cādence®

PSpice Advanced Analysis Help

Product Version 16.5
May 2011

Contents

Advanced Analysis Workflow

Advanced Analysis is an add-on program for PSpice A/D and AMS Simulator[1]. Use these four Advanced Analysis tools to improve circuit performance, reliability, and yield:

- Sensitivity identifies which components have parameters critical to the measurement goals of your circuit design. For more information see The Sensitivity Tool on page 27.
- The four Optimizer engines optimize the parameters of key circuit components to meet your performance goals. For more information see The Optimizer Tool on page 51.
- Smoke warns of component stress due to power dissipation, increase in junction temperature, secondary breakdowns, or violations of voltage / current limits. For more information see The Smoke Analysis Tool on page 97.
- Monte Carlo estimates statistical circuit behavior and yield. For more information see The Monte Carlo Tool on page 121.

See the following topics for more information on using Advanced Analysis:

- Setting up a project
- Working with parameterized components
- Using Advanced Analysis libraries
- Preparing your design for Advanced Analysis
- Measurement Expressions
- Advanced Analysis Engines
- Troubleshooting

1. The content of this manual is true for both PSpice A/D and AMS Simulator. Depending on the license and installation, either PSpice A/D or AMS Simulator is installed in the Cadence hierarchy.

Setting up a project

Before you begin an Advanced Analysis project, you need:

- Circuit components that are Advanced Analysis-ready
- A circuit drawn in a schematic editor[1] and successfully simulated in PSpice or AMS Simulator[2].
- PSpice or AMS measurements that check circuit behavior critical to your design.

Creating measurement expressions

Sensitivity, Optimizer, and Monte Carlo require measurement expressions as input. You should create these measurements expressions in PSpice so you can test the results.

You can also create measurement expressions in Sensitivity, Optimizer, or Monte Carlo which can be exported to each other, but these measurements cannot be exported to Advanced Analysis for testing.

Validating the intial circuit

Before you use Advanced Analysis:

1. Make your circuit components Advanced-Analysis ready for the components you want to analyze.
2. Set up a Advanced Analysis simulation.

   The Advanced Analysis tools use the following simulations:

This tool...	Works on these PSpice simulations...
Sensitivity	Time Domain (transient)
	DC Sweep
	AC Sweep/Noise

1. In this manual schematic editor refers to either OrCAD Capture or Design Entry HDL depending on the license or installation.
2. Depending on the licenseand installation, either PSpice or AMS Simulator is installed. However, all information for PSpice provided in this manual is also true for AMS Simulator.

This tool...	**Works on these PSpice simulations...**
Optimizer	Time Domain (transient)
	DC Sweep
	AC Sweep/Noise
Smoke	Time Domain (transient)
Monte Carlo	Time Domain (transient)
	DC Sweep
	AC Sweep/Noise

3 Simulate the circuit and make sure the results and waveforms are what you expect.

4 Define measurements in PSpice to check the circuit behaviors that are critical for your design. Make sure the measurement results are what you expect.

For information on setting up simulations, see your *PSpice Simulator User's Guide*.

For information on setting up measurements, see: Creating measurement expressions.

Introducing Advanced Analysis files

The principal files used by Advanced Analysis are:

- PSpice simulation profiles (.sim)
- Advanced Analysis profiles (.aap)

Advanced users may also use these files:

- Device property files (.prp)
- Custom derating files for Smoke (.drt)
- Discrete value tables for Optimizer (.table)

Introducing the numerical conventions

PSpice ignores units such as Hz, dB, Farads, Ohms, Henrys, volts, and amperes. It adds the units automatically, depending on the context.

