACE_12	Specifies the color of the twelfth trace. The default setting is PURPLE.

### **PSPICE section**

The [PSPICE] section contains settings that are specific to PSpice . The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the PSPICE.INI file.



#### **Caution**

***All of the command lines listed below containing xxxxxCMD are only used when working with PSpice Schematics as the front-end design tool for PSpice . The default settings for these should normally not be changed.***

Command line options for the PSpice applications are explained in much more detail in the online PSpice Reference Guide.

Keyword	Description
BACKUP	<p>Specifies the directory in which backup copies are placed. The default is set at installation. This line can be edited in the form:</p> <p><b>BACKUP=&lt;directory&gt;</b></p> <p>where &lt;directory&gt; specifies the directory where backups will be written. If you are installing a network version, this directory cannot be write-protected and should be set to a local directory, not a network directory.</p>

## PSpice Help

### Files and Commands

---

LIBPATH	<p>Specifies the directories where the model, part, and package libraries are located. The default is set at installation. This line can be edited in the form:</p> <p>LIBPATH=&lt;directory&gt;</p> <p>where &lt;directory&gt; specifies the directory where the library files are located. (For more information, see the section on LIBPATH below.)</p>
MATHEXCEPTIONS	<p>Allows reporting of math exception errors. (The keyword is not included in the netlist.) Valid settings are Off or On. The default setting is ON.</p>
OPTIMIZERCMD	<p>Specifies the command used to run PSpice Optimizer. This line can be edited in the form:</p> <p>OPTIMIZERCMD=&lt;optimize r&gt; [options]</p> <p>where &lt;optimizer&gt; indicates the executable for starting Optimizer and [options] can be any of the available Optimizer switches.</p>
PARTSCMD*	<p>Specifies the command used to run PSpice Model Editor. This line can be edited in the form:</p> <p>PARTSCMD=&lt;modeled&gt; [options]</p> <p>where &lt;modeled&gt; indicates the executable for starting Model Editor and [options] can be any of the available Model Editor switches.</p>
PROBECMD*	<p>Specifies the command to run Probe. This line can be edited in the form:</p> <p>PROBECMD=&lt;probe&gt; [options]</p> <p>where &lt;probe&gt; indicates the executable for starting Probe, and [options] can be any of the available Probe switches.</p>
PSPICECMD*	<p>Specifies the command to run PSpice . This line can be edited in the form:</p> <p>PSPICECMD=&lt;PSpice &gt; [options]</p> <p>where &lt;PSpice &gt; indicates the executable for starting PSpice , and [options] can be any of the available PSpice switches.</p>

## PSpice Help

### Files and Commands

---

PSPICECMDLINE	<p>Specifies what command line options should be used when starting up PSpice . This line can be edited in the form:</p> <p>PSPICECMDLINE= -&lt;switch&gt; &lt;filename&gt;</p> <p>where -&lt;switch&gt; indicates the and &lt;filename&gt; is the name of the corresponding file to be opened. Valid switches are: -i = custom .INI file -p = measurement function file</p>
SET_OPTION_FLAG	<p>Forces the use of SPICE2 time steps when simulating transmission lines. This is not included in the default settings. This line can be inserted in the form:</p> <p>SET_OPTION_FLAG= OLDTLINETSTEP</p>
STMEDCMD*	<p>Specifies the command to run PSpice Stimulus Editor. This line can be edited in the form: S</p> <p>TMEDCMD=&lt;stmed&gt; [options]</p> <p>where &lt;stmed&gt; indicates the executable for starting Stimulus Editor, and [options] can be any of the available Stimulus Editor switches.</p>
TEXTEDITCMD*	<p>Specifies the text editor to use for browsing the netlist and output files. This line can be edited in the form:</p> <p>TEXTEDITCMD=&lt;editor name&gt; %f</p> <p>where &lt;editor name&gt; specifies the text editor and %f specifies where the file name will be substituted in the command line. The default text editor is Notepad.</p>

\* Used with PSpice Schematics only; not used with Capture.

### **LIBPATH**

This variable is originally set during installation to the directory:

LIBPATH=C:\Program Files\Orcad\Capture\ Library\PSpice

If you install the PSpice applications into another directory, then a corresponding subdirectory will be created under your named directory.

For example, if you specify C:\Mydir as the directory where PSpice should be installed, then your LIBPATH variable will be set to:

LIBPATH=C:\Mydir\Orcad\Capture\Library\ PSpice

You can specify more than one directory for LIBPATH by separating the paths with a semicolon (;). When a library file is referenced, PSpice will search for the file in the directories in the order specified by this variable. You can change the LIBPATH variable by editing this line in the form:

LIBPATH=<directory>;<directory>;...;<directory>

For example, if you want PSpice to search for model library files in more than one place, you could set the variable to:

LIBPATH=C:\Program Files\Orcad\Capture\Library\PSpice ; D:\ProjectA\Library

In this case, PSpice will look first for a library file in C:\Program Files\Orcad\Capture\Library\PSpice . If it finds the specified library file here, it will stop searching. Otherwise, PSpice will then search for the specified library file in D:\ProjectA\Library.



***If you chose to install PSpice Schematics as an alternate front-end design entry tool for PSpice , the following additional settings will appear in the [PSpice ] section of the .INI file: SCHEMATICSCMD, MSGVIEWCMD, BOMOUTPUTTYPE, COMPDESCFIE. These are specific settings used by PSpice Schematics, and normally should not be changed.***

### **PSpice NETLIST section**

The [PSpice NETLIST ] section contains settings that specify which model library files should be processed when generating a PSpice simulation netlist.

The typical setting created during installation will appear in the form:

LINE1=.lib "nom.lib"

You can specify more than one library file to include in the netlist generation process. The library files you specify will be searched in the order in which they are listed.

For example, if you have three different sets of model libraries that should be searched and processed during netlist generation, list them as follows:

LINE1=.lib "nom.lib"

LINE2=.lib "projectA.lib"

```
LINE3=.lib "projectB.lib"
```

**Note:** For more specific information on how to set up Library and Include files for processing simulation netlists, see the online Help and User's Guide for the particular front-end tool you are using (Capture or PSpice Schematics).

### **SCHEMATICS section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [SCHEMATICS] section defines the configuration settings used by PSpice Schematics. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
DFLTTITLESYM	Specifies the default symbol to be used for the title block. The default is titleblk.

### **SCHEMATICS BORDER section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [SCHEMATICS BORDER] section defines the window size and position that will be used when PSpice Schematics is launched. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
HEIGHT	Specifies the height of the PSpice Schematics window. The default is 530.
LEFT	Specifies the left position of the PSpice Schematics window. The default is 54.
TOP	Specifies the top position of the PSpice Schematics window. The default is 54.

WIDTH	Specifies the width of the PSpice Schematics window. The default is 768.
ZOOMED	Specifies the zoom factor of the PSpice Schematics window. The default is 0.

### **SCHEMATICS INTERFACES section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [SCHEMATICS INTERFACES] section defines the configuration settings used by PSpice Schematics. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
ECOEXT	Specifies the file extension to be used when generating an ECO file. The default is .BCO.
MAPFILE1	Specifies the name of the default mapping file for generating package type correspondence when creating a layout netlist. The default is PCBOARDS.XNT.
PAREX	Specifies the default name to be used when generating a file. The default is MSIM.
PCBOARDS=EXT	Specifies the file extension to be used when generating a layout netlist in the MicroSim PCBoards format. The default is .NLF.
RDBEXT	Specifies the file extension to be used when generating a file. The default is .si.
REFPINSEP	Specifies the spacing used to separate the pin numbers from the wires on the schematic page. The default is 2.

### **SCHEMATICS LAYERS section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [SCHEMATICS LAYERS] section defines the configuration settings used by PSpice Schematics. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

Keyword	Description
DISPLAY_INFO	Specifies whether command information is displayed in the command line. Valid settings are On or Off. The default setting is ON.
HIDDENPINS	Specifies whether overbars are allowed on names for hidden pins. Valid settings are Overbarsallowed or Off. The default setting is OVERBARSALLOWED.
OVERBARSALLOWED	Specifies whether overbars are allowed on wire labels (node names). Valid settings are On or Off. The default setting is ON.
SIM_VOLTAGES	Specifies whether simulation voltages are printable. Valid settings are Print or Off. The default setting is PRINT.

### **SCHEMATICS MRP LIST section**



***If you did not choose to install PSpice Schematics, this section will not appear in the pspice.ini file.***

The [SCHEMATICS MRP LIST] section defines how large the PSpice Schematics MRP (Most Recently Placed) list can be. This is a listing of the parts that have been placed most recently in the schematic page editor. The entry takes the form:

MAXMRPLISTSIZE=<number>

where <number> specifies how many entries should be displayed in the drop down MRP list in PSpice Schematics. The default number is 10.

### **STIMULUS EDITOR DISPLAY COLORS section**

The [STIMULUS EDITOR DISPLAY COLORS] section defines the color settings used by PSpice Stimulus Editor. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

To change the settings in the [STIMULUS EDITOR DISPLAY COLORS] section, use the format:



`<item name>=<color>`

where `<item name>` specifies the Stimulus Editor item and `<color>` specifies the color.

For example, the entry `FOREGROUND=DARKGREEN` results in graph axes being drawn in dark green, Or, `BACKGROUND=CYAN` results in the screen background changing to the color cyan instead of the default color black.

Alternatively, you may specify varying degrees of color by using the RGB (red, green, blue) value of the color you desire. For example, `TRACE_2=255 0 0` is the same as `TRACE_2=RED`.

The available colors are: black, blue, brown, brightwhite, cyan, darkblue, darkcyan, darkgray, darkgreen, darkmagenta, darkred, green lightgray, magenta, red, and yellow.

In some cases you may want to limit the number of colors used for drawing. For a super VGA display, the default maximum number of colors (`NUMTRACECOLORS`) is six.

Keyword	Description
BACKGROUND	Specifies the background color. The default setting is BLACK.
FOREGROUND	Specifies the foreground color. The default setting is WHITE.
NUMTRACECOLORS	Specifies the maximum number of trace colors. The default setting is 6. After reaching the maximum number, Stimulus Editor begins displaying the next trace color by repeating the first trace color, then the second, etc.
TRACE_1	Specifies the color of the first trace. The default setting is BRIGHTGREEN.
TRACE_2	Specifies the color of the second trace. The default setting is BRIGHTRED.
TRACE_3	Specifies the color of the third trace. The default setting is BRIGHTBLUE.
TRACE_4	Specifies the color of the fourth trace. The default setting is BRIGHTYELLOW.
TRACE_5	Specifies the color of the fifth trace. The default setting is BRIGHTMAGENTA.
TRACE_6	Specifies the color of the sixth trace. The default setting is BRIGHTCYAN.

### ***STIMULUS EDITOR PRINTER COLORS section***

The [STIMULUS EDITOR PRINTER COLORS] section defines the color settings used for printing from PSpice Stimulus Editor. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.

To change the settings in the [STIMULUS EDITOR PRINTER COLORS] section, use the format:

`<item name>=<color>`

where <item name> specifies the Stimulus Editor item and <color> specifies the color.

For example, the entry FOREGROUND=DARKGREEN results in graph axes being printed in dark green. The default maximum number of colors is twelve.

The available colors are: black, blue, brown, brightwhite, cyan, darkblue, darkcyan, darkgray, darkgreen, darkmagenta, darkred, green lightgray, magenta, red, and yellow.

If you are using a four pen plotter, you will get the best results if you set the maximum number of colors to three. The first three trace colors will be used for the colors of digital traces.

Keyword	Description
BACKGROUND	Specifies the background color. The default setting is BRIGHTWHITE.
FOREGROUND	Specifies the foreground color. The default setting is BLACK.
NUMTRACECOLORS	Specifies the maximum number of trace colors. The default setting is 6. After reaching the maximum number, Stimulus Editor begins printing the next trace color by repeating the first trace color, then the second, etc.
TRACE_1	Specifies the color of the first trace. The default setting is GREEN.
TRACE_2	Specifies the color of the second trace. The default setting is RED.
TRACE_3	Specifies the color of the third trace. The default setting is BLUE.
TRACE_4	Specifies the color of the fourth trace. The default setting is YELLOW.
TRACE_5	Specifies the color of the fifth trace. The default setting is MAGENTA.

TRACE\_6                      Specifies the color of the sixth trace. The default setting is CYAN.

### ***SUBCKT SETTING section***

The [SUBCKT SETTING] section defines how various parameters in subcircuits are processed when a hierarchical netlist is generated. The following table describes the different settings. The settings are listed here in alphabetical order for easy reference, but they may appear in any order in the pspice.ini file.



***This section must be added in order to disable the automatic assignment of the global pin name prefix “\$G\_” to hidden pin names when hierarchical netlists are generated.***

Keyword	Description
CREATESUBCKTFORMULT DEVICE	Controls whether subcircuits are created for multiple devices template when a hierarchical netlist is generated. Valid settings are Yes or No. The default setting is YES.
LOCALIZEHIDDENPIN	Controls whether hidden pin names for subcircuits are localized when a hierarchical netlist is generated. Valid settings are Yes or No. The default setting is NO.
SORTPIN	Controls whether pin names for subcircuits are sorted when a hierarchical netlist is generated. Valid settings are Yes or No. The default setting is YES.



---

## Descriptions of menus

---

The main PSpice menu bar is shown below. For a detailed description of a particular menu, click on the name of that menu.

**Note:** Before you open a simulation file, the Edit, Trace and Plot menus are not visible. After a simulation file is opened, all of the menus are accessible.

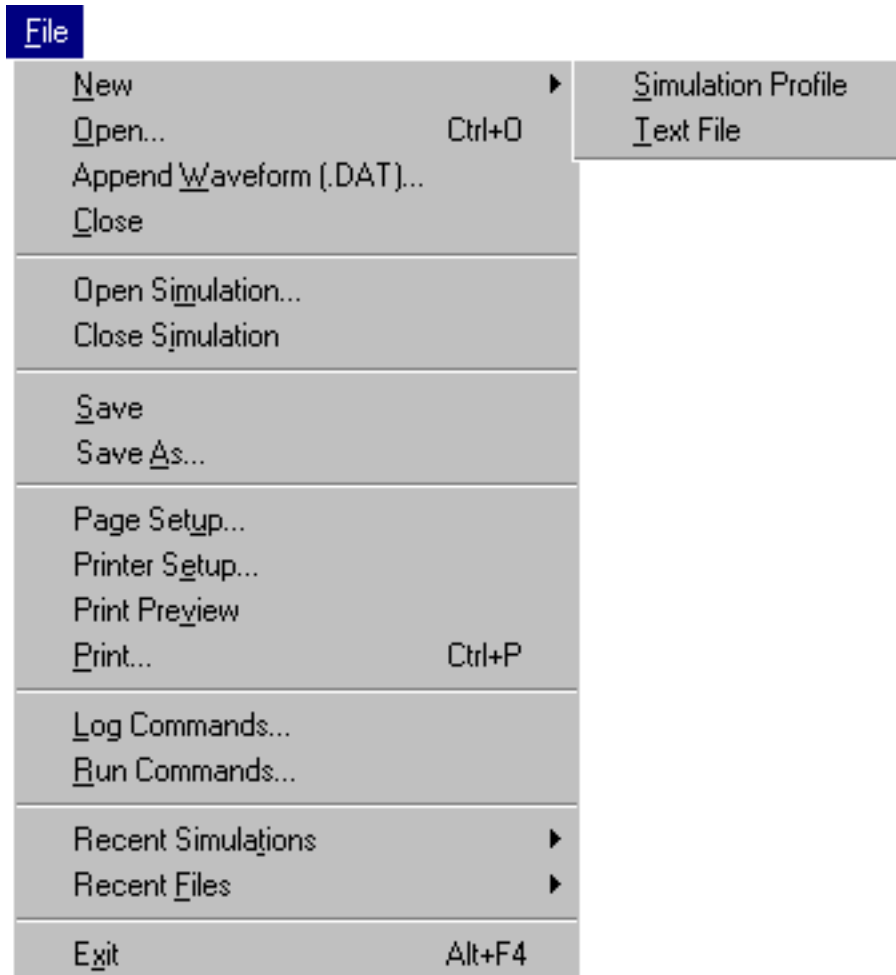
### The File menu

The File menu provides typical file management functions.

## PSpice Help

### Descriptions of menus

---




Menu command...	Function...
New	
..Simulation	Creates a new simulation profile.
..Profile Text File	Creates a new text file.
Open	Opens an existing file.
Append Waveform (.DAT)	Appends a Probe data file to the current file.
Close	Closes the current file.
Open Simulation	Opens an existing simulation file.
Close Simulation	Closes the current simulation file.

## PSpice Help

### Descriptions of menus

---

Save	Saves the current file.
Save As	This menu is available only if a .DAT file is loaded. Saves the current file under a new name.
Export	
Probe Data (.dat file)	Exports the selected data into a new data file.
Stimulus library (.stl file)	Exports the selected data into a new stimulus library file.
Text (.txt file)	Exports the selected data into a new text file.
	
Import	Imports traces stored in .txt or .csv format in PSpice .
Page Setup	Provides page setup options.
Printer Setup	Provides printer setup options.
Print Preview	Previews the current print setup.
Print	Prints the current file.
Log Commands	Records the command sequence in a log file (.CMD).
Run Commands	Runs the commands in a previously saved log file (.CMD).
Recent Simulations	Lists the most recently used simulation files (.SIM).
Recent Files	Lists the most recently used data files (.DAT).
Exit	Exits PSpice .

## The Edit menu

The Edit menu provides standard text editing and navigation functions for use when editing a simulation text file. Most of the commands in the Edit menu are grayed out and unavailable unless you are currently working in a text file.

**Note:** The Undo and Redo commands only apply to actions performed within the text editor. These do not affect any actions performed within Probe or PSpice .

## PSpice Help

### Descriptions of menus

---

Edit	
U <u>ndo</u>	
R <u>edo</u>	
C <u>u</u> t	Ctrl+X
C <u>o</u> py	Ctrl+C
P <u>a</u> ste	Ctrl+V
D <u>e</u> lete	Del
Select A <u>l</u> l	
F <u>i</u> nd...	Ctrl+F
Find N <u>e</u> xt	F3
R <u>e</u> place...	Ctrl+H
G <u>o</u> to Line...	Ctrl+G
I <u>n</u> sert File...	
T <u>o</u> GGLE B <u>o</u> okmark	
N <u>e</u> xt B <u>o</u> okmark	
P <u>r</u> ev <u>i</u> ous B <u>o</u> okmark	
C <u>l</u> ear B <u>o</u> ok <u>m</u> arks	
M <u>o</u> lify O <u>b</u> ject...	

Menu	command...	Function...
Undo		Returns the current file to the state it was in prior to the last executed command ("undoes" the last command).
Redo		Reinstates the state of the file before the last undo command ("undoes" the undo command).
Cut		Copies the selected area to the clipboard and removes it from the text file.
Copy		Copies the selected area to the clipboard.
Paste		Pastes the contents of the clipboard to the cursor location.
Delete		Deletes the selected objects.
Select All		Selects all objects in the text file.



## **PSpice Help**

### Descriptions of menus

---

Find	Provides a means of searching for a particular string of text. See Find dialog box for more information.
Find Next	Finds the next instance of the defined search string.
Replace	Provides a means of searching for and replacing a particular string of text with a new string.
Goto Line	Jumps to the specified line number.
Insert File	Inserts an existing file into the current file.
Toggle Bookmark	Toggles on or off a bookmark at the selected location.
Next Bookmark	Jumps to the next bookmark.
Previous Bookmark	Jumps back to the previous bookmark.
Clear Bookmarks	Clears all of the defined bookmarks.
Modify Object	Provides options for modifying objects.

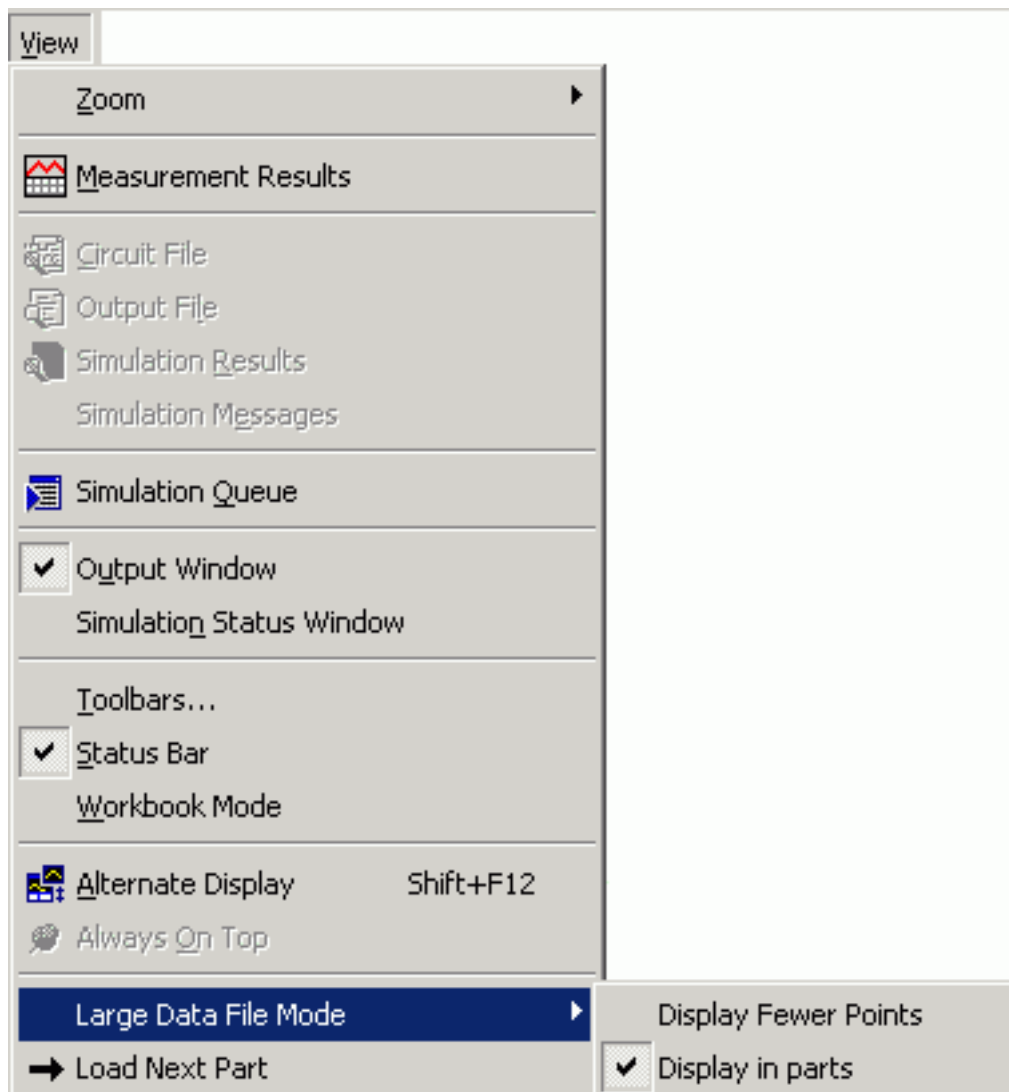
## **The View menu**

The View menu provides controls for defining what portion of a Probe window is displayed, for moving around the window, and for defining what toolbars are displayed.

## PSPICE Help

### Descriptions of menus

---



Menu command...

Function...

## PSpice Help

### Descriptions of menus

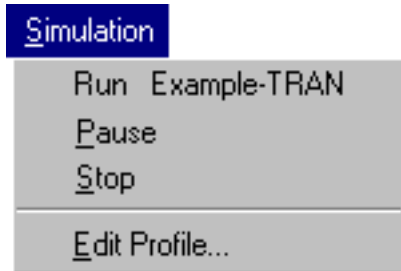
---

Zoom	Provides the following zoom functions:
Fit	fits the entire plot in the current window
In	magnifies the view at the selected point
Out	expands the view at the selected point
Area	magnifies the view to the selected area
Previous	restores the previous view
Redraw	refreshes the display
Pan - New Center	pans to the selected center point; maintains the view
Measurement Results	Displays the measurement results for the current simulation.
Circuit File	Displays the circuit file for the current simulation.
Output File	Displays the output file for the current simulation.
Simulation Results	Displays the results of the current simulation.
Simulation Messages	Displays the messages for the current simulation.
Simulation Queue	Provides a listing of the current simulations in the queue.
Output Window	Toggle for turning on or off the Output Window.
Simulation Status Window	Toggle for turning on or off the Status Window.
PSpice_toolbars	Toggle for turning on or off the toolbars.
Status Bar	Toggle for turning on or off the Status toolbar.
Workbook Mode	Toggle for turning on or off the Workbook Mode.
Alternate Display	Displays only the Probe window with the plotted waveforms. Additional simulation data provided by PSpice is not displayed. See Toggling between display modes .
Large Data File Mode	Toggle for switching between two mode for displaying large data files
Display fewer points	Displays the complete trace constructed using fewer data points from the large data file.  Though the full trace is displayed, it is not very accurate.
Display in parts	Displays a part of the trace at a time. The complete trace described by the large data file is divided into multiple parts. At given point of time, only one part is loaded for viewing.

Load Next Part	Loads and displays the next part of the trace. Enabled only when the Display in parts mode is selected for loading and viewing the large data file.
----------------	---

## The Simulation menu

The Simulation menu provides controls for starting, stopping, and changing a simulation.



Menu command...	Function...
Run <current simulation>	Runs the current (named) simulation.
Pause	Momentarily interrupts the current simulation.
Stop	Halts the current simulation.
Edit Profile	Provides access to the simulation profile for editing the simulation setup.

