cādence[®]

PSpice Help

Product Version 17.4-2020 June 2020 © 2020 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

About PSpice 15
What is PSpice?
What is Probe?
Types of analyses you can run with PSpice
AC sweep and noise
DC sweep & other DC calculations
Transient and Fourier
Parametric and temperature 20
Monte Carlo and sensitivity/worst-case
Frequency Response Analysis
Files used by PSpice (input files) 22
Files Generated by Schematic Editors or Design Entry Programs
Other Input Files
Model libraries
Stimulus files
Include files
Preparing and configuring input files
Files generated by PSpice (output files) 25
Documentation
Additional sources of information about PSpice
Preparing your design for simulation 29
Creating designs for simulation
Placing stimulus sources
Creating a simulation netlist
Setting up analyses
Setting up an AC analysis
Setting up the loading of bias points
Setting up the saving of bias points
Setting up a DC analysis
Setting up a Monte Carlo/worst-case analysis
Setting the bias point detail

Setting digital options
Setting up a parametric analysis
Setting up a sensitivity analysis 40
Setting the temperature
Setting up a transient analysis 42
Simulating your circuit
Interacting with a simulation
Extending a transient analysis
Interrupting a simulation
Scheduling changes to runtime parameters
Running multiple simulations
Using the Simulation Manager
Understanding the Simulation Manager 55
Available functionality of the Simulation Manager
Error message handling by the Simulation Manager
Setting up multiple simulations 58
Starting, stopping, and pausing simulations
Attaching PSpice to a simulation 60
Setting options in the Simulation Manager 60
The Simulation Manager File menu 62
The Simulation Manager Edit menu 63
The Simulation Manager View menu 63
The Simulation Manager Simulation menu 64
The Simulation Manager Tools menu
The Simulation Manager Toolbar 66
Entering distributions
<u>Using markers</u>
Limiting waveform data file size
Setting data collection options
Suppressing data from a transient run
Assigning marker colors
Viewing results
Viewing results as you simulate
Configuring PSpice Display of Simulation Results
Viewing Monte Carlo histograms
Defining part properties needed for simulation

Handling unmodeled pins
Saving a copy of your project
Setting up your design for simulation
Files needed for simulation 77
Files that Design Entry Programs generate
Other files that you can configure for simulation
Files that PSpice generates
Checklist for simulation setup 80
When netlisting fails or the simulation does not start
Using parts that you can simulate
Part naming conventions
Finding the part that you want
Passive parts
Breakout parts
Behavioral parts
Defining part properties needed for simulation
Specifying values for part properties
Using global parameters and expressions for values
Expressions
Defining stimuli
Things to watch for
Performing Circuit Analysis 109
Analyzing waveforms with PSpice 109
What is waveform analysis? 109
Performing post-simulation analysis of the results
Pinpointing design errors in digital circuits
What are the features of PSpice simulation profiles? 110
Increased reusability of simulation profiles 110
Support for profile level configuration 110
Creating a new simulation profile 111
Using a simulation profile
Editing a simulation profile
Deleting a simulation profile 114
Viewing the simulation queue
General simulation settings for simulation profiles 115

Analysis settings for simulation profiles
Configuration Files Tab
Stimulus settings for simulation profiles 117
Library settings for simulation profiles 118
Include files settings for simulation profiles 119
Options for simulation profiles
Data collection options for simulation profiles
Probe windows settings for simulation profiles
<u>Traces</u>
Adding traces
Viewing trace information
Editing trace display properties
Setting grid display properties
Setting plot edge properties
Defining analog trace expressions
Defining digital trace expressions
Narrowing the list of output variables 144
Deleting traces
Setting the digital plot size
Using cursors
Moving cursors along a trace
Changing views
Creating a Fourier Transform
Cautions when using FFTs
Changing axis settings
Adding a new Y axis
Deleting a Y axis
Using multiple plots
Using Probe windows
Toggling between display modes
Keeping the Probe window visible at all times
<u>To print plots</u>
Using Display Control
Using plot window templates
<u>Creating a plot window template</u>
Modifying a plot window template

Deleting a plot window template 170	0
Copying a plot window template 17	1
Restoring a plot window template	1
Viewing the properties of a plot window template	3
Loading a plot window template 174	4
Placing plot window template markers 174	4
Labeling plots	7
Editing labels	0
Copying Probe data to other applications	2
Loading large data file	3
Importing traces	3
<u>Import Traces</u>	4
Using performance analysis and measurements 187	7
Using Performance Analysis	7
Using Measurement Expressions	7
Composing Measurement Expressions	8
Measurement Expression Example 189	9
Viewing Measurement Results	0
Evaluating a measurement	0
Measurement Results Example	1
Measurement Definitions Included with PSpice	2
Creating Custom Measurement Definitions 194	4
Composing a New Measurement Definition	5
Managing Measurements	6
Copying a Measurement Definition	6
Editing a Measurement Definition 197	7
Measurement Definition Example	7
Measurement Definition Syntax 198	8
Measurement Name Syntax	9
Marked Point Expression Syntax	0
Comments Syntax	0
Search Command Syntax	1
Limiting a Measurement Expression to a Specific Range of Data	6
Control Elements in Dialog Boxes	7
Introducing the Performance Analysis Wizard	9
Selecting a Measurement	0

Selecting Measurement Arguments	210
Testing the Measurement	211
<u>Setting Options</u>	213
Setting PSpice Application Theme	213
Setting Probe window options	213
Setting large data file options	214
Selecting a Printer	215
Using Print Preview	215
Setting up the Page	216
Printing in PSpice	217
Setting the Width of Printed Plot Lines	217
Changing the Screen Colors	218
Header and footer codes	219
Customizing toolbars	219
Customizing Commands	221
Generating a Report	221
Reference Information	223
Using files	223
PSpice default keyboard shortcuts	224
Invalid node names	225
Limits in PSpice and Probe	226
Files and Commands	227
Using .PRB files	227
Loading .PRB files	227
PRB file	228
Moving data to other applications	228
Logging commands	229
Creating and changing macros	230
<u>CSDF</u>	232
Binary	233
Specifying default command line options	233
Configuring the pspice.INI file	233
Calculating Hysteresis Core Loss	244
Descriptions of menus	249
The File menu	249

The Edit menu
<u>The View menu</u>
The Simulation menu
<u>The Trace menu</u>
The Plot menu
<u>The Tools menu</u>
The Window menu
<u>Terms used in File</u>
Export Data
Find dialog box
Regular expressions
The Large Data File dialog box
Using the Quick Reference Card 263
Keyboard shortcuts
PSpice toolbars
Index of PSpice symbol and part properties
Introduction to device equations
Making device model changes
Changing a parameter name
Giving a parameter an alias
Adding a parameter
Changing the device equations
Functional subsections of the device source file
Adding a new device
Specifying new internal device structure
Recompiling and linking the device equations option
Personalizing your DLL
Simulating with the device equations option
Selecting which models to use from a Device Equations DLL
<u>Popup Menu Items</u>
PSpice Errors and Solutions
.PROBE and .ALIAS must agree on /CSDF
Invalid device type
Maximum number of alias nodes exceeded
Unable to open index file
Model type unknown

Duplicate library entry for amedalnemes
<u>Onable to open stimulus life</u>
Model references form circular list. For example:
Unable to open probe file
Unable to make index for library file
<u>Model < modelname> referenced by model < modelname>, is undefined</u>
Subcircuit <inename> used by <inename> is underined</inename></inename>
Unable to open library
Making new index file for library file
Missing model name in library 385 Missing model type in library 385
Missing model type in library
Missing subcircuit name
<u>IVON - VOFF too small for VSWITCH model</u>
RON or ROFF less than or equal to zero for VSWITCH model
RON or ROFF greater than 1/GMIN for VSWITCH model
RON = ROFF for VSWITCH model 386
ION - IOFFI too small for ISWITCH model
RON or ROFF less than or equal to zero for ISWITCH model
RON or ROFF greater than 1/GMIN for ISWITCH model
RON = ROFF for ISWITCH model
<pre><param/> not a subcircuit param</pre>
Less than 2 connections at node
Node is floating
Invalid radix, expecting BIN (1), OCT (3), or HEX (4)
Unrecognized parameter
Tolerances on model <modelname> ignored due to <tolerance></tolerance></modelname>
MC or .WCASE ignored (No < analysis type> command in circuit)
No models had tolerancesMC or .WCASE ignored
The circuit matrix is singular and cannot be solved
The circuit matrix is too close to being singular to solve
Convergence problem
Convergence problem
Time step is too small in Transient Analysis at xxx
Missing or invalid expression

Missing expression
Bad radix spec
LABEL invalid in REPEAT loop
Missing goto label
GOTO invalid in REPEAT loop
HREPEAT missing FOR or FOREVER
Attempt to redefine builtin name
<u>Must be D</u>
Must be I or V or D
<u>Must be I or V</u>
<u>Must be V</u>
Must be I or V, D not allowed
Expression not allowed here
Unknown parameter
Probability must not be less than 0
At least two pairs of numbers necessary 395
Please simplify distribution too complicated
Use RLGC & LEN for lossy line
Use Z0 & TD or F/NL for ideal line
Z0 or RLGC parameters must be specified
TD or F must be specified
BadTransferFunction
Missing REPEAT iteration count
Symbols Table overflow
Voltage Source and/or Inductor Loop Involving xxx
Convergence problem
Convergence problem
Invalid Outside of .SUBCKT
Library Index File Does Not Have the Correct Format
Unable to Find Library File
Library File Has Changed Since Index File Was Created
The Timestamp Changed from xxx to yyy 399
Model <modelname> Used by <filename> Is Undefined</filename></modelname>
Missing param name in library 400
There Are No Devices in This Circuit (This Message Will Be Printed)
Only one .TEMP value allowed with .STEP 400

Only one .TEMP, .DC TEMP, or .STEP TEMP permitted
Unable to open file
Missing .ENDS in .SUBCKT 401
Name on .ENDS does not match .SUBCKT 401
Invalid device in subcircuit
<u>Subcircuit <filename> is Undefined</filename></u> 402
Incorrect Number of Interface Nodes for <filename></filename>
Digital Simulator Option not present 402
Cannot Open Temporary Digital File 403
Missing model
Missing number of nodes
Too few output nodes specified
Bad or missing parameter 404
Invalid value
Undefined parameter used in expression 404
Undefined Parameter: <pre>oparameter></pre>
I(node) is not valid
Must be independent source (I or V) 405
Digital node table overflow
Missing parameter
Not a valid parameter for model type 406
<u>Must be 'l' or 'V'</u>
Missing node number
Missing device name
Analog simulator option not present
Invalid parameter
Inductor part of this K device 407
Inductor part of another core device 408
Transmission line part of this K device
Invalid specification 408
Bad value
Invalid number
No analog devicesDC sweep ignored 409
No analog devicessmall-signal analysis Ignored
<u>Missing value</u>
EOF in subcircuit

Errors and Solutions	. 411
Unable to write to disk: check if disk is full	. 411
Unable to read from file - improper mode	. 411
Unable to write to file - improper mode	. 411
Cannot open file: filename	. 411
Cannot open directory for backup directory name	. 412
File error, fseek failed	. 412
File error, ftell failed	. 412
File error, cannot reopen	. 412
Previous error in opening file	. 412
Backup failed: Permission denied to file	. 413
Backup failed: Bad file number	. 413
Backup failed: Cannot write to file	. 413
Extension can only have up to three characters	. 414
Filenames can have only up to eight characters	. 414
File name contains an invalid character	. 414
Blanks are not allowed in file names	. 414
Directory in path does not exist	. 414
No filename?!	. 414
Cannot open temporary file	. 415

About PSpice

What is PSpice?

PSpice¹ 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 PSpice Simulator is installed. However, all information about PSpice provided in this manual is true for PSpice 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…	Click this topic
What Probe is	What is Probe?
The types of analyses you can run with PSpice …	Types of analyses you can run with PSpice
More resources and training for PSpice	Additional sources of information about PSpice

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.

PSpice Help About PSpice

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…	<u>Using Probe windows</u>

Types of analyses you can run with PSpice

Basic analyses

Click any of the following analysis types for more information:

- AC sweep and noise
- <u>DC sweep & other DC calculations</u>
- Transient and Fourier

Advanced multi-run analyses

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

Click any of the following analysis types for more information:

- Parametric and temperature
- Monte Carlo and sensitivity/worst-case
- Frequency Response Analysis

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

Caution **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	Small-signal DC gain, input resistance, and output resistance
transfer	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	Voltages, currents, and digital states tracked over time
	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
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.

For this You can step one of these... analysis...

Parametric	global parameter
-	model parameter
-	component value
	DC source
-	operational temperature
Temperature	operational temperature

Monte Carlo and sensitivity/worst-case

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

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.