Name	Numerical value	User types in:	Or:	Example Uses
femto-	10^{-15}	F, f	1e-15	2f
				2F
				2e-15
pico-	10^{-12}	P, p	1e-12	40p
				40P
				40e-12
nano-	10^{-9}	N, n	1e-9	70n
				70N
				70e-9
micro-	10^{-6}	U, u	1e-6	20u
	.000001			20U
				20e-6
milli-	$\mathbf{10^{-3}}$	**M, m**	**1e-3**	**30m**
	.001			**30M**
				30e-3
				.03
kilo-	10^{3}	K, k	1e+3	2k
	1000			2K
				2e3
				2e+3
				2000

Name	Numerical value	User types in:	Or:	Example Uses
mega-	**10^6** 1,000,000	**MEG, meg**	**1e+6**	**20meg** **20MEG** 20e6 20e+6 20000000
giga-	10^9	G, g	1e+9	25g 25G 25e9 25e+9
tera-	10^{12}	T, t	1e+12	30t 30T 30e12 30e+12

Working with parameterized components

PSpice[1] ships with over 30 Advanced Analysis libraries containing over 4,300 components. Separate library lists are provided for Advanced Analysis libraries and standard PSpice libraries. The components in the Advanced Analysis libraries are listed in the *Advanced Analysis library list*. The Advanced Analysis libraries contain parameterized and standard components. The majority of the components are parameterized. Standard components in the Advanced Analysis libraries are similar to components in the standard PSpice libraries and will not be discussed further in this document.

Parameterized components

A parameter is a physical characteristic of a component that controls behavior for the component model. In schematic editor, a parameter is called a **property**. A parameter value is either a number or a variable. When the parameter value is a variable, you have the option to vary its numerical solution within a mathematical expression and use it in optimization.

Design EntryWhen the parameter value is a variable, you have the option to vary its numerical solution within a mathematical expression and use it in optimization.In the Advanced Analysis libraries, components may contain one or more of the following parameters:

- Tolerance parameters

  For example, for a resistor the positive tolerance could be POSTOL = 10%.

- Distribution parameters

  For example, for a resistor the distribution function used in Monte Carlo analysis could be DIST = FLAT.

- Optimizable parameters

  For example, for an opamp the gain bandwidth could be GBW = 10 MHz.

- Smoke parameters

  For example, for a resistor the power maximum operating condition could be POWER = 0.25 W.

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

To analyze a circuit component with an Advanced Analysis tool, make sure the component contains the following parameters:

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

Tolerance parameters

Tolerance parameters define the positive and negative deviation from a component's nominal value. In order to include a circuit component in a Sensitivity or Monte Carlo analysis, the component must have tolerances for the parameters specified. Use the *Advanced Analysis library list* to identify components with parameter tolerances.

In Advanced Analysis, tolerance information includes:

- Positive tolerance

  For example, POSTOL for RLC is the amount a value can vary in the plus direction.

- Negative tolerance

  For example, NEGTOL for RLC is the amount a value can vary in the negative direction.

Tolerance values can be entered as percents or absolute numbers.

Distribution parameters

Distribution parameters define types of distribution functions. Monte Carlo uses these distribution functions to randomly select tolerance values within a range.

For example, in the schematic editor's property editor, a resistor could provide the following information:

Property	Value
DIST	FLAT

Optimizable parameters

Optimizable parameters are any characteristics of a model that you can vary during simulations. In order to include a circuit component in an Optimizer analysis, the component must have optimizable parameters. Use the *Advanced Analysis library list* to identify components with optimizable parameters.

For example, in schematic editor's property editor, an opamp could provide the following gain bandwidth:

Property	Value
GBW	1e7

Note that the parameter is available for optimization only if you add it as a property on the schematic instance and assign it a value.

During Optimization, the GBW can be varied between any user-defined limits to achieve the desired specification.

Smoke parameters

Smoke parameters are maximum operating conditions for the component. To perform a Smoke analysis on a component, define the smoke parameters for that component. You can still use non-smoke-defined components in your design, but the smoke test ignores these components. Use the online *Advanced Analysis library list* to identify components with smoke parameters.

Most of the analog components in the standard PSpice libraries also contain smoke parameters. Use the online *PSpice library list* to identify components in the standard PSpice libraries that have smoke parameters.