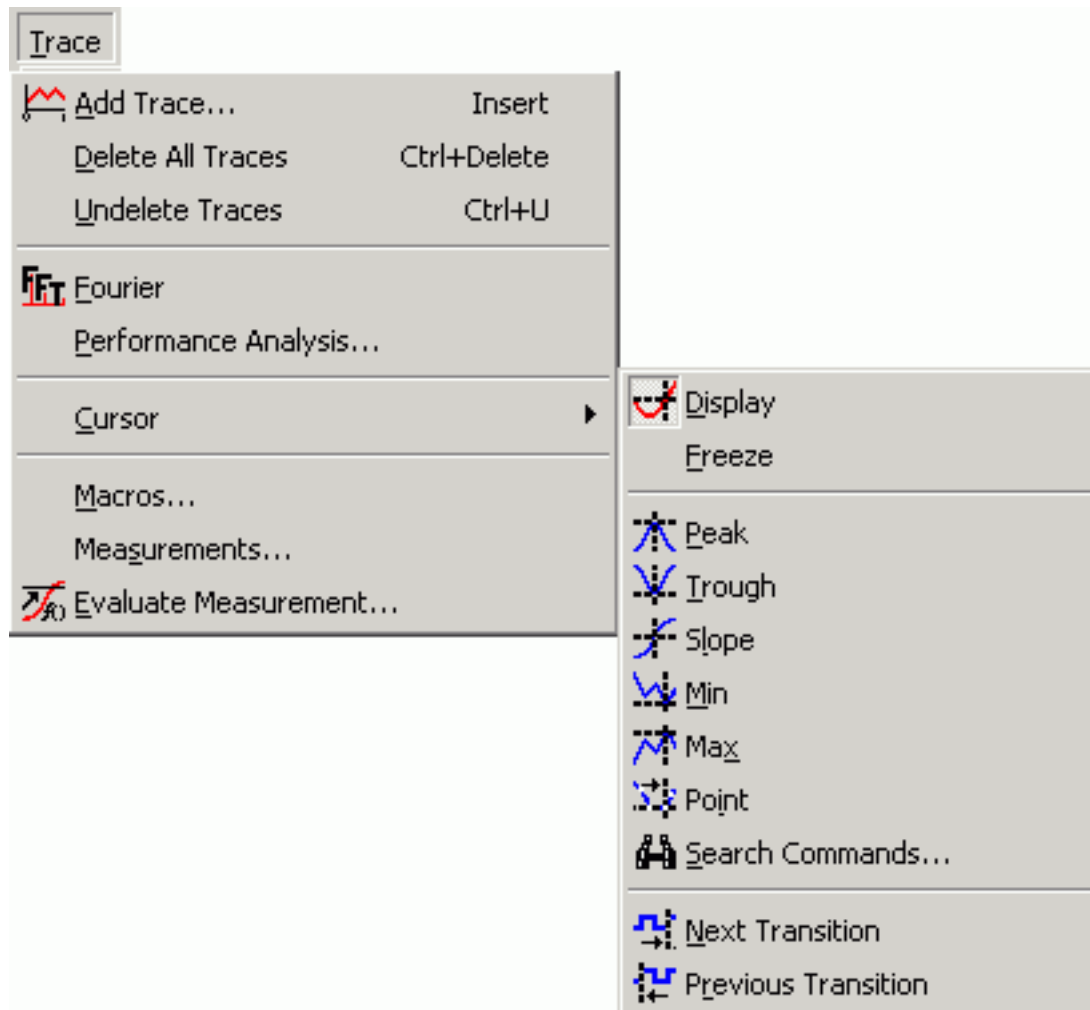
## The Trace menu

The Trace menu provides controls for displaying traces and cursors, and for setting up specialized analyses such as Fourier, performance analysis and measurement expressions.

## PSpice Help

### Descriptions of menus

---



Menu command...	Function...
Add Trace	Adds a trace to the display.
Delete All Traces	Deletes all the traces from the display.
Undelete Traces	Restores deleted traces.
Fourier	Toggles on or off the display of the Fourier transform of analog traces.
Performance Analysis	Toggles on or off performance analysis.

## PSpice Help

### Descriptions of menus

---

Cursor	Provides the following cursor controls:
Display	toggles on or off the cursors
Freeze	locks the cursor at its current position
Peak	jumps the cursor to the next peak on the plot
Trough	jumps the cursor to the next trough on the plot
Slope	jumps the cursor to the next slope on the plot
Min	jumps the cursor to the next minimum value
Max	jumps the cursor to the next maximum value
Point	jumps the cursor to the next data point on the plot
Search Commands	jumps the cursor to a specified point
Next Transition	jumps the cursor to the next digital transition point
Previous Transition	jumps the cursor back to the previous digital transition
Macros	Provides a way to define macros in PSpice .
Measurements...	Provides options for setting up measurement expressions.
Evaluate Measurements...	Provides options for evaluating measurement expressions.

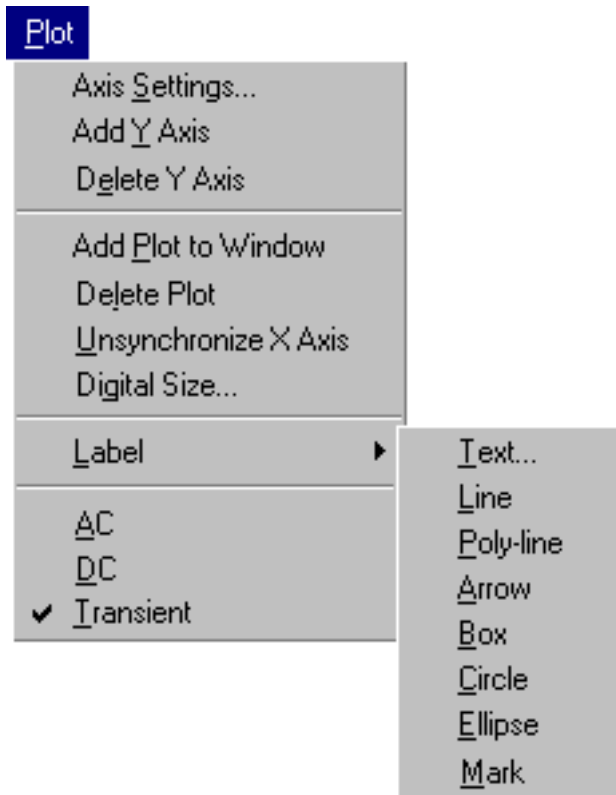
## The Plot menu

The Plot menu provides standard text editing functions for use when editing a simulation text file.

## PSpice Help

### Descriptions of menus

---



Menu command...	Function...
Axis Settings	Provides options for defining the X and Y axes.
Add Y Axis	Adds a new Y axis.
Delete Y Axis	Deletes the second Y axis.
Add Plot to Window	Adds a new plot to the Probe window.
Delete Plot	Deletes the currently selected plot.
Unsynchronize X Axis	Unsynchronizes the X axis.
Digital Size	

## PSpice Help

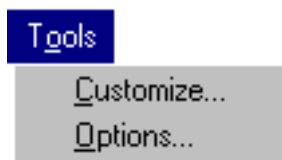
### Descriptions of menus

---

Label	Provides the following drawing functions:
Text	inserts text
Line	draws a line segment
Poly-line	draws a multi-segment poly-line
Arrow	inserts an arrow
Box	draws a rectangular box
Circle	draws a circle
Ellipse	draws an ellipse
Mark	inserts a mark
AC	If checked, identifies the current simulation as an AC analysis.
DC	If checked, identifies the current simulation as a DC analysis.
Transient	If checked, identifies the current simulation as a Transient analysis.

## The Tools menu

The Tools menu provides ways to customize the work environment for PSpice .

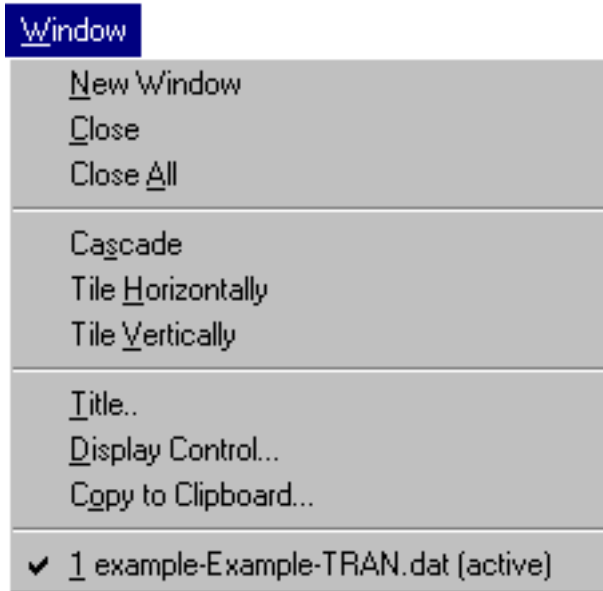


Menu command...	Function...
Customize	Provides customization options for the PSpice environment.
Options	
General tab	Provides general option settings for default values.
Large Dat tab	Provides options settings for opening a large data file in PSpice .



## The Window menu

The Window menu provides typical controls for how different Probe windows are displayed.



Menu command...	Function...
New Window	Opens a new window in Probe.
Close	Closes the current window.
Close All	Closes all windows.
Cascade	Cascades the windows.
Tile Horizontally	Tiles the windows horizontally.
Tile Vertically	Tiles the windows vertically.
Title	Changes the title of the Probe window.
Display Control	Provides a means of saving and restoring the display settings.
Copy to Clipboard	Copies the contents of the current Probe window to the clipboard.
<listing of currently displayed windows in Probe>	

## Terms used in File

### Export Data

Data displayed on the PSpice window can be exported either into a data file, a stimulus library file or into a text file.

The Export Probe Data dialog box is used for generating a data (.dat) file, the Export Stimulus Library dialog box is used for generating a stimulus file, and using Export Text Data dialog box generates a text file.

Output Variables to Export	Lists the variables for which traces will be present in the text file to be generated.
File Name	Specify the name of the new text file to be generated.
Available Output Variables	Lists the output variables for which data is available.
Analog	Select this to list all Analog variables in the Available Output Variables list box.
Digital	
Voltages	Select this if you want only the voltage output variables to be listed in the Available Output Variables list box.
Currents	Select this if you want only the current output variables to be listed in the Available Output Variables list box.
Power	Select this if you want only the power output variables to be listed in the Available Output Variables list box.
Noise (V <sup>2</sup> /Hz)	
Alias Names	Select this to list Alias Names in the Available Output Variables list box.
Subcircuit Nodes	Select this to list the subcircuit nodes in the Available Output Variables list box.
Add	Use this add a new trace variable. Clicking this moves the selected variable from the Available Output Variables list box to Output Variables to Export list box.
Add All	Use this add all the variables listed in the Available Output Variables list box to Output Variables to Export list box.

## PSpice Help

### Descriptions of menus

---

Remove	Select this to remove a variable, for which you do not want the reference file.
Remove All	Select this to remove all the variables form the Output Variables to Export list box.
Data Compression	Data Compression options are available only for the transient analysis. For AC and DC analysis these options are not enabled.
Enable	Select this check box to enable data compression. Selecting this check box enables the Tolerance fields. Increasing the tolerance value decreases the number of data files in the reference waveform. As a result the size of the dat file generated is also reduced.
Absolute Tolerances	Use this to specify the Absolute tolerance to be used while generating the reference data.
Relative Tolerances	Use this to specify the Relative tolerance to be used while generating the reference data. The number of data points in the reference waveform and size of the dat file generated is inversely proportional to the absolute and the relative tolerance.
Reset	Select this to Reset the tolerance values to the default values.

## Find dialog box

The Find dialog box is used to find text in the text window in PSpice .

The options in the Find dialog box are described below:

Match Case	Select this to find text that matches the case of the search string
Regular expression	Select this to find text using regular expressions.
Wrap around search	<p>If this check box is selected, the search wraps around to the start of the document.</p> <p>If this check box is not selected, PSpice displays the “string not found” error message if the string is not found between the current cursor position and the end of the file.</p>
Mark All	Click this button to mark all the lines where the search string exists.

## Regular expressions

You can use regular expressions to find text using the Find dialog box.

Regular expressions are made up of normal characters and metacharacters. Normal characters include upper and lower case letters and digits. The metacharacters have special meanings and are described in detail below.

Metacharacter	Description
.	The period (.) matches any single character except the newline character <code>^</code> . For example the regular expression <code>r.t</code> will match the strings <code>rat</code> , <code>rut</code> , <code>r t</code> , but not <code>root</code> .
[ ]	<p>Use the <code>[ ]</code> brackets to match any one of the characters between the brackets. For example, the regular expression <code>r[aou]t</code> matches <code>rat</code>, <code>rot</code>, and <code>rut</code>, but not <code>ret</code>.</p> <p>Ranges of characters can be specified by using a hyphen. For example, the regular expression <code>[0-9]</code> means match any digit. Multiple ranges can be specified as well. The regular expression <code>[A-Za-z]</code> means match any upper or lower case letter.</p> <p>To match any character except those in the range, use the caret as the first character after the opening bracket. For example, the expression <code>[^269A-Z]</code> will match any characters except <code>2</code>, <code>6</code>, <code>9</code>, and upper case letters.</p>
\$	The dollar sign (\$) matches the end of a line. For example, the regular expression <code>professional\$</code> will match the end of the line "He is a professional" but not the end of the line "They are professionals"
^	The caret (^) matches the beginning of a line. For example, the regular expression <code>^When in</code> will match the beginning of the line "When in the course of human events" but will not match the beginning of the line "What and When in the"
*	The asterisk (*) matches zero or more occurrences of the character immediately preceding. For example, the regular expression <code>.*</code> matches any number of any characters. The regular expression <code>[a-z]*</code> matches zero or more lower-case characters.

## PSpice Help

### Descriptions of menus

---

\	A backslash (\) followed by any metacharacter matches the literal character itself. That is, the backslash “escapes” the metacharacter. For example, \\$ is used to match the dollar sign \$ rather than the end of a line. Similarly, the expression \. is used to match the period character rather than any single character.
	Matches two conditions. For example the regular expression himlher matches the line "it belongs to him" and matches the line "it belongs to her" but does not match the line "it belongs to them"
?	A question mark (?) is an optional element. For example, xy?z matches either xyz or xz.
+	Matches one or more occurrences of the character or regular expression immediately preceding. For example, the regular expression 9+ matches 9, 99, 999 and so on. The regular expression [a-z]+ matches one or more lower-case characters.

## The Large Data File dialog box

The Large Data File dialog box appears when you try to open a large data file in PSpice . A large data file is a .dat file in which the number of data points per trace is greater than the limit specified in PSpice . By default, this limit is set to 1 million data points per trace.

Options	Used for...
Use fewer data points to display the complete trace	<p>Select this when you want the complete trace to be displayed. In this case all data points are not used to construct the trace, only a few data points are used to construct the trace.</p> <p>The number of points used to construct the trace can be specified in the Probe Setting dialog box. The number of data points to be used is specified in the Data points in one part text box in the Large data file tab of the Probe Settings dialog box.</p> <p>For example, if in the .dat file the complete trace is defined using 4 million data points and the value specified in the Data points in one part text box is 1 million, then the trace displayed on the Probe window will be created using only 1 million data points, which is one-fifth of the total number of data points.</p>

## PSpice Help

### Descriptions of menus

---

Use all data points to display trace in parts.

Select this when you want to load and display the trace in parts. In this case, all the data points are used to construct the part of the trace.

The size of each part of the trace is decided by the number specified in the Data points in one part text box in the Large data file tab of the Probe Settings dialog box.

For example, if the complete trace from 0 to 5 seconds has 500000 data points and the value specified in the Data points in one part text box is 100000, then the complete trace will be divided into 5 parts, each constructed using 100000 data points.

Ignore this warning

Select this if you want to load the complete trace in one go using all the data points.

When you select this option the data file may or may not get loaded depending on the system memory. If the memory required for loading the complete data file is not sufficient, an error message stating Out of Memory appears and the file is not loaded.

Always use this option

Select this check box, if you want that one of the three options selected by you should be set as default.

---

# Using the Quick Reference Card

---

The Quick Reference Card (QRC) is an online description of the shortcut keys and toolbars for PSpice . The shortcut keys provide an alternative, keyboard-based method of executing the most common commands. The toolbars also duplicate the most common menu commands and allow you to start them with just one click of the mouse.

Click on the button to jump to the listings of shortcuts or toolbars you are interested in.

[Keyboard shortcuts](#)

[PSpice toolbars](#)

[File toolbar in PSpice](#)

[Edit toolbar in PSpice](#)

[Simulate toolbar in PSpice](#)

[Probe toolbar in PSpice](#)

[Cursor toolbar in PSpice](#)

## Keyboard shortcuts

Key	Mode or user interface item	Function or menu command
CTRL+ A	Zoom menu (on the View menu)	Area
CTRL+ C	Edit menu	Copy
CTRL+ SHIFT+ C	Cursor menu (on the Trace menu)	Display
CTRL+ F	Edit menu	Find
CTRL+ SHIFT+ F	Cursor menu (on the Trace menu)	Freeze
CTRL+ G	Edit menu	Goto Line

## PSpice Help

### Using the Quick Reference Card

---

CTRL+ H	Edit menu	Replace
CTRL+ I	Zoom menu (on the View menu)	In
CTRL+T	Zoom menu (on the View menu)	Out
CTRL+ SHIFT+ I	Cursor menu (on the Trace menu)	Point
CTRL+ L	Zoom menu (on the View menu)	Redraw
CTRL+ SHIFT+ L	Cursor menu (on the Trace menu)	Slope
CTRL+ SHIFT+ M	Cursor menu (on the Trace menu)	Min
CTRL+ N	With a PSpice window active	Creates a new text file
CTRL+ SHIFT+ N	Cursor menu (on the Trace menu)	Next Transition
CTRL+ O	File menu	Open
CTRL+ P	File menu	Print
CTRL+ SHIFT+ P	Cursor menu (on the Trace menu)	Peak
CTRL+ SHIFT+ R	Cursor menu (on the Trace menu)	Previous Transition
CTRL+ SHIFT+ S	Cursor menu (on the Trace menu)	Search Commands
CTRL+ SHIFT+ T	Cursor menu (on the Trace menu)	Trough
CTRL+ U	With a waveform window active	Restores the last deleted traces
CTRL+ V	Edit menu	Paste
CTRL+ X	Edit menu	Cut
CTRL+ SHIFT+ X	Cursor menu (on the Trace menu)	Max
CTRL+ Y	With a waveform window active	Add a Y axis



## PSpice Help

### Using the Quick Reference Card

---

CTRL+ SHIFT+ Y	With a waveform window active	Delete a Y axis
F1	Help menu	Help Topics
F3	Edit menu	Find Next
ALT+ F4	File menu	Exit
F12	With a waveform window active	Restores the last waveform window
INSERT	With a waveform window active	Opens the Add Traces dialog box
DELETE	Edit menu	Delete
CTRL+ DELETE	With a waveform window active	Deletes all traces in the waveform window

## PSpice toolbars

### PSpice toolbar in Capture

Tool	Name	Description
	New simulation profile	Opens the New Simulation dialog box. Equivalent to the New Simulation Profile command on the PSpice menu.
	Edit simulation settings	Opens the Simulation Settings dialog box. Equivalent to the Edit Simulation Settings command on the PSpice menu.
	Run	Runs the simulation. Equivalent to the Run command on the PSpice menu.
	View simulation results	Opens the simulation window. Equivalent to the View Simulation Results command on the PSpice menu.

## PSpice Help

### Using the Quick Reference Card

---

Voltage level marker	Places a voltage level marker on the schematic page. Equivalent to the Voltage Level command on the Markers menu (on the PSpice menu).
Current into pin marker	Places a current into pin marker on the schematic page. Equivalent to the Current Into Pin command on the Markers menu (on the PSpice menu).
Voltage differential marker	Places a voltage differential marker on the schematic page. Equivalent to the Voltage Differential command on the Markers menu (on the PSpice menu).

#### File toolbar in PSpice

Tool	Name	Description
	New	Opens a new simulation file or a new text file. Equivalent to the Simulation Profile command or Text File command on the New menu (on the File menu).
	Open	Opens a data file. Equivalent to the Open command on the File menu.
	Append file	Opens the Append dialog box, which you use to append a data file to the current waveform data. Equivalent to the Append Waveform (.DAT) command on the File menu.
	Save	Saves the active file. Equivalent to the Save command on the File menu.

## PSpice Help

### Using the Quick Reference Card

---

#### Print

Prints the active file. Similar to the Print command on the File menu.

### Edit toolbar in PSpice

**Note:** Most of the commands in the Edit toolbar are grayed out and unavailable unless you are currently working in a text file. The Undo and Redo commands only apply to actions performed within the text editor. These do not affect any actions performed within Probe or PSpice .

Tool	Name	Description
	Cut	Removes the selected object and places it on the Clipboard. Equivalent to the Cut command on the Edit menu.
	Copy	Copies the selected object to the Clipboard. Equivalent to the Copy command on the Edit menu.
	Paste	Pastes the contents of the Clipboard at the cursor. Equivalent to the Paste command on the Edit menu.
	Undo	Undoes the last command performed in the text editor only. Equivalent to the Undo command on the Edit menu.
	Redo	Redoes the last command performed in the text editor only. Equivalent to the Redo command on the Edit menu.
	Toggle bookmark	Toggles a bookmark at the current line. Equivalent to the Toggle Bookmark command on the Edit menu.

## PSpice Help

### Using the Quick Reference Card

---

Next bookmark	Moves to the next bookmark. Equivalent to the Next Bookmark command on the Edit menu.
Previous bookmark	Moves to the previous bookmark. Equivalent to the Previous Bookmark command on the Edit menu.
Clear all bookmarks	Clears all bookmarks in the active window. Equivalent to the Clear Bookmarks command on the Edit menu.

#### Simulate toolbar in PSpice

Tool	Name	Description
	Run	Runs the current simulation. Equivalent to the Run command on the Simulation menu.
	Pause	Pauses the simulation run. Equivalent to the Pause command on the Simulation menu.

#### Probe toolbar in PSpice

Tool	Name	Description
	Zoom in	Zooms in on a specified point. Equivalent to the In command on the Zoom menu (on the View menu).

## PSpice Help

### Using the Quick Reference Card

---

Zoom out	Zooms out from a specified point. Equivalent to the Out command on the Zoom menu (on the View menu).
Zoom area	Zooms in on a selected area of a graph. Equivalent to the Area command on the Zoom menu (on the View menu).
Zoom fit	Zooms to show all traces and labels. Equivalent to the Fit command on the Zoom menu (on the View menu).
Toggle Large Data File Mode	Toggles between two modes for loading a large data file, Display Fewer points and Display in parts.
Load Next Part	Is enabled when the Display in parts mode for loading Large Data File is selected.

Loads and displays the next part of the trace. Select the down-arrow button when you want load any one of the next 5 or the previous 5 parts.

## PSPICE Help

### Using the Quick Reference Card

---

Log X axis	Toggles the X axis between log and linear scaling. Equivalent to selecting the Log option or Linear option in the X Axis tab in the Axis Settings dialog box (accessed by choosing the Axis Settings command on the Plot menu).
Fourier	Toggles between displaying the Fourier transform of all analog traces in the selected plot. Equivalent to the Fourier (or End Fourier) command on the Trace menu.
Performance analysis	Toggles the performance analysis on and off. Equivalent to the Performance Analysis command on the Trace menu.
Log Y axis	Toggles the Y axis between log and linear scaling. Equivalent to selecting the Log option or Linear option in the Y Axis tab in the Axis Settings dialog box (accessed by choosing the Axis Settings command on the Plot menu).
Add trace	Opens the Add Traces dialog box, which you use to add a trace (or traces) to a selected plot. Equivalent to the Add Trace command on the Trace menu.
Evaluate measurement	Opens the Evaluate Measurement dialog box, which you use to evaluate a measurement expression's value. Equivalent to the Evaluate Measurement command on the Trace menu.

## PSpice Help

### Using the Quick Reference Card

---

Text label	Opens the Text Label dialog box, which you use to add a text label to a graph. Equivalent to the Text command on the Label menu (on the Plot menu).
Mark data points	Marks data points on analog traces. Equivalent to selecting the Mark Data Points option in the Probe Options dialog box (accessed by choosing the Options command on the Tools menu).
Toggle cursor	Toggles the display of the Probe cursor on and off. Equivalent to the Display command on the Cursor menu (on the Trace menu).

### Cursor toolbar in PSpice

Tool	Name	Description
	Cursor peak	Positions the cursor at the next peak value. Equivalent to the Peak command on the Cursor menu (on the Trace menu).
	Cursor trough	Positions the cursor at the next trough. Equivalent to the Trough command on the Cursor menu (on the Trace menu).
	Cursor slope	Positions the cursor at the next slope. Equivalent to the Slope command on the Cursor menu (on the Trace menu).

## PSpice Help

### Using the Quick Reference Card

---

Cursor min	Positions the cursor at the next minimum value. Equivalent to the Min command on the Cursor menu (on the Trace menu).
Cursor max	Positions the cursor at the next maximum value. Equivalent to the Max command on the Cursor menu (on the Trace menu).
Cursor point	Positions the cursor at the next data point. Equivalent to the Point command on the Cursor menu (on the Trace menu).
Cursor search	Opens the Search Command dialog box, which you use to position the cursor at a specific place along a trace. Equivalent to the Search Commands command on the Cursor menu (on the Trace menu).
Cursor next tran	Positions the cursor at the next digital transition. Equivalent to the Next Transition command on the Cursor menu (on the Trace menu).
Cursor prev tran	Positions the cursor at the previous digital transition. Equivalent to the Previous Transition command on the Cursor menu (on the Trace menu).
Mark label	Marks the current cursor location with a value. Equivalent to the Mark command on the Label menu (on the Plot menu).



**PSpice Help**  
Using the Quick Reference Card

---

**PSpice Help**  
Using the Quick Reference Card

---

## Index of PSpice symbol and part properties

The following index lists every symbol/part property that can be used with PSpice, its associated symbol/part, what libraries the symbol/part can be found in, and a brief description of what the property is used for. This list should be used when preparing a circuit for simulation by setting up property assignments for symbols/parts that are placed in Capture. The properties are listed alphabetically for easy searching.