Frequency Response Analysis

Frequency Response Analysis is typically used to get frequency response of the non-linear switching circuits having varying operating point. This analysis injects transient signals into

PSpice Help About PSpice

the loop of interest and then extracts frequency data using Fourier analysis to plot gain/phase response.

Following options have been added in Capture – PSpice flow to enable Frequency Response Analysis:

MINSIMPTS

Specifies minimum number of time points used for a transient simulation. More the number of points better would be the output resolution.

To use MINSIMPTS, add the following comment text on the Capture Schematic:

@PSpice:

.options MINSIMPTS=1000

■ .PROBE64 P(<>)

Writes parameter defined using .PARAM command on the probe.

To use .PROBE64 command, add the following comment text on the Capture Schematic:

@PSpice:

.PROBE64 P(FREQ)

Note: The text following <code>@PSpice:</code> is treated as SPICE directive.

For more information on the usage of MINSIMPTS option and .PROBE64 command in Frequency Response Analysis, see the example at <Installation Directory> \tools\pspice\capture_samples\anasim\fra.

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 editors¹, others come from libraries, and still others are user-defined.

Files Generated by Schematic Editors or Design Entry Programs

Capture is a design entry program in which 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

For more information about designing circuits with Capture, 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.

^{1.} Schematic editor or design entry programs refer to OrCAD Capture.

Related Topics

For information about	Click this topic
What model libraries are	Model libraries
What stimulus files are	<u>Stimulus files</u>
What include files are	Include files
Preparing and configuring input files	Preparing and configuring input files

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
- subcrcuit 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.

Stimulus files

A stimulus file contains time-based definitions for analog and/or digital input waveforms. You can create a stimulus file using a standard text editor, to create the definition (a typical file extension is .STM).

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 Capture as the primary design entry program. Entering the design in Capture 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 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 VPLOT1) that you place and connect to nets in your design

Documentation

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

- 1. From the programs folder in the Windows Start menu, choose *PSpice for TI 2020 PSpice for TI 2020 Help*.
- 2. From the Library pane, 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.

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.

PSpice Help About PSpice

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

In order to simulate a design with PSpice that is created in the design entry programs, Capture, 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 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.

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.

or

 directly from within Capture 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 detailed information about generating simulation netlists from design entry programs, refer to Capture User Guide.

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 <u>Creating a</u> <u>new simulation profile</u> 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:
 - □ <u>Setting up an AC analysis</u>
 - □ <u>Setting up a parametric analysis</u>
 - Setting up the loading of bias points
 - □ <u>Setting up a sensitivity analysis</u>
 - Setting up the saving of bias points
 - □ <u>Setting the temperature</u>
 - □ <u>Setting up a DC analysis</u>
 - □ <u>Setting the bias point detail</u>
 - Setting up a Monte Carlo/worst-case analysis
 - □ <u>Setting digital options</u>
 - □ <u>Setting up a transient analysis</u>
- 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 <u>Linear</u> or Logarithmic. (If Logarithmic, also select <u>Decade</u> or <u>Octave</u>.) 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:
 - D Primary Sweep value
 - □ <u>Secondary Sweep value</u>
 - Parametric Sweep value
 - <u>Monte Carlo options</u>
 - <u>Temperature Sweep temperature</u>
- 6. Under Options, select Do Not Save Subcircuit Voltages and Currents if you do not want to save the node voltages and currents for subcircuits.
- 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 <u>Voltage source</u>, <u>Current source</u>, <u>Global parameter</u>, <u>Model parameter</u>, or <u>Temperature</u>.
- 5. Under Sweep type, choose <u>Linear</u>, Logarithmic, or <u>Value list</u>. (If Logarithmic, also select <u>Decade</u> or <u>Octave</u>.)
- 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:

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

<none></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 <u>Vary both DEV and LOT</u>, Vary <u>DEV</u>, Vary <u>LOT</u>.
- 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 <u>Hi</u> or <u>Low</u>.
- **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
- Let 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
 - □ <u>Calculate small-signal DC gain</u>

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:
 - □ <u>Voltage source</u>
 - □ <u>Current source</u>
 - Global parameter
 - □ <u>Model parameter</u>
 - □ <u>Temperature</u>
- **5.** Under Sweep type, choose one of the following:
 - □ <u>Linear</u>
 - Logarithmic: <u>Decade</u> or <u>Octave</u>
 - 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 subcrcuit
- **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 <u>Creating a new simulation profile</u> 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 <u>Probe windows settings</u> 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.
- □ 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 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	Interacti	ve simulation functionality
PSpice		extend transient analysis
		interrupt a simulation, change parameters, and resume the simulation
		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 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 blankor- 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.
Running a single AC/DC simulation using a	PSpice will stop (terminate) after a successful simulation.
profile or a circuit	-or-
circuit.	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	If the old simulation has completed, PSpice will load the new simulation and run it.
in PSpice.	-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 <u>Running</u> <u>multiple simulations</u>.

To extend a transient analysis

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



2. Click 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 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
- D ITL1
- □ ITL2
- D ITL4
- □ Autoconvergence
- Enable Advanced Convergence Algorithm

Note: For more details about using these runtime parameters, see the 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 <u>Using the Simulation Manager</u>.)

Caution

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 Runtime Settings				—
	Use Original Value	Change To		
Relative accuracy of V's and I's:	📝 0.001	0.001	(RELTOL)	ОК
Best accuracy of currents [amps]:	📝 1e-012	1e-012	(ABSTOL)	OK & Resume Simulation
Best accuracy of voltages [volts]:	📝 1e-006	1e-006	(VNTOL)	Cancel
Minimum conductance for any branch [1/ohm]:	📝 1e-012	1e-012	GMIN)	
Run to time [seconds]:	V 1	1	(TSTOP)	Help
Maximum step size [seconds]:	\checkmark		(TMAX)	
DC and bias "blind" iteration limit:	V 150	150	(ITL1)	
DC and bias "best guess" iteration limit:	V 20	20	(ITL2)	
Transient time point iteration limit:	V 10	10	(ITL4)	
Autoconverge		Settings		
Enable Advanced Convergence Algorithms		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.

- D RELTOL
- □ ABSTOL
- D VNTOL
- GMIN
- D ITL4

Note: For more details about using these runtime parameters, please see the 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:

```
.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 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.

Related Topics

For information about…	Click this topic
Using the Simulation Manager	Using the Simulation Manager
Setting up multiple simulations	Setting up multiple simulations
Starting, stopping, and pausing simulations	Starting, stopping, and pausing simulations
Attaching PSpice to a simulation	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.

PSpice Simulation N	Manager			×
File Edit View Sir	nulation Tools Help		cāden	ce
9 🛃 🕹 🏠	🗙 🍫 🕨 🕨	III 🐺 🖻		
Simulation	Schedule	Status	Percent Complete	
tran.sim	stopped	Time= .014	1%	
tran.sim	stopped	complete - no errors	100%	
📕 tran.sim	stopped	complete - no errors	100%	
Ready				h.

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

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 the name of that menu in the image.



Related Topics

For information about	Click this topic
Understanding the status of jobs in the queue	Understanding the Simulation Manager
What the various versions of PSpice support	Available functionality of the Simulation Manager
How the Simulation Manager handles errors	Error message handling by the Simulation Manager
Setting up multiple simulations	Setting up multiple simulations
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

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.

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=""></status>	Basic status information about the progress of the analysis will be displayed for a simulation that is currently running.
paused	The simulation has been paused either manually or automatically by the Simulation Manager.
	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.
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></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	 One simulation may be running and multiple simulations may be paused. 	
	The queue is run sequentially.	

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.



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.



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 <u>Setting options in the Simulation Manager</u>.

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 <u>Setting options in the Simulation</u> <u>Manager</u>.

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.

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.

Specify the default response when attempting to start a simulation from another application while a simulation is currently running:	OK Cance
Oisplay the simulation in the queue.	
Pause the current simulation and run the new one.	
Stop the current simulation and run the new one.	
When a simulation completes, the first paused simulation	
in the list should:	
Hesume simulating.	
At the second	

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.)
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.



Menu command... Function...

Add Simulation	Opens a simulation file (.SIM) or circuit file (.CIR) and adds it to the queue.
	Shortcut: INS
<mru list=""></mru>	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 <u>The Simulation Manager Toolbar</u>.



Menu command	Function
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.

Vie	w
R ,	Simulation <u>R</u> esults
B	Output Fi <u>l</u> e
	<u>S</u> tatus Bar
9	Always <u>O</u> n Top

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.



64

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.

Tools	No.
<u>Options</u>	
Custom	ize

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

Related Topics

For information Click this topic... about... Options that can be Setting options in the Simulation Manager...

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	
9	Keeps the Simulation Manager on top of all other open applications.	
Always On Top		
<u>~</u>	Adds a file to the queue. Shortcut: INS	
Add Simulation		
₽	Moves the selected file down one position in the queue	
Move Down arrow		
†	Moves the selected file up one position in the queue.	
Move Up arrow		
×	Deletes the selected file from the queue. Shortcut: DEL	
Delete		

Edit Simulation Profile	Opens the simulation profile for the selected simulation and allows you to change the analysis settings.
	Runs all of the simulations that are queued. The simulations will run in the order in which they are listed in the queue.
Run Queued Items	
	Runs the selected simulation(s).
Run Selected	
	Stops the selected simulation(s).
Stop Selected	
	Pauses the selected simulation(s).
00	
Pause Selected	
No.	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:

```
(<deviation>, <probability>)
```

- **4.** To remove a distribution from the current simulation profile, under *Existing distributions*, select the distribution name and click *Delete*.
- 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.

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 subcrcuit 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 <u>Suppressing data from a transient run</u>.

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 <u>Setting data collection</u> 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.
- 2. Click the *Data Collection* tab.
- **3.** In the *Data Collection Options* section, 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 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.

4. Select *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. Choose *PSpice Edit Simulation Profile* to open the Simulation Settings dialog box.
- 2. Select the Analysis tab.
- 3. From the Analysis Type list, select the Time Domain (Transient) option.
- 4. In the Start saving data after text box, specify a delay time.
- 5. Click *OK* to close the Simulation Settings dialog box.
- 6. Choose *PSpice 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 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. Choose *PSpice Edit Simulation Profile* 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:
 - Choose *PSpice Edit Simulation Profile*.
 - 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 *PSpice – Edit Simulation Profile*, click the *Probe Window* tab, and then select one 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.
- 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.

PSpice Help Preparing your design for simulation



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

1From the Window menu, choose Copy to Clipboard.

The histogram graph will be transferred to the Windows clipboard.

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
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	the pin property specifying the net name to which a power (invisible) pin is connected.
	■ Whenever you define a hidden pin for a part, the part editor automatically creates an PSPICEDEFAULTNET property.

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 <u>Index of PSpice</u> symbol and part properties on page 275

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. Right-click and select View PSpice Model.

The model definition opens in a new tab and cannot be edited.

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. **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.

Note:

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, others come from libraries, 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.

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
- subcrcuit 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, using a text editor like Notepad

Stimulus file

You can create a stimulus file by using a standard text editor to create the definition (a typical file extension is .STM).

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.



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.

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

PSpice Help Setting up your design for simulation

	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.

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 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.

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	Your operating system manual
containing your	
design has write	
permission.	