For example, in schematic editor's property editor, a resistor could provide the following smoke parameter information:

Property	Value
POWER	RMAX
MAX_TEMP	RTMAX

Use the design variables table to set the values of RMAX and RTMAX to 0.25 Watts and 200 degrees Centigrade, respectively.

Advanced Analysis libraries location

Schematic Editor symbol libraries

<Target_directory>\Capture\Library\PSpice\AdvAnls\

PSpice Advanced Analysis model libraries

<Target_directory> \ PSpice \ Library

Using Advanced Analysis libraries

In the schematic editor, there are three ways to quickly identify if a component is from an Advanced Analysis library:

- Looking in the online *Advanced Analysis library list*
- Using the library tool tip in the **Place Part** dialog box of Capture or **Component Browser** in Design Entry HDL
- Using the Parameterized Part icon in the **Place Part** dialog box

Using the Advanced Analysis library list

The *Advanced Analysis library list* contains the names of parameterized and standard libraries. Most of the libraries are parameterized. Standard components in the Advanced Analysis libraries are similar to standard PSpice[1] library components. Each library contains the following items:

- Component names and part numbers
- Manufacturer names
- Lists of component parameters for each component
    - Tolerance parameters
    - Optimizable parameters
    - Smoke parameters

Some component libraries, primarily opamp libraries, contain components with all of the parameter types.

Examples from the library list are shown below:

Device Type	Generic Name	Part Name	Part Library	Mfg. Name	TOL	OPT	SMK
Opamp	AD101A	AD101A	OPA	Analog Devices	Y	Y	Y

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

Device Type	Generic Name	Part Name	Part Library	Mfg. Name	TOL	OPT	SMK
Bipolar Transistor	2N1613	2N1613	BJN	Motorola	N	Y	Y
Analog Multiplier	AD539	AD539	DRI	Analog Devices	N	N	N

The parameter columns are the three columns on the right in the list. The abbreviations in the parameter columns have the following meanings:

This library list column heading...	With the following notation...	Means the component...
TOL	Y	Has tolerance parameters in the model
TOL	N	Does not have tolerance parameters in the model
OPT	Y	Has optimizable parameters in the model
OPT	N	Does not have optimizable parameters in the model
SMK	Y	Has smoke parameters in the model
SMK	N	Does not have smoke parameters in the model
DIST	Y	Has a distribution parameter associated with the model
DIST	N	Does not have a distribution parameter associated with the model

You can find the online *Advanced Analysis library list* from your Windows Start menu.

1. From the Windows Start menu, choose the *Cadence – Release 16.5 – Cadence Help*.

   The Cadence Help Window appears.

2. Expand either the *Allegro AMS Simulator* or the *PSpice* node in the list of products.
3. Double-click *PSpice Advanced Analysis library list* to load the document into the viewer pane of Cadence Help.
4. Click the link *PSpice Advanced Analysis Library List* to open PDF file.

Using the library tool tip

One easy way to identify if a component comes from an Advanced Analysis library is to use the tool tip in the *Place Part* dialog box of OrCAD Capture.

1 From the Place menu, choose *Part*.

 The *Place Part* dialog box appears.

2 Enter a component name in the *Part* text box.

3 Hover your mouse over the highlighted component name.

 A library path name appears in a tool tip.

4 Check for `ADVANLS` in the path name.

 If `ADVANLS` is in the path name, the component comes from an Advanced Analysis library.

Using the Parameterized Part icon

Another easy way to identify if a component comes from an Advanced Analysis library is to use the Parameterized Part icon in the **Place Part** dialog box.

1 From the Place menu, select **Part**.

 The **Place Part** dialog box appears.

2 Enter a component name in the **Part** text box.

 Or:

 Scroll through the **Part List** text box

3 Look for  in the lower right corner of the dialog box.

 This is the Parameterized Part icon. If this icon appears when the part name appears in the Part text box, the component comes from an Advanced Analysis library.