Property	Symbol(s)/Part(s)	PSpice Library	Description	Required	Default	Units
AC	VPWL, VSIN, VPWL_F_RE_FOREVER, VPWL_RE_N_TIMES, VPULSE, VPWL_ENH, VPWL_RE_FOREVER, VPWL_FILE, VSFFM, VSRC, VPWL_F_RE_N_TIMES, VSTIM	SOURCE, SOURCSTM	AC magnitude (used in AC sweep analysis only)	No	0	V
AC	IPWL, ISIN, IPWL_F_RE_FOREVER, IPWL_RE_N_TIMES, IPULSE, IPWL_ENH, IPWL_RE_FOREVER, IPWL_FILE, ISFFM, ISRC, IPWL_F_RE_N_TIMES, ISTIM	SOURCE, SOURCSTM	AC magnitude (used in AC sweep analysis only)	No	0	A
AC	IPRINT, VPRINT1, VPRINT2	SPECIAL	Print AC sweep results for signal to output file (YIN)	No	N	none

## PSpice Help

ACMAG	IAC	SOURCE	AC magnitude (used in AC sweep analysis only)	Yes	0	A
ACPHASE	IAC	SOURCE	AC phase (used in AC sweep analysis only)	No	0	Degrees
ACMAG	VAC	SOURCE	AC magnitude (used in AC sweep analysis only)	Yes	0	V
ACPHASE	VAC	SOURCE	AC phase (used in AC sweep analysis only)	No	0	Degrees
AD	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Drain diffusion area	No	DEF AD	m <sup>2</sup>
AGD	ZbreakN	BREAKOUT	Gate-drain overlap area	No	5.0e- 6	m <sup>2</sup>
ANALYSIS	PRINT1, WATCH1	SPECIAL	Analysis type (ACIDCITRAN)	No	TRA N	none
AREA	DbreakCR, JbreakN, QbreakP3, DbreakW, JbreakP, QbreakP4, QbreakL, QbreakN3, QbreakN4, QbreakN, QbreakP, DbreakZ, Dbreak, Bbreak, ZbreakN	BREAKOUT	Area value (for scaling)	No	1.0	none

## PSPice Help

ARM_MASS	Relay_SPDT_phy	ANL_MISC	Contact arm mass (moment)	Yes	0.5	g*cm*sec*sec
AS	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Source diffusion area	No	DEF AS	m^2
ASSERTION	HOLD<SIZE>	DIG_MISC	Assert hold violation when low to high or high to low (LHHL)	Yes	LH	none
ASSERTION	SETUP<SIZE>	DIG_MISC	Assert setup violation when low to high or high to low (LHHL)	Yes	LH	none
BACKEMF	BLDCMTR	MIX_MISC	Back EMF constant	Yes	0.12	Volt*sec/rev
BIT	VECTOR<SIZE>	SPECIAL	Bit position within a single hex or octal digit (see .VECTOR)	No	none	none
C	TLOSSY, TLURC<SIZE>, TLUMP<SIZE>	ANALOG, TLINE	Capacitance value per unit length of transmission line	Yes	1	F
C	T<SIZE>coupled	TLINE	Capacitance value per unit length of coupled transmission line	Yes	-1	F
C1	T<SIZE>coupledX	TLINE	Capacitance value per unit length of first coupled transmission line	Yes	-1	F

## PSpice Help

C2	T<SIZE>coupledX	TLINE	Capacitance value per unit length of second coupled transmission line	Yes	-1	F
C3	T<SIZE>coupledX	TLINE	Capacitance value per unit length of third coupled transmission line	Yes	-1	F
C4	T<SIZE>coupledX	TLINE	Capacitance value per unit length of fourth coupled transmission line	Yes	-1	F
C5	T<SIZE>coupledX	TLINE	Capacitance value per unit length of fifth coupled transmission line	Yes	-1	F
CAP	DIGCAP	DIG_MISC	Capacitance value for digital I/O model	Yes	10P	F
CAPACITANCE	BLDCMTR	MIX_MISC	Winding capacitance to ground	Yes	0.001u	F
CIN	ADCPAR<SIZE>	DATA CONV	Input capacitance	Yes	10P	F
CIN	ADCMIC<SIZE>	DATA CONV	Input capacitance	Yes	22p	F
CKDPDMAX	ADCSER<SIZE>	DATA CONV	Maximum clock to data delay	Yes	80n	sec
CKDPDMIN	ADCSER<SIZE>	DATA CONV	Minimum clock to data delay	Yes	25n	sec
CLKHI	ADCSER<SIZE>	DATA CONV	Minimum clock high pulse width	Yes	40n	sec
CLKLO	ADCSER<SIZE>	DATA CONV	Minimum clock low pulse width	Yes	60n	sec
CLKMAX	ADCSER<SIZE>	DATA CONV	Maximum clock frequency	Yes	5Meg	Hz

## PSpice Help

CLKMIN	ADCSER<SIZE>	DATA CONV	Minimum clock frequency	Yes	Hz	
	E>				178.571K	
CM	T<SIZE>coupled, T<SIZE>coupledX	TLINE	Mutual capacitance per unit length	Yes	0	F
CM	Kcouple2	TLINE	Mutual coupling capacitance per unit length	Yes	none	F
CM12	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 2	Yes	0	F
CM13	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 3	Yes	0	F
CM14	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 4	Yes	0	F
CM15	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 5	Yes	0	F
CM21	Kcouple<SIZE>	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 1	Yes	none	F
CM23	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 3	Yes	0	F
CM24	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 4	Yes	0	F

## PSpice Help

---

CM25	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 5	Yes	0	F
CM31	Kcouple<SIZE>	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 1	Yes	none	F
CM32	Kcouple<SIZE>	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 2	Yes	none	F
CM34	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 4	Yes	0	F
CM35	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 5	Yes	0	F
CM41	Kcouple<SIZE>	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 1	Yes	none	F
CM42	Kcouple<SIZE>	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 2	Yes	none	F
CM43	Kcouple<SIZE>	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 3	Yes	none	F
CM45	T<SIZE>coupledX	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 5	Yes	0	F



## PSpice Help

---

CM51	Kcouple<SIZE >	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 1	Yes	none	F
CM52	Kcouple<SIZE >	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 2	Yes	none	F
CM53	Kcouple<SIZE >	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 3	Yes	none	F
CM54	Kcouple<SIZE >	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 4	Yes	none	F
CO	QRLSZCS	SWIT_RAV	Resonant capacitor value	Yes	.1u	F
COEFF	EPOLY	ANALOG	Voltage gain	Yes	1	V/V
COEFF	FPOLY	ANALOG	Current gain	Yes	1	A/A
COEFF	GPOLY	ANALOG	Transconductance	Yes	1	A/V
COEFF	HPOLY	ANALOG	Transresistance	Yes	1	V/A
COMMAN D<LINE >	STIM1	SOURCE	(time, bit value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0	pair
COMMAN D<LINE >	STIM16	SOURCE	(time, hex value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0000	pair
COMMAN D<LINE >	STIM4	SOURCE	(time, bit value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0000	pair

## PSpice Help

COMMAN D<LINE >	STIM8	SOURCE	(time, bit value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0000 0000	pair
CONTA T_MATC H	Relay_DPDT_ b, Relay_DPDT_ nb	MIX_MISC	Matching of contact make/break/bounce times: 1 = exact	Yes	0.95	none
CONVWH I	ADCSER<SIZ E>	DATA CONV	Minimum high width of convert pulse	Yes	40n	sec
CONVWH I	ADCPAR<SIZ E>	DATA CONV	Minimum high width of convert pulse	Yes	45n	sec
CONVWH I	ADCMIC<SIZ E>	DATA CONV	Minimum high width of convert pulse	Yes	50n	sec
CONVWL O	ADCPAR<SIZ E>	DATA CONV	Minimum low width of convert pulse	Yes	45n	sec
CONVWL O	ADCMIC<SIZ E>	DATA CONV	Minimum low width of convert pulse	Yes	50n	sec
CONVWL O	ADCSER<SIZ E>	DATA CONV	Minimum low width of convert pulse	Yes	60n	sec
COUPLI NG	XFRM_LINEAR, XFRM_NONLI NEAR, K_LINEAR, kbreak	ANALOG, BREAKOUT	Coupling coefficient	Yes	1	none
COUT	DACCUR<SIZ E>	DATA CONV	Output capacitance	Yes	10p	F
CSMINL O	DACPAR<SIZ E>	DATA CONV	Minimum low width of CS	Yes	90n	sec
D	CMSSCCM	SWIT_RAV	Duty cycle	Yes	0.75	none
D	VMSSCCM	SWIT_RAV	Duty cycle	Yes	0.4	none
DAMPIN G	BLDCMTR	MIX_MISC	Damping and eddy current losses (linear torque with speed)	Yes	0.36	g*cm *sec/ rad

## PSpice Help

---

DAMPING	Relay_SPDT_phy	ANL_MISC	Limit damping rate	Yes	1000	g*cm*sec/rad
DB	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in DB in output file (YIN)	No	N	none
DC	VDC	SOURCE	DC voltage magnitude (used for bias point and in transient analysis)	Yes	0	V
DC	IDC	SOURCE	DC current magnitude (used for bias point and in transient analysis)	Yes	0	A
DC	VPWL, VSIN, VPWL_F_RE_FOREVER, VPWL_RE_N_TIMES, VPULSE, VPWL_ENH, VPWL_RE_FOREVER, VPWL_FILE, VSFFM, VSRC, VPWL_F_RE_N_TIMES, VSTIM, VAC	SOURCE, SOURCSTM	DC voltage magnitude (used for bias point)	No	0	V

## PSpice Help

DC	IPWL, ISIN, IPWL_F_RE_ FOREVER, IPWL_RE_N_ TIMES, IPULSE, IPWL_ENH, IPWL_RE_FO REVER, IPWL_FILE, ISFFM, ISRC, IPWL_F_RE_ N_TIMES, ISTIM, IAC	SOURCE, SOURCST M	DC current magnitude (used for bias point)	No	0	A
DC	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write DC analysis results to output file (YIN)	No	N	none
DEADTIME	SG1525A/ 25C, SG1526B, SG1525, SG1524B, SG1529	SWIT_REG	Dead time	Yes	1u	sec
DEADTIME	SG1842, SG1843, SG1844, SG1845, SG1846	SWIT_REG	Dead time	Yes	2u	sec
DEADTIME	SG1825	SWIT_REG	Dead time	Yes	5E-08	sec
DELAY	DIGCLOCK	SOURCE	Delay before clock starts	No	0	sec
DELAY	EFREQ, FTABLE, GFREQ	ABM	Phase delay to apply to data in frequency table	No	0	sec
DELAY	DELAY	DIG_MISC	Digital delay	Yes	50n	sec

## PSpice Help

---

DENOM	LAPLACE	ABM	Denominator of Laplace transform	Yes	1 + s	none
DETENT	BLDCMTR	MIX_MISC	Magnetic detent torque	Yes	2.9	g*cm
DF	VSIN, ISIN	SOURCE	Damping factor	No	0	none
DHOLD	DACSER<SIZE>	DATACONV	Data hold time	Yes	10n	sec
DIG_GND	STIM<SIZE>	SOURCE	Digital ground reference node (change for user power supply)	Yes	\$G_DGN	none
DIG_POWER	STIM<SIZE>	SOURCE	Digital power reference node (change for user power supply)	Yes	\$G_DPWR	none
DRAG	Relay_SPDT_phy	ANL_MISC	Air and other contact arm drag	Yes	1	g*cm / sec^2
DSET	DACSER<SIZE>	DATACONV	Data setup time	Yes	100n	sec
ERRORLIMIT	RELEASE<SIZE>, HOLD<SIZE>, CONSTRAINT<SIZE>, SETUP<SIZE>, WIDTH_HI, WIDTH_LO, MAXFREQ	DIG_MISC	Maximum number of constraint checker violations to report	No	20	none

## PSpice Help

EXP<LINE>	PWR, PWRS, ABM2, ABM3, ABM1, ABM1/I, ABM3/I, ABM2/I, ABM/I, ABM	ABM	ABM expression line containing circuit variables and math functions and operators forming controlling expression. Start with lowest number. Lines are concatenated.	Yes	1	none
EXPR	ELAPLACE, GTABLE, EFREQ, ETABLE, GVALUE, GFREQ, EVALUE, GLAPLACE	ABM	ABM expression line containing circuit variables and math functions and operators forming controlling expression. Default is signal connected to input pins of symbol.	No		instance V(%IN+, %IN-)
F	T	ANALOG	Frequency for NL parameter (see Transmission Line device)	No	none	Hz
F0	BANDREJ, BANDPASS	ABM	Lower stopband frequency of filter	Yes	10	Hz
F1	BANDREJ, BANDPASS	ABM	Lower passband frequency of filter	Yes	100	Hz
F2	BANDREJ, BANDPASS	ABM	Upper passband frequency of filter	Yes	300	Hz
F3	BANDREJ, BANDPASS	ABM	Upper stopband frequency of filter	Yes	1000	Hz
FC	VSFFM, ISFFM	SOURCE	Carrier frequency	Yes	none	Hz

## PSpice Help

---

FILE	VPWL_F_RE_ FOREVER, IPWL_F_RE_ N_TIMES, VPWL_FILE, IPWL_F_RE_ FOREVER, VPWL_F_RE_ N_TIMES	SOURCE	Input file containing PWL source data, (time, analog value) pairs	Yes	none	none
FILE	VECTOR<SIZ E>	SPECIAL	Output file for .VECTOR results	No	none	none
FILENAME	FileStim<SIZE >	SOURCE	Input file containing STIM source data, (time, digital value) pairs	Yes	none	none
FILENAME	LIB	SPECIAL	Library file name to be included	Yes	none	none
FILENAME	INCLUDE	SPECIAL	Include file name to be included	Yes	none	none
FIRST_ NPAIRS	IPWL_RE_FO REVER, VPWL_RE_N_ TIMES, VPWL_ENH, VPWL_RE_F OREVER, IPWL_ENH, IPWL_RE_N_ TIMES	SOURCE	First line containing PWL data, (time, analog value) pairs. Lines will be concatenated.	Yes	none	pairs
FLOAT			Property that should be added on an unconnected pin. Add this property on unconnected pins instead of using a No Connect symbol.			

## PSpice Help

---

See  
Using  
the  
FLOAT  
proper  
ty

FM	VSFFM, ISFFM	SOURCE	Modulation frequency	Yes	none	Hz
FORMAT	STIM1	SOURCE	Digital data format (1=bit, 4=hex).	No	1	binar y
FORMAT	STIM4	SOURCE	Digital data format (1=bit, 4=hex)	No	1111	binar y
FORMAT	STIM8	SOURCE	Digital data format (1=bit, 4=hex)	No	1111 1111	binar y
FORMAT	STIM16	SOURCE	Digital data format (1=bit, 4=hex)	No	4444	hex
FP	HIPASS	ABM	Pass band cutoff frequency	Yes	100	Hz
FP	LOPASS	ABM	Pass band cutoff frequency	Yes	10	Hz
FREQ	VSIN, ISIN	SOURCE	Frequency of sinusoid (transient analysis only)	Yes	none	Hz
FRICTI ON	BLDCMTR	MIX_MISC	Friction/drag losses (constant torque losses)	Yes	0.72	g*cm



## PSpice Help

FRQ	COAX, RG6/U, TLINE	Frequency to evaluate expressions for R and G (conductor and dielectric losses). If specified, R and G are constant in the model. No assignment to FRQ (leaving the value empty) will select Laplace expression for R and G to model skin effect.	No	Hz
	RG58A/U, RG59B/U, RG8A/U, RG9B/U, RG174/U, RG8/U, RG12A/U, RG58C/U, RG6A/U, RG11A/U, RG59/u=, RG22B/U, RG55B/U, RG58/U+, RG8/u+, RG62/U RG179B/U, RG178B/U, RG188A/U, RG212/U, RG223/U, RG11/u+, RG62A/U, RG63B/U, RG187A/U, RG213/U, RG214/U, RG11/U, RG55/U, RG196A/U, RG215/U, RG71B/U, RG195A/U, RG217/U, RG58/U, RG218/U, RG59/U, RG219/U, RG180B/U		100 Meg	

## PSpice Help

FRQ	TP19AWG, TP26AWG, TP24AWG, TP22AWG	TLINE	Frequency to evaluate expressions for R and G (conductor and dielectric losses). If specified, R and G are constant in the model. No assignment to FRQ (leaving the value empty) will select Laplace expression for R and G to model skin effect (use for AC Sweep analysis only).	No	5k	Hz
FS	LOPASS	ABM	Stop band cutoff frequency	Yes	100	Hz
FS	CMLSCCM	SWIT_RAV	Operating frequency	Yes	100k	Hz
FS	CMSSCCM	SWIT_RAV	Operating frequency	Yes	100k	Hz
FS	HIPASS	ABM	Stop band cutoff frequency	Yes	10	Hz
FS	VMLSDCM	SWIT_RAV	Operating frequency	Yes	50k	Hz
FS	VMCCMDCM	SWIT_RAV	Operating frequency	Yes	50k	Hz
FSOFFS ET	QRLSZCS	SWIT_RAV	Frequency from Vco at zero Vc	Yes	280k	Hz
G	T<SIZE>coupled, TLUMP<SIZE>	TLINE	Per unit length conductance	Yes	0	Siemens
G	TLOSSY	ANALOG	Per unit length conductance	Yes	1	Siemens
G1	T<SIZE>coupledX	TLINE	Per unit length conductance of first conductor	Yes	0	Siemens

## PSpice Help

G2	T<SIZE>coupledX	TLINE	Per unit length conductance of second conductor	Yes	0	Siemens
G3	T<SIZE>coupledX	TLINE	Per unit length conductance of third conductor	Yes	0	Siemens
G4	T<SIZE>coupledX	TLINE	Per unit length conductance of fourth conductor	Yes	0	Siemens
G5	T<SIZE>coupledX	TLINE	Per unit length conductance of fifth conductor	Yes	0	Siemens
GAIN	E	ANALOG	Voltage gain	Yes	1	V/V
GAIN	F	ANALOG	Current gain	Yes	1	A/A
GAIN	G	ANALOG	Transconductance	Yes	1	I/V
GAIN	H	ANALOG	Transresistance	Yes	1	V/I
GAIN	DIFFER	ABM	Prescaling factor before differentiation	Yes	1	instance
GAIN	INTEG	ABM	Prescaling factor before integration	Yes	1	instance
GAIN	GAIN	ABM	Gain	Yes	1000	instance
GAIN	GLIMIT	ABM	Prescaling factor before limiting	Yes	1k	instance
GAIN	HILO	ABM	Prescaling factor before limiting	Yes	1k	instance
GAIN	SOFTLIM	ABM	Prescaling factor before limiting	Yes	1k	instance
GAIN_REF	ADCPAR<SIZE>	DATA CONV	Upper input voltage limit	Yes	0.4	none
GAIN_REF_GND	ADCPAR<SIZE>	DATA CONV	Lower input voltage limit	Yes	-0.4	none
HCTAU	BULB	OPTO	Heat capacity time constant	Yes	4.082	sec

## PSpice Help

HI	WATCH1	SPECIAL	Upper limit value for .WATCH	Yes	none	V
HI	GLIMIT, HILO, SOFTLIM	ABM	Upper limit	Yes	10	V
HOLDTIME	HOLD<SIZE>	DIG_MISC	Hold time for constant checking	Yes	none	sec
I_DROP	BOUNCE, NO_BOUNCE, Relay_DPDT_ b, Relay_SPDT_ b, Relay_SPDT_ nb, Relay_DPDT_ nb	ANL_MISC	Drop-out current	Yes	25m	A
I_PULL	BOUNCE, NO_BOUNCE, Relay_DPDT_ b, Relay_SPDT_ b, Relay_SPDT_ nb, Relay_DPDT_ nb	ANL_MISC	Pull-in current	Yes	35m	A
I1	IPULSE, IEXP	SOURCE	First current level	Yes	none	A
I2	IPULSE, IEXP	SOURCE	Second current level	Yes	none	A
I1	IPWL	SOURCE	First PWL point (0s, current value)	Yes	none	A
I2	IPWL	SOURCE	Second PWL point (time, current value)	No	none	A
I3	IPWL	SOURCE	Third PWL point (time, current value)	No	none	A
I4	IPWL	SOURCE	Fourth PWL point (time, current value)	No	none	A

## PSpice Help

I5	IPWL	SOURCE	Fifth PWL point (time, current value)	No		A none
I6	IPWL	SOURCE	Sixth PWL point (time, current value)	No		A none
I7	IPWL	SOURCE	Seventh PWL point (time, current value)	No		A none
I8	IPWL	SOURCE	Eighth PWL point (time, current value)	No		A none
IAMPL	ISIN	SOURCE	Current amplitude of sinusoid (transient analysis only)	Yes		A none
IAMPL	ISFFM	SOURCE	Current amplitude of SFFM (transient analysis only)	Yes		A none
IC	Cbreak, C	BREAKOUT , ANALOG	Initial voltage condition on capacitor	No	0	V
IC	Lbreak, L	BREAKOUT , ANALOG	Initial current condition on inductor	No	0	A
IC	INTEG	ABM	Initial condition for integrator	Yes	0	instance
IC	VMSSCCM	SWIT_RAV	Current flowing from terminal C	Yes	1	A
IC	CMSSCCM	SWIT_RAV	Current flowing from terminal C	Yes	100	A
IMAG	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in IMAGINARY format in output file (YIN)	No	N	none
INDUCTANCE	BLDCMTR	MIX_MISC	Winding inductance	Yes	3m	Hz
INERTIA	BLDCMTR	MIX_MISC	Moment of inertia of rotor	Yes	0.30	g*cm *sec* sec

## PSpice Help

IO_MOD EL	DACSER<SIZE> E> DACCUR<SIZE> E> ADCSER<SIZE> E> ADCPAR<SIZE> E> DACPAR<SIZE> E> ADCMIC<SIZE> E>	DATA CONV	I/O model to use for and D/A converter models	No	IO_H CT	none
IO_MOD EL	STIM<SIZE>, DIGCLOCK, DIGSTIM	SOURCE	I/O model to use for digital stimulus	No	IO_S TM	none
IOFF	ISIN, ISFFM	SOURCE	Offset current of transient analysis source	Yes		A none
ISINK	DACPAR<SIZE> E>	DATA CONV	Typical output sink current	Yes	- 400u	A
ISOURCE	DACPAR<SIZE> E>	DATA CONV	Typical output source current	Yes	5m	A
K_COEF	3phase	ANL_MISC	Mutual coupling coefficient	Yes	0.99 99	none
KP	ZbreakN	BREAKOUT	MOS transconductance for IGBT	No	0.38	A/ V^2
L	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Length	no	DEF L	m
L	TLOSSY, TLUMP<SIZE> >	ANALOG, TLINE	Per unit length inductance	Yes	1	H

## PSpice Help

L	T<SIZE>coupled	TLINE	Per unit length inductance	Yes	-1	H
L_COIL	BOUNCE, NO_BOUNCE, Relay_DPDT_b, Relay_SPDT_b, Relay_SPDT_nb, Relay_DPDT_nb, Relay_SPDT_phy	ANL_MISC, MIX_MISC	Coil inductance	Yes	5m	H
L1	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of first coupled inductor	Yes	L1	refdes
L1	T<SIZE>coupledX	TLINE	Per unit length inductance of first conductor	Yes	-1	H
L1_TURNS	XFRM_NONLINEAR	BREAKOUT	Number of turns for first inductor	Yes	none	none
L1_VALUE	XFRM_LINEAR	ANALOG	Inductance value of first inductor	Yes	none	H
L2	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of second coupled inductor	Yes	L2	refdes
L2	T<SIZE>coupledX	TLINE	Per unit length inductance of second conductor	Yes	-1	H
L2_TURNS	XFRM_NONLINEAR	BREAKOUT	Number of turns for second inductor	Yes	none	none
L2_VALUE	XFRM_LINEAR	ANALOG	Inductance value of second inductor	Yes	none	H
L3	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of third coupled inductor	Yes	L3	refdes

## PSpice Help

---

L3	T<SIZE>coupledX	TLINE	Per unit length inductance of third conductor	Yes	-1	H
L4	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of fourth coupled inductor	Yes	L4	refdes
L4	T<SIZE>coupledX	TLINE	Per unit length inductance of fourth conductor	Yes	-1	H
L5	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of fifth coupled inductor	Yes	L5	refdes
L5	T<SIZE>coupledX	TLINE	Per unit length inductance of fifth conductor	Yes	-1	H
L6	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of sixth coupled inductor	Yes	L6	refdes
LDACMINLO	DACPAR<SIZE>	DATACONV	Minimum pulse width low for LDAC	Yes	90n	sec



## PSpice Help

LEN	COAX, RG6/U, TLINE, RG58A/U, ANALOG RG59B/U, RG8A/U, RG9B/U, RG174/U, RG8/U, RG12A/U, RG58C/U, RG6A/U, RG11A/U, RG59/u=, RG22B/U, RG55B/U, RG58/U+, RG8/u+, RG62/U RG179B/U, RG178B/U, RG188A/U, RG212/U, RG223/U, RG11/u+, RG62A/U, RG63B/U, RG187A/U, RG213/U, RG214/U, RG11/U, RG55/U, RG196A/U, RG215/U, RG71B/U, RG195A/U, RG217/U, RG58/U, RG218/U, RG59/U, RG219/U, RG180B/U, TWSTPAIR, TP24AWG, TP22AWG, TP19AWG, TP16AWG, TLOSSY, TLURC<SIZE >, TLUMP<SIZE	Length of transmission line. The characteristic parameters R, L, G, and C are defined per unit length. For coax and twisted pair models, the length must be specified in meters.	Yes	none	m
-----	---	---	-----	------	---