```
Your system has Your operating system manual sufficient free memory and disk space.
```

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.

Libraries are provided with PSpice: Standard PSpice 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 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\psice\library\

You can find some information about a particular library (.lib) when you open it in a text editor and read the comments in the file.

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 the part browser in Capture and restricting the parts list to those names that match a specified wildcard text string.

From Capture, select *Place – PSpice Component – Search*. The PSpice Part Search window opens.

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.

= Tip

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

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.

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 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	С
L	inductor	L
R R_VAR	resistor	R
XFRM_LINEAR K_LINEAR	transformer	K and L
Т	ideal transmission line	Т
TLOSSY	Lossy transmission line	Т
TnCOUPLED TnCOUPLEDX ¹ KCOUPLE ¹	coupled transmission line	T and K

1. For these device types, the PSpice libraries supply several parts. Refer to the 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.

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 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	В
CBREAK	capacitor	С
DBREAKx	diode	D
JBREAKx*	JFET	J
KBREAK	inductor coupling	К
LBREAK	inductor	L
MBREAKx*	MOSFET	М
QBREAKx*	bipolar transistor	Q
RBREAK	resistor	R
SBREAK	voltage-controlled switch	S
TBREAK	transmission line	Т
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 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></id>	Value of <id>. Error if no <id> property or if no value assigned.</id></id>
& <id></id>	Value of <id> if <id> is defined.</id></id>
? <id>ss</id>	Text between ss separators if <id> is defined.</id>
? <id>ssss</id>	Text between the first ss separators if $\langle id \rangle$ is defined, else the second ss clause.
~ <id>ss</id>	Text between ss separators if <id> is undefined.</id>
~ <id> ssss</id>	Text between the first ss separators if $\langle id \rangle$ is undefined, else the second ss clause.
# <id>ss</id>	Text between ss separators if <id> is defined, but delete rest of template if <id> is undefined.</id></id>

* 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.



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.

This pin name character	Is replaced with this
<	l (L)
>	g

PSpice Help Setting up your design for simulation

=	е
\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.

PSpice Help Setting up your design for simulation

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.

Assign this value	To use this interface subcircuit (level)
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.

Assign this value	To use this propagation delay
0	circuit-wide default
1	minimum
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 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?1K. If you specify the tolerance value as an absolute number, say 10, the distribution values will be taken in the range of 10K?1C.

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:
 - □ 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*.

Caution

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
	Ι	boolean OR
	٨	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 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 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
periodic pulse	VPULSE
piecewise-linear	VPWL
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
sine wave	VSIN

** 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.

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
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 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 PSpice Reference Guide.

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.

PSpice Help Setting up your design for 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.

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 the pin to open the Property Editor spreadsheet.
- **2.** Click 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.
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 <u>Using</u> <u>Performance Analysis</u> 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 <u>Creating a new simulation profile</u>.

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 use PSpice to create a new simulation profile only by importing settings from a simulation profile that exists in the same project or in another project.

To import settings from an existing simulation profile to create a new simulation profile, do the following:

1. From the *File* menu in PSpice, point to *New* and choose *Simulation Profile*.

The New Simulation dialog box appears.

- 2. In the *Profile name* text box, specify a name for the profile (such as the name of the analysis type for the new profile).
- **3.** Click the Browse button to specify the location and name of an another simulation profile (.sim) to import its settings into the new profile.
- 4. Click *Create* to create the profile and display the Simulation Settings dialog box.

A new simulation profile is created and can bee seen under *PSpice Resources* – *Simulation Profiles* of the project which you had selected to import the existing simulation profile in the *Inherit from an existing profile* text box.

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

General	General simulation settings for simulation profiles
Analysis	Analysis settings for simulation profiles
Configuration Files	Library settings for simulation profiles
	Stimulus settings for simulation profiles
	Include files settings for simulation profiles
Options	Options for simulation profiles
Data Collection	Data collection options for simulation profiles
Probe Window	Probe windows settings for simulation profiles

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.

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	General simulation settings for simulation profiles
Analysis	Analysis settings for simulation profiles
Configuration Files	Library settings for simulation profiles
	Stimulus settings for simulation profiles
	Include files settings for simulation profiles
Options tab	Options for simulation profiles
Data Collection tab	Data collection options for simulation profiles
Probe window	Probe windows settings for simulation profiles

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:
 - D 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 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* drop-down list box, select the basic analysis type you want to use. For more information about these, see <u>Setting up analyses</u>.

Analysis Options

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

Configuration Files Tab

Use the *Configuration Files* tab of the Simulation Settings dialog box to configure and edit the following:

- <u>Stimulus settings for simulation profiles</u>
- Library settings for simulation profiles
- Include files settings for simulation profiles

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	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.
	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.
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 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.

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 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
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.
	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.

PSpice Help Performing Circuit Analysis		
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.	
★ and	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:\Cadence\SPB_17.4\tools\pspice\library then C:\MYLIBS, type the following:

"C:\Cadence\SPB_17.4\tools\pspice\library";"c:\mylibs"

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.

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, Analog Advanced, Gate-level Simulation, or Output file to display the settings for each option category.

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 in the text box correspond to the option names used in the PSpice .OPTIONS command. For more information about this command, refer to the PSpice Reference Guide.

Click this button	To do this
General	Enter values for speed level, tolerances, and minimum conductance.
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.
AutoConverge	Suggest relaxed limits for various options that PSpice can modify during a simulation to achieve convergence.
Restart	Restart the convergence calculation
PREODER	Presorts the matrix diagonal by Markowitz counts.

Options	Description	Units	Default
General section			

Ontions	Description	Unite	Default
		Unita	
SPEED_LEVEL	ncreases simulation performance by optimizing switching behavior of models. If increase in simulation performance is not needed, set SPEED_LEVEL=0.	-	3
RELTOL	relative accuracy of V and I		0.001
VNTOL	best accuracy of voltages	volt	1.0 uV
ABSTOL	best accuracy of currents	amp	1.0 pA
CHGTOL	best accuracy of charges	coulomb	0.01 pC
GMIN	minimum conductance used for any branch	ohm ⁻¹	1.0E-12
Auto Converge section			
ITL1	Maximum number of iterations of convergence calculation during bias point analysis		150.0
ITL2	Maximum number of iterations of convergence calculation during DC analysis		20
ITL4	Maximum number of iterations of convergence calculation at each time step during transient analysis		10
TNOM	Nominal Temperature value	Celsius	27 degrees Celsius
THREADS	When THREADS=0, PSpice uses the number of threads that are equal to the number of cores detected in the system.		0
	When THREADS is anything other than 0, PSpice uses the exact number of threads as specified.		
	It is recommended to set THREADS to 0 for PSpice to decide the optimal number of threads to be used.		
PIVTOL	absolute magnitude required for pivot in matrix solution		1.0E-13

Options	Description	Units	Default
MOSFET Section			
DEFAD	MOSFET default drain area (AD).	meter ²	0.0
DEFAS	MOSFET default source area (AS).	meter ²	0.0
DEFL	MOSFET default length (L).	meter	100.0 u
DEFW	MOSFET default width (W).	meter	100.0 u

Analog Advanced options

Use the Analog Simulation settings to enter values for the total transient iteration limit, relative magnitude for matrix pivot, and absolute magnitude for matrix pivot.

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

Click this	To do this
General	Enter values for speed level, tolerances, and minimum conductance.
Bias Point	Suggest relaxed limits for various options that PSpice can modify during a simulation to achieve convergence.
Transient	Enter values for the default drain area, default source area, default length, and default width.

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

Flag option	Meaning
NOGMINI	Specifies not to add GMIN across current sources.
BRKDEPSRC	Sets automatic break-points for behavioral sources.
CONVAID	Generates .10P file for debugging purpose when convergence fails.

Flag option,	Meaning, continued
STEPGMIN	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.
NOSTEPSRC	Do not run source stepping algorithm for bias point convergence.
NOSTEPDEP	Do not step dependent sources during source stepping algorithm for bias point convergence.
GMINSRC	Enables step GMIN inside source-stepping
PSEUDOTRAN	Uses Pseudo-Transient Method.

Options	Description	Units	Default
ITL5	total repeating limit for all points for transient analysis (ITL5=0 means ITL5=infinity)	-	0.0
PIVREL	relative magnitude required for pivot in matrix solution	-	1.0E-3
PIVTOL	absolute magnitude required for pivot in matrix solution	-	1.0E-13
SOLVER	performance package solution algorithm	-	1
	(Solver = 0 selects the original solution algorithm;		
	Solver = 1 selects the advanced solution algorithm)		
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 $.0001$.		
WCDEVIATION	worst case deviation. It can have double values between 0 and 1.	-	Same as RELTOL

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
DIODECJO	Minimum value for Diode junction capacitance	ohm	0
DIODERS	Minimum value for Diode ohmic resistance	ohm	0
ВЈТСЈ	minimum value for BJT Base-collector zero- bias depletion capacitance (Cjc), Base-emitter zero-bias depletion capacitance (Cje), and zero-bias collector substrate capacitance (Cjs)	farad	0
GMINSTEPS	the GMIN stepping size in integer (any positive value). Set to 0 for engine default.	-	Same as ITL1
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
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
method	integration method (values can be either TRAPEZOIDAL or GEAR)	-	-
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
CSHUNT	shunt capacitance added from all nodes of the design to GND. Recommended value is 1pF.	farad	0
TRANCONV	Enables alternate path search if transient simulation fails.		

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
General	Enter values to set delay or initial state in flip-flops or latches.
Advanced options	Enter values for the minimum output drive resistance, maximum output drive resistance, overdrive ratio, default delay calculation, and error message limits.

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

Flag option	Meaning
NOPRBMSG	Suppresses simulation error messages in Probe data file.

DIGMNTYMX	default delay selector: 1=min, 2-typical, 3=max, 4=min/max	-	2.0
DIGINITSTATE	sets initial state of all flip-flops and latches in circuit: 0=clear, 1=set, 2=X	-	2.0
DIGIOLVL	default digital I/O level: 1-4;	-	1.0
DIGDRVF	minimum drive resistance (Input/Output UIO type model, DRVH (high) and DRVL (low) values)	ohm	2.0
DIGDRVZ	maximum drive resistance (UIO type model, DRVH and DRVL values)	ohm	20K
DIGOVRDRV	ratio of drive resistances required to allow one output to override another driving the same node	-	3.0
DIGMNTYSCALE	scale factor used to derive minimum delays from typical delays	-	0.4
DIGTYMXSCALE	scale factor used to derive maximum delays from typical delays	-	1.6

DIGERRDEFAULT	default error limit per digital constraint device	-	20.0
DIGERRLIMIT	maximum digital error message limit	-	0

Output file options

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

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).	
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).	
NOMODE	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.	
NUMDGT	Number of digits in printed values. This is 4 by default.	
WIDTH	same as the .WIDTH OUT= statement (can be set to either 80 or 132)	
	Default is 80.	

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.

Simulation	n Settings -	TRAN				×
General	Analysis	Configuration Files	Options	Data Collection	Probe Window	
- Data (Collection O	ptions				
	Voltages:	All		-		
	Currents:	All		+		
	Power:	All		•		
	Digital:	All		•]	
	Noise:	Al		-	·	
Probe	Data: ve data in th	⊚ 32-bit ne CSDF format (.CS!	ම 64-b С)	it		
			ОК	Cancel	Apply	Help

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, see <u>Setting data</u> <u>collection options</u>.

Data Collection Options

Option	Description
A11	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 <u>Setting Probe window options</u>.

Probe window display options

Use this option...

To do this...

Show the traces:

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

Display the Probe windows when:

the simulation is finished.

Display Probe window:

- during simulation
- the simulation is running, and update the waveforms as the simulation progresses.
- after simulation has completed.

Show

- All markers on open schematics
- 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.
- Last plot

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.



You can add traces that use data from individual data files loaded into PSpice. Click here <u>To use a function or arithmetic operator</u> 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 <u>Narrowing the list of output variables</u> 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 <u>Defining analog trace expressions</u>for more information on using operators, functions, and macros for analog traces.
 - □ Click here <u>Defining digital trace expressions</u> 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 Settings 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, 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**
- □ <u>To use a sweep variable</u>

^{__} Tip

You can add traces that use data from individual data files that you loaded into PSpice. Click here <u>To use a function or arithmetic operator</u> for an example.



Device noise is available in addition to total input and output noise. These are generated only if you run a noise analysis. Click here <u>Device noise variables</u> to display the device noise output variables.

Note: Click here <u>How noise units are reported</u> for information on how noise units are reported.

Analog Operators

Valid analog arithmetic operators:

()	grouping
* /	multiplication/division
+ -	addition/subtraction
G	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 <i>Add Trace</i> window.

Analog Functions

Valid analog arithmetic functions:

ABS(x)	x
ARCTAN(x)	arc tangent of x with results in radians

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	
	Note: 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)	In(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	
	Note: 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 (VOut1) 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.

U(<node_name>) or V(<node_name_1>, <node_name_2>)

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

Vx(<device_name>) and lx(<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.
- \Box 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 <u>Device noise variables</u> to display the device noise variables. Click here <u>How noise units are reported</u> 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)

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)
		NTOT (total noise)
М	(MOSFET)	NFID (Idrain flicker noise)
		NRB (RB thermal noise)
		NRD (RD thermal noise)
		NRG (RG thermal noise)
		NRS (RS thermal noise)
		NSID (Idrain shot noise)
		NTOT (total noise)
Ν	(Digital Input)	NRHI (resistance noise between the digital device output and its PWR pin)
		NRLO (resistance noise between the digital device output and its GND pin)
		NTOT (total noise)
0	(Digital Output)	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 Is reported in these units...
device contribution of the (volts)^2/(Hz)
form Nxxx
total input or output noise (volts)/(\sqrt{Hz})
of the forms V(ONOISE) or
V(INOISE)
```

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:

- □ <u>To add a digital signal</u>
- To add a bus
- □ <u>To add a digital signal or bus constant</u>
- □ <u>To use a digital operator</u>
- To add a digital signal or bus constant
- □ <u>To use a digital operator</u>

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.
```

```
<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:

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

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

Display name	(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
0	octal
В	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 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

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
<u>`1</u>	high
Ϋ́F	falling
`R	rising
`Χ	unknown
Ϋ́Ζ	high impedance

bus constant radix

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

h or x	hexadecimal
d	decimal
0	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 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:

- □ <u>To display both cursors</u>
- □ <u>To move the cursors using the mouse</u>
- <u>To apply the cursors to a different trace</u>
- <u>To change the number of digits displayed</u>
- □ <u>To move the cursors using the keyboard</u>

To display both cursors

Do one of the following:

- On the toolbar, click the Toggle Cursor button.
- From the Trace menu, choose *Cursor Display*.