Preparing your design for Advanced Analysis

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

You may create a new design or use an existing design for Advanced Analysis. There are several steps for making your design Advanced Analysis-ready.

Creating new designs for Advanced Analysis

If you create a new design, perform the following steps:

1. Select parameterized components
2. Set parameter value for each parameterized component
3. Add additional parameters

Selecting a parameterized component

Select parameterized components from Advanced Analysis libraries.

1 Open the online *Advanced Analysis library list* found in Cadence Online Documentation.

2 Find a component marked with a **Y** in the **TOL**, **OPT**, or **SMK** columns of the *Advanced Analysis library list*.

Components marked in this manner are parameterized components.

3 For that component, write down the **Part Library** and **Part Name** from the *Advanced Analysis library list*.

4 Add the library to your design in your schematic editor.

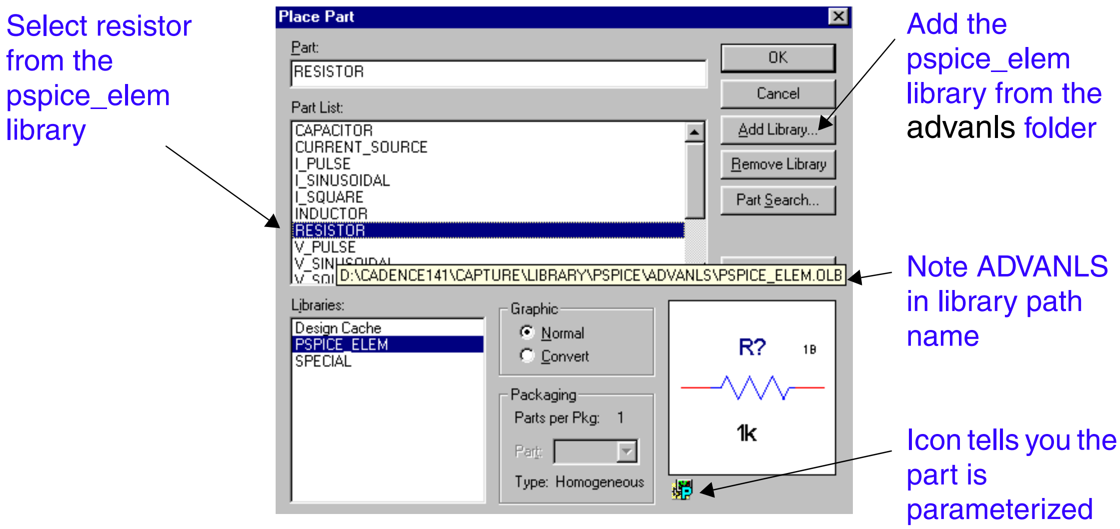
5 Place the parameterized component on your schematic.

For example, select the **resistor** component from the **pspice_elem** Advanced Analysis library.

The `pspice_elem` library on the *Advanced Analysis library list* contains a resistor component with tolerance, optimizable, and smoke parameters. The following example uses that component:

1 In Capture, from the Place menu, select **Part.** Similarly, in Design Entry HDL use the Component Browser.

The **Place Part** dialog box appears.



2 Use the Add Library browse button to add the **pspice_elem** library from the advanls folder to the **Libraries** text box.

3 Select **Resistor** and click **OK**.

The resistor appears on the schematic.

Setting a parameter value

For each parameterized component in your design, set the parameter value individually on the component using your schematic editor.

A convenient way to add parameter values on a global basis is to use the design variable table.