## PSpice Help

LFIL	VMLSDCM, VMCCMDCM	SWIT_RAV	Filter inductance	Yes	500u	H
LFIL	CMLSCCM, CMSSCCM	SWIT_RAV	Filter inductance	Yes	5u	H
LIM_K	Relay_SPDT_ phy	ANL_MISC	Limit spring rate	Yes	1000 000	g/ sec
LM	T<SIZE>coupl ed, T<SIZE>coupl edX	TLINE	Mutual inductance per unit length	Yes	0	H
LM	Kcouple2	TLINE	Mutual coupling inductance per unit length	Yes	none	H
LM12	T<SIZE>coupl edX, Kcouple<SIZE >	TLINE	Mutual coupling inductance per unit length between tlines 1 and 2	Yes	0	H
LM13	T<SIZE>coupl edX, Kcouple<SIZE >	TLINE	Mutual coupling inductance per unit length between tlines 1 and 3	Yes	0	H
LM14	T<SIZE>coupl edX, Kcouple<SIZE >	TLINE	Mutual coupling inductance per unit length between tlines 1 and 4	Yes	0	H
LM15	T<SIZE>coupl edX, Kcouple<SIZE >	TLINE	Mutual coupling inductance per unit length between tlines 1 and 5	Yes	0	H
LM21	Kcouple<SIZE >	TLINE	Mutual coupling inductance per unit length between tlines 2 and 1	Yes	none	H
LM23	T<SIZE>coupl edX, Kcouple<SIZE >	TLINE	Mutual coupling inductance per unit length between tlines 2 and 3	Yes	0	H

## PSpice Help

---

LM24	T<SIZE>coupledX, Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 2 and 4	Yes	0	H
LM25	T<SIZE>coupledX, Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 2 and 5	Yes	0	H
LM31	Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 3 and 1	Yes	none	H
LM32	Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 3 and 2	Yes	none	H
LM34	T<SIZE>coupledX, Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 3 and 4	Yes	0	H
LM35	T<SIZE>coupledX, Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 3 and 5	Yes	0	H
LM41	Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 4 and 1	Yes	none	H
LM42	Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 4 and 2	Yes	none	H
LM43	Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 4 and 3	Yes	none	H

## PSPice Help

LM45	T<SIZE>coupledX, Kcouple<SIZE>	TLINE	Mutual coupling inductance per unit length between tlines 4 and 5	Yes	0	H
LO	WATCH1	SPECIAL	Lower voltage value for .WATCH	Yes	none	none
LO	GLIMIT, HILO, LIMIT, SOFTLIM	ABM	Lower limit voltage	Yes	0	V
LO	QRLSZCS	SWIT_RAV	Resonant inductor value	Yes	20n	H
M	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Multiplier	No	1	none
MAG	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in MAGNITUDE format in output file (YIN)	No	Y	none
MAGUNITS	EFREQ, FTABLE, GFREQ	ABM	Units for magnitude table entries (MAGIDB)	No	MAG	none
MAXFREQ	MAXFREQ	DIG_MISC	Maximum frequency for constraint checker	Yes	none	Hz
MAXFREQ	ADCPAR<SIZE>	DATA CONV	Maximum clock frequency	Yes	11Meg	Hz
MAXP	Relay_SPDT_phy	ANL_MISC	Maximum permeance (when arm is close to coil)	Yes	5	gauss*m^2/A
MINFREQ	MINFREQ	DIG_MISC	Minimum frequency for constraint checker	Yes	none	Hz

## PSpice Help

---

MINFRE Q	ADCPAR<SIZE>	DATA CONV	Minimum clock frequency	Yes	10K	Hz
MINHOLD	ADCMIC<SIZE>	DATA CONV	R/Cbar to CSbar hold time	Yes	10n	sec
MINLO	ADCMIC<SIZE>	DATA CONV	R/Cbar minimum pulse width	Yes	50n	sec
MINP	Relay_SPDT_phy	ANL_MISC	Minimum permeance (when arm is close to coil)	Yes	1	gauss*m^2/A
MINPER	ADCMIC<SIZE>	DATA CONV	Minimum time between conversions	Yes	10u	sec
MINSET	ADCMIC<SIZE>	DATA CONV	R/Cbar to CSbar setup time	Yes	10n	sec
MINW	DACSER<SIZE>	DATA CONV	LDAC low minimum pulse width	Yes	50n	sec
MOD	VSFFM, ISFFM	SOURCE	Modulation Index	Yes	none	none
MUTUAL_IND	BLDCMTR	MIX_MISC	Adjacent winding mutual coupling factor	Yes	0.5	none
N	QRLSZCS	SWIT_RAV	Full-wave=1; half-wave=2	Yes	2	none
NL	T	ANALOG	Number of wavelengths	No	none	none
NRB	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Multiplier of RSH to get RB	No	0	none

## PSpice Help

---

NRD	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Multiplier of RSH to get RD	No	0	none
NRG	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Multiplier of RSH to get RG	No	0	none
NRS	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Multiplier of RSH to get RS	No	0	none
NUM	LAPLACE	ABM	Numerator of Laplace transform	Yes	1	none
OFFTIME	DIGCLOCK	SOURCE	Time clock is low	Yes	.5u	sec
ONTIME	DIGCLOCK	SOURCE	Time clock is high	Yes	.5u	sec
OPPVAL	DIGCLOCK	SOURCE	State to transition to from low (usually 1)	Yes	1	none
PD	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Perimeter of drain	No	0	m

## PSpice Help

PER	IPULSE, VPULSE	SOURCE	Period	No	TST OP	sec
PERIOD	SG1525A/ 25C, SG1526B, SG1525, SG1524B, SG1529	SWIT_REG	Period of internal oscillator	Yes	1m	sec
PERIOD	SG1825	SWIT_REG	Period of internal oscillator	Yes	2.5u	sec
PERIOD	SG1825, SG1842, SG1843, SG1844, SG1845, SG1846	SWIT_REG	Period of internal oscillator	Yes	22.5 u	sec
PERMEAN- CE_RATIO	3phase	ANL_MISC	Ratio of Permeance_inner/ Permeance_outer	Yes	1	none
PHASE	IPRINT, IPLOT, VLOT1, VLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in PHASE format in output file (YIN)	No	N	none
PHASEUN- ITS	EFREQ, FTABLE, GFREQ	ABM	Units for phase table entries (DEG RAD)	No	DEG	none
PNOM	BULB	OPTO	Nominal power	Yes	100	W
POS	VECTOR<SIZ E>	SPECIAL	Column position (see .VECTOR)	No	see .VEC TOR	none
PRI_IN- D	3phase	ANL_MISC	Primary inductance	Yes	200 m	H
PRI_R	3phase	ANL_MISC	Primary resistance	Yes	0.1	Ohm s

## PSpice Help

---

PS	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Perimeter of source	No	0	m
PULSE	74LS122, 74LS123	1_SHOT	Pulsewidth	Yes	116n	sec
PULSE	CD4098B	1_SHOT	Pulsewidth	Yes	1u	sec
PULSE	CD4538B	1_SHOT	Pulsewidth	Yes	20u	sec
PULSE	74121	1_SHOT	Pulsewidth	Yes	30n	sec
PULSE	54L121	1_SHOT	Pulsewidth	Yes	35n	sec
PULSE	74122	1_SHOT	Pulsewidth	Yes	45n	sec
PULSE	74123	1_SHOT	Pulsewidth	Yes	45n	sec
PULSE	54L122, 54L123	1_SHOT	Pulsewidth	Yes	90n	sec
PW	IPULSE, VPULSE	SOURCE	Pulsewidth	No	TST OP	sec
QUIESC UR	SG1842, SG1843	SWIT_REG	Quiescent current	Yes	11m	A
R	T<SIZE>coupl ed, TLUMP<SIZE >, TLOSSY, TLURC<SIZE >	TLINE, ANALOG	Per unit length resistance	Yes	0	Ohm s



## PSpice Help

R_CLOSE	BOUNCE, Relay_SPDT_ phy, NO_BOUNCE, Relay_DPDT_ B, Relay_SPDT_ b, Relay_SPDT_ nb, Relay_DPDT_ nb	ANL_MISC, MIX_MISC	Closed state resistance	Yes	0.05	Ohms
R_COIL	BOUNCE, Relay_SPDT_ phy, NO_BOUNCE, Relay_DPDT_ B, Relay_SPDT_ b, Relay_SPDT_ nb, Relay_DPDT_ nb	ANL_MISC, MIX_MISC	Series resistance of coil	Yes	10	Ohms
R_I	EFREQ, FTABLE, GFREQ	ABM	Use real and imaginary (set value to R_I). Default is magnitude and phase	No	magnitude/ phase	none
R_OPEN	BOUNCE, Relay_SPDT_ phy, NO_BOUNCE, Relay_DPDT_ B, Relay_SPDT_ b, Relay_SPDT_ nb, Relay_DPDT_ nb	ANL_MISC, MIX_MISC	Closed state resistance	Yes	100 MEG	Ohms

## PSpice Help

R_REF_IN	ADCPAR<SIZE>	DATA CONV	Input resistance of reference	Yes	5K	Ohms
R_REF_OUT	ADCPAR<SIZE>	DATA CONV	Output resistance of reference out	Yes	12	Ohms
R_REF_OUT	ADCMIC<SIZE>	DATA CONV	Output resistance of reference out	Yes	4K	Ohms
R1	T<SIZE>coupledX	TLINE	Per unit length resistance of first conductor	Yes	0	Ohms
R2	T<SIZE>coupledX	TLINE	Per unit length resistance of second conductor	Yes	0	Ohms
R3	T<SIZE>coupledX	TLINE	Per unit length resistance of third conductor	Yes	0	Ohms
R4	T<SIZE>coupledX	TLINE	Per unit length resistance of fourth conductor	Yes	0	Ohms
R5	T<SIZE>coupledX	TLINE	Per unit length resistance of fifth conductor	Yes	0	Ohms
RADIX	VECTOR<SIZE>	SPECIAL	Radix of values of the specified nodes	No	see .VECTOR	none
RCLOSED	Sw_tClose, Sw_tOpen	ANL_MISC	Closed state resistance	Yes	0.01	Ohms
RD	VMSSCCM	SWIT_RAV	Diode on resistance	Yes	0.000001	Ohms
RE	VMSSCCM	SWIT_RAV	Models ripple across esr of cap	Yes	0.000001	Ohms
REAL	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in REAL format in output file (YIN)	No	N	none

## PSpice Help

---

REF_AS SERTION	RELEASE<SI ZE>	DIG_MISC	Reference edge of clock	Yes	LH	none
REF_VO LT_GAIN	ADCMIC<SIZ E>	DATA CONV	Reference voltage gain	Yes	4	none
REFERENCE	CD4000_PWR , DIGIFPWR	SPECIAL	Negative power supply value	Yes	0	V

## PSpice Help

---

REL_CT R	A4N27, A4N28, CNY17-1, CNY17-2, PS2561, CNY17-3, H11A520, PS1001, A4N49A, BPW32, A4N48A, A4N47A, A4N32, A4N25A, MRD510, PS2565-1, PS2621, A4N33, H11A2, MRD500, H11AV1, MCT2, H11A3, PS2501-1, PS2601, H11AV2, H11A4, H11AV3, A4N25, A4N26, MCT2E, MLED96, PS2505-1, MOC1005, MOC1006, IL300, SLD1121VS	OPTO	Relative current transfer ratio	Yes	0.5	none
RELEASE ETIME	RELEASE<SI ZE>	DIG_MISC	Minimum time between signal inactive and clock edge	Yes	none	sec

## PSpice Help

REPEAT _VALUE	VPWL_RE_N_ TIMES, IPWL_F_RE_ N_TIMES, VPWL_ENH, VPWL_FILE, IPWL_ENH, IPWL_RE_N_ TIMES, VPWL_F_RE_ N_TIMES	SOURCE	Number of repetitions of specified window of signal	No	1	none
RESIST ANCE	BLDCMTR	MIX_MISC	Winding resistance	Yes	6	Ohm s
RI	CMLSCCM, CMSSCCM	SWIT_RAV	Current feedback coefficient	Yes	0.01	none
RIN	ADCPAR<SIZ E>	DATA CONV	Input resistance of analog input	Yes	50K	Ohm s
RIPPLE	HIPASS, LOPASS, BANDREJ, BANDPASS	ABM	Ripple specification	Yes	1dB	none
RM	VMSSCCM	SWIT_RAV	Resistance modeling the base storage effects	Yes	0.00 0001	Ohm s
RMPHIT E	VMLSCCM, VMLSDCM, VMSSCCM, VMCCMDCM	SWIT_RAV	External ramp height	Yes	2	V
ROFF	Sbreak	BREAKOUT	Off-state resistance	Yes	1000 000	Ohm s
RON	Sbreak	BREAKOUT	On-state resistance	Yes	1	Ohm s
ROPEN	Sw_tClose, Sw_tOpen	ANL_MISC	Open-state resistance	Yes	1Meg	Ohm s
ROW1	FTABLE	ABM	First row of data triplets (freq, mag, phase)	Yes	(0Hz , 0, 0)	triple t

## PSpice Help

ROW1	TABLE	ABM	First row of data pairs (input, output)	Yes	(0v, 0v)	pair
ROW2	FTABLE	ABM	Second row of data triplets (freq, mag, phase); rows are concatenated	No	(10Hz, -3, -30)	triple t
ROW2	TABLE	ABM	Second row of data pairs (input, output); rows are concatenated	No	(1v, 1v)	pair
ROW3	FTABLE	ABM	Third row of data triplets (freq, mag, phase); rows are concatenated	No	(20Hz, -6, -90)	triple t
ROW3	TABLE	ABM	Third row of data pairs (input, output); rows are concatenated	No	(2v, 4v)	pair
ROW4	FTABLE	ABM	Fourth row of data triplets (freq, mag, phase); rows are concatenated	No	(30Hz, -10, -120)	triple t
ROW4	TABLE	ABM	Fourth row of data pairs (input, output); rows are concatenated	No	(3v, 9v)	pair
ROW5	FTABLE	ABM	Fifth row of data triplets (freq, mag, phase); rows are concatenated	No	(40Hz, -15, -150)	triple t
ROW5	TABLE	ABM	Fifth row of data pairs (input, output); rows are concatenated	No	(4v, 16v)	pair
RSW	VMSSCCM	SWIT_RAV	Switch on resistance	Yes	0.00001	Ohm s
RTR_POLE_PAIRS	BLDCMTR	MIX_MISC	Number of north poles on the rotor	Yes	2	none

## PSpice Help

SCHOLD	DACSER<SIZE>	DATA CONV	SYNC to SCLK hold time	Yes	120n	sec
SCLK	DACSER<SIZE>	DATA CONV	SCLK cycle time	Yes	200n	sec
SCSET	DACSER<SIZE>	DATA CONV	SYNC to SCLK setup time	Yes	50n	sec
SE	CMLSCCM, CMSSCCM	SWIT_RAV	External ramp slope	Yes	100000	V/sec
SEC_R	3phase	ANL_MISC	Secondary winding resistance	Yes	0.1	Ohms
SECOND_NPAIRS	IPWL_RE_FOREVER, VPWL_RE_N_TIMES, VPWL_ENH, VPWL_RE_FOREVER, IPWL_ENH, IPWL_RE_N_TIMES	SOURCE	Second line containing PWL data, (time, analog value) pairs. Lines will be concatenated.	No	none	pairs
SET	R_VAR, C_VAR, POT	ANALOG, BREAKOUT	Position of slider between minimum and maximum value (linear interpolation)	Yes	0.5	none
SETUP TIME	SETUP<SIZE>	DIG_MISC	Setup time	Yes	none	sec
SIG_EDGE	RELEASE<SIZE>	DIG_MISC	Signal edge for constraint checking	Yes	LH	none
SIGNAL NAME	FileStim<SIZE>	SOURCE	Signal name in file	No	none	none
SIGNAL NAMES	VECTOR<SIZE>	SPECIAL	Names of signals which appear in the header of the vector file	No	node names	none
SN	CMSSCCM	SWIT_RAV	Current sense ramp slope	Yes	100000	V/sec

## PSpice Help

---

SPR	Relay_SPDT_ phy	ANL_MISC	Contact arm spring force	Yes	40	g_c m/ sec^ 2
SR	DACPAR<SIZ E>	DATA CONV	Slew rate of output	Yes	2.5M eg	V/ sec
STARTV AL	DIGCLOCK	SOURCE	First digital value of clock	Yes	0	none
STOP	HIPASS, LOPASS, BANDREJ, BANDPASS	ABM	Stopband attenuation	Yes	50dB	none
T_BOUN CE	BOUNCE, Relay_DPDT_ b, Relay_SPDT_ b	ANL_MISC, MIX_MISC	Bounce time (after contact is closed)	Yes	5m	sec
T_BREA K	BOUNCE, Relay_DPDT_ b, Relay_SPDT_ b, NO_BOUNCE, Relay_DPDT_ nb, Relay_SPDT_ nb	ANL_MISC, MIX_MISC	Time for contact to open when current is turned off/on	Yes	10m	sec
T_MAKE	BOUNCE, Relay_DPDT_ b, Relay_SPDT_ b, NO_BOUNCE, Relay_DPDT_ nb, Relay_SPDT_ nb	ANL_MISC, MIX_MISC	Time for contact to close when current is turned off/on	Yes	20m	sec



## PSpice Help

---

T1	VPWL, IPWL	SOURCE	Time for first PWL pair (should be 0)	Yes	none	sec
T1	Kcouple<SIZE >	TLINE	Reference designator of first coupled transmission line	Yes	T1	refdes
T10	VPWL, IPWL	SOURCE	Time for tenth PWL pair	Yes	none	sec
T2	VPWL, IPWL	SOURCE	Time for second PWL pair	Yes	none	sec
T2	Kcouple<SIZE >	TLINE	Reference designator of second coupled transmission line	Yes	T2	refdes
T3	VPWL, IPWL	SOURCE	Time for third PWL pair	Yes	none	sec
T3	Kcouple<SIZE >	TLINE	Reference designator of third coupled transmission line	Yes	T3	refdes
T4	VPWL, IPWL	SOURCE	Time for fourth PWL pair	Yes	none	sec
T4	Kcouple<SIZE >	TLINE	Reference designator of fourth coupled transmission line	Yes	T4	refdes
T5	VPWL, IPWL	SOURCE	Time for fifth PWL pair	Yes	none	sec
T5	Kcouple<SIZE >	TLINE	Reference designator of fifth coupled transmission line	Yes	T5	refdes
T6	VPWL, IPWL	SOURCE	Time for sixth PWL pair	Yes	none	sec
T7	VPWL, IPWL	SOURCE	Time for seventh PWL pair	Yes	none	sec
T8	VPWL, IPWL	SOURCE	Time for eighth PWL pair	Yes	none	sec
T9	VPWL, IPWL	SOURCE	Time for ninth PWL pair	Yes	none	sec

## PSpice Help

TABLE	EFREQ, GFREQ	ABM	Triplets (freq, mag, phase) for frequency table	Yes	(0,0, 0) (1Me g,- 10,9 0)	triple ts
TABLE	ETABLE, GTABLE	ABM	Pairs (input, output) for nonlinear table	Yes	(- 15,- 15) (15,1 5)	pairs
TAU	ZbreakN	BREAKOUT	Ambipolar recombination lifetime	No	7.1e- 6	sec
TBUSY	ADCMIC<SIZ E>	DATA CONV	BUSYbar low time	Yes	8u	sec
TBUSYRC	ADCMIC<SIZ E>	DATA CONV	TBUSYRC - BUSYbar Delay from R/Cbar low	Yes	83n	sec
TC1	IEXP, VEXP	SOURCE	Rise (fall) time constant	No	TST EP	sec
TC2	IEXP, VEXP	SOURCE	Fall (rise) time constant	No	TST EP	sec
TCLOSE	Sw_tClose	ANL_MISC	Time at which switch closes	Yes	0	sec
TD	IPULSE, VPULSE	SOURCE	Delay	No	0	sec
TD	T	ANALOG	Propagation delay	No	none	sec
TD	ISIN, VSIN	SOURCE	Delay	No	0	sec
TD1	IEXP, VEXP	SOURCE	Rise (fall) delay	No	0	sec
TD2	IEXP, VEXP	SOURCE	Fall (rise) delay	No	<td1 >+ TST EP	sec
TF	IPULSE, VPULSE	SOURCE	Fall time	No	TST EP	sec

## PSpice Help

THDGMN	DACPAR<SIZE>	DATA CONV	Data valid to Write hold time	Yes	10n	sec
THIRD_NPAIRS	IPWL_RE_FOREVER, VPWL_RE_N_TIMES, VPWL_ENH, VPWL_RE_FOREVER, IPWL_ENH, IPWL_RE_N_TIMES	SOURCE	Second line containing PWL data, (time, analog value) pairs. Lines will be concatenated.	No	none	pairs
TIMESTEP	STIM<SIZE>	SOURCE	Number of seconds per clock cycle, or step when using the "C" suffix	No	0	sec
TOL	DELAY	DIG_MISC	Tolerance for delay in percent	Yes	10	%
TOLERANCE	R, L, C	ANALOG	Tolerance (e.g. 5%) for monte carlo/worst case analysis	No	none	none
TOPEN	Sw_tOpen	ANL_MISC	Time at which switch opens	Yes	0	sec
TORQUE_CONST	BLDCMTR	MIX_MISC	Torque constant	Yes	300	g*cm/amp
TPADHMN	ADCPAR<SIZE>	DATA CONV	Minimum output delay high	Yes	10n	sec
TPADHTY	ADCPAR<SIZE>	DATA CONV	Typical output delay high	Yes	20n	sec
TPADLMN	ADCPAR<SIZE>	DATA CONV	Minimum output delay low	Yes	10n	sec
TPADLTY	ADCPAR<SIZE>	DATA CONV	Typical output delay low	Yes	20n	sec
TPHZMN	ADCMIC<SIZE>	DATA CONV	Minimum bus relinquish time H-Z	Yes	10n	sec
TPHZMX	ADCMIC<SIZE>	DATA CONV	Maximum bus relinquish time H-Z	Yes	83n	sec

## PSpice Help

---

TPHZTY	ADCMIC<SIZE>	DATA CONV	Typical bus relinquish time H-Z	Yes	35n	sec
TPHZTY	ADCPAR<SIZE>	DATA CONV	Typical output float delay H-Z	Yes	50n	sec
TPLZMN	ADCMIC<SIZE>	DATA CONV	Minimum bus relinquish time L-Z	Yes	10n	sec
TPLZMX	ADCMIC<SIZE>	DATA CONV	Maximum bus relinquish time L-Z	Yes	83n	sec
TPLZTY	ADCMIC<SIZE>	DATA CONV	Typical bus relinquish time L-Z	Yes	35n	sec
TPLZTY	ADCPAR<SIZE>	DATA CONV	Typical output float delay L-Z	Yes	50n	sec
TPZHMN	ADCMIC<SIZE>	DATA CONV	Minimum data access time Z-H	Yes	10n	sec
TPZHMX	ADCMIC<SIZE>	DATA CONV	Maximum data access time Z-H	Yes	83n	sec
TPZH TY	ADCMIC<SIZE>	DATA CONV	Typical data access time Z-H	Yes	35n	sec
TPZH TY	ADCPAR<SIZE>	DATA CONV	Typical data access time Z-H	Yes	50n	sec
TPZLMN	ADCMIC<SIZE>	DATA CONV	Minimum data access time Z-L	Yes	10n	sec
TPZLMX	ADCMIC<SIZE>	DATA CONV	Maximum data access time Z-L	Yes	83n	sec
TPZLTY	ADCMIC<SIZE>	DATA CONV	Typical data access time Z-L	Yes	35n	sec
TPZLTY	ADCPAR<SIZE>	DATA CONV	Typical data access time Z-L	Yes	50n	sec
TR	IPULSE, VPULSE	SOURCE	Rise time	No	TST EP	sec
TRAN	ISRC, VSRC	SOURCE	Transient specification - EXP(), PULSE(), PWL(), SFFM() OR SIN()	No	none	none

## PSpice Help

TRAN	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write TRAN analysis results to output file (YIN)	No	Y	none
TSC0	ADCSER<SIZ E>	DATA CONV	CONVST/EOC* to CLOCK Skew (typical)	Yes	40n	sec
TSF	IPWL_RE_FO REVER, VPWL_F_RE_ FOREVER, VPWL_RE_N_ TIMES, IPWL_F_RE_ N_TIMES, IPWL_F_RE_ FOREVER, VPWL_ENH, VPWL_RE_F OREVER, VPWL_FILE, IPWL_ENH, IPWL_RE_N_ TIMES, VPWL_F_RE_ N_TIMES	SOURCE	Time scaling factor (multiplies time values)	No	1	none
TSUDGM N	DACPAR<SIZ E>	DATA CONV	Data valid to Write setup time	Yes	90n	sec
TTRAN	Sw_tClose	ANL_MISC	Transition time for switch to close	Yes	1u	sec
TTRAN	Sw_tOpen	ANL_MISC	Transition time for switch to open	Yes	1u	sec
TURN S	Relay_SPDT_ phy	ANL_MISC	Number of turns	Yes	100	none
V_RATI O	3phase	ANL_MISC			1	

## PSpice Help

V1	VPULSE, VEXP	SOURCE	First voltage level	Yes	none	V
V2	VPULSE, VEXP	SOURCE	Second voltage level	Yes	none	V
V1	VPWL	SOURCE	First PWL point (0s, current value)	Yes	none	V
V2	VPWL	SOURCE	Second PWL point (time, current value)	No	none	V
V3	VPWL	SOURCE	Third PWL point (time, current value)	No	none	V
V4	VPWL	SOURCE	Fourth PWL point (time, current value)	No	none	V
V5	VPWL	SOURCE	Fifth PWL point (time, current value)	No	none	V
V6	VPWL	SOURCE	Sixth PWL point (time, current value)	No	none	V
V7	VPWL	SOURCE	Seventh PWL point (time, current value)	No	none	V
V8	VPWL	SOURCE	Eighth PWL point (time, current value)	No	none	V
V9	VPWL	SOURCE	Ninth PWL point (time, current value)	No	none	V
V10	VPWL	SOURCE	Tenth PWL point (time, current value)	No	none	V
VALLEY V	VMLSCCM, VMLSDCM, VMCCMDCM	SWIT_RAV	Valley voltage of external ramp	Yes	1	V
VALUE	IC1, IC2	SPECIAL	.IC value	Yes	0	V
VALUE	NODESET1, NODESET2	SPECIAL	.NODESET value	Yes	0	V
VALUE	R, Rbreak	ANALOG, BREAKOUT	Resistance	Yes	1K	Ohm s
VALUE	L, Lbreak	ANALOG, BREAKOUT	Inductance	Yes	1m	H