The dockable cursor window 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 this cursor window 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.

To freeze the cursor, do one of the following:

→ Choose *Trace – Cursor – Freeze* to freeze the cursor locations on the current trace.

OR

1. Choose *Tools – Options*.

The Probe Settings dialog box opens.

- 2. Select the Cursor Settings tab.
- 3. Select Show non-dockable(old) cursor window.

The Probe Cursor window appears.

4. Click the Probe Cursor window (below the title bar) to freeze the cursor locations on the current trace.

To change the number of digits displayed

- **1.** Choose *Tools Options*.
- 2. Select the *Cursor Settings* tab.
- 3. In the *Number of cursor digits* text box, type a number between 2 and 18.
- **4.** Click *OK*.

To move the cursors using the keyboard

Use the following key combinations (directional keys on the keyboard) 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

PSpice Help Traces		
Cursor 1 to the previous or the next trace	Ctrl + right arrow or	
	Ctrl + left arrow	
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 to the beginning or the end of the	Shift + Home or	
trace	Shift + End	

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 on which the cursor is on, click a different trace symbol in the legend.

Caution

Cursor movements are not recorded during command logging.

For more information, see the following topics:

- □ <u>To turn cursors on or off</u>
- □ <u>To move the cursor to the next peak</u>
- To move the cursor to the next trough
- □ <u>To move the cursor to the next slope</u>
- D To move the cursor to a minimum or maximum point
- **To move the cursor to the next data point generated by the simulator**

- <u>To move the cursor to the next transition</u>
- To move the cursor to the previous transition
- □ <u>To mark every data point with a symbol</u>
- □ <u>To search for a specific cursor location</u>

To turn cursors on or off

→ On the toolbar, click the Toggle Cursor button.

To move the cursor to the next peak

→ On the toolbar, click the Cursor Peak button.

A first peak is a location where the points on each side of the peak have a lower Y value.

Peak is defined as the first local maxima (local peak) of the selected trace. If you click the Cursor Peak button again, cursor moves to the next local peak of the same trace in the forward direction. If you move the cursor in the reverse direction using mouse or arrow key, and then click the Cursor Peak button, cursor moves to the local peak in the that direction.

To move the cursor to the next trough

→ On the toolbar, click the Cursor Trough button.

A trough is the opposite of a peak. A first trough is a location where the points on each side of the trough have a higher Y value.

A trough is defined as the first local minima (local trough) of the selected trace. If you click Cursor Trough button again, it moves to the next local trough of the same trace in the forward direction. If you move the cursor in the reverse direction using mouse or arrow key, and then click the Cursor Trough button, cursor moves to the local trough in that direction.

To move the cursor to the next slope

→ On the toolbar, click the Cursor Slope button.

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 Cursor Min button.
 The minimum value is the minimum Y value of the selected trace.
- To see the maximum value, on the toolbar, click the Cursor Max button.
 The maximum value is the maximum Y value of the selected trace.

To move the cursor to the next data point generated by the simulator

On the toolbar, click the Cursor Point button.
 Clicking this button takes the cursor to the next data point generated by the simulator.

To move the cursor to the next transition

On the toolbar, click the Cursor Next Tran button.
 Clicking this button takes the cursor to the next transition point in a digital trace.

To move the cursor to the previous transition

→ On the toolbar, click the Cursor Prev Tran button.

Clicking this button takes the cursor to the previous transition point in a digital trace.

To mark every data point with a symbol

→ On the toolbar, click the Mark Label button.

To search for a specific cursor location

Do the following:

- **1.** Choose *Trace Cursor Display* or click the Toggle Cursor button.
- Choose Trace Cursor Search Commands or click the Cursor Search button.
 The Search Command window opens.
- 3. Specify the command sfxv() or sfle() to move the cursor to specific X and Y locations respectively.

For example:

- **To move the cursor to X = 10ns, specify** sfxv(10ns).
- **To move the cursor to** Y = 10V, specify sfle(10v).
- **4.** Click *OK*.

Important

By default, the sfxv() and sfle() commands search in the forward direction from current cursor location.

To move the cursor at a specific location in the reverse direction, use the following commands:

```
□ sbxv()
```

□ sble()

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.

- □ <u>To zoom in or out</u>
- □ <u>To change the center point</u>
- □ <u>To display the selected area</u>
- □ <u>To redraw the screen</u>
- □ <u>To see the previous view</u>
- □ <u>To fit the view</u>
- □ <u>To display the Toolbar</u>
- □ <u>To display the status bar</u>

Note: The new items were formerly part of the Probe Settings 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 check mark next to the menu command indicates that the Toolbar is displayed.

To display the status bar

From the View menu, choose Status Bar.

A check mark 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:
 - **D** To save the current display, type a name in the New Name box. Click Save.
 - **D** To save to a different location, click the Save To button. Type a new name. Click OK.
 - **D** 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.

Display Control	X
New Name LAST DISPLAY	Save
Displays Templates	Save To
LAST DISPLAY LAST SESSION	Сору То
	Delete
	Restore
	Properties
	Load
	Help
	Close

- 2. Click 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 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:

- <u>To set the x-axis</u>
- <u>To set the y-axis</u>
- <u>To set the x-grid</u>
- <u>To set the y-grid</u>

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 <u>Defining analog trace expressions</u> 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.

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:

- □ <u>To add a new plot</u>
- □ <u>To select the current plot</u>
- □ <u>To delete a plot</u>
- □ <u>To unsynchronize plots</u>

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.

Display Control	— ×
New Name LAST DISPLAY	Save
Displays Templates	Save To
LAST DISPLAY LAST SESSION	Сору То
	Delete
	Restore
	Properties
	Load
	Help
	Close

For more information, see the following topics:

- □ <u>To save a display</u>
- □ <u>To copy a display</u>
- □ <u>To delete a display</u>
- □ <u>To use a saved display</u>
- □ <u>To load displays from another .PRB file</u>

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 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 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 the Displays tab.
- **3.** Do one of the following:
 - **D** 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.

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.

Display Control	×
New Name Average	Save
Displays Templates	Save To
Average Derivative	Сору То
Falltime of Step Response (multi-run) First Peak (multi-run)	Delete
Fourier Transform Integral Overshoot of Step Besponse [multi-run]	Restore
Period [multi-run] Pulsewidth [multi-run]	Properties
Risetime of Step Response (multi-run) Rms	Load
	Help
	Close

Related Topics

For information about	Click this topic
Creating a new template	Creating a plot window template
Modifying an existing template…	Modifying a plot window template
Deleting a template…	Deleting a plot window template
Copying a template	Copying a plot window template
Restoring a template…	Restoring a plot window template
Viewing the properties of a template	Viewing the properties of a plot window template
Loading a template	Loading a plot window template
Placing plot window template markers	Placing plot window template markers

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.

Display Control	×
New Name Average	Save
Displays Templates	Save To
Average Derivative	Сору То
Falltime of Step Response [multi-run]	Delete
Fourier Transform Integral Overshoot of Step Response [multi-run]	Restore
Period [multi-run] Pulsewidth [multi-run]	Properties
Risetime of Step Response [multi-run] Rms	Load
	Help
	Close

- **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.

Save Plot Window Te	emplate - Step 1 of 2
Name	
Bode Plot d6	
Description	
Bode Plot for d6	* *
Store Template In	
Local File	D:\Cadence\SPB_16.6\tools\pspice\capture_samples\anasim\v
🔘 Global File	D:\Cadence\SPB_16.6\tools\PSpice\Common\pspice.prb
🔘 Other File 🛛 🗡	
	Next> Cancel Help

5. In the Description text box, type in a description for the template, if you would like one. (This is optional.)

- 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.

Ar	gument Assignment(s):	×++
#	Node/Pin Name	Туре	Description
1	V(out1)	voltage 🔻	
2		voltage	
3		any	
		Back Fi	nish Cancel Help

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 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 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.

est	ore Plot Window Tem	plate	
N	ame		
A	verage		
D	escription		
R	unning average over tim	e	A
			+
#	Node/Pin Name	Type	-
#	Node/Pin Name	Type any	Description Voltage, current, or power test point

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 the name of the plot window template you want to view.
- 4. Click Properties.

The Plot Window Template Properties dialog box appears.

Falltime of Step Response [multi-run] Description Measures the falltime of a step response; the first point is assumed to be the "hig value, and the last point the stable "low" value. Template Stored In Global File: D:\Cadence\SPB_16.6\tools\PSpice\Common\pspice.prb Arguments # Type Description		ep Response [multi-run]	ltime of Step	Fal
Description Measures the falltime of a step response; the first point is assumed to be the "hig value, and the last point the stable "low" value. Template Stored In Global File: D:\Cadence\SPB_16.6\tools\PSpice\Common\pspice.prb Arguments # Type Description				
Measures the falltime of a step response; the first point is assumed to be the "hig value, and the last point the stable "low" value. Template Stored In Global File: D:\Cadence\SPB_16.6\tools\PSpice\Common\pspice.prb Arguments # Type Description			scription	De
Template Stored In Global File: D:\Cadence\SPB_16.6\tools\PSpice\Common\pspice.prb Arguments # Type Description	i to be the "high" -	falltime of a step response; the first p e last point the stable "low" value.	asures the ue, and the	Me val
Global File: D:\Cadence\SPB_16.6\tools\PSpice\Common\pspice.prb Arguments # Type Description		red In	mplate Stor	Ter
Arguments # Type Description	ospice.prb	D:\Cadence\SPB_16.6\tools\PS	Global File:	1
# Type Description			juments	Arg
		Description	Туре	#
1 any Step response test point		Step response test point	any	1

- **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.

Derivative	Â	Place
First Peak [multi-run]		Cancel
Fourier Transform Integral	Ξ	
Overshoot of Step Response [mult		<u> </u>
Pulsewidth [multi-run]		Help
Risetime of Step Response [multi-ri Rms	-	
4 III +		
Description:		
Running average over time		*

- 2. Click 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	Click this topic
Plot window templates in PSpice 	Using plot window templates
Working with current, voltage or power markers…	<u>Using markers</u>
Defining simulation profiles	Creating a new simulation profile

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	<u>To place text</u>
Line	<u>To draw a line</u>
Box	<u>To draw a box</u>
circle	<u>To draw a circle</u>
poly-line	<u>To draw a poly-line</u>
Arrow	<u>To draw an arrow</u>
Ellipse	<u>To draw an ellipse</u>
mark	To mark the cursor location

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
- □ <u>To copy and paste an item</u>
- □ <u>To modify a label or an ellipse angle</u>

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 commaseparated (.csv) file. Using the import feature you can import waveforms generated by measuring instruments such as digital oscilloscope to PSpice.

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 text box.
- 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.

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.
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.
	Note: 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

PSpice Help Traces

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. 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 predefined 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
- 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 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 <u>Measurement Expression Example</u> 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

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 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 <u>Viewing Measurement Results</u>.

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 column.

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.

- 2. Select the measurement that you want to evaluate by clicking one of the entries in the *Measurements* list.
- **3.** 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 <u>Measurement</u> <u>Definitions Included with PSpice</u>.

To view information on how to create new measurements, see <u>Composing Measurement</u> <u>Expressions</u>.

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 check mark 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_XRa nge	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_XRan ge	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_NoOvershoo t	Falltime with no overshoot.
Falltime_StepRespon se	Falltime of a negative-going step response curve
Falltime_StepRespon se_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 \boldsymbol{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_NoOvershoo t	Risetime of a step response curve with no overshoot
Risetime_StepRespon se	Risetime of a step response curve
Risetime_StepRespon se_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</begin_x>
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</begin_x>
SlewRate_Fall	Slew rate of a negative-going step response curve
SlewRate_Fall_XRang e	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_XRang e	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_NegativeSlo pe	Value of X corresponding to the nth negative slope crossing of the given Y_value, for the specified waveform
XatNthY_PercentYRan ge	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=Ymin+(Ymax-Ymin)*Y_pct/100
XatNthY_Positive Slope	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

- **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.

a

Work in the Simulation Results view in PSpice. In the side toolbar, click

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

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
{
    l| search_commands_and_marked_points_for_expression_1;
```

```
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 $<\!\!\text{condition}\!\!>$, then the last $<\!\!\text{condition}\!\!>$ 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)
A db value relative to startvalue	3db (3db above startvalue)
	.+3db (3db below 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)

Decimal point (.)

A decimal point (.) represents the Y value of the last point found using a search on the current trace expression of the measurement definition. If this is the first search command, then it represents the Y value of the start point of the search.