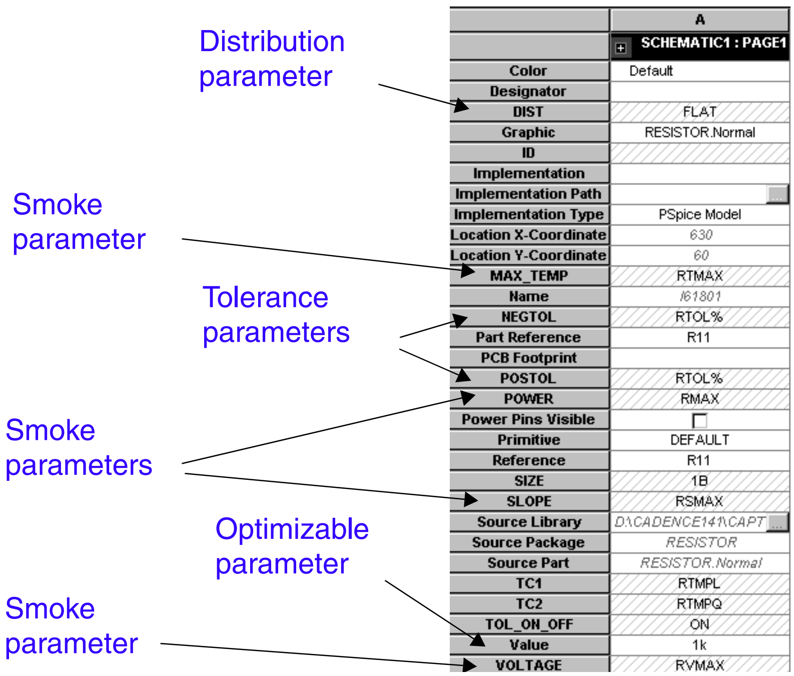
Note: If you set a value for POSTOL and leave the value for NEGTOL blank, Advanced Analysis will automatically set the value of NEGTOL equal to the value of POSTOL and perform the analysis.

Note: As a minimum, you must set a value for POSTOL. If you set a value for NEGTOL and leave the POSTOL value blank, Advanced Analysis will not include the parameter in Sensitivity or Monte Carlo analyses.

The following example shows how to set parameter values:

1 Double click on the Resistor symbol.

The Property Editor appears. Note the Advanced Analysis parameters already listed for this component.



2 Verify that all the parameters required for Sensitivity, Optimizer, Smoke, and Monte Carlo are visible on the symbol.

3 Set the resistor **VALUE** parameter to 10k.

4 Set the resistor **POSTOL** parameter to **RTOL%**.

Adding additional parameters

Part	**Tolerance Property Name**	**Value**
Resistor	POSTOL	RTOL%
Resistor	NEGTOL	RTOL%
Inductor	POSTOL	LTOL%
Inductor	NEGTOL	LTOL%
Capacitor	POSTOL	CTOL%
Capacitor	NEGTOL	CTOL%

For RLC components, the parameter required for Advanced Analysis Optimizer is the value for the component. Examples are listed below:

Part	Optimizable Property Name	Value
Resistor	VALUE	10K
Inductor	VALUE	33m
Capacitor	VALUE	0.1u

For example: For RLC components, the parameters required for Advanced Analysis Smoke are listed below. The values shown are those that can be set using the design variables table.

Part	Smoke Property Name	Value
Resistor	VOLTAGE	RVMAX

If you use RLC components from the “analog” library, you will need to add parameters and set values; however, instead of setting values for the POSTOL and NEGTOL parameters, you set the values for the TOLERANCE parameter. The positive and negative tolerance values will use the value assigned to the TOLERANCE parameter.

If the component does not have Advanced Analysis parameters visible on the symbol, add the appropriate Advanced Analysis parameters using your schematic editor.

For example: For RLC components, the parameters required for Advanced Analysis Sensitivity and Monte Carlo are listed below. The values shown are those that can be set using the design variables table.

Using the design variables table

The design variables table is a component available in the installed libraries that allows you to set global values for parameters. For example, using the design variables table, you can easily set a 5% positive tolerance on all your circuit resistors. The default information available in the design variables table includes variable names for tolerance and smoke parameters. For example, RTOL is a variable name in the design variables tables, which can be used to set POSTOL (and NEGTOL) tolerance values on all your circuit resistors.