## PSpice Help

VALUE	C, Cbreak	ANALOG, BREAKOUT	Capacitance	Yes	1n	F
VAMPL	VSIN	SOURCE	Voltage amplitude of sinusoid (transient analysis only)	Yes	none	V
VAMPL	VSFFM	SOURCE	Voltage amplitude of SFFM (transient analysis only)	Yes	none	V
VAP	VMSSCCM, CMSSCCM	SWIT_RAV	Voltage across terminal A P	Yes	20	V
VCC1	ECL_10K_PW R, ECL_100K_P WR	SPECIAL	First power supply voltage for ECL supply	Yes	0	V
VCC2	ECL_10K_PW R, ECL_100K_P WR	SPECIAL	Second power supply voltage for ECL supply	Yes	0	V
VCOCOE FF	QRLSZCS	SWIT_RAV	Coefficient for voltage to frequency conversion	Yes	20k	none
VEE	ECL_100K_P WR	SPECIAL	VEE	Yes	-4.5	V
VEE	ECL_10K_PW R	SPECIAL	VEE	Yes	-5.2	V
VNOM	BULB	OPTO	Nominal operating voltage	Yes	120	V
VOFF	VSIN, VSFFM	SOURCE	Offset voltage	Yes	none	V
VOFF	Sbreak	BREAKOUT	Voltage lower threshold	Yes	0	V
VOLTAGE	CD4000_PWR , DIGIFWR	SPECIAL	Digital power supply voltage	Yes	5	V
VON	Sbreak	BREAKOUT	Voltage upper threshold	Yes	1	V

## PSpice Help

VSF	IPWL_RE_FO REVER, VPWL_F_RE_ FOREVER, VPWL_RE_N_ TIMES, IPWL_F_RE_ N_TIMES, IPWL_F_RE_ FOREVER, VPWL_ENH, VPWL_RE_F OREVER, VPWL_FILE, IPWL_ENH, IPWL_RE_N_ TIMES, VPWL_F_RE_ N_TIMES	SOURCE	Voltage scaling factor (multiplies voltages)	No	1	none
VSINK	DACPAR<SIZ E>	DATA CONV	Output sink voltage (@ISINK)	Yes	-0.1	V
VSOURCE	DACPAR<SIZ E>	DATA CONV	Output source voltage (@ISOURCE)	Yes	10	V
VTT	ECL_10K_PW R, ECL_100K_P WR	SPECIAL	VTT, termination voltage	Yes	-2	V
W	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Length	No	DEF W	m
WB	ZbreakN	BREAKOUT	Metallurgical base width	No	9.0e- 5	m



## PSpice Help

WHEN	RELEASE<SIZE>, HOLD<SIZE>, CONSTRAINT <SIZE>, SETUP<SIZE>, WIDTH_HI, WIDTH_LO, MINFREQ, MAXFREQ	DIG_MISC	Used to define a boolean expression describing a condition for a constraint checker primitive	No	none	none
WIDTH	WIDTH_HI	DIG_MISC	Minimum width high constraint	Yes	none	sec
WIDTH	WIDTH_LO	DIG_MISC	Minimum width low constraint	Yes	none	sec
WIDTH	STIM1	SOURCE	Number of bits	Yes	1	none
WIDTH	STIM16	SOURCE	Number of bits	Yes	16	none
WIDTH	STIM4	SOURCE	Number of bits	Yes	4	none
WIDTH	STIM8	SOURCE	Number of bits	Yes	8	none
WIDTH_ MIN_HI	ADCPAR<SIZE>	DATA CONV	Minimum width high	Yes	45n	sec
WIDTH_ MIN_LO	ADCPAR<SIZE>	DATA CONV	Minimum width low	Yes	45n	sec
WRMINLO	DACPAR<SIZE>	DATA CONV	Minimum width low of WR	Yes	90n	sec
XFORM	ELAPLACE, GLAPLACE	ABM	Laplace transform	Yes	1/s	none
Z0	T	ANALOG	Characteristic impedance	Yes	none	Ohms



## **PSpice Help**

### **Menu Bar**

Click each menu bar item to see its pull-down menu.

### **Open button**

Click to open a data file.

### **Append button**

Click to add data from another data file to the existing data.

### **Print button**

Click to print one copy of the current plot window.

### **Cut button**

Click to cut the currently selected item.

### **Copy button**

Click to copy the currently selected item.

### **Paste button**

Click to paste the currently copied or cut item.

### **Zoom buttons**

Click to zoom the displayed view in or out.

### **View Area button**

Click to zoom in to the selected area.

### **View Fit button**

Click to zoom out so that all the data is visible in the selected plot.

### **X Scale button**

Click to toggle the X axis between log and linear scaling.

### **Fourier Transform button**

Click to display the Fourier Transform of all traces in the selected plot.

### **Performance Analysis button**

Click to enter Performance Analysis.

### **Y Scale button**

Click to toggle the Y axis between log and linear scaling.

### **Add Trace button**

Click to add a trace.

### **Eval Measurement Function button**

Click to evaluate the measurement function or expression of measurement functions on data from one PSpice run.

### **Text button**

Click to type text to place on the displayed plot.

### **Display Cursor button**

Click to turn the data cursor on or off.

### **Peak button**

Click to move the cursor to the next peak.

### **Trough button**

Click to move the cursor to the next trough.

### **Slope button**

Click to move the cursor to the next point of maximum slope.

### **Min button**

Click to move the cursor to the minimum Y value.

### **Max button**

Click to move the cursor to the maximum Y value.

### **Point button**

Click to move the cursor to the next data point.

### **Search button**

Click to display the Search dialog box.

### **Next Transition button**

Click to move the cursor to the next digital transition.

### **Previous Transition button**

Click to move the cursor to the previous digital transition.

### **Mark Data Points button**

Click to view data points on the plot.

### **X Axis Data Range**

Click Auto Range to let Probe set the range, or click User Defined and type a specific data range to display.

### **Processing Options**

Click to select either Fourier Transform or Performance Analysis:

- Fourier Transform displays a line graph.
- Performance Analysis displays a histogram.

### **Variable**

Click to display the Axis Variable dialog box.

### **Text Area**

The definition of the measurement function is shown here.

### **This Measurement Function is Saved in the File**

The location of the measurement function on your system.

### **Use Symbols**

Click to select when and how Probe uses symbols when displaying traces.

### **Use Scroll Bars**

Click to set scroll bar usage.

### **Trace Color Scheme**

Click to set how Probe uses color to display traces.

### **Highlight Error States**

Select to enable automatic highlighting of error states when digital traces are displayed.

### **Number of Histogram Divisions**

Type the number of histogram divisions to be used when Probe displays a Performance Analysis result from a Monte Carlo simulation.

### **Number of Cursor Digits**

Type the number of digits Probe will display when the cursor position is shown.

### **New Name**

Type a name to save this display as.

### **List**

Click to select one of the available displays shown here.

### **Save**

Click to save the display with the assigned name.

### **Save To**

Click to save the display to a specific place.

### **Copy To**

Click to copy the selected display.

### **Delete**

Click to delete the selected display.

You can only delete local displays using the Delete button. To delete a display from a remote or global .PRB file, click Delete From.

### **Delete From**

Click to delete a display from a remote or global .PRB file.

### **Restore**

Click to use the selected display.

### **Load**

Click to load another file with displays.

### **Close**

Click to close the Save/Restore dialog box.

### **Box**

Type the search command in this box.

### **Cursor To Move**

Click to select the cursor to search.

### **List**

Click to select one of the available traces and measurement functions listed here.

### **Analog**

Click to select whether analog node names are listed.



### **Digital**

Click to select whether digital node names are listed.

### **Voltages**

Click to select whether voltage node names are listed.

### **Currents**

Click to select whether current node names are listed.

### **Alias Names**

Click to select whether alias node names are listed.

### **Internal Subcircuit Nodes**

Click to select whether Internal Subcircuit Nodes names are listed.

### **Measurement Functions**

Click to select whether measurement functions are listed.

### **Trace Command**

Type the command or expression to use with the selected trace or measurement function.

### **Margins**

Type the margins in inches in the text boxes.

### **Plots Per Page**

Click to select the number of plots to print on each page.

### **Orientation**

Click to select the orientation of the page.

### **Cursor Information**

Click to select where the cursor information is printed.

### **Draw Border**

Click to select whether a border is used when the trace is printed.

### **Draw Plot Title**

Click to select whether the plot title is printed on the page.

### **Header**

Click to display the Header dialog box.

### **Footer**

Click to display the Footer dialog box.

### **Printer Setup**

Click to display the Printer Setup dialog box.

### **Printer Select**

Click to display the Printer Select dialog box.

### **Set Default**

Click to set the current settings as the default settings.

### **Reset Default**

Click to reset the current settings to the default settings.

### **New Measurement Function Name**

Type the name of the new measurement function.

### **File to Keep Measurement Function In**

Click to select the location of the saved measurement function.

### **Box**

Type the name of the window title.

### **Definition**

Type the name and definition of the macro.

### **List**

The available macros and the definitions are listed here.

### **Save**

Click to save the current macro and definition.

### **Save To**

Click to save the current macro and definition to a specific location on your system.

### **Delete**

Click to delete the current macro and definition.

You can only delete local macros using the Delete button. To delete a macro from a remote or global .PRB file, click Delete From.

### **Delete From**

Click to delete a macro and definition from remote or global .PRB file.

### **Load**

Click to load another file with macros and definitions.

### **Close**

Click to close the Macros dialog box.

### **Left Side**

Type the text for the left side of the header.

### **Center**

Type the text for the center of the header.

### **Right Side**

Type the text for the right side of the header.

### **OK**

Click to use the current header definitions and close the Header dialog box.

### **Cancel**

Click to close the window and abandon your changes.

### **Set Default**

Click to set the current header settings as the default.

### **Reset Default**

Click to reset the header to the default settings.

### **Date Run button**

Click to insert the date in the selected header area.

### **Header Date & Time button**

Click to insert the date and time in the selected header area.

### **Header Time Run button**

Click to insert the time the trace was run in the selected header area.

### **Header Temperature button**

Click to insert the temperature information in the selected header area.

### **Left Side**

Type the text for the left side of the footer.

### **Center**

Type the text for the center of the footer.

### **Right Side**

Type the text for the right side of the footer.

### **OK**

Click to use the current footer settings and close the Footer dialog box.

### **Cancel**

Click to close the window and abandon your changes.

### **Set Default**

Click to set the current footer settings as the default.

### **Reset Default**

Click to reset the footer to the default settings.

### **Date Run button**

Click to insert the time the trace was run in the selected footer area.

### **Date & Time button**

Click to insert the date and time in the selected header area.

### **Time Run button**

Click to insert the time the trace was run in the selected header area.

### **Temperature button**

Click to insert the temperature information in the selected header area.

### **List**

The available measurement functions are listed.

### **New**

Click to create a new measurement function.

### **Copy**

Click to copy a measurement function.

### **View**

Click to view a measurement function definition.

### **Edit**

Click to edit an existing measurement function.

### **Delete**

Click to delete the selected measurement function.

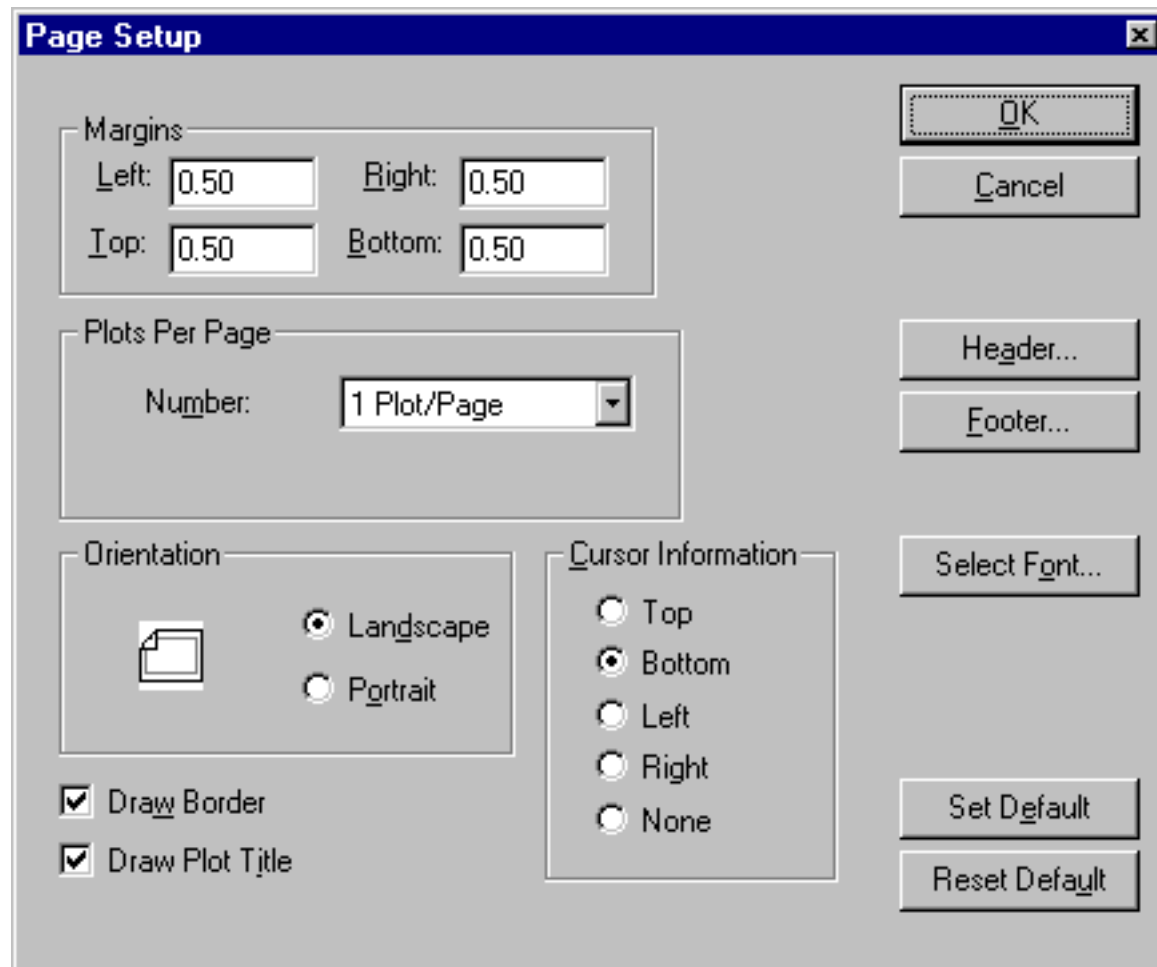
### **Eval**

Click to evaluate the selected measurement function.

### **Load**

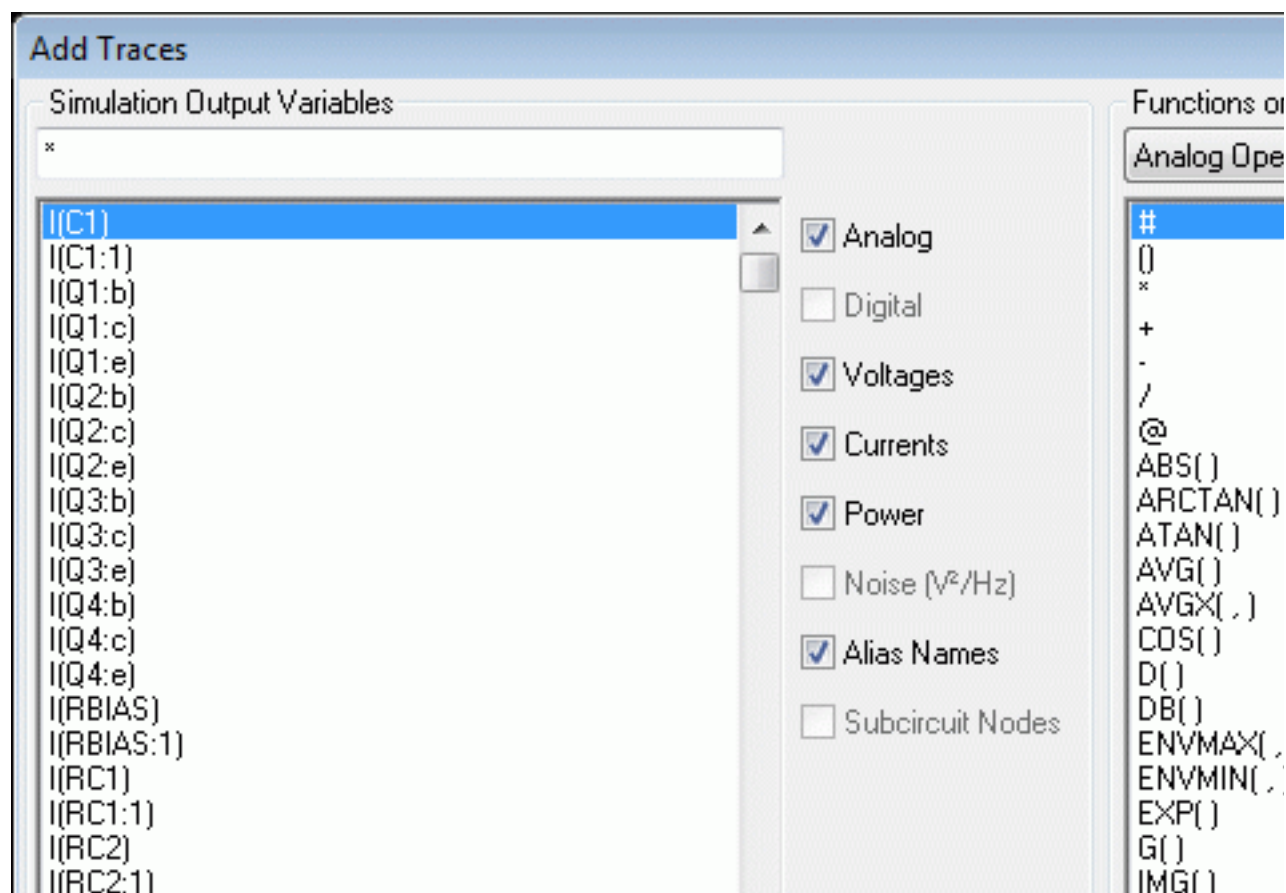
Click to load another file with measurement function definitions.