[,posneg]

Finds the next Y value crossing at the specified level. This can be between real data points, in which case an interpolated artificial point is created.

At least [#consecutive_points#]-1 points following the level crossing point must be on the same side of the level crossing for the first point to count as the level crossing.

[,posneg] can be Positive (P), Negative (P), or Both (B). The default is Both.

SLope[(posneg)]

Finds the next maximum slope (positive or negative as specified) in the specified direction.

[(posneg)] refers to the slope going Positive (P), Negative (N), or Both (B). If more than the next [#consecutive_points#] points have zero or opposite slope, the Slope function does not look any further for the maximum slope.

Positive slope means increasing Y value for increasing indices of the X expression.

The point found is an artificial point halfway between the two data points defining the maximum slope.

The default [(posneg)] is Positive.

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

A percentage of full range	50%
A marked point Or	X1
an expression of	Y1
marked points	(x1-x2)/2
A value relative to	3 (startvalue -3)
startvalue	. +3 (startvalue +3)
A db value relative	3db (3db below startvalue)
to startvalue	.+3db (3db above startvalue)
A value relative to	Max-3 (maxrng –3)
max or min	Min+3 (minrng +3)
A db value relative	Max-3db (3db below maxrng)
to max or min	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.

June 2020 © 2020

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) = $x^2 - x^1$

 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.

Help Cancel	<back< th=""><th>Next></th><th>Finish</th><th></th></back<>	Next>	Finish	
-------------	--	-------	--------	--

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 PSpice Application Theme

To set the theme of PSpice:

1. From the *Tools* menu, select *Options*.

The Probe Settings dialog box opens.

- 2. Select the Color Settings tab.
- 3. Select Dark or Light from the Application Theme drop-down list.
- **4.** Click *OK*.

Note: The standard Windows operating system dialog boxes do not follow the application theme when launched from PSpice.

Setting Probe window options

Use the Probe Settings 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 Settings 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 Settings dialog box.

Setting large data file options

- 1. From the Tools menu, select Options to display the Probe Settings 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 (100000).

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 than 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.

PSpice Help Setting Options

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:

background	The colors for the window background
foreground	The default color for items not explicitly specified
trace 1 through trace 12	Colors the traces with specified color when they appear on the Probe window. You can specify up to 12 colors.

Valid color names

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

Specify varying degrees of color by using the R, G, B value of the color. The range is from 0-255. For example, <a tribute>=255 0 0 is the same as <a tribute>=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
Т3	specifies the current time in the header and/or footer
&Ν	specifies where page numbering appears in print job
A&	prints simulation date from current data file
M&	prints simulation time from current data file
I3	prints simulation title from current data file
&E	prints simulation temperature from data file
&Р	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 add menu option or button as a new menu or in the toolbar

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 to be displayed.

The selected toolbar appears.

4. Click *Close* to close the dialog box.

To add menu option or button as a new menu or in the toolbar

- **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 or menu options in the *Commands* frame.
- 4. To add the selected button or option to a toolbar or menu bar, drag it from the Customize dialog box to any toolbar or menu bar displayed in the program window.
- 5. Click *Close* to close the dialog box.

To remove buttons from toolbars

→ 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:
 - □ Click *Reset All*.
 - □ Click *Reset*.
- 4. Click *Close* to close the dialog box.

Customizing Commands

To customize commands:

- **1.** Choose *Tools Customize*.
- 2. In the *Commands* tab, select a menu name from the *Categories* list.

The buttons for the selected menu are displayed in the *Commands* frame.

3. Drag a button or menu option and place it in the toolbar.

Generating a Report

Using the Generate Report option, you can generate an HTML report based on transient analysis data. The HTML report displays the transient analysis data in the following template:

- 1. General Information
 - **a.** Simulation Data (.dat) File
 - **b.** Analysis Name
 - C. Circuit Name
 - **d.** Simulation Time
 - e. PSpice Version
- 2. Circuit Information
 - **a.** List of Nets
 - **b.** List of Devices
- 3. Simulation Data
 - **a.** Current Data
 - **b.** Voltage Data
 - **C.** Power Data

The template above can be customized by modifying the TCL (orPspReport.tcl) file located at the following path: <Installation Directory>\tools\pspice\tclscripts.

Note: As you generate the report, remember the following points:

■ The maximum size limit of the HTML file is 800 MB.

■ To generate report, the simulation data file must have transient analysis data of analog devices.

Reference Information

Using files

Opening and closing 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.

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
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

PSpice Help Reference Information

Ctrl+N	create a new text file
F12	restore the last Probe window session
Ctrl+P	display the Print dialog box
F1	view 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.

Ģ	'at' symbol
<u>8</u>	percent sign
<u>ک</u>	ampersand
*	asterisk
+	plus sign (only as initial character)
<space></space>	Space
?	Question mark
~	Tilde
#	Hash
^	Power of
	"Double quotes
$\backslash \backslash$	Back slashes
!	Exclamation mark
(Opening braces
)	Closing braces
	'Single quotes

PSpice Help Reference Information

<	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
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.PRB 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.PRB.

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.

Caution

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:

- □ <u>To create a new macro</u>
- □ <u>To modify an existing macro</u>
- □ <u>To load a macro</u>
- □ <u>To delete a macro</u>
- □ <u>To delete a macro from another file</u>
- 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.
 - **D** 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



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.

Caution 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

- $\Box \quad ADD(A,B) = A+B$
- **D** 10X(A) = 10*A; here is an in-line comment
- \Box 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 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_17.4\tools\pspice\library"

indicates that the LIBPATH setting will point to the directory C:\Cadence\SPB_17.4\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.)

PART LIBS section

PART LIBS section

PART LIBS section

PART LIBS section

PROBE section

PROBE DISPLAY COLORS section

PROBE PRINTER COLORS section

PSPICE section

PSpice NETLIST section

SUBCKT SETTING section

PART LIBS section

The [PART LIBS] section lists the part libraries that are configured for use in PSpice Schematic to Capture translation. 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. You may list multiple extensions.

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.
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.
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.
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= <installation_directory>\tools\PSpic e\Common\pspice.prb</installation_directory>

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.
TRACEWIDTH	Specifies the trace width in the probe. The default is 1.

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 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 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.

Command line options for the PSpice applications are explained in much more detail in the PSpice Reference Guide.

Keyword	Description
BACKUP	Specifies the directory in which backup copies are placed. The default is set at installation. This line can be edited in the form:
	BACKUP= <directory></directory>
	where <directory> 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.</directory>

LIBPATH	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:
	where <directory< directory="" files<="" library="" specifies="" td="" the="" where=""></directory<>
	are located. (For more information, see the section on LIBPATH below.)
MATHEXCEPTIONS	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.
OPTIMIZERCMD	Specifies the command used to run PSpice Optimizer. This line can be edited in the form:
	OPTIMIZERCMD= <optimize r=""> [options]</optimize>
	where <optimizer> indicates the executable for starting Optimizer and [options] can be any of the available Optimizer switches.</optimizer>
PSPICECMDLINE	Specifies what command line options should be used when starting up PSpice. This line can be edited in the form:
	PSPCIECMDLINE= - <switch> <filename></filename></switch>
	where - <switch> indicates the and <filename> is the name of the corresponding file to be opened. Valid switches are: $-i =$ custom .INI file $-p =$ measurement function file</filename></switch>
SET_OPTION_FLAG	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:
	SET OPTION FLAG= OLDTLINETSTEP

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.

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 help and user guide for the particular front-end tool you are using.

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.

Calculating Hysteresis Core Loss

The Hysteresis Core Loss application calculates the area of the B-H curve, which is the integrated average of B-H loops, on the basis of the specified time range, that is, the start time and end time.

If the specified time range covers more than 50 B-H loops, the area of the B-H curve is calculated using the first 50 B-H loops. If the specified time range covers less than 50 B-H loops, the area of the B-H curve is calculated using the number of the loops covered within the specified time range.

The following figures illustrate an example where the circuit is in the transient state.



Figure 12-1 Transient State of the circuit

Note that in <u>Figure 12-1</u> on page 244, the time range for the B-H loops is from 0 to 4 seconds and the B-H loops, as shown in <u>Figure 12-2</u> on page 245, do not overlap as the B-H values are different for each loop.



Figure 12-2 B-H Curve of the Circuit's Transient State

The following figures illustrate an example of the steady-state behavior of the circuit in the specified time range of 4 to 10 seconds.

Note that the B-H loops, as shown in the <u>Figure 12-4</u> on page 247, overlap as the circuit is in its steady state. The core loss of the circuit's steady state is calculated if the specified time range is from 4 to 10 seconds.





Figure 12-3 Steady State of the circuit

Figure 12-4 B-H Curve of the circuit's steady state

Note: For accurate core loss calculation, specify a value for the maximum step size that generates optimal number of data points.

PSpice Help Files and Commands

Descriptions of menus

The main PSpice menu bar is shown below. For a detailed description of a particular menu, click 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.

Menu command	Function
New	
Simulation	Creates a new simulation profile.
Profile Text File	Creates a new text file.
Open	Opens an existing file.
Close	Closes the current file.
Open Simulation	Opens an existing simulation file.
Close Simulation	Closes the current simulation file.
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).
Change Product	Changes the product license, if available.
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.

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.
Сору	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.

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 Line Number Display	Toggles on or off line number display in .cir and .out files.
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.

Menu command	Function
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.

PSpice Help Descriptions of menus

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.
Cursor Window	Toggle for turning on or off the Cursor Window.
Simulation Status Window	Toggle for turning on or off the Status Window.
PSpice_toolbars	Toggle for turning on or off the toolbars.
Command Window	Toggle for turning on or off the Command Window.
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=""></current>	Runs the current (named) simulation.
Pause	Momentarily interrupts the current simulation.
Save	Saves the current simulation.
Stop	Halts the current simulation.
Edit Profile	Provides access to the simulation profile for editing the simulation setup.
Edit Runtime Settings	Provides access to runtime simulation settings.

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.

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.

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.

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	
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	Provides different options for Probe settings.
General tab	Provides general option settings for default values.
Large Dat tab	Provides options settings for opening a large data file in PSpice.
Cursors Settings tab	Provides options for cursor settings
Color Settings tab	Provides different options for color settings
Font Settings tab	Provides different options for font settings
FRA	Provides option to enable Frequency Response Analysis.

Generate Report	Provide option to generate report based on transient analysis data
PSpice Calculator	
Hysteresis Core Loss	provides option to calculate hysteresis core loss
Matlab	provides option the following MATLAB options:
Set Matlab Path	- Sets the MATLAB path
Start Engine	- Starts the MATLAB Engine
Export All Traces	- Exports all traces in MATLAB
Export Selected Traces	- Exports the selected traces in MATLAB
Co-Simulation	- Launches MATLAB for PSpice Simulink Co-Simulation

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<="" td=""><td></td></listing>	

<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^2/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.

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 % $ \label{eq:select}$
Regular expression	Select this to find text using regular expressions.
Wrap around search	If this check box is selected, the search wraps around to the start of the document.
	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.
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 ^. For example the regular expression r.t will match the strings rat, rut, r t, but not root.
[]	Use the [] brackets to match any one of the characters between the brackets. For example, the regular expression r[aou]t matches rat, rot, and rut, but not ret.
	Ranges of characters can specified by using a hyphen. For example, the regular expression [0-9] means match any digit. Multiple ranges can be specified as well. The regular expression [A-Za-z] means match any upper or lower case letter.
	To match any character except those in the range, use the caret as the first character after the opening bracket. For example, the expression [^269A-Z] will match any characters except 2, 6, 9, and upper case letters.
\$	The dollar sign (\$) matches the end of a line. For example, the regular expression professional\$ 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 ^When in 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 .* matches any number of any characters. The regular expression [a-z]* matches zero or more lower-case characters.

\	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	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.
	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.
	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.

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.

Note: The following two options are disabled for marching waveforms:

- Use fewer data points to display the complete trace
- Use all data points to display trace in parts

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 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

Кеу	Mode or user interface item	Function or menu command
CTRL+ A	Zoom menu (on the View menu)	Area
CTRL+ C	Edit menu	Сору
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

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 Profile	Opens the Simulation Settings dialog box. Equivalent to the Edit Simulation Profile 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.
	Save	Saves the active file. Equivalent to the Save command on the File menu.
	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.
	Сору	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.
	Next bookmark	Moves to the next bookmark. Equivalent to the Next Bookmark command on the Edit menu.

PSpice Help Using the Quick Reference Card

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).
	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 downarrow button when you want load any one of the next 5 or the previous 5 parts.

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.

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 Settings 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
1001	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).

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).

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.