1 From Capture's Place menu, select **Part**. Similarly, for Design Entry HDL use the Component Browser.

2 Add the PSpice SPECIAL library to your design libraries.

3 Select the **Variables** component from the PSpice SPECIAL library.

4 Click **OK**.

A design variable table of parameter variable names will appear on the schematic.

5 Double click on a number in the design variable table.

The **Display Properties** dialog box will appear.

6 Edit the value in the **Value** text box.

7 Click **OK**.

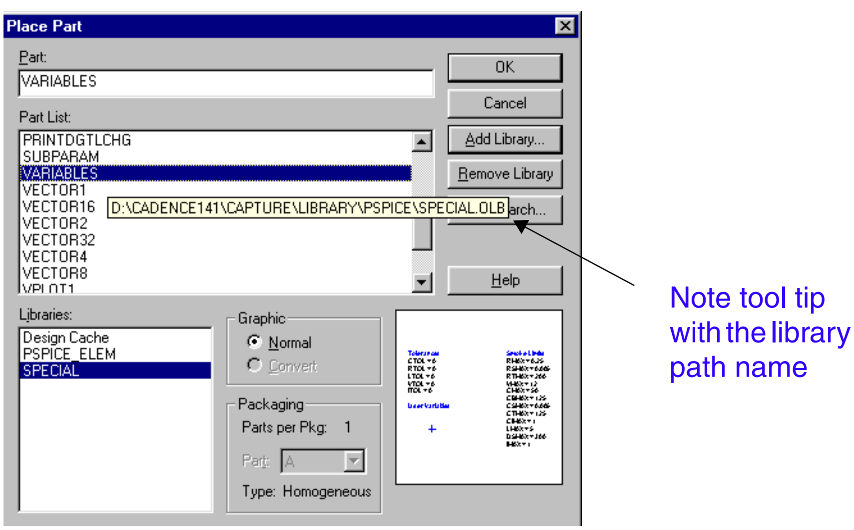
The new numerical value will appear on the design variables table on the schematic and be used as a global value for all applicable components.

Parameter values set on a component instance will override values set in the design variables table.

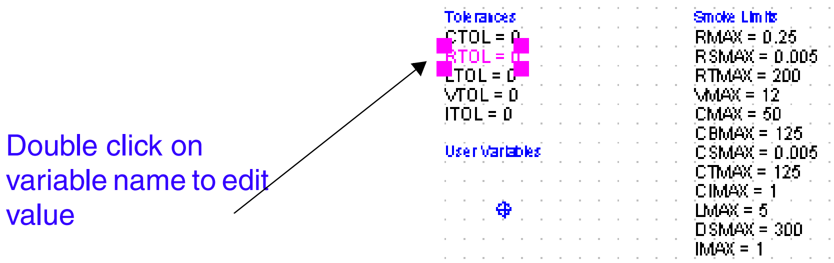
In the following example, you will set the resistor parameter values using the design variables table.

You will set one parameter for this resistor.

1 Select the **Variables** part from the PSpice SPECIAL library.

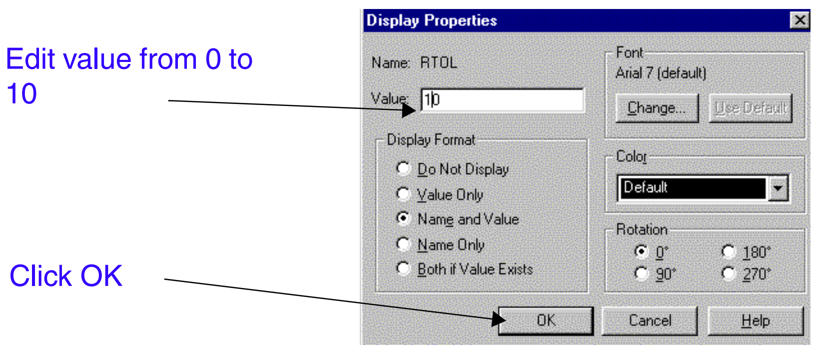


The design variables table appears on the schematic.