### Page Setup dialog box

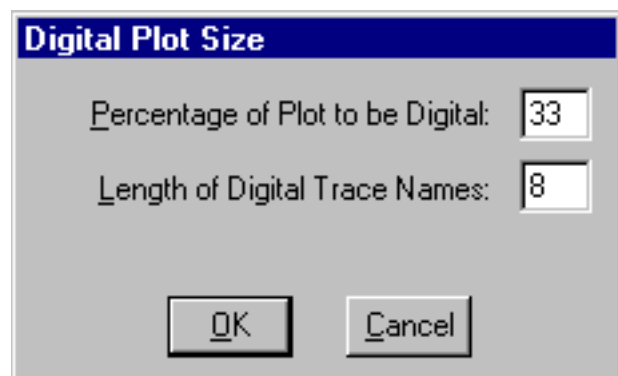




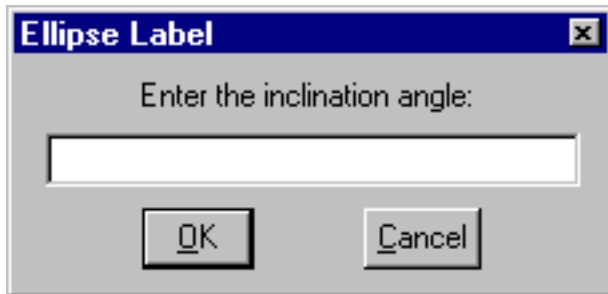
## Add Trace dialog box



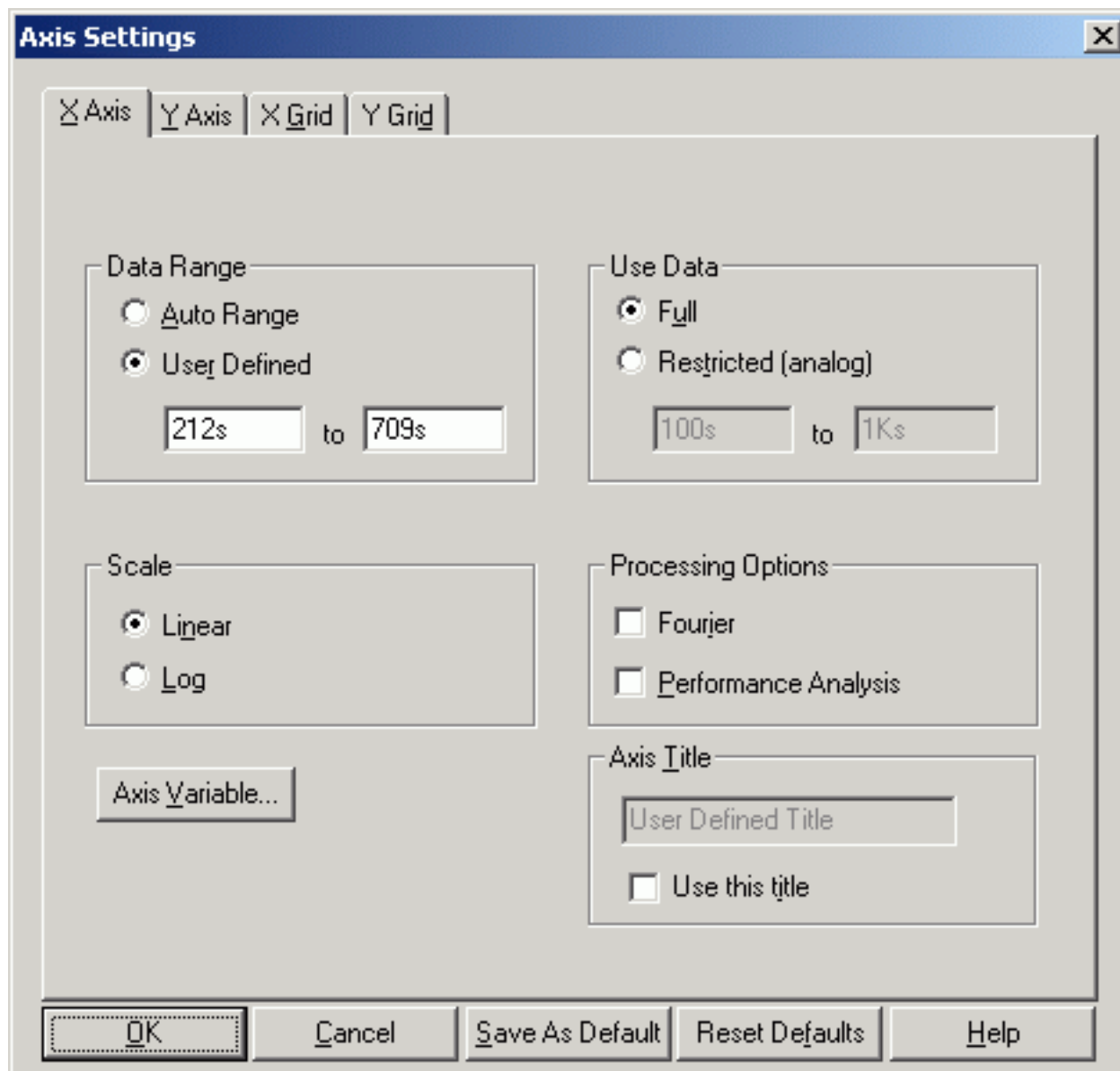
## Digital Size dialog box



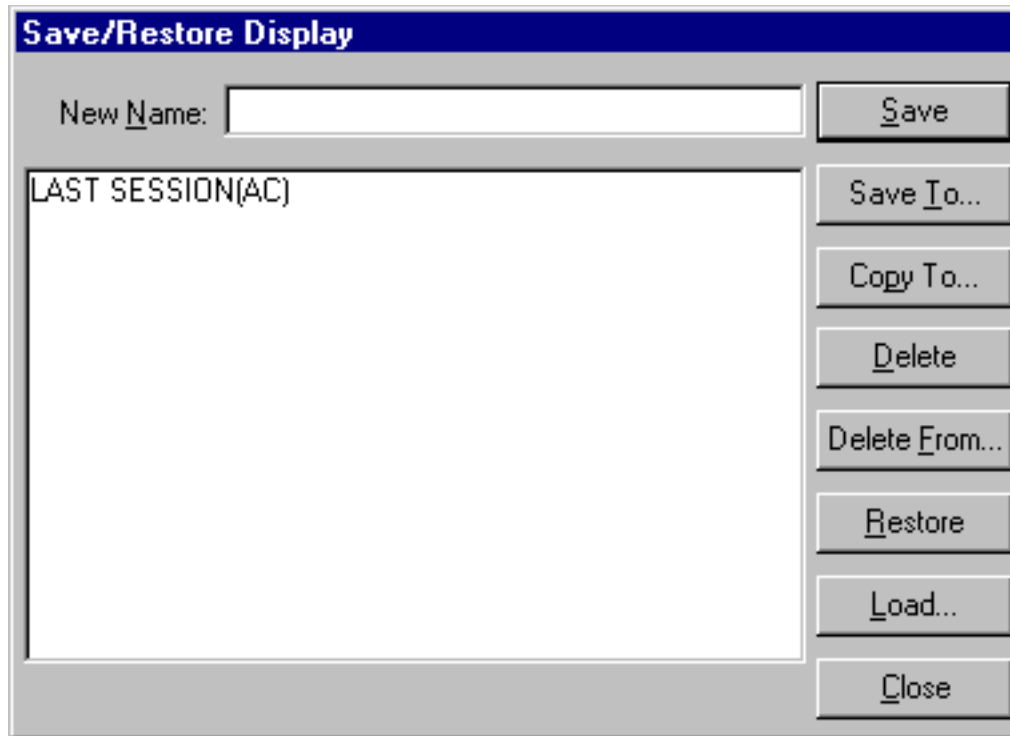
### Ellipse dialog box



## X Axis tab



### Display Control dialog box



### Open File button



### Append File button



### Printer button



### Zoom In button



### Zoom Out button



### Area button



### Cursor button



### Copy button



### Fourier Transform button



### Text button



### Paste button



### Cut button



### Fit button



### Peak button



### Trough button



### Slope button



### Point button



### Min button



### Max button



### Next Transition button



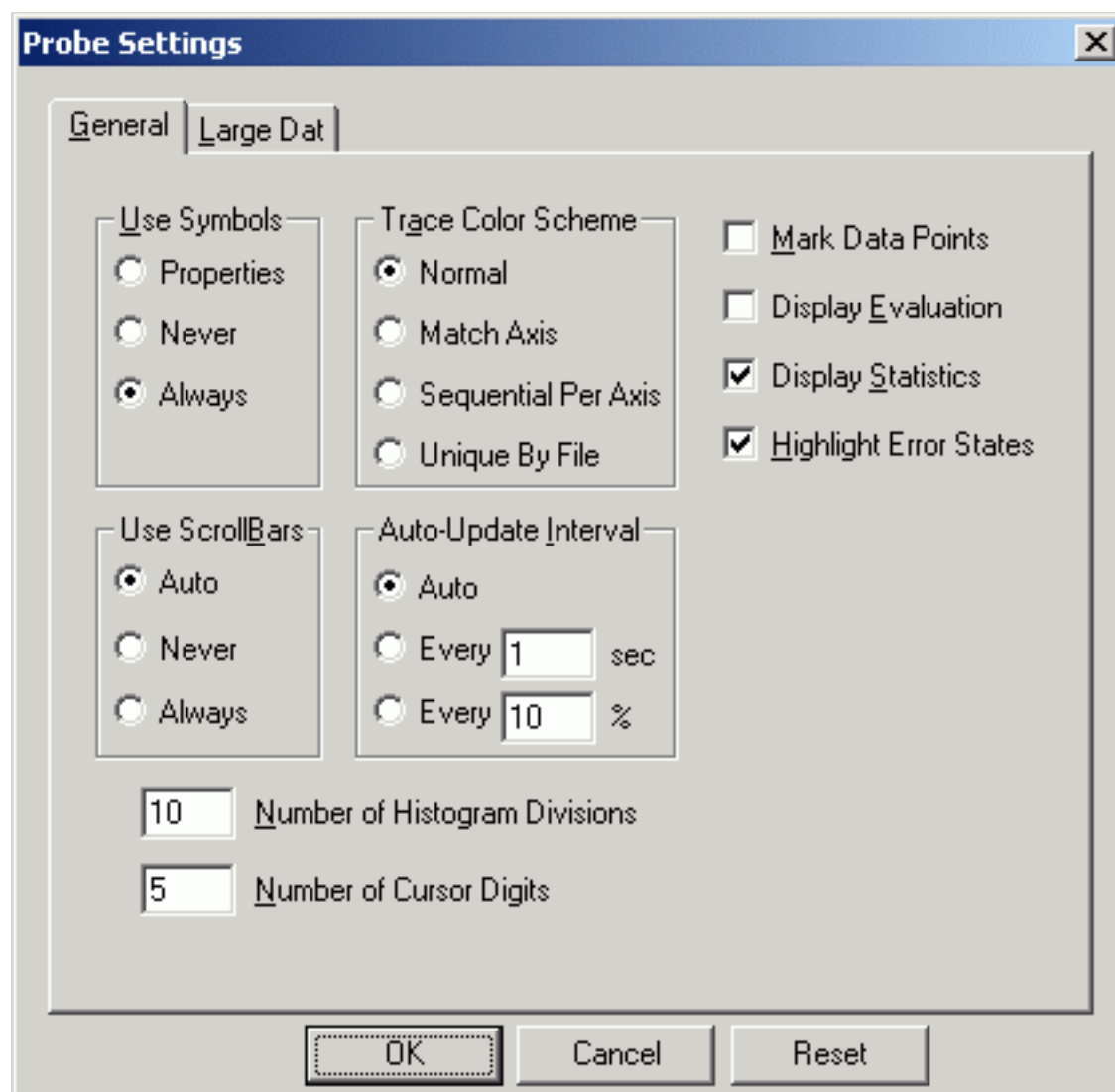
### Previous Transition button



### Add Trace button

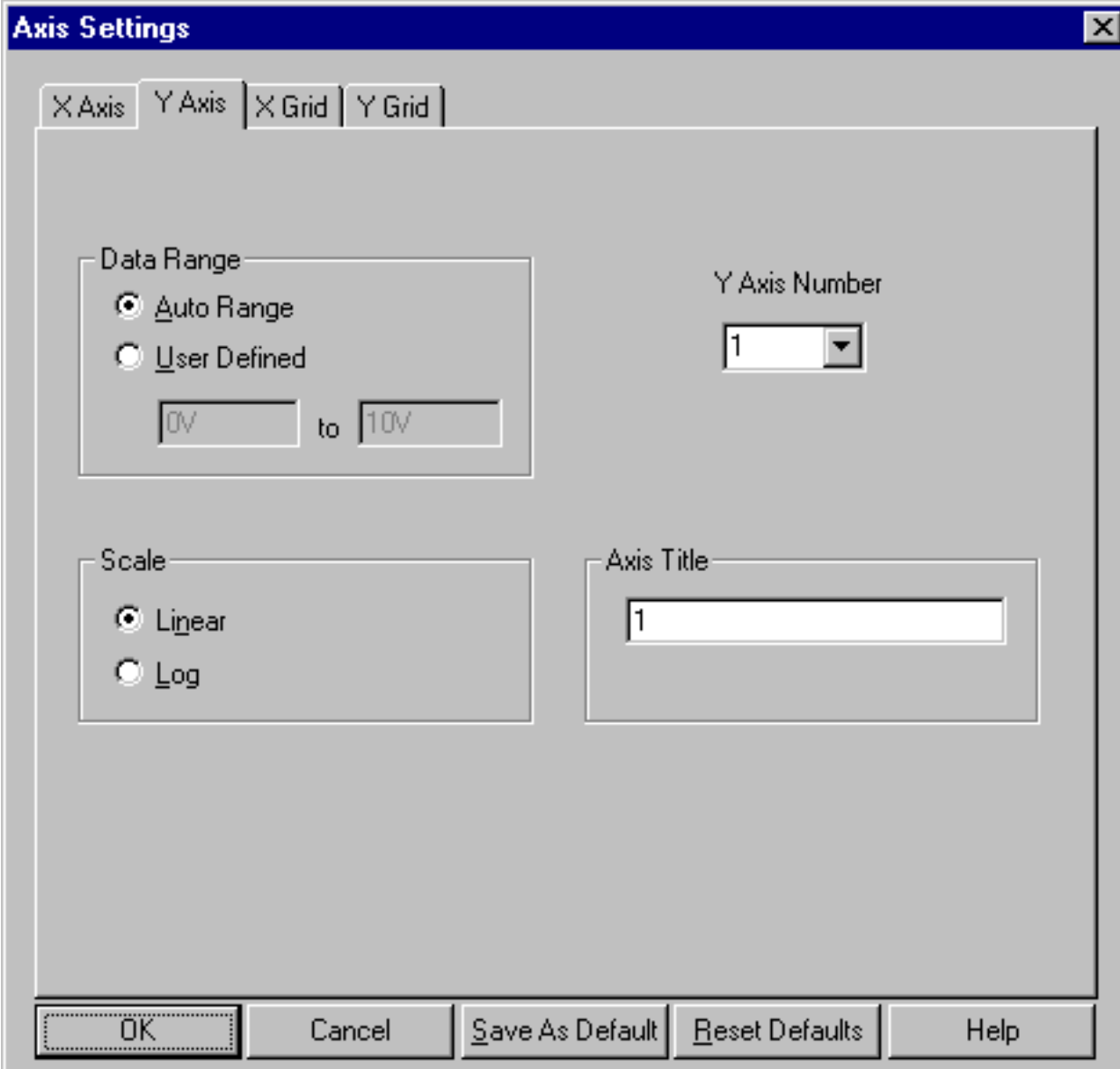


### Probe Options dialog box





Y Axis tab



The image shows the 'Axis Settings' dialog box with the 'Y Axis' tab selected. The dialog has a title bar with a close button. Below the title bar are four tabs: 'X Axis', 'Y Axis' (selected), 'X Grid', and 'Y Grid'. The main area contains four sections: 'Data Range' with radio buttons for 'Auto Range' (selected) and 'User Defined' (with input fields for '0V' and '10V'); 'Y Axis Number' with a dropdown menu showing '1'; 'Scale' with radio buttons for 'Linear' (selected) and 'Log'; and 'Axis Title' with a text input field containing '1'. At the bottom are five buttons: 'OK', 'Cancel', 'Save As Default', 'Reset Defaults', and 'Help'.

**Axis Settings**

X Axis Y Axis X Grid Y Grid

Data Range

☒ Auto Range

☐ User Defined

0V to 10V

Y Axis Number

1

Scale

☒ Linear

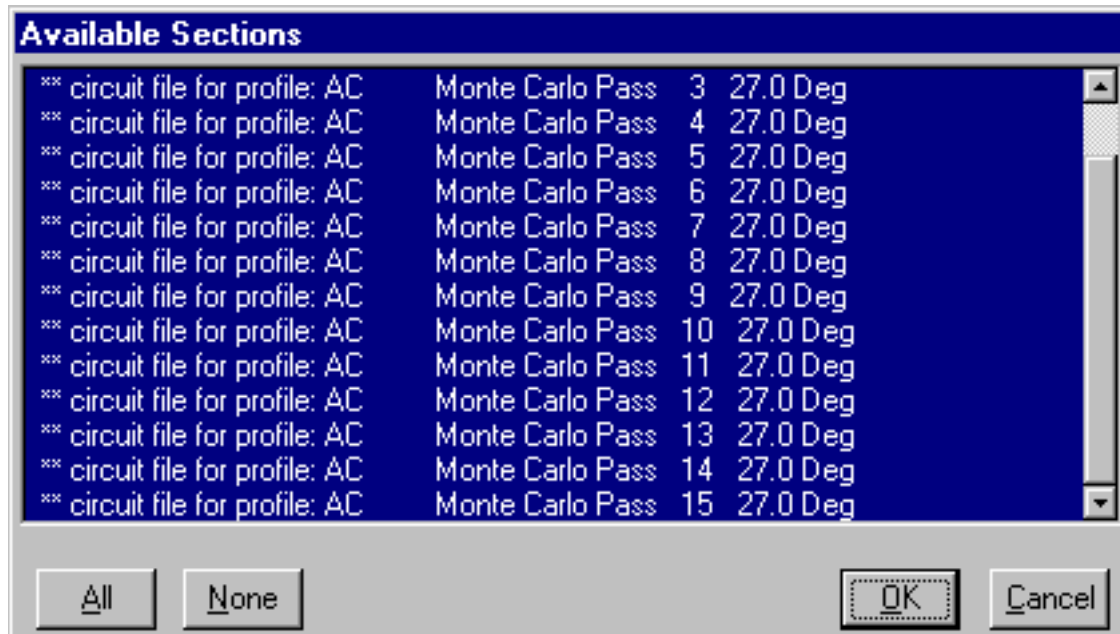
☐ Log

Axis Title

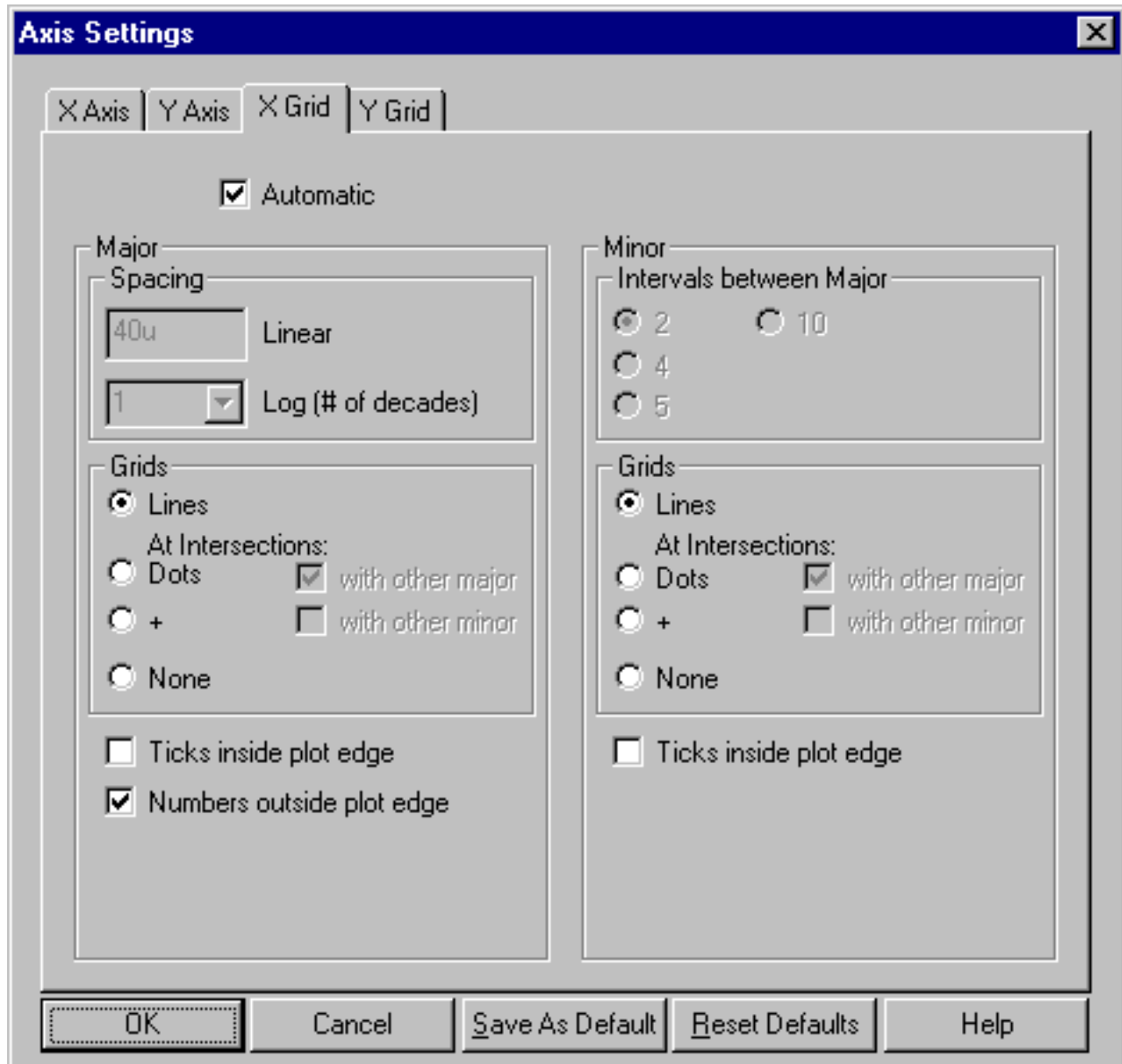
1

OK Cancel Save As Default Reset Defaults Help

### Available Sections dialog box

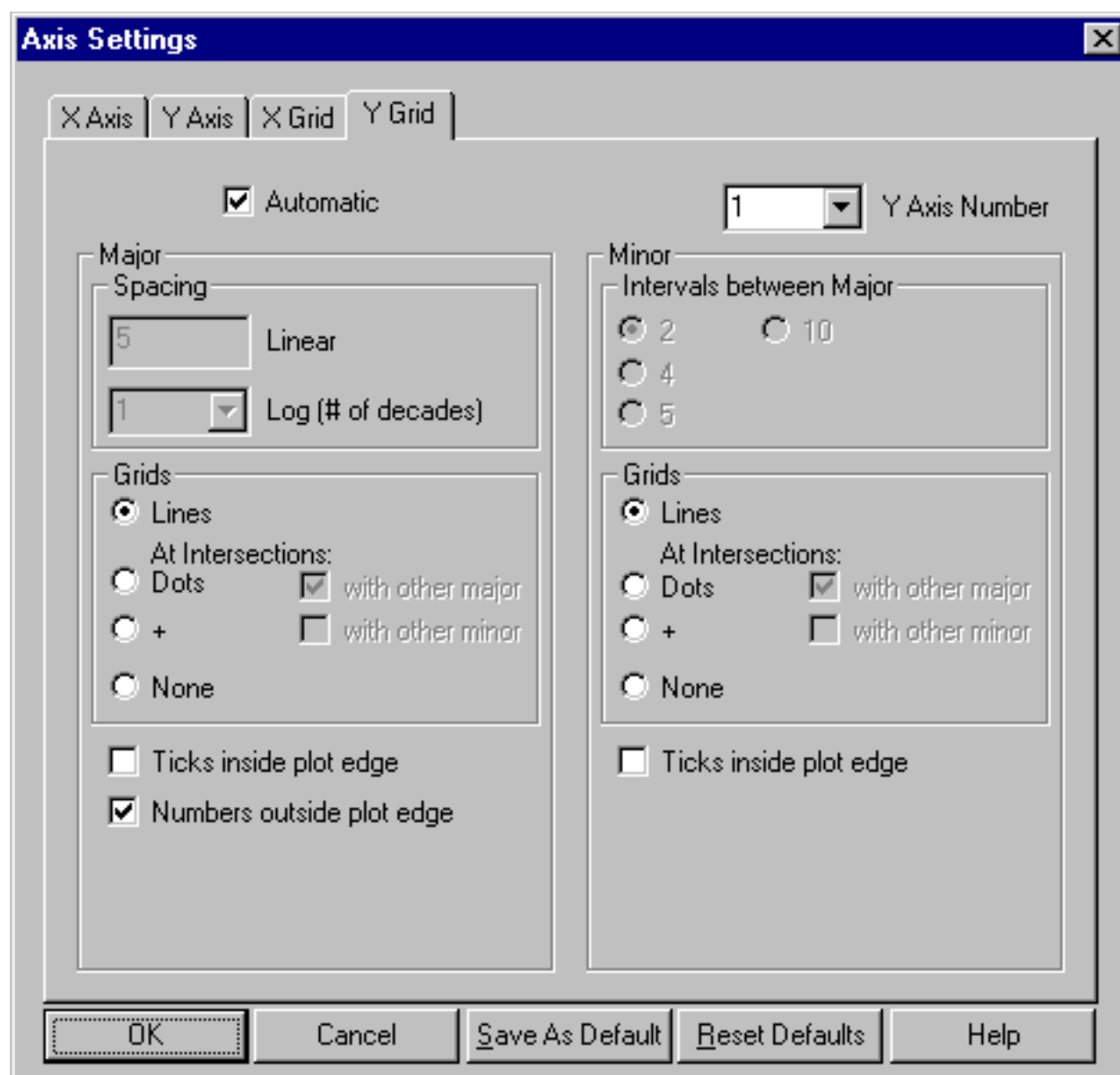


## X Grid tab



## Y Grid tab

{bmc BM101.SHG}



## **PSpice Help**

### **Voltage source**

This sets the source's voltage to the sweep value during the sweep.

In the Name text box, type a reference designator of an independent voltage source, such as V1.

### **Current source**

This sets the source's current to the sweep value during the sweep.

In the Name text box, type the name of an independent current source.

### **Global parameter**

This sets the value to the sweep value and all expressions are re-evaluated.

In the Parameter name text box, type a global parameter name.

### **Model parameter**

This sets the parameter in the model to the sweep value.

From the Model type list, select a model type. In the Model Name text box, type the model name. In the Parameter name text box, type a parameter name.

### **Temperature**

This sets the temperature to the sweep value. For each value in the sweep, the model parameters of all the circuit components are updated to that temperature.

### **Linear**

Indicates a linear sweep. The swept variable is swept linearly from the starting to the ending value. The Increment value is the step size.

### **Octave**

Indicates sweep by octaves. The sweep variable is swept logarithmically by octaves.

### **Decade**

Indicates sweep by decades. The sweep variable is swept logarithmically by decades.

### **Value list**

Uses a list of values. In this case, there are no start and end values. Instead, the numbers you type in the Values List text box are the values that the sweep variable is set to.

### **YMAX**

Finds the greatest difference in each waveform from the nominal run.

### **MAX**

Finds the maximum value of each waveform.

### **MIN**

Finds the minimum value of each waveform.

### **RISE\_EDGE**

Finds the first occurrence of the waveform crossing above the threshold value. Type a threshold value in the Threshold value text box.

### **FALL\_EDGE**

Finds the first occurrence of the waveform crossing below the threshold value. Type a threshold value in the Threshold value text box.

### **Low**

Specifies the lower limit of the range over which the function is evaluated.

### **Hi**

Specifies the upper limit of the range over which the function is evaluated.

### **None**

Forces the nominal run to produce output.

### **All**

Forces all output to be generated, including the nominal run.

### **First**

Generates output only during the first n runs. Type the value for n in the Runs text box.

### **Every**

Generates output every nth run. Type the value for n in the Runs text box.

### **Runs**

Performs an analysis and generates output only for listed runs. Up to 25 values can be specified in the Runs text box. Prints out at the beginning of each run the model parameter values actually used for each component during that run.

### **Random number seed**

Defines the seed for the random number generator within the Monte Carlo analysis. You must type an odd integer ranging from 1 to 32767. If the seed value is not set, it defaults to 17533.

### **Output All**

Requests output from the sensitivity runs, after the first run. The sensitivity and worst case runs are done with variations on model parameters as specified by the DEV and LOT tolerances. The default is to vary by BOTH.

### **Vary both DEV and LOT, Vary DEV, Vary LOT**

Vary DEV and Vary LOT limit the devices analyzed to only the device types that have a DEV tolerance or a LOT tolerance.

Vary both DEV and LOT includes all the device types in the analysis.

### **Limit devices to type(s)**

In the text box, type a list of the specific device types you want included in the analysis. The list is a string containing the initial letters of PSpice primitives.

### **Primary Sweep value**

The first DC sweep value at which the bias point is to be saved. If there is only one sweep value, type a value in the Primary Sweep value text box. If there are two sweep variables, then Primary Sweep value specifies the first sweep value.

### **Secondary Sweep value**

The second DC sweep value at which the bias point is to be saved. If there is only one sweep value, type a value in the Primary Sweep value text box. If there are two sweep variables, then Secondary Sweep value specifies the second sweep value.

### **Parametric Sweep value**

The step value at which the bias point is to be saved for parametric analyses.

### **Number of runs**

The number of the Monte Carlo or worst case analyses run for which the bias point is to be saved.

### **Use distribution**

This option is the default distribution for Monte Carlo deviations.

From the list, select Uniform or Gaussian, or click the Distributions button to enter your own distribution.

### **Initialize flip-flops to X, 0, or 1**

If set to X, all flip-flops and latches produce an X (unknown state) until explicitly set or cleared, or until a known state is clocked in.

If set to 0, all such devices are cleared.



If set to 1, all such devices are preset.

### **default propagation delay mode**

You can change the mode for an individual part in your design by changing the part's MNTYMXDLY property. By default, this part value is set to 0, which tells PSpice to use the default value set in the Options tab.

Enter this...	To set this mode as the default
1	minimum
2	typical
3	maximum
4	worst-case (min/max)

### **Temperature Sweep temperature**

Defines the temperature at which the bias point is to be saved for temperature analyses.

### **Include detailed bias point information for nonlinear controlled sources and semiconductors**

This option saves the small-signal (linearized) parameters of all the nonlinear controlled sources and all the semiconductor devices to the output file.

This is equivalent to the .OP (bias point) PSpice circuit file command.

### **Perform Sensitivity analysis**

In the Output Variable(s) text box, type

This option is equivalent to the .SENS (DC sensitivity) PSpice circuit file command.

### **Calculate small-signal DC gain**

This option calculates the small-signal DC gain by linearizing the circuit around the bias point.

In the From Input Source Name text box, type

In the To Output Variable text box, type

This option is equivalent to the .TF (small-signal DC transfer function) PSpice circuit file command.

### Data collection parameters

Choose this option... To do this...

All voltages, currents, and digital states	Save Probe data for all nodes and devices in the circuit. This is the default.
All but internal subcircuit data	Save data for all nodes and devices, except internal subcircuit nodes and devices.
At Markers only	Save the Probe data at those nodes and devices where markers are placed.
None	Disable Probe data collection.

### Text Data File Format (CSDF)

Select the Save data in the CSDF format (.CSD) option to write Probe data in text format rather than binary format. This option is not available if the Run Probe During Simulation option is used.

### Introduction to device equations

The purpose of the Device Equations option is to change the built-in model equations for one or more of the semiconductor devices (GaAsFET, Diode, Junction FET, MOSFET, Bipolar transistor, and IGBT). This means you can extend PSpice to support user-defined or proprietary native device models.

This option is not an addition to PSpice : it is a different packaging of the program that includes the source code for the device model subroutines. You need a Device Equations license to modify and extend PSpice code, but you do not need a Device Equations license to use the modified code.

There are several kinds of changes that can be made using the Device Equations option. These include, in ascending order of complexity:

- Changing a parameter name
- Giving a parameter an alias
- Adding a parameter
- Changing the device equations
- Adding a new device
- Specifying new internal device structure

You need a supported C++ compiler to compile Device Equations extensions; for Windows 95/98 and NT, you need Microsoft Visual C++ 6.0 or later.