Proper ty	Symbol(s)/ Part(s)	PSpice Library	Description	Req uired	Defa ult	Units
AC	VPWL, VSIN, VPWL_F_RE_ FOREVER, VPWL_RE_N_ TIMES, VPULSE, VPWL_ENH, VPWL_RE_F OREVER, VPWL_FILE, VSFFM, VSRC, VPWL_F_RE_ N_TIMES, VSTIM	SOURCE	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_FO REVER, IPWL_FILE, ISFFM, ISRC, IPWL_F_RE_ N_TIMES, ISTIM	SOURCE, SOURCST M	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
ACMAG	IAC	SOURCE	AC magnitude (used in AC sweep analysis only)	Yes	0	A
ACPHAS E	IAC	SOURCE	AC phase (used in AC sweep analysis only)	No	0	Degr ees
ACMAG	VAC	SOURCE	AC magnitude (used in AC sweep analysis only)	Yes	0	V
ACPHAS E	VAC	SOURCE	AC phase (used in AC sweep analysis only)	No	0	Degr ees
AD	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP, MbreakP3, Mbreak P4, MbreakN3	BREAKOUT	Drain diffusion area	No	DEF AD	m^2
AGD	ZbreakN	BREAKOUT	Gate-drain overlap area	No	5.0e- 6	m^2
ANALYS IS	PRINT1, WATCH1	SPECIAL	Analysis type (ACIDCITRAN)	No	TRA N	none

AREA	DbreakCR, JbreakN, QbreakP3, DbreakW, JbreakP, QbreakP4, QbreakN4, QbreakN4, QbreakN4, QbreakR, DbreakZ, Dbreak, Bbreak, ZbreakN	BREAKOUT	Area value (for scaling)	No	1.0	none
ARM_MA SS	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
ASSERT ION	HOLD <size></size>	DIG_MISC	Assert hold violation when low to high or high to low (LHIHL)	Yes	LH	none
ASSERT ION	SETUP <size ></size 	DIG_MISC	Assert setup violation when low to high or high to low (LHIHL)	Yes	LH	none
BACKEM F	BLDCMTR	MIX_MISC	Back EMF constant	Yes	0.12	Volt* sec/ rev
BIT	VECTOR <siz E></siz 	SPECIAL	Bit position within a single hex or octal digit (see .VECTOR)	No	none	none

С	TLOSSY, TLURC <size >, TLUMP<size< th=""><th>ANALOG, TLINE</th><th>Capacitance value per unit length of transmission line</th><th>Yes</th><th>1</th><th>F</th></size<></size 	ANALOG, TLINE	Capacitance value per unit length of transmission line	Yes	1	F
	>			Ň	4	_
С	ed	ILINE	capacitance value per unit length of coupled transmission line	Yes	-1	F
C1	T <size>coupl edX</size>	TLINE	Capacitance value per unit length of first coupled transmission line	Yes	-1	F
C2	T <size>coupl edX</size>	TLINE	Capacitance value per unit length of second coupled transmission line	Yes	-1	F
С3	T <size>coupl edX</size>	TLINE	Capacitance value per unit length of third coupled transmission line	Yes	-1	F
C4	T <size>coupl edX</size>	TLINE	Capacitance value per unit length of fourth coupled transmission line	Yes	-1	F
C5	T <size>coupl edX</size>	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
CAPACI TANCE	BLDCMTR	MIX_MISC	Winding capacitance to ground	Yes	0.00 1u	F
CIN	ADCPAR <siz E></siz 	DATACONV	Input capacitance	Yes	10P	F
CIN	ADCMIC <siz E></siz 	DATACONV	Input capacitance	Yes	22p	F

CKDPDM AX	ADCSER <siz E></siz 	DATACONV	Maximum clock to data delay	Yes	80n	sec
CKDPDM IN	ADCSER <siz E></siz 	DATACONV	Minimum clock to data delay	Yes	25n	sec
CLKHI	ADCSER <siz E></siz 	DATACONV	Minimum clock high pulse width	Yes	40n	sec
CLKLO	ADCSER <siz E></siz 	DATACONV	Minimum clock low pulse width	Yes	60n	sec
CLKMAX	ADCSER <siz E></siz 	DATACONV	Maximum clock frequency	Yes	5Me g	Hz
CLKMIN	ADCSER <siz E></siz 	DATACONV	Minimum clock frequency	Yes	178. 571K	Hz
СМ	T <size>coupl ed, T<size>coupl edX</size></size>	TLINE	Mutual capacitance per unit length	Yes	0	F
СМ	Kcouple2	TLINE	Mutual coupling capacitance per unit length	Yes	none	F
СМ12	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 2	Yes	0	F
СМ13	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 3	Yes	0	F
CM14	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 4	Yes	0	F
CM15	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 1 and 5	Yes	0	F

CM21	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 1	Yes	none	F
CM23	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 3	Yes	0	F
CM24	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 4	Yes	0	F
CM25	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 2 and 5	Yes	0	F
CM31	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 1	Yes	none	F
CM32	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 2	Yes	none	F
СМ34	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 4	Yes	0	F
CM35	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 3 and 5	Yes	0	F
CM41	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 1	Yes	none	F

CM42	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 2	Yes	none	F
CM43	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 3	Yes	none	F
CM45	T <size>coupl edX</size>	TLINE	Mutual coupling capacitance per unit length between tlines 4 and 5	Yes	0	F
СМ51	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 1	Yes	none	F
СМ52	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 2	Yes	none	F
СМ53	Kcouple <size ></size 	TLINE	Mutual coupling capacitance per unit length between tlines 5 and 3	Yes	none	F
СМ54	Kcouple <size ></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 ></line 	STIM1	SOURCE	(time, bit value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0	pair
COMMAN D <line ></line 	STIM16	SOURCE	(time, hex value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0000	pair
COMMAN D <line ></line 	STIM4	SOURCE	(time, bit value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0000	pair
COMMAN D <line ></line 	STIM8	SOURCE	(time, bit value) pairs of stimuli to be generated (see Stimulus Generator)	Yes	0s 0000 0000	pair
CONTAC 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></siz 	DATACONV	Minimum high width of convert pulse	Yes	40n	sec
CONVWH I	ADCPAR <siz E></siz 	DATACONV	Minimum high width of convert pulse	Yes	45n	sec
CONVWH I	ADCMIC <siz E></siz 	DATACONV	Minimum high width of convert pulse	Yes	50n	sec
CONVWL O	ADCPAR <siz E></siz 	DATACONV	Minimum low width of convert pulse	Yes	45n	sec
CONVWL O	ADCMIC <siz E></siz 	DATACONV	Minimum low width of convert pulse	Yes	50n	sec
CONVWL O	ADCSER <siz E></siz 	DATACONV	Minimum low width of convert pulse	Yes	60n	sec

COUPLI NG	XFRM_LINEA R, XFRM_NONLI NEAR, K_LINEAR, kbreak	ANALOG, BREAKOUT	Coupling coefficient	Yes	1	none
COUT	DACCUR <siz E></siz 	DATACONV	Output capacitance	Yes	10p	F
CSMINL O	DACPAR <siz E></siz 	DATACONV	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
DAMPIN G	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	Ν	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_F OREVER, VPWL_FILE, VSFFM, VSRC, VPWL_F_RE_ N_TIMES, VSTIM, VAC	SOURCE, SOURCST M	DC voltage magnitude (used for bias point)	No	0	V
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	Ν	none
DEADTI ME	SG1525A/ 25C, SG1526B, SG1525, SG1524B, SG1529	SWIT_REG	Dead time	Yes	1u	Sec

DEADTI ME	SG1842, SG1843, SG1844, SG1845, SG1846	SWIT_REG	Dead time	Yes	2u	Sec
DEADTI ME	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
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 <siz E></siz 	DATACONV	Data hold time	Yes	10n	Sec
DIG_GN D	STIM <size></size>	SOURCE	Digital ground reference node (change for user power supply)	Yes	\$G_ DGN D	none
DIG_PW R	STIM <size></size>	SOURCE	Digital power reference node (change for user power supply)	Yes	\$G_ DPW R	none
DRAG	Relay_SPDT_ phy	ANL_MISC	Air and other contact arm drag	Yes	1	g*cm / sec^ 2
DSET	DACSER <siz E></siz 	DATACONV	Data setup time	Yes	100n	Sec

ERRORL IMIT	RELEASE <si ZE>, HOLD<size>, CONSTRAINT <size>, SETUP<size >, WIDTH_HI, WIDTH_LO, MAXFREQ</size </size></size></si 	DIG_MISC	Maximum number of constraint checker violations to report	No	20	none
EXP <li NE></li 	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	V(%I N+, %IN-)	insta nce
F	Т	ANALOG	Frequency for NL parameter (see Transmission Line device)	No	none	Hz
FO	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
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></siz 	SPECIAL	Output file for .VECTOR results	No	none	none
FILENA ME	FileStim <size ></size 	SOURCE	Input file containing STIM source data, (time, digital value) pairs	Yes	none	none
FILENA ME	LIB	SPECIAL	Library file name to be included	Yes	none	none
FILENA ME	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.			

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
FRQ	COAX, RG6/U, RG58A/U, RG59B/U, RG8A/U, RG9B/U, RG174/U, RG174/U, RG12A/U, RG12A/U, RG58C/U, RG58C/U, RG58/U, RG55B/U, RG55B/U, RG55B/U, RG55B/U, RG62/U RG179B/U, RG178B/U, RG178B/U, RG178B/U, RG11/u+, RG212/U, RG11/u+, RG62A/U, RG11/u+, RG63B/U, RG11/U, RG187A/U, RG11/U, RG213/U, RG213/U, RG213/U, RG214/U, RG11/U, RG215/U, RG196A/U, RG195A/U, RG215/U, RG215/U, RG215/U, RG215/U, RG217/U, RG217/U, RG218/U, RG218/U, RG218/U, RG218/U, RG218/U, RG218/U, RG219/U, RG219/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	100 Meg	Hz
-----	---	-------	---	----	------------	----
	RG180B/U					

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>coupl ed, TLUMP<size ></size </size>	TLINE	Per unit length conductance	Yes	0	Siem ens
G	TLOSSY	ANALOG	Per unit length conductance	Yes	1	Siem ens
G1	T <size>coupl edX</size>	TLINE	Per unit length conductance of first conductor	Yes	0	Siem ens

G2	T <size>coupl edX</size>	TLINE	Per unit length conductance of second conductor	Yes	0	Siem ens
G3	T <size>coupl edX</size>	TLINE	Per unit length conductance of third conductor	Yes	0	Siem ens
G4	T <size>coupl edX</size>	TLINE	Per unit length conductance of fourth conductor	Yes	0	Siem ens
G5	T <size>coupl edX</size>	TLINE	Per unit length conductance of fifth conductor	Yes	0	Siem ens
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	н	ANALOG	Transresistance	Yes	1	V/I
GAIN	DIFFER	ABM	Prescaling factor before differentiation	Yes	1	insta nce
GAIN	INTEG	ABM	Prescaling factor before integration	Yes	1	insta nce
GAIN	GAIN	ABM	Gain	Yes	1000	insta nce
GAIN	GLIMIT	ABM	Prescaling factor before limiting	Yes	1k	insta nce
GAIN	HILO	ABM	Prescaling factor before limiting	Yes	1k	insta nce
GAIN	SOFTLIM	ABM	Prescaling factor before limiting	Yes	1k	insta nce
GAIN_R EF	ADCPAR <siz E></siz 	DATACONV	Upper input voltage limit	Yes	0.4	none
GAIN_R EF_GND	ADCPAR <siz E></siz 	DATACONV	Lower input voltage limit	Yes	-0.4	none
HCTAU	BULB	ΟΡΤΟ	Heat capacity time constant	Yes	4.08 2	Sec

HI	WATCH1	SPECIAL	Upper limit value for .WATCH	Yes	none	V
HI	GLIMIT, HILO, SOFTLIM	ABM	Upper limit	Yes	10	V
HOLDTI ME	HOLD <size></size>	DIG_MISC	Hold time for constraint 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	А
12	IPULSE, IEXP	SOURCE	Second current level	Yes	none	А
I1	IPWL	SOURCE	First PWL point (0s, current value)	Yes	none	А
12	IPWL	SOURCE	Second PWL point (time, current value)	No	none	А
I3	IPWL	SOURCE	Third PWL point (time, current value)	No	none	А
I4	IPWL	SOURCE	Fourth PWL point (time, current value)	No	none	А

15	IPWL	SOURCE	Fifth PWL point (time, current value)	No	none	A
IG	IPWL	SOURCE	Sixth PWL point (time, current value)	No	none	A
I7	IPWL	SOURCE	Seventh PWL point (time, current value)	No	none	A
18	IPWL	SOURCE	Eighth PWL point (time, current value)	No	none	A
IAMPL	ISIN	SOURCE	Current amplitude of sinusoid (transient analysis only)	Yes	none	A
IAMPL	ISFFM	SOURCE	Current amplitude of SFFM (transient analysis only)	Yes	none	A
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	insta nce
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	Ν	none
INDUCT ANCE	BLDCMTR	MIX_MISC	Winding inductance	Yes	3m	Hz
INERTI A	BLDCMTR	MIX_MISC	Moment of inertia of rotor	Yes	0.30	g*cm *sec* sec