Device Equations extensions are implemented using a dynamic-link library, which means you can share your models with other users by distributing just a DLL.

If you want to run PSpice on Windows 95 or NT with a Device Equations DLL developed by someone else, then you do not need a compiler or a Device Equations license. Just copy the DLL into the directory with your PSpice program file. For more information, see [Simulating with the device equations option](#).

### Making device model changes

To get started, look at the files M.H and MOS.CPP, which implement the MOSFET equations. The other devices have similar structures.

M.H contains two important class definitions:

- the class for the MOS transistor (class M\_Device)

- the class for the MOS model (class M\_Model)

During read-in, the simulator creates an instance of the transistor class for every MOSFET in the circuit and an instance of the model class for every .MODEL statement of type NMOS or PMOS. The transistor instance is set up using information particular to that transistor, such as the nodes to which it is connected, its length and width, and the locations of its entries in the circuit's conductance matrix. All parameters of the model object are set up using the values from the .MODEL statement, if one exists; otherwise, the default values are used.

The transistor object corresponds to the LOC, LOCV, and LX tables in U.C. Berkeley SPICE2. The model object corresponds to the LOC and LOCM tables in SPICE.

**Note:** Do not change the transistor object (class M\_Device), except when changing the internal device topology. It is included only to allow compiling of MOS.CPP.

The simulator needs to associate each entry in the model class with a model parameter name (and default value) in the .MODEL statement. You can accomplish this by using the ASSOCIATE macro. Just below the device class in M.H there is a list of all the parameters, each in an ASSOCIATE macro. The occurrence of ASSOCIATE binds together the class entry, the parameter name, and the default value. The read-in section of the simulator uses this information to parse the .MODEL statement.

For more details on how to change parameters, click the following:

Changing a parameter name

Giving a parameter an alias

Adding a parameter

### Changing a parameter name

This is the easiest change. Find the parameter in the list of ASSOCIATE macros. Change the parameter's name (last item on the line) and/or the default value (middle item). The names and defaults of the model parameters that are supplied can be changed, as well as those parameters that are added.

When the simulator runs, it prints the parameter values for each .MODEL statement unless the NOMOD option is used in the .OPTIONS statement. Normally only parameters which have not been defaulted are listed. A parameter can be forced to be listed, whether or not it has been defaulted, by preceding its name using an asterisk (\*). For example, VTO is listed that way in M.H.

### Giving a parameter an alias

Sometimes a parameter requires an alternate name (an alias). Several bipolar model parameters, such as ISE, already have alternate names. The alias for ISE is C2. Look in Q.H at the occurrences of the parameters ISE and C2 in the ASSOCIATE macros for an example of how this is accomplished. There is only one entry in the model class (Q\_ise) for the parameter, but there are two ASSOCIATE entries. This means that either name (ISE or C2) on the .MODEL statement can put a number into the class entry Q\_ise.

**Note:** When model parameters are listed, the first name found in the ASSOCIATE list (searching downward) is the name which is echoed on the output.

Insert the new name first if it is the name to be printed.

### Adding a parameter

Adding a parameter is probably the most common case. The parameter must be added to both the model class (e.g., class M\_Model) and the corresponding ASSOCIATE list. It is recommended to follow the PSpice naming convention (e.g., M\_wd and M\_vto), but it is not required.

Model parameters are set forth as pairs of elements instead of simple floating point values. This is to provide the use of expressions for model parameters. Because of this, when adding a parameter (for example, M\_new), the following line is required:

```
MXPR( M_new, Mx_new );
```

instead of

```
float M_new;
```

**Note:** Do not modify the value of the Mx\_new class element.

The read-in mechanism can handle expressions for user-added parameters. By the time the model code is called, the expressions have been evaluated and their value placed in the appropriate fields. See the include file m.h for further examples and comments.

When the simulator is doing a read-in, model parameters are listed for each .MODEL statement (unless NOMOD has been specified on the .OPTIONS statement). Normally, only those parameters that have not been defaulted are listed. A parameter can be forced to be listed, even if it has been defaulted, by preceding its name using an asterisk (\*) in the ASSOCIATE macro. For instance, VTO in M.H is listed in that manner.

The default value, OMITTED, is used by the simulator to force the calculation of a parameter's value during read-in. For instance, VTO is calculated from other values if it is not given a

value. These calculations are built into the read-in and are fixed. Cadence recommends that parameters that you add be given a normal default value and not be computed by using OMITTED.

Once the parameter has been added, the model class becomes one parameter longer, and the read-in section of PSpice places a value in its entry. The parameter can now be used in the device code (e.g., MOS.CPP).

### Changing the device equations

The device equations are in the file that has the same name as the type of device (DIODE.CPP, BJT.CPP, JFET.CPP, MOS.CPP, GASFET.CPP). The files D.CPP, Q.CPP, J.CPP, M.CPP, and B.CPP contain auxiliary functions that implement the AC equations, matrix setup, temperature updating, etc. The code in these subroutines use the model parameters and the device's terminal voltages to calculate the branch currents and conductances, and, during transient analysis, the terminal charges and branch capacitances. These equations are neither simple nor easy. A good understanding of U.C. Berkeley's SPICE2G is recommended before making such a change. Two useful references are:

1. Nagel, L.W., "SPICE2: A Computer Program to Simulate Semiconductor Circuits", Memorandum No. M520, May 1975.
2. Cohen, Ellis, "Program Reference for SPICE2", Memorandum No. M592, June 1976.

which are available from:

Software Distribution Office

EECS/ERL Industrial Liaison Program

205 Cory Hall #1770

University of California

Berkeley, CA 94720-1770

(510) 643-6687

For more details about device source files, see Functional subsections of the device source file.

### Functional subsections of the device source file

The code in each of the device source files is arranged into separate functional subsections. Each subsection occurs at least once, but can occur several times for devices that have more than one level. The subsections required are outlined below.

Subsection	Description
Initialization	This consists of locating and binding the device instance and its model, initializing any local variables, and obtaining appropriate values for the device branch voltages. The branch voltages (e.g., vds, vgs) are set differently depending upon whether there are user-specified initial conditions (using IC= or .IC), and on whether the present Newton Raphson cycle has finished or not.
Computing new nonlinear branch voltage:	This is needed to monitor progress towards a Newton Raphson solution.
Test if the solution has changed:	If there is not significant change bypass the rest of the computation. Otherwise, continue.
Limit any nonlinear branch voltages:	This code uses the macro PNJLIM() to insure that the branch voltages are in the appropriate operating region.
Compute currents and conductances:	This is the meat of the Device Equations code, and involves obtaining all the branch currents (e.g., ibs, ibd) as well as all the derivatives to be used in the conductance matrix.
Charge calculations:	Internal charges are calculated and updated.
Check convergence:	Check to see if the nonlinear device branches now have values that are within a small tolerance range of those obtained in the last repeat cycle, and set a return flag to signal whether the device converged.
Load the current vector and conductance matrix:	The macro Y_MATRIX () is used to obtain handles to the proper matrix elements, and the elements are assigned their values based on the present evaluation of the device equations and derivatives.

SPICE2G is written in FORTRAN, whereas PSpice is in C/C++. For the device subroutines, as much correspondence as possible has been maintained between the two. Because of FORTRAN, SPICE kept integer and real numbers in different tables: NODPLC (indexed by LOC) and VALUE (indexed by LOCV or LOCM). In PSpice, these have been combined into one object (e.g., class M\_Device).

The state vector information is constructed somewhat differently, though the overall pattern is similar. In SPICE the state vector information is kept in a set of vectors in VALUE. There is one vector for each time point “remembered” (from 4 to 7, depending on the order of the integration method). Each device’s LOC table contains an offset, LX, to its portion of the information in each state vector. In PSpice the number of state vectors is fixed, and each device’s state information is kept in its own device object (e.g., class M\_Device).

For example, for MOSFETs the state vectors are an array, struct msv\_def m\_sv[MSTVCT] in class M\_Device. MSTVCT is the number of state vectors and is defined in TRAN.H to be equal to 4. The definition of msv\_def (also in M.H) lists the various currents, conductances, charges, and capacitances that are in the state vector. Finally, M.H contains a set of #defines, which allows accessing of the entries to the state vectors by name. It is these (uppercase) names which are then used in MOS.CPP. This may seem like a roundabout way of constructing the state vector information, but the actual usage (in MOS.CPP) is quite straightforward and is similar to that in SPICE.

### Adding a new device

The Device Equations option does not allow the addition of an entirely new device. However, in many cases the same thing can be achieved by making use of an existing device.

Suppose, for example, that a lightning arrester device is to be added. The lightning arrester has two terminals, therefore it can be built into the diode equations, because the diode also has two terminals. This means that in the circuit (.CIR) file the lightning arresters would use the letter D to start and would refer to a .MODEL statement of the type D.

At first glance it appears that this would preclude using diodes in circuits, since they have been replaced by lightning arresters. This problem is avoided by keeping all the diode model parameters, adding the lightning arrester parameters, adding a LEVEL parameter, and giving the LEVEL parameter a default of 1. In the diode subroutine (in DIODE.CPP), a large if test would select all the old diode code if LEVEL=1 and all the new lightning arrester code otherwise. The new LEVEL parameter would switch between diode and lightning arrester.

This approach can be extended to as many devices as wanted. This could be:

- LEVEL=1 as a diode
- LEVEL=2 as a lightning arrester



### ■ LEVEL=3 as a gas discharge tube

And so on. The restriction is that all of the devices added to the diode must have two terminals. If the device to be added has three terminals, it must be built into a three terminal device, such as the JFET. The highest number of terminals that can be modeled is four, using the MOSFET. There is not a good way to add devices, such as pentodes, that have five or more terminals.

## Specifying new internal device structure

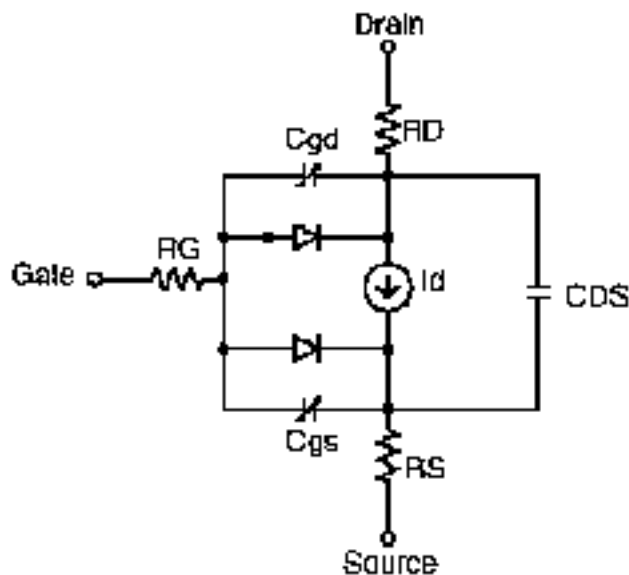
You may want to change the topology of a device in order to accommodate a more elaborate set of parasitic resistances and/or capacitances. To do this requires that positions in the conductance matrix be assigned to include the terms that the additional equations generate. This requires five steps:

1. Ensuring that all of the new internal nodes and matrix conductance terms are added to the device class in the device header file
2. Allocating the new matrix elements
3. Providing handles to access the new matrix elements and to bind the nodes to the branches
4. Including logic, if needed, to support device model parameter checking and updating
5. Adding the new device equations to the device code

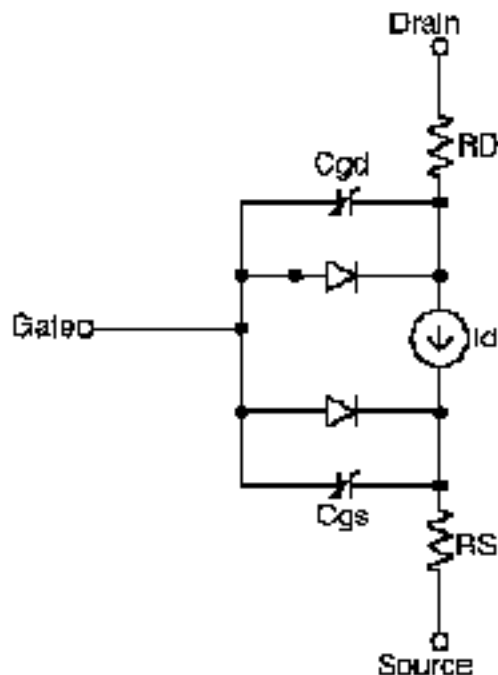
## Example

This process can be illustrated by looking at the PSpice JFET and GaAsFET devices, as shown in the procedure below. The topologies of these two devices are nearly identical, except that the GaAsFET has an additional internal capacitance, CDS, between the source and drain, and an additional internal resistance, RG, at the gate. This gives the GaAsFET topology one additional node where RG joins the rest of the structure and two additional internal branches.

**GaAsFET Model**



**JFET Model**



## Procedure

### ***Step one: editing the device header file***

These differences are reflected in the device class definitions in J.H and B.H. Each of the device nodes is given a name and declared to be of type CKT\_IDX.

The JFET device class, J\_Device, lists the two internal nodes j\_d and j\_s, while the GaAsFET device class, B\_Device, has three internal nodes b\_d, b\_s, and a new one, b\_g. The two additional branches in the GaAsFET require three new matrix conductance terms.

The conductance terms are declared type MTX\_IDX and are listed immediately following the internal nodes.

The JFET has a term j\_GG, which appears on the matrix diagonal for the external gate node.

The GaAsFET has an additional gate node which requires one additional matrix diagonal conductance term, b\_gg, along with two off-diagonal conductance terms, b\_gg and b\_gG. These are used by the source code in GASFET.CPP to designate where the conductance

terms associated with RG go when the matrix is loaded. CDS doesn't need any additional nodes or matrix terms because the items required are already in place to accommodate the parallel current source, id.

With the nodes and conductance terms taken care of in the device header file, the first step is completed.

### ***Step two: setting up memory allocation for the new matrix elements***

You can set up memory allocation to properly incorporate the new equations into the conductance matrix by modifying J.CPP. In this file is the function J\_Device::MatPtr(), while B.CPP contains B\_Device::MatPtr(). These functions call the function Reserve() once for each conductance matrix term that was declared in the header file. For instance, when b\_gg, b\_Gg, and b\_gG are added for the GaAsFET, these require corresponding code in B\_Device::MatPtr() as follows:

```
flag &= Reserve (ng,ng);
```

```
flag &= Reserve (nG,ng);
```

```
flag &= Reserve (ng,nG);
```

The arguments ng and nG are local variables that serve as aliases for the respective device nodes, b\_g and b\_G, and are assigned at the beginning of B\_Device::MatPtr() as follows:

```
ng = bloc -> b_g;
```

```
nG = bloc -> b_G;
```

### ***Step three: binding the nodes and branches***

The mechanics of step three, binding the nodes and branches, are very similar to the mechanics of step two. The functions of interest are J\_Device::MatLoc() and B\_Device::MatLoc(), and they now call Indxcl() instead of Reserve(). The GaAsFET again has three more lines of code:

```
flag &= Indxcl (&(bloc->b_gg),ng,ng);
```

```
flag &= Indxcl (&(bloc->b_Gg),nG,ng);
```

```
flag &= Indxcl (&(bloc->b_gG),ng,nG);
```

### ***Step four: handling model parameters***

Step four, handling model parameters, is basically the same as it would be for a case not involving topology changes, with one significant exception: this requires handling the case where the parasitics associated with an internal node can be zero. In this case the node must be generated conditionally. An instance of this is the GaAsFET internal resistance  $R_G$ . If  $R_G$  is zero, the parasitic resistance between the internal node  $b_g$  and the external node  $b_G$  can be removed from the circuit. This is accomplished in the function `B_Device::AddInternalNodes()` in `B.CPP`, using the following line of code:

```
INTERNAL_NODE(P->B_rg,b_g,b_G);
```

`INTERNAL_NODE()` is a macro that performs the required logic, depending on whether the model parameter `B_rg` is zero or not. The other two calls to this macro in `B_Device::AddInternalNodes()` correspond to the  $R_D$  and  $R_S$  resistances that also exist for the JFET.

### ***Step five: implementing the new device equations***

The final step does not involve any further topological considerations and is carried out just as it would be if the device internal topology weren't being changed.

## **Recompiling and linking the device equations option**

The source files needed to create the Device Equations DLL can be copied from the CD to any directory you choose, though it is recommended that you create a new empty directory. The MSVC++ project files, `DEVEQ.DSP` and `DEVEQ.MAK`, are included to compile and link the DLL.

For information on obtaining the Microsoft compiler, contact Microsoft Corporation directly.

### **To create a new `deveq.dll`:**

1. Load `DEVEQ.DSP` into the Visual C++ development environment.
2. From the Build menu, select Build `Deveq.dll`.
3. The project supports debug and release versions of the build target.
4. After `DEVEQ.DLL` is built, copy it to the directory that contains PSpice .EXE.

For details on how to personalize your `DEVEQ.DLL` file, click [Personalizing your DLL](#).

### Personalizing your DLL

The function `DLLMain()` in `DEVEQDLL.CPP` contains the following line of code:

```
DEVEQVERSIONINFO("Device Equations",VERSIONNUM);
```

To personalize your DLL, change the first argument to a string which identifies you as the author of the DLL, as in:

```
DEVEQVERSIONINFO("(c)Copyright 1998\nMyCorp\n123 MyAddress\nMyCity, ST 12345",  
"9.0.1");
```

You can leave the `VERSIONNUM` argument alone, in which case it will match the version number of your PSpice release, or you can substitute your own version numbers. It is useful to be able to relate the DLL to the PSpice release it was built from, so you should use `VERSIONNUM` unless there is a compelling need to change it.

### Simulating with the device equations option

After you obtain a working Device Equations DLL, place it in the directory that contains PSpice .EXE.

PSpice will locate and load `DEVEQ.DLL` when you start the program, provided the .INI file entry is specified correctly. For instructions on modifying the .INI file, click [Selecting which models to use from a Device Equations DLL](#).

The code in the DLL will be substituted for the device model code that ships with the plain version of PSpice. The presence of the DLL is also noted in the Devices tab of the PSpice Simulation Status Window and in the .OUT file.

If PSpice doesn't find the DLL, it runs as the normally configured PSpice.

### Selecting which models to use from a Device Equations DLL

You can tell PSpice which device models to use from a custom DLL by adding an entry to the `pspice.ini` configuration file; for any device type you do not specify, PSpice uses the normally configured PSpice models.

To specify which models to use from a custom DLL:

1. In a standard text editor (such as Notepad), open `pspice.ini`, located in your Windows directory.

## PSpice Help

---

2. Find the [PSpice ] section and add this line to the section:

```
USE_DEVEQ_MODELS="<device letters>"
```

where <device letters> is any or all of the following:

For this device type...	Use this device letter...
GaAsFET	B
Diode	D
Junction FET	J
MOSFET	M
Bipolar transistor	Q

For example, to use all of the possible device models from your custom DLL, type the following:

```
USE_DEVEQ_MODELS="BDJMQ"
```

3. Save pspice.ini.
4. Start PSpice and run a simulation.

## Popup Menu Items

### **Y axis number**

Click to select a number from the list or type a number.

### **Attributes**

Draws symbols on the traces if the Show symbols check box is enabled in the Properties dialog box. To access the Properties dialog box, right-click the trace and choose Properties.

### **Never**

Never use symbols.

### **Always**

Always display trace symbols.

### **Normal**

Use a different color for each trace (number of different colors permitting).

### **Match Axis**

Use the same color for all the traces that belong to the same Y axis.

### **Sequential Per Axis**

Use the available colors in sequence for each Y axis.

### **Unique by File**

Use the same color for all the traces that belong to the same file. (This applies to analog traces only—digital traces have their own color differentiation scheme.)

### **Auto**

Use scroll bars when an axis is zoomed in.

### **Never**

Never use scroll bars.

### **Always**

Always use scroll bars.

### **Auto**

Allow Probe to update traces each time it gets new data from the simulation.

### **Every sec**

Update traces at regular time intervals. Time intervals can be specified in seconds.

### **Every %**

Update based on the percentage of completion of the analysis. Type the value as the percentage of completion.

### **Auto-Update Interval**

Select how often Probe updates a displayed trace.



### **Display Status Line**

Click to select whether Probe shows the status line at the bottom of the screen.

### **Mark Data Points**

Click to select whether data points are automatically marked on the trace.

### **Display Evaluation**

Click to select whether Probe displays the traces and marked points when you choose Evaluate Measurement from the Trace menu.

### **Display Statistics**

Click to select to enable automatic display of statistics about each histogram.

### **Highlight Error States**

Click to select whether Probe automatically highlights error states when it displays a trace.

### **Display Toolbar**

Click to select whether Probe automatically displays the toolbar at the top of the window.

### **Data Range**

Click Auto Range to let Probe set the range automatically, or click User Defined and type a specific data range to display.

### Scale

Click to select either Linear or Log.

#### linear

Linear is the default for all analyses, except for AC analysis, which is logarithmic.

#### logarithmic

For AC analysis, the X axis starts out as logarithmic. You cannot put the axis into a log scale if either end of the axis range is zero or negative.

### Use Data

Click Full to make the entire range of raw data available, or click Restricted and type a specific range of raw data to use.

### Processing Options

Click to select either Fourier Transform or Performance Analysis.

- Fourier Transform changes an AC analysis X axis from Frequency to Time, or a Transient X axis from Time to Frequency.
- Performance Analysis displays a histogram for Monte Carlo Analysis, or displays a line graph of the measurement Evaluation value versus the parameter that changes between multiple PSpice runs.

### Axis Variable

Click to see the Axis Variable dialog box. This looks and functions much like the Add Traces dialog box. It allows you to define a particular variable, or set of variables, specific to the X axis.

### Number

Click to select the Y axis number from the list or type a number. You can have 1 to 3 axes.

### Title

Type a title in the box. This text is shown vertically, to the left of the plot.

### Display statistics

The statistics that are displayed are as follows:

<code>n samples</code>	The number of Monte Carlo runs selected for analysis in Probe.
<code>n divisions</code>	The number of divisions that the X axis is divided into to create the histogram (i.e., the number of vertical bars that make up the histogram).
<code>mean</code>	The arithmetic average of all the measurement expression values.
<code>sigma</code>	The standard deviation
<code>minimum</code>	The minimum measurement expression value.
<code>10th percentile</code>	A number representing measurement expression values such that 10% of the measurement expression values are less than or equal to the number, and also at least 90% of the measurement expression values are greater than or equal to the number. If there is more than one measurement expression value that satisfies this criteria, then the 10th percentile is the midpoint of the interval between the measurement expression values that satisfy the criteria.

## PSpice Help

---

median	Same as 10th percentile, except for 50%.
90th percentile	Same as 10th percentile, except for 90%.
maximum	The maximum measurement expression value.

### Percentage of Plot to be Digital

Specify the percentage of the plot that should be allocated for the digital traces. Select a valid displayed range. The default is 33 percent.

### Length of Digital Trace Name

Specify the length of the digital signal's display name.

### OK

Click to accept the values in the dialog box and close the dialog box.

### Cancel

Click to close the dialog box without saving any changes.

### Help

Click to see the help system.

### **Close**

Click to close the dialog box.

### **Reset**

Click to set the options to the default configuration.

### **Print**

Click to print one page using the current settings.

### **Enter the Inclination Angle**

Type the angle at which you want to place the ellipse.

### **Toolbar**

Click to perform a common function in Probe.

### **Display**

The traces and plots are shown here.

### **Message Bar**

Messages appear here about what you are currently doing.

### **Legend**

Double-click the trace name to display the Modify Traces dialog box.

Double-click the symbol (or one of the symbols) to the left of a trace name to view information about the trace.

### **Move to the next transition**

Shift+Control+N

### **Add a Y axis**

Control+Y

### **Move to the next trough**

Shift+Control+T

### **Copy the selected item**

Control+C

### **Move to the previous transition**

Shift+Control+R

### **Cut the selected item**

Control+X

### **Open a file**

Control+F12

### **Delete the selected item**

Delete

### **Paste the cut or copied item**

Control+V

### **Delete a Y axis**

Shift+Control+Y

### **Open the Print dialog box**

Control+Shift+F12

### **Exit Probe**

Alt+F4

### **Redraw the screen**

Control+L

### **Go back to the previous view**

Control+P

### **Search command**

Shift+Control+S

### **Move to the next maximum value**

Shift+Control+X

### **Turn the data cursor on or off**

Shift+Control+C

### **Move to the next minimum value**

Shift+Control+M

### **Zoom in to the selected area**

Control+A



### **Move to the next peak**

Shift+Control+P

### **Zoom out so that all data on the screen is shown**

Control+N

### **Move to the next point**

Control+Shift+I

### **Zoom in around a specified point**

Control+I

### **Zoom out around a specified point**

Control+O

### **Delete all traces in the selected plot**

Control+Delete

### **Undelete last deleted**

Control+U

### Move to the next slope

Shift+Control+L

### Freeze the cursor

Shift+Control+F

### expression

An expression is a single output variable or combination of output variables using arithmetic operators, Probe functions, or macros.

### alias

Node or device pin names that represent the same point in a circuit.

### Major

The Major frame allows you to customize the display of the major grid axes.

#### ■ Spacing

Choose Linear to define the linear spacing of the grids.

Choose Log (# of decades) to set the decade interval for a logarithmic grid display.

**Note:** The Spacing settings can only be defined if the Automatic check box is disabled.

#### ■ Grids

You can display the grids as either Lines, Dots (At intersections), + (At intersections) or None. If you choose Dots or + (At intersections), you can display these either with other major grids, with other minor grids, with both or with none.

### ■ Ticks inside plot edge

When this option is enabled, tick marks are displayed at each major axis. Normally these are not visible when the grids are displayed as Lines.

### ■ Numbers outside plot edge

When this option is enabled (default), the unit labels are displayed for each major axis.

## Minor

The Minor frame allows you to customize the display of the minor grid axes.