IO_MOD EL	DACSER <siz E>, DACCUR<siz E>, ADCSER<siz E>, ADCPAR<siz E>, DACPAR<siz E>, ADCMIC<siz E></siz </siz </siz </siz </siz </siz 	DATACONV	I/O model to use for and D/A converter models	No	IO_H CT	none
IO_MOD EL	STIM <size>, DIGCLOCK, DIGSTIM</size>	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	none	A
ISINK	DACPAR <siz E></siz 	DATACONV	Typical output sink current	Yes	- 400u	A
ISOURC E	DACPAR <siz E></siz 	DATACONV	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 >,</size 	ANALOG, TLINE	Per unit length inductance	Yes	1	Н

L	T <size>coupl ed</size>	TLINE	Per unit length inductance	Yes	-1	Н
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	Η
L1	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of first coupled inductor	Yes	L1	refde s
L1	T <size>coupl edX</size>	TLINE	Per unit length inductance of first conductor	Yes	-1	Н
L1_TUR NS	XFRM_NONLI NEAR	BREAKOUT	Number of turns for first inductor	Yes	none	none
L1_VAL UE	XFRM_LINEA R	ANALOG	Inductance value of first inductor	Yes	none	Н
L2	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of second coupled inductor	Yes	L2	refde s
L2	T <size>coupl edX</size>	TLINE	Per unit length inductance of second conductor	Yes	-1	Н
l2_tur NS	XFRM_NONLI NEAR	BREAKOUT	Number of turns for second inductor	Yes	none	none
L2_VAL UE	XFRM_LINEA R	ANALOG	Inductance value of second inductor	Yes	none	Н
L3	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of third coupled inductor	Yes	L3	refde s

L3	T <size>coupl edX</size>	TLINE	Per unit length inductance of third conductor	Yes	-1	Н
L4	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of fourth coupled inductor	Yes	L4	refde s
L4	T <size>coupl edX</size>	TLINE	Per unit length inductance of fourth conductor	Yes	-1	Н
L5	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of fifth coupled inductor	Yes	L5	refde s
L5	T <size>coupl edX</size>	TLINE	Per unit length inductance of fifth conductor	Yes	-1	Н
L6	K_LINEAR, kbreak	ANALOG, BREAKOUT	Reference Designator of sixth coupled inductor	Yes	L6	refde s
LDACMI NLO	DACPAR <siz E></siz 	DATACONV	Minimum pulse width low for LDAC	Yes	90n	sec

	COAX, RG6/U, RG58A/U, RG59B/U, RG8A/U, RG8A/U, RG174/U, RG174/U, RG8/U, RG12A/U, RG58C/U, RG6A/U, RG6A/U, RG59/u=, RG22B/U, RG55B/U, RG55B/U, RG55B/U, RG179B/U, RG179B/U, RG178B/U, RG178B/U, RG178B/U, RG11/u+, RG62A/U, RG11/U, RG23/U, RG11/U, RG63B/U, RG187A/U, RG213/U, RG11/U, RG213/U, RG196A/U, RG196A/U, RG195A/U, RG195A/U, RG195A/U, RG215/U, RG195A/U, RG215/U, RG195A/U, RG217/U, RG218/U, RG218/U, RG218/U, RG218/U, RG219/U, RG219/U, RG180B/U, TWSTPAIR, TP22AWG, TP22AWG, TP22AWG,	TLINE, ANALOG	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
June 2020 © 2020	TP19AWG, TP16AWG, TLOSSY, TLURC <size >, TLUMP<size< td=""><td></td><td>297</td><td></td><td>Product Version 17.4- All Rights Reserve</td></size<></size 		297		Product Version 17.4- All Rights Reserve

-2020 d.

LFIL	VMLSDCM, VMCCMDCM	SWIT_RAV	Filter inductance	Yes	500u	Н
LFIL	CMLSCCM, CMSSCCM	SWIT_RAV	Filter inductance	Yes	5u	Η
LIM_K	Relay_SPDT_ phy	ANL_MISC	Limit spring rate	Yes	1000 000	g/ sec
LM	T <size>coupl ed, T<size>coupl edX</size></size>	TLINE	Mutual inductance per unit length	Yes	0	Н
LM	Kcouple2	TLINE	Mutual coupling inductance per unit length	Yes	none	н
LM12	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 1 and 2	Yes	0	Η
LM13	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 1 and 3	Yes	0	Н
LM14	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 1 and 4	Yes	0	Н
LM15	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 1 and 5	Yes	0	н
LM21	Kcouple <size ></size 	TLINE	Mutual coupling inductance per unit length between tlines 2 and 1	Yes	none	н
LM23	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 2 and 3	Yes	0	Η

LM24	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 2 and 4	Yes	0	н
LM25	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 2 and 5	Yes	0	Н
LM31	Kcouple <size ></size 	TLINE	Mutual coupling inductance per unit length between tlines 3 and 1	Yes	none	н
LM32	Kcouple <size ></size 	TLINE	Mutual coupling inductance per unit length between tlines 3 and 2	Yes	none	н
LM34	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 3 and 4	Yes	0	н
LM35	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 3 and 5	Yes	0	н
LM41	Kcouple <size ></size 	TLINE	Mutual coupling inductance per unit length between tlines 4 and 1	Yes	none	н
LM42	Kcouple <size ></size 	TLINE	Mutual coupling inductance per unit length between tlines 4 and 2	Yes	none	н
LM43	Kcouple <size ></size 	TLINE	Mutual coupling inductance per unit length between tlines 4 and 3	Yes	none	Н

LM45	T <size>coupl edX, Kcouple<size ></size </size>	TLINE	Mutual coupling inductance per unit length between tlines 4 and 5	Yes	0	Н
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	Н
М	MbreakN4, MbreakN, MbreakN3, MbreakP4, MbreakP3, 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	Υ	none
MAGUNI TS	EFREQ, FTABLE, GFREQ	ABM	Units for magnitude table entries (MAGIDB)	No	MAG	none
maxfre Q	MAXFREQ	DIG_MISC	Maximum frequency for constraint checker	Yes	none	Hz
maxfre Q	ADCPAR <siz E></siz 	DATACONV	Maximum clock frequency	Yes	11M eg	Hz
MAXP	Relay_SPDT_ phy	ANL_MISC	Maximum permeance (when arm is close to coil)	Yes	5	gaus s*m^ 2/A
MINFRE Q	MINFREQ	DIG_MISC	Minimum frequency for constraint checker	Yes	none	Hz

MINFRE Q	ADCPAR <siz E></siz 	DATACONV	Minimum clock frequency	Yes	10K	Hz
MINHOL D	ADCMIC <siz E></siz 	DATACONV	R/Cbar to CSbar hold time	Yes	10n	sec
MINLO	ADCMIC <siz E></siz 	DATACONV	R/Cbar minimum pulse width	Yes	50n	sec
MINP	Relay_SPDT_ phy	ANL_MISC	Minimum permeance (when arm is close to coil)	Yes	1	gaus s*m^ 2/A
MINPER	ADCMIC <siz E></siz 	DATACONV	Minimum time between conversions	Yes	10u	sec
MINSET	ADCMIC <siz E></siz 	DATACONV	R/Cbar to CSbar setup time	Yes	10n	sec
MINW	DACSER <siz E></siz 	DATACONV	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	Т	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

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, MbreakP3, 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
OFFTIM E	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

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
PERMEA NCE_RA TIO	3phase	ANL_MISC	Ratio of Permeance_inner/ Permeance_outer	Yes	1	none
PHASE	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in PHASE format in output file (YIN)	No	Ν	none
PHASEU NITS	EFREQ, FTABLE, GFREQ	ABM	Units for phase table entries (DEGIRAD)	No	DEG	none
PNOM	BULB	OPTO	Nominal power	Yes	100	W
POS	VECTOR <siz E></siz 	SPECIAL	Column position (see .VECTOR)	No	see .VEC TOR	none
PRI_IN D	3phase	ANL_MISC	Primary inductance	Yes	200 m	Н
PRI_R	3phase	ANL_MISC	Primary resistance	Yes	0.1	Ohm s

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 ></size </size </size>	TLINE, ANALOG	Per unit length resistance	Yes	0	Ohm s

R_CLOS E	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	Ohm s
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	Ohm s
R_I	EFREQ, FTABLE, GFREQ	ABM	Use real and imaginary (set value to R_I). Default is magnitude and phase	No	mag nitud e/ phas e	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	Ohm s

R_REF_ IN	ADCPAR <siz E></siz 	DATACONV	Input resistance of reference	Yes	5K	Ohm s
R_REF_ OUT	ADCPAR <siz E></siz 	DATACONV	Output resistance of reference out	Yes	12	Ohm s
R_REF_ OUT	ADCMIC <siz E></siz 	DATACONV	Output resistance of reference out	Yes	4K	Ohm s
R1	T <size>coupl edX</size>	TLINE	Per unit length resistance of first conductor	Yes	0	Ohm s
R2	T <size>coupl edX</size>	TLINE	Per unit length resistance of second conductor	Yes	0	Ohm s
R3	T <size>coupl edX</size>	TLINE	Per unit length resistance of third conductor	Yes	0	Ohm s
R4	T <size>coupl edX</size>	TLINE	Per unit length resistance of fourth conductor	Yes	0	Ohm s
R5	T <size>coupl edX</size>	TLINE	Per unit length resistance of fifth conductor	Yes	0	Ohm s
RADIX	VECTOR <siz E></siz 	SPECIAL	Radix of values of the specified nodes	No	see .VEC TOR	none
RCLOSE D	Sw_tClose, Sw_tOpen	ANL_MISC	Closed state resistance	Yes	0.01	Ohm s
RD	VMSSCCM	SWIT_RAV	Diode on resistance	Yes	0.00 0001	Ohm s
RE	VMSSCCM	SWIT_RAV	Models ripple across esr of cap	Yes	0.00 0001	Ohm s
REAL	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write signal in REAL format in output file (YIN)	No	N	none

REF_AS SERTIO N	RELEASE <si ZE></si 	DIG_MISC	Reference edge of clock	Yes	LH	none
REF_VO LT_GAI N	ADCMIC <siz E></siz 	DATACONV	Reference voltage gain	Yes	4	none
REFERE NCE	CD4000_PWR , DIGIFPWR	SPECIAL	Negative power supply value	Yes	0	V

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, H11AV2, H11A4, H11AV2, H11A4, H11AV3, A4N25, A4N26, MCT2E, MLED96, PS2505-1, MOC1005, MOC1006, IL300, SLD1121VS	OPTO	Relative current transfer ratio	Yes	0.5	none
RELEAS ETIME	RELEASE <si ZE></si 	DIG_MISC	Minimum time between signal inactive and clock edge	Yes	none	Sec

REPEAT _VALUE	VPWL_RE_N_ TIMES, IPWL_F_RE_ N_TIMES, VPWL_ENH, VPWL_FILE, 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></siz 	DATACONV	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	1Me g	Ohm s
ROW1	FTABLE	ABM	First row of data triplets (freq, mag, phase)	Yes	(0Hz , 0, 0)	triple t

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	(10H z, -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	(20H z,-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	(30H z, - 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	(40H z, - 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	0.00 0001	Ohm s
RTR_PO LE_PAI RS	BLDCMTR	MIX_MISC	Number of north poles on the rotor	Yes	2	none

SCHOLD	DACSER <siz E></siz 	DATACONV	SYNC to SCLK hold time	Yes	120n	sec
SCLK	DACSER <siz E></siz 	DATACONV	SCLK cycle time	Yes	200n	sec
SCSET	DACSER <siz E></siz 	DATACONV	SYNC to SCLK setup time	Yes	50n	sec
SE	CMLSCCM, CMSSCCM	SWIT_RAV	External ramp slope	Yes	1000 00	V/ sec
SEC_R	3phase	ANL_MISC	Secondary winding resistance	Yes	0.1	Ohm s
SECOND _NPAIR S	IPWL_RE_FO REVER, VPWL_RE_N_ TIMES, VPWL_ENH, VPWL_RE_F OREVER, 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
SETUPT IME	SETUP <size ></size 	DIG_MISC	Setup time	Yes	none	sec
SIG_ED GE	RELEASE <si ZE></si 	DIG_MISC	Signal edge for constraint checking	Yes	LH	none
SIGNAM E	FileStim <size ></size 	SOURCE	Signal name in file	No	none	none
SIGNAM ES	VECTOR <siz E></siz 	SPECIAL	Names of signals which appear in the header of the vector file	No	node nam es	none
SN	CMSSCCM	SWIT_RAV	Current sense ramp slope	Yes	1000 00	V/ sec