### ■ Intervals between Major

You can define the intervals for the minor axes as either 2, 4, 5 or 10 between each major axis.

**Note:** The Intervals between Major settings can only be defined if the Automatic check box is disabled.

### ■ Grids

You can display the grids as either Lines, Dots (At intersections), + (At intersections) or None. If you choose Dots or + (At intersections), you can display these either with other major grids, with other minor grids, with both or with none.

### ■ Ticks inside plot edge

When this option is enabled, tick marks are displayed at each major axis. Normally these are not visible when the grids are displayed as Lines.

## Axis Title

Allows users to specify a name for the X-axis. Select the Use this title check box. In the text box that gets enabled, enter the new title for the X-Axis.

### **Axis Position**

Use this option to specify whether the Y-axis should be located to the left or to the right of the Probe window. This option is useful only in case of multiple Y-axis. In case you have only one Y-axis, it needs to be on the left of the Probe window.

## PSpice Errors and Solutions

### **.PROBE and .ALIAS must agree on /CSDF**

Either a .PROBE/CSDF and .ALIAS or a .PROBE and .ALIAS/CSDF were entered.

#### **Solution**

1. Correct the conflict by specifying /CSDF on both or neither statement.

### **Invalid device type**

The first character of all devices defines what kind of device it is. The first character is not known to be a valid device.

#### **Solution**

1. Correct the spelling of the name.

### **Maximum number of alias nodes exceeded**

Too many nodes have alias names associated with them.

#### **Solution**

1. Eliminate the aliases for some nodes.

### **Unable to open index file**

The index file could not be created. Either it is open by another program or the system-wide limit on the number of files which can be concurrently open has been exceeded.

#### **Solution**

1. If another program has the file open, close that program.

### Model type unknown

In a .MODEL statement the model type was probably mistyped.

#### Solution

1. Make sure that the model type is valid. The valid model types for each device are listed in the Reference Manual section dealing with the device.

### Duplicate library entry for <modelname>

The identified model appears in more than one library. Only the first one will be used.

#### Solution

1. Remove the definition from all the other libraries, or change their names to make them unique.

### Out of Memory

Insufficient memory for circuit.

#### Solution

1. Allocate PSpice more memory:
  - ☐ If you are running other programs concurrently with PSpice , close them and try again.
  - ☐ Install more memory for your computer.

### Unrecognizable command

The first character on a line was a period (.), indicating a command, but the subsequent characters did not define a valid command.

### **Solution**

1. Correct the spelling of the command.

### **Unable to open stimulus file**

The identified file was listed in a STIMULUS device, but could not be found.

### **Solution**

1. Correct the spelling of the file name.

### **Model references form circular list. For example:**

A model with an AKO: references another model with an AKO: which in turn references the first.

### **Solution**

1. Break the loop of AKO: references.

### **Unable to open probe file**

The Probe Data file could not be created. Either it is open by another program or the system-wide limit on the number of files which can be concurrently open has been exceeded.

### **Solution**

1. If another program has the file open, close that program.

### **Unable to make index for library file**

An index file had to be created for a library file. However, the system did not permit it.

### **Solution**

1. An index file will reside in the same directory as the library file. Make sure that you have permission to write to that directory.
2. If the index file already exists, make sure that you have permission to modify it.

### **Model <modelname> referenced by model <modelname>, is undefined**

There was an AKO: reference (.MODEL first\_name ... AKO: second\_name ...) The second model is not defined in either the circuit file or any of the referenced libraries.

#### **Solution**

1. Check for correct spelling of the second model name.
2. If necessary, add the name of the library in which it can be found.

### **Subcircuit <filename> used by <filename> is undefined**

The identified subcircuit is not defined in either the circuit file or any of the referenced libraries.

#### **Solution**

1. Check for correct spelling of the model name.
2. If necessary, add the name of the library in which it can be found.

### **Unable to open library**

The specified library could not be found.

#### **Solution**

1. Check the spelling of the file name.



### Making new index file for library file

The index file associated with the indicated library file is being rebuilt.

This is for your information only, and is not an error. The index file is being rebuilt because it either did not exist or the library file was changed.

### Missing model name in library

During the process of building the index file for a library file, a .MODEL statement was encountered which did not have a model name.

#### Solution

1. Correct the .MODEL statement, using the following basic syntax:

```
.MODEL <name> <type> <optional parameters>
```

### Missing model type in library

During the process of building the index file for a library file, a .MODEL statement was encountered which did not have a model type.

#### Solution

1. Correct the .MODEL statement, using the following basic syntax:

```
.MODEL <name> <type> <optional parameters>
```

### Missing subcircuit name

During the process of building the index file for a library file, a .SUBCKT statement was encountered which did not have a name.

#### Solution

1. Correct the .SUBCKT statement, using the following basic syntax:

```
.SUBCKT <name> [<nodes> ...]
```

### **IVON - VOFF too small for VSWITCH model**

The ON and OFF voltages specified are too close in value to each other.

#### **Solution**

1. Change value to be more reasonable.

### **RON or ROFF less than or equal to zero for VSWITCH model**

Either the ON resistance or OFF resistance is not positive.

#### **Solution**

1. All resistances must be positive. Fix the incorrect value.

### **RON or ROFF greater than 1/GMIN for VSWITCH model**

Either the ON resistance or OFF resistance is too large.

#### **Solution**

1. All resistances must be less than 1/GMIN. Fix the incorrect value.

### **RON = ROFF for VSWITCH model**

The ON resistance is the same as the OFF resistance.

#### **Solution**

1. They must be different to be a meaningful device. Change one of them.

### **ION - IOFFI too small for ISWITCH model**

The ON and OFF currents specified are too close in value to each other.

#### **Solution**

1. Change either to be more reasonable.

### **RON or ROFF less than or equal to zero for ISWITCH model**

Either the ON resistance or OFF resistance is not positive.

#### **Solution**

1. All resistances must be positive. Fix the incorrect value.

### **RON or ROFF greater than 1/GMIN for ISWITCH model**

Either the ON resistance or OFF resistance is too large.

#### **Solution**

1. All resistances must be less than  $1/GMIN$ . Fix the incorrect value.

### **RON = ROFF for ISWITCH model**

The ON resistance is the same as the OFF resistance.

#### **Solution**

1. Change one of them. They must be different to be a meaningful device.

### **<param> not a subcircuit param**

One of the optional parameters listed in the PARAMS: section of a subcircuit call is not defined.

#### **Solution**

1. Correct the spelling to match those defined by the subcircuit.

### **Less than 2 connections at node**

This node connects to only one device terminal in the circuit or in a subcircuit model. A port may be mislabeled.

#### **Solution**

1. Starting with the node indicated in the message, verify the circuit connections.

### **Node is floating**

The voltage of this node cannot be determined. The node or the circuitry containing it may be isolated from power supplies or ground by capacitors, or the node's connections to power supplies or ground is missing.

#### **Solution**

1. Starting with the node indicated in the message, verify the circuit connections.

### **Invalid radix, expecting BIN (1), OCT (3), or HEX (4)**

In a .VECTOR command, the optional RADIX parameter was entered, but its value was not 1, 3, 4, B, O, or H.

#### **Solution**

1. Correct the RADIX value.

### Unrecognized parameter

In a .VECTOR command, a parameter was entered which was not valid.

#### Solution

1. Correct the statement.

### Tolerances on model <modelname> ignored due to <tolerance>

A model which is the target of a .STEP PARAM or .DC PARAM had tolerances. This is a warning only.

The tolerances are ignored.

### MC or .WCASE ignored (No <analysis type> command in circuit)

A .MC command was encountered specifying an analysis which was missing.

#### Solution

1. Add a .DC, .TRAN, or .AC command as appropriate.

### No models had tolerances. .MC or .WCASE ignored

A .MC command was encountered, but none of the models had tolerances specified.

#### Solution

1. Either add tolerances to one or more models, or remove the .MC command.

### **The circuit matrix is singular and cannot be solved.**

The matrix is singular (has a 0.0 value along the diagonal.)

#### **Solution**

1. Check for the following frequent causes:
  - ☐ floating nodes
  - ☐ a closed path through the circuit which has zero resistance

### **The circuit matrix is too close to being singular to solve.**

The matrix is nearly singular (has value almost 0.0 along the diagonal.)

#### **Solution**

1. Check for the following frequent causes:
  - ☐ a path to ground which is very high impedance
  - ☐ a path with a very large gain

### **Convergence problem**

PSpice could not derive values for the node voltages or device currents that satisfy the convergence criteria used.

#### **Solution**

1. Check circuit connections, device model parameters used, expected operating regions of the devices, etc.
2. Set initial conditions, relax the tolerance parameters, use GMIN stepping (for DC convergence problems), etc.

### Convergence problem

PSpice could not derive values for the node voltages or device currents that satisfy the convergence criteria used.

#### Solution

1. Check circuit connections, device model parameters used, expected operating regions of the devices, etc.
2. Set initial conditions, relax the tolerance parameters, use GMIN stepping (for DC convergence problems), etc.

### Time step is too small in Transient Analysis at xxx

The simulation must terminate due to the time step needed for convergence becoming too small.

#### Solution

1. Check for the following possible causes:
  - ☐ waveforms with very fast rise/fall times
  - ☐ model parameters completely wrong

### Missing or invalid expression

An expression was expected, but was either missing or invalid.

#### Solution

1. Make sure that expressions are enclosed in '{' and '}' characters and that any '(' characters have a matching ')' character.
2. Make sure that continuation lines have a '+' character in the first column.
3. If the netlist was generated by Capture, contact the Customer Support.

### Missing expression

An expression was expected, but was either missing or invalid.

#### Solution

1. Make sure that expressions are enclosed in '{' and '}' characters and that any '(' characters have a matching ')' character.
2. Make sure that continuation lines have a '+' character in the first column.
3. If the netlist was generated by Capture, contact the Customer Support.

### Bad radix spec

In a digital stimulus device, the radix specified was not one of the permitted values: 1, 3 or 4.

#### Solution

1. Correct the radix.

### LABEL invalid in REPEAT loop

In a digital stimulus device, a LABEL: appeared in a REPEAT ... ENDREPEAT construct.

#### Solution

1. Remove the label.

### Missing goto label

In a digital stimulus device, a GOTO was encountered but the target label was missing.

#### Solution

1. Correct the statement.



### **GOTO invalid in REPEAT loop**

In a digital stimulus device, a GOTO appeared in a REPEAT ... ENDREPEAT construct.

#### **Solution**

1. Remove the GOTO.

### **HREPEAT missing FOR or FOREVER**

In a digital stimulus device, a REPEAT appeared but one of the two required keywords FOR or FOREVER was missing.

#### **Solution**

1. Correct the specification.

### **Attempt to redefine builtin name**

In a .FUNC statement, a reserved macro name such as 'SQRT()' was being redefined.

#### **Solution**

1. Refer to the Reference manual for the list of reserved macro names and avoid redefining them.

### **Must be D**

In a .PRINT/DCTLCHG statement, a request for a voltage or current was indicated.

#### **Solution**

1. Make sure only digital nodes are printed with a .PRINT/DGTLCHG statement.

### **Must be I or V or D**

In a .PRINT TRAN statement, only currents, voltages, and digital states can be printed.

#### **Solution**

1. Correct the statement.

### **Must be I or V**

In a .PRINT AC or .PRINT DC, only currents and voltages can be printed.

#### **Solution**

1. Correct the statement.

### **Must be V**

In a .PRINT NOISE statement, only voltages can be printed.

#### **Solution**

1. Correct the statement.

### **Must be I or V, D not allowed**

In a .PRINT AC or .PRINT DC, only currents and voltages can be printed.

#### **Solution**

1. Correct the statement.

### **Expression not allowed here**

In a PLSYN device, a TESTVECTOR parameter was specified as a {parameter}.

#### **Solution**

1. Change the {parameter} to a literal constant.

### **Unknown parameter**

In a RAM, ROM or PLD device, an unknown parameter was entered.

#### **Solution**

1. Check the spelling of all parameters and correct as necessary.

### **Probability must not be less than 0.**

In a .DISTRIBUTION statement a probability was less than 0.

#### **Solution**

1. Make sure all probabilities are between 0 and 1 inclusive.

### **At least two pairs of numbers necessary**

In a .DISTRIBUTION statement, at least two (probability, value) pairs must be entered.

#### **Solution**

1. Make sure you have at least two pairs.

### **Please simplify .. distribution too complicated**

In a .DISTRIBUTION statement, more than 100 (probability, value) pairs were entered.

#### **Solution**

1. Reduce the complexity of the expression.

### **Use RLGC & LEN for lossy line**

A lossy transmission line was specified and a either Z0 or TD parameter was entered.

#### **Solution**

1. Remove the Z0 or TD parameter; lossy transmission lines are characterized with the R, L, G, G and LEN parameters.

### **Use Z0 & TD or F/NL for ideal line**

An ideal transmission line was specified and a Lossy parameter was entered.

#### **Solution**

1. Remove the Lossy parameter; ideal transmission lines are characterized with the Z0, TD, F and NL parameters.

### **Z0 or RLGC parameters must be specified**

Transmission lines require either Z0 or R, L, G, and C to be set.

#### **Solution**

1. Enter a value for Z0 if ideal, or for R, L, G, and C if lossy.

### **TD or F must be specified**

An ideal transmission line was specified, but neither TD nor F was entered.

#### **Solution**

1. Enter a value for either TD or F.

### **BadTransferFunction**

An E or G device was entered and LAPLACE was specified for it. The laplace expression was invalid.

#### **Solution**

1. Correct the expression.

### **Missing REPEAT iteration count**

In a PWL type Voltage or Current source, the REPEAT iteration count was missing.

#### **Solution**

1. Correct the device specification.

### **Symbols Table overflow**

You have too many devices to simulate with the memory available.

#### **Solution**

1. Give PSpice more memory:
  - ☐ If you are running other programs concurrently with PSpice , close them and try again.
  - ☐ Add more memory for your computer.

### **Voltage Source and/or Inductor Loop Involving xxx**

This may be caused by a loop constructed with a combination of voltage sources and/or inductors.

#### **Solution**

1. Break the loop by adding a series resistance.

### **Convergence problem**

PSpice could not derive values for the node voltages or device currents that satisfy the convergence criteria used.

#### **Solution**

1. Check circuit connections, device model parameters used, expected operating regions of the devices, etc.
2. Set initial conditions, relax the tolerance parameters, use GMIN stepping (for DC convergence problems), etc.

### **Convergence problem**

PSpice could not derive values for the node voltages or device currents that satisfy the convergence criteria used.

#### **Solution**

1. Check circuit connections, device model parameters used, expected operating regions of the devices, etc.
2. Set initial conditions, relax the tolerance parameters, use GMIN stepping (for DC convergence problems), etc.

### **Invalid Outside of .SUBCKT**

A .ENDS statement was encountered, but a subcircuit was not being defined.

#### **Solution**

1. Remove the .ENDS statement or, if it was intended to mark the end of the circuit, change it to .END.

### **Library Index File Does Not Have the Correct Format**

The index file shown is corrupt.

#### **Solution**

1. Delete the index file shown. This will cause it to be rebuilt in the correct format.

### **Unable to Find Library File**

The specified library could not be found.

#### **Solution**

1. Check the spelling of the file name.

### **Library File Has Changed Since Index File Was Created**

The specified library file was modified since its index file was last created.

#### **Solution**

PSpice will automatically recreate the associated index file.

### **The Timestamp Changed from xxx to yyy**

The specified library file was modified since its index file was last created.

#### **Solution**

PSpice will automatically recreate the associated index file.

### **Model <modelname> Used by <filename> Is Undefined**

The identified model is not defined in either the circuit file or any of the referenced libraries.

#### **Solution**

1. Make sure the model name is spelled correctly.
2. If necessary, add the name of the library in which it can be found.

### **Missing param name in library**

During the process of building the index file for a library file, a .PARAM statement was encountered which did not have a parameter name.

#### **Solution**

1. Correct the .PARAM statement. The basic syntax is:

```
.PARAM name = value
```

### **There Are No Devices in This Circuit (This Message Will Be Printed)**

Multiple circuits may be simulated from a single file. After each .END statement is encountered and the simulation has been completed, PSpice will continue to read the input



file for any subsequent circuits. If any data is read but no valid devices exist when the end of file is reached, this message is issued.

### **Solution**

1. Remove blank lines after the last .END statement in the circuit file.

### **Only one .TEMP value allowed with .STEP**

If you have a .STEP command, there may be a .TEMP command specifying only a single temperature.

### **Solution**

1. Remove all but the first temperature listed in the .TEMP statement.

### **Only one .TEMP, .DC TEMP, or .STEP TEMP permitted**

The temperature(s) at which a circuit may be simulated can be set in exactly one of three ways. Two or more different ways have been specified.

### **Solution**

1. Remove all but one of the .TEMP, .DC TEMP and .STEP TEMP statements.

### **Unable to open file**

In a .INCLUDE ... FILE = ... statement, the specified file name could not be found.

### **Solution**

1. Correct the spelling of the file name.

### Missing .ENDS in .SUBCKT

The file ended during the processing of a subcircuit definition before the .ENDS statement was encountered.

#### Solution

1. Insert a .ENDS statement at the appropriate place in the file.

### Name on .ENDS does not match .SUBCKT

A subcircuit was being processed. The last line of a subcircuit is the .ENDS statement, which may optionally have a name. This name must be identical to the subcircuit name if it is present.

#### Solution

1. Remove the offending name from the .ENDS statement.

### Invalid device in subcircuit

While processing a subcircuit, an unknown device was encountered.

#### Solution

1. Correct the spelling of the device name.

### Subcircuit <filename> is Undefined

The indicated subcircuit could not be found in the circuit file or any of the libraries.

#### Solution

1. Check the spelling and correct as required.
2. Add the name of the library in which it is defined to the library list.

### **Incorrect Number of Interface Nodes for <filename>**

The subcircuit was defined with a different number of interface nodes than were listed when an instance was placed.

#### **Solution**

1. Make sure that the definition and the reference to the subcircuit have the same number of nodes.

### **Digital Simulator Option not present**

The Digital Simulator Option must be purchased to use this feature.

#### **Solution**

1. Contact Cadence to purchase the Digital Simulator Option.

### **Cannot Open Temporary Digital File**

One or more temporary files required by the digital simulator could not be opened.

#### **Solution**

1. Increase the FILES= value in the C:\CONFIG.SYS file.
2. If this does not solve the problem, contact Customer Support.

### **Missing model**

A model name was expected on a device statement, but was missing.

#### **Solution**

1. Correct the statement.

### Missing number of nodes

In a Pin Delay Device (U... PINDLY), the number of nodes parameter was missing.

#### Solution

1. Correct the statement.

### Too few output nodes specified

In a Pin Delay Device (U... PINDLY), the number of output nodes parameter was missing.

#### Solution

1. Correct the statement.

### Bad or missing parameter

In a Digital Stimulus Device (U... STIM ...), one of the required parameters was either invalid or missing entirely.

#### Solution

1. Correct the statement.

### Invalid value

In a Digital Stimulus Device (U... STIM ...), one of the states specified was not 0, 1, R, F, X, or Z.

#### Solution

1. Correct the state value.

### Undefined parameter used in expression

In an expression, a reference was made to a parameter which was neither one of the pre-defined ones, nor one defined in a .PARAM statement.

#### Solution

1. Make sure the spelling of all parameters within the expression is correct.

### Undefined Parameter: <parameter>

In an expression, a reference was made to this parameter which was neither one of the pre-defined ones, nor one defined in a .PARAM statement.

#### Solution

1. Make sure the spelling of all parameters within the expression is correct.

### I(node) is not valid

In a .PRINT ... statement, an attempt was made to print a current at a node.

#### Solution

1. Correct the statement. You can only print currents through a device and voltages at nodes or device pins.

### Must be independent source (I or V)

In a .PRINT ... statement, only a voltage source or current source is allowed.

#### Solution

1. Correct the statement.

### Digital node table overflow

There are too many digital nodes to simulate with the memory available.

#### Solution

1. Allocate more memory to PSpice by doing one of the following:
  - ☐ If you are running other programs concurrently with PSpice , close them and try to simulate again.
  - ☐ Purchase more memory for your computer.

### Missing parameter

A parameter was expected in a .AC or .DC statement, but was missing.

#### Solution

1. Correct the statement.

### Not a valid parameter for model type

A parameter in a .MODEL statement was misspelled.

#### Solution

1. Correct the spelling.

### Must be 'I' or 'V'

In a .DC or .STEP DC, an attempt was made to sweep some device other than a voltage or current source.

#### Solution

1. Change the device you want swept to a voltage or current source.

### Missing node number

In an .IC or .NODESET statement, the node number to be set was not specified.

#### Solution

1. Correct the statement.

### Missing device name

In an .IC or .NODESET statement, the device whose node was to be set was not specified.

#### Solution

1. Correct the statement.

### Analog simulator option not present

The statement requires analog simulator option, but that option is not installed.

#### Solution

1. Contact Cadence to purchase the analog simulator option.

### Invalid parameter

For a device other than a MOSFET or IGBT, a parameter was entered that is specific to these two devices.

#### Solution

1. Correct the device.

### **Inductor part of this K device**

In a K (Coupling) device, the same Inductor was entered twice.

#### **Solution**

1. Delete the second inductor reference.

### **Inductor part of another core device**

The same inductor appears in more than one K (Core) device.

#### **Solution**

1. Remove the reference in the second K device.

### **Transmission line part of this K device**

In a K (Coupling) device, the same Transmission Line was entered twice.

#### **Solution**

1. Delete the second reference.

### **Invalid specification**

In a voltage source or current source, the transient specification was not EXP, PULSE, PWL, SFFM, or SIN.

#### **Solution**

1. Correct the transient specification.



### **Bad value**

A floating point value was expected, but an invalid number was encountered.

#### **Solution**

1. See the online PSpice Reference Manual for the format of valid numbers, and correct the statement.

### **Invalid number**

An invalid floating point number was generated in the process of a calculation. The most common cause of this is a .MODEL parameter which is too far out of line.

#### **Solution**

1. Check all .MODEL parameters for correct scaling. If the problem persists, contact Customer Support.

### **No analog devices--DC sweep ignored**

The circuit has only digital devices. Digital devices are only simulated in the Transient (time-domain) analysis.

#### **Solution**

1. Remove the .DC statement.

### **No analog devices--small-signal analysis Ignored**

The circuit has only digital devices. Digital devices are only simulated in the Transient (time-domain) analysis.

#### **Solution**

1. Remove the .AC, .OP, .SENS, .TF or .NOISE statement.

### **Missing value**

An expected value is missing.

#### **Solution**

1. Make sure that continuation lines have a + character in the first column.
2. If the netlist was generated by Capture, contact Customer Support.

### **EOF in subcircuit**

During the definition of a subcircuit, an end-of-file condition was encountered. The file is probably corrupt.

#### **Solution**

1. Regenerate the file.

## Errors and Solutions

### Unable to write to disk: check if disk is full

The program could not completely save a file.

#### Solution

Using Windows Explorer, check if the disk is full. If so, make room by deleting files and re-try the operation.

### Unable to read from file - improper mode

This is an internal error.

#### Solution

Contact Customer Support.

### Unable to write to file - improper mode

This is an internal error.

#### Solution

Contact Customer Support.

### Cannot open file: filename

This error can occur while trying to load or save a file.

#### Solution

If loading a file, check:

- That the file exists
- That you have read access to the file

If saving a file, check:

- That the existing file is not write-protected

- That you have write privilege to the drive and/or directory

### **Cannot open directory for backup directory name**

While creating a backup copy of a file, the backup directory specified by the BACKUP setting in the pspice.ini configuration file did not exist. The program attempts to create such a directory. If it is unable to create it, you will get this error.

#### **Solution**

Check the BACKUP setting in your pspice.ini file. Create the directory then re-try the operation.

### **File error, fseek failed**

This is an internal error.

#### **Solution**

Contact Customer Support.

### **File error, ftell failed**

This is an internal error.

#### **Solution**

Contact Customer Support.

### **File error, cannot reopen**

This is an internal error.

#### **Solution**

Contact Customer Support.

### **Previous error in opening file**

This is an internal error.

### **Solution**

Contact Customer Support.

### **Backup failed: Permission denied to file**

The program was unable to complete making a backup copy of a file. Possible causes:

- The disk or directory where the backup is being made is write-protected
- Another user is making a backup of the same file at the same time

### **Solution**

Check that you have write privileges to the backup directory.

Check if another user is editing a schematic with the same name. In general, each user should use a separate backup directory (specified in your pspice.ini file).

### **Backup failed: Bad file number**

This is an internal error.

### **Solution**

Contact Customer Support.

### **Backup failed: Cannot write to file**

The program was unable to complete making a backup copy of a file. Possible causes:

- The disk where the backup is being made (specified by the BACKUP entry in pspice.ini) is full
- The disk or directory where the backup is being made is write-protected

### **Solution**

Check if the disk is full. If so, free up space and re-try the operation.

Check that you have write privileges to the backup directory.

### **Extension can only have up to three characters**

You specified a file name with an extension of more than three characters .

#### **Solution**

Specify an extension with 3 or less characters following the '.'.

### **Filenames can have only up to eight characters**

The program only supports file names with up to eight characters, not including the extension.

#### **Solution**

Specify a name with 8 or less characters.

### **File name contains an invalid character**

A file name with a '.' (other than the one preceding the extension) was specified.

#### **Solution**

Remove the '.' and re-try the operation

### **Blanks are not allowed in file names**

File names with embedded spaces are not supported

#### **Solution**

Remove the spaces and re-try the operation

### **Directory in path does not exist**

The directory specified for the file does not exist.

### **No filename?!**

You must specify a file name.

### **Cannot open temporary file**

The program could not create a temporary file.

#### **Solution**

Check the following:

1. The TMP environment (set in your AUTOEXEC.BAT) setting to be sure it is a directory that exists
2. Check that you have write access to the drive/directory if you are on a network.

If you are still unable to create temporary files, call Customer Support.