SPR	Relay_SPDT_ phy	ANL_MISC	Contact arm spring force	Yes	40	g_c m/ sec^ 2
SR	DACPAR <siz E></siz 	DATACONV	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

т1	VPWL, IPWL	SOURCE	Time for first PWL pair (should be 0)	Yes	none	sec
Τ1	Kcouple <size ></size 	TLINE	Reference designator of first coupled transmission line	Yes	T1	refde s
т10	VPWL, IPWL	SOURCE	Time for tenth PWL pair	Yes	none	sec
т2	VPWL, IPWL	SOURCE	Time for second PWL pair	Yes	none	Sec
Т2	Kcouple <size ></size 	TLINE	Reference designator of second coupled transmission line	Yes	T2	refde s
Т3	VPWL, IPWL	SOURCE	Time for third PWL pair	Yes	none	sec
Т3	Kcouple <size ></size 	TLINE	Reference designator of third coupled transmission line	Yes	Т3	refde s
Т4	VPWL, IPWL	SOURCE	Time for fourth PWL pair	Yes	none	sec
Τ4	Kcouple <size ></size 	TLINE	Reference designator of fourth coupled transmission line	Yes	T4	refde s
Т5	VPWL, IPWL	SOURCE	Time for fifth PWL pair	Yes	none	sec
Т5	Kcouple <size ></size 	TLINE	Reference designator of fifth coupled transmission line	Yes	Т5	refde s
т6	VPWL, IPWL	SOURCE	Time for sixth PWL pair	Yes	none	sec
 Т7	VPWL, IPWL	SOURCE	Time for seventh PWL pair	Yes	none	sec
Т8	VPWL, IPWL	SOURCE	Time for eighth PWL pair	Yes	none	sec
Т9	VPWL, IPWL	SOURCE	Time for ninth PWL pair	Yes	none	sec

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></siz 	DATACONV	BUSYbar low time	Yes	8u	Sec
TBUSYR C	ADCMIC <siz E></siz 	DATACONV	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	т	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</td1 	sec
TF	IPULSE, VPULSE	SOURCE	Fall time	No	TST EP	sec

THDGMN	DACPAR <siz E></siz 	DATACONV	Data valid to Write hold time	Yes	10n	Sec
THIRD_ NPAIRS	IPWL_RE_FO REVER, VPWL_RE_N_ TIMES, VPWL_ENH, VPWL_RE_F OREVER, IPWL_ENH, IPWL_RE_N_ TIMES	SOURCE	Second line containing PWL data, (time, analog value) pairs. Lines will be concatenated.	No	none	pairs
TIMEST EP	STIM <size></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	%
TOLERA NCE	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
TPADHM N	ADCPAR <siz E></siz 	DATACONV	Minimum output delay high	Yes	10n	sec
TPADHT Y	ADCPAR <siz E></siz 	DATACONV	Typical output delay high	Yes	20n	sec
TPADLM N	ADCPAR <siz E></siz 	DATACONV	Minimum output delay low	Yes	10n	sec
TPADLT Y	ADCPAR <siz E></siz 	DATACONV	Typical output delay low	Yes	20n	sec
TPHZMN	ADCMIC <siz E></siz 	DATACONV	Minimum bus relinquish time H-Z	Yes	10n	sec
TPHZMX	ADCMIC <siz E></siz 	DATACONV	Maximum bus relinquish time H-Z	Yes	83n	Sec

TPHZTY	ADCMIC <siz E></siz 	DATACONV	Typical bus relinquish time H-Z	Yes	35n	sec
TPHZTY	ADCPAR <siz E></siz 	DATACONV	Typical output float delay H-Z	Yes	50n	sec
TPLZMN	ADCMIC <siz E></siz 	DATACONV	Minimum bus relinquish time L-Z	Yes	10n	sec
TPLZMX	ADCMIC <siz E></siz 	DATACONV	Maximum bus relinquish time L-Z	Yes	83n	sec
TPLZTY	ADCMIC <siz E></siz 	DATACONV	Typical bus relinquish time L-Z	Yes	35n	sec
TPLZTY	ADCPAR <siz E></siz 	DATACONV	Typical output float delay L-Z	Yes	50n	sec
TPZHMN	ADCMIC <siz E></siz 	DATACONV	Minimum data access time Z-H	Yes	10n	sec
TPZHMX	ADCMIC <siz E></siz 	DATACONV	Maximum data access time Z-H	Yes	83n	sec
TPZHTY	ADCMIC <siz E></siz 	DATACONV	Typical data access time Z-H	Yes	35n	sec
TPZHTY	ADCPAR <siz E></siz 	DATACONV	Typical data access time Z-H	Yes	50n	sec
TPZLMN	ADCMIC <siz E></siz 	DATACONV	Minimum data access time Z-L	Yes	10n	sec
TPZLMX	ADCMIC <siz E></siz 	DATACONV	Maximum data access time Z-L	Yes	83n	sec
TPZLTY	ADCMIC <siz E></siz 	DATACONV	Typical data access time Z-L	Yes	35n	sec
TPZLTY	ADCPAR <siz E></siz 	DATACONV	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

TRAN	IPRINT, IPLOT, VPLOT1, VPLOT2, VPRINT1, VPRINT2	SPECIAL	Write TRAN analysis results to output file (YIN)	No	Y	none
TSC0	ADCSER <siz E></siz 	DATACONV	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_ENH, IPWL_ENH, IPWL_ENH, 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></siz 	DATACONV	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
TURNS	Relay_SPDT_ phy	ANL_MISC	Number of turns	Yes	100	none
V_RATI O	3phase	ANL_MISC			1	

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	Н

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	ΟΡΤΟ	Nominal operating voltage	Yes	120	V
VOFF	VSIN, VSFFM	SOURCE	Offset voltage	Yes	none	V
VOFF	Sbreak	BREAKOUT	Voltage lower threshold	Yes	0	V
VOLTAG E	CD4000_PWR , DIGIFPWR	SPECIAL	Digital power supply voltage	Yes	5	V
VON	Sbreak	BREAKOUT	Voltage upper threshold	Yes	1	V

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></siz 	DATACONV	Output sink voltage (@ISINK)	Yes	-0.1	V
VSOURC E	DACPAR <siz E></siz 	DATACONV	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

WHEN	RELEASE <si ZE>, HOLD<size>, CONSTRAINT <size>, SETUP<size >, WIDTH_HI, WIDTH_LO, MINFREQ, MAXFREQ</size </size></size></si 	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 <siz E></siz 	DATACONV	Minimum width high	Yes	45n	Sec
WIDTH_ MIN_LO	ADCPAR <siz E></siz 	DATACONV	Minimum width low	Yes	45n	Sec
WRMINL O	DACPAR <siz E></siz 	DATACONV	Minimum width low of WR	Yes	90n	sec
XFORM	ELAPLACE, GLAPLACE	ABM	Laplace transform	Yes	1/s	none
Z0	Т	ANALOG	Characteristic impedance	Yes	none	Ohm s

Menu Bar

Click each menu bar item to see its pull-down menu.

Open button

Click to open a data file.

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.

Сору То

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.

ΟΚ

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.

ΟΚ

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.

Сору

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

Page Setup		×
Margins Left: 0.50 Bight: 0.50 Iop: 0.50 Bottom: 0.50		<u>DK</u> <u>C</u> ancel
Plots Per Page Number: 1 Plot/Page		He <u>a</u> der <u>F</u> ooter
Orientation C Landscape O Portrait	C Top Bottom	Select F <u>o</u> nt
✓ Draw Border ✓ Draw Plot Title	◯ Right ◯ None	Set D <u>e</u> fault Reset Defa <u>u</u> lt

PSpice Help

Add Trace dialog box

Add Traces		
Simulation Output Variables		Functions or
×		Analog Ope
I(C1) I(C1·1)	🔺 🔽 Analog	# 0
I(Q1:b) I(Q1:c)	Digital	* +
I(Q1:e) I(Q2:b)	Voltages	7
I(Q2:c) I(Q2:e)	Currents	@ ABS()
I(Q3:b) I(Q3:c)	Power	ARCTAN() ATAN()
I(Q3:e) I(Q4:b)	Noise (V²/Hz)	AVG() AVGX(,)
I(Q4:c) I(Q4:e)	Alias Names	COS() D()
I(RBIAS) I(RBIAS:1)	Subcircuit Nodes	DB() ENVMAX(,
I(RC1) I(RC1:1)		ENVMIN() EXP()
I(RC2) I(RC2:1)		G() IMG()

Digital Size dialog box



Ellipse dialog box

Ellipse Label	×
Enter the in	clination angle:
<u>0</u> K	<u>C</u> ancel

X Axis tab

Axis Settings	×
≚Axis Y Axis X <u>G</u> rid Y Gri <u>d</u>	
Data Range	Use Data
C Auto Range	⊙ Full
User Defined	C Restricted (analog)
212s to 709s	100s to 1Ks
Scale	Processing Options
• Linear	Fourjer
C Log	Performance Analysis
	Axis <u>T</u> itle
Axis <u>V</u> ariable	User Defined Title
	, □ Use this title
	s Default Reset De <u>f</u> aults <u>H</u> elp

Display Control dialog box

Save/Restore Display	
New <u>N</u> ame:	<u>S</u> ave
LAST SESSION(AC)	Save <u>T</u> o
	Со <u>р</u> у То
	<u>D</u> elete
	Delete <u>F</u> rom
	<u>R</u> estore
	<u>L</u> oad
	<u>C</u> lose

Open 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 Settings dialog box

Probe Settings			×
General Large Data File	Cursor Settings Color Se	ettings Font Settings	
Use Symbols Properties	Trace Color Scheme	Mark Data Points	
Never	Match Axis	Display Evaluation	
Always	Sequential Per Axis	Display Statistics	
	Unique By File	Highlight Error States	
Use ScrollBars Auto-Update Intervals • Auto • Auto • Never • Every • Always • Every 10 Number of Histogram Divisions 1 Default Trace Width			
OK Cancel Reset			

Y Axis tab

Axis Settings			×
X Axis Y Axis X Grid Y Grid			
Data Range		Y Axis Number	
Scale © Li <u>n</u> ear © Log	Axis T	itle	
OK Cancel	Save As Default	<u>R</u> eset Defaults	Help

Available Sections dialog box

Available Sections				
** circuit file for profile: AC	Monte Carlo Pass	3	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	- 4	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	5	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	6	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	-7	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	8	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	9	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	10	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	11	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	12	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	13	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	-14	27.0 Deg	
** circuit file for profile: AC	Monte Carlo Pass	15	27.0 Deg	-
<u>All N</u> one			<u> </u>	<u>C</u> ancel

X Grid tab

Axis Settings	×
XAxis XAxis XGrid YGrid	
Automatic	Minor
40u Linear 1 Log (# of decades)	C 4 C 5
Grids ● Lines At Intersections: ● Dots ● + ■ with other major ● + ■ with other minor	Grids C Lines At Intersections: C Dots V with other major C + With other minor None
 Ticks inside plot edge Numbers outside plot edge 	Ticks inside plot edge
OK Cancel <u>S</u> ave A	As Default Reset Defaults Help

Y Grid tab

{bmc BM101.SHG}

PSpice Help

Axis Settings XAxis YAxis XGrid YGrid	
Automatic Major Spacing 5 Linear 1 Log (# of decades) Grids Grids Lines At Intersections: At Intersections: At Intersections: O Dots At Intersections: O Dots Mith other major O + With other minor None Ticks inside plot edge	Minor Intervals between Major 1 2 10 4 5 Grids Lines At Intersections: Dots With other major + with other minor None
✓ Numbers outside plot edge OK Cancel Save A	As Default Reset Defaults Help

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 VI.

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.

PSpice Help

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<link>.

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

357

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.

PSpice Help



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:

 $ng = bloc \rightarrow b_g;$

 $nG = bloc \rightarrow b_G;$

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 RG. If RG 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 RD and RS 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.

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	В
Diode	D
Junction FET	J
MOSFET	Μ
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.

PSpice Help

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:

n samples	The number of Monte Carlo runs selected for analysis in Probe.
n divisions	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).
mean	The arithmetic average of all the measurement expression values.
sigma	The standard deviation
minimum	The minimum measurement expression value.

10th percentile	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.
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.

ΟΚ

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 if 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 Help

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

June 2020 © 2020 **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

June 2020 © 2020 **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 systemwide 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.

- **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 - VOFFI too small for VSWITCH model

The ON and OFF voltages specified are too close in value to each other.

Solution

June 2020 © 2020 **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 that 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 that 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.)

- 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.

- **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.

- 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

June 2020 © 2020 **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.

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.

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.

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 predefined 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 predefined 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.

D 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 'l' 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 (timedomain) 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 (timedomain) 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.

PSpice Help

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.

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.