2 Double click on the RTOL number **0** in the design variables table.

The **Display Properties** dialog box appears.



3 Edit the value in the **Value** text box.

4 Click **OK**.

The new numerical value will appear on the design variable table on the schematic.

Advanced Analysis will now use the resistor with a positive tolerance parameter set to 10%. If we added more resistors to this design, we could then set the POSTOL resistor parameter values to RTOL% and each resistor would immediately apply the 10% value from the design variables table.

Note: Values set on the component instance override values set with the design variables table.

Modifying existing designs for Advanced Analysis

Perform Advanced Analysis on the parameterized components. To make sure specific components are Advanced Analysis-ready (parameterized), do the following steps:

- Set tolerances for the RLC components

  Note: For standard RLC components, the TOLERANCE property can be used to set tolerance values required for Sensitivity and Monte Carlo. Standard RLC components can also be used in the Optimizer.

- Replace active components with parameterized components from the Advanced Analysis libraries
- Add smoke parameters and values to RLC components

The Sensitivity Tool

Sensitivity identifies which components have parameters critical to the measurement goals of your circuit design.

Note: Sensitivity analysis is available with the following products:

- PSpice[1] Advanced Optimizer Option
- PSpice Advanced Analysis

The Sensitivity Analysis tool examines how much each component affects circuit behavior by itself and in comparison to the other components. It also varies all tolerances to create worst-case (minimum and maximum) measurement values.

You can use Sensitivity to identify the sensitive components, then export the components to Optimizer to fine-tune the circuit behavior.

You can also use Sensitivity to identify which components affect yield the most, then tighten tolerances of sensitive components and loosen tolerances of non-sensitive components. With this information you can evaluate yield versus cost trade-offs.

Absolute and relative sensitivity

Sensitivity displays the absolute sensitivity or the relative sensitivity of a component. Absolute sensitivity is the ratio of change in a measurement value to a one unit positive change in the parameter value.

For example: There may be a 0.1V change in voltage for a 1 Ohm change in resistance.

Relative sensitivity is the percentage of change in a measurement based on a one percent positive change of a component parameter value.

For example: For each 1 percent change in resistance, there may be 2 percent change in voltage.

Since capacitor and conductor values are much smaller than one unit of measurement (Farads or Henries), relative sensitivity is the more useful calculation.

Absolute sensitivity should be used when the tolerance limits are not tight or have wide enough bandwidth. Where as relative sensitivity should be used when the tolerance limits are

---

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

tight enough or have less bandwidth. The tolerance variations are assumed to be linear in this case.

Sensitivity strategy

If Sensitivity analysis shows that the circuit is highly sensitive to a single parameter, adjust component tolerances on the schematic and rerun the analysis before continuing on to Optimizer.

Optimizer works best when all measurements are initially close to their specification values and require only fine adjustments.

Plan ahead

Sensitivity requires:

- Circuit components that are Advanced Analysis-ready
- A circuit design, that is working and can be simulated in PSpice
- Measurements set up in PSpice

Any circuit components you want to include in the Sensitivity data need to be Advanced Analysis-ready, with their tolerances specified.

You can see the following for more information:

Creating measurement expressions	Composing measurement expression
Checking measurement expressions in PSpice	Viewing results of measurements

Setting up for Sensitivity in the schematic editor

Start with a working circuit in the schematic editor. Circuit components you want to include in the Sensitivity data need to have the tolerances of their parameters specified. Circuit simulations and measurements should already be set up.

The simulations can be Time Domain (transient), DC Sweep, and AC Sweep/Noise analyses.

To set up sensitivity in the schematic editor:

1 Open your circuit from your schematic editor.

2 Run a PSpice simulation.

3 Check your key waveforms in PSpice and make sure they are what you expect.

4 Check your measurements and make sure they have the results you expect.

The Advanced Analysis examples folder contains several demonstration circuits. This example uses the RFAmp circuit.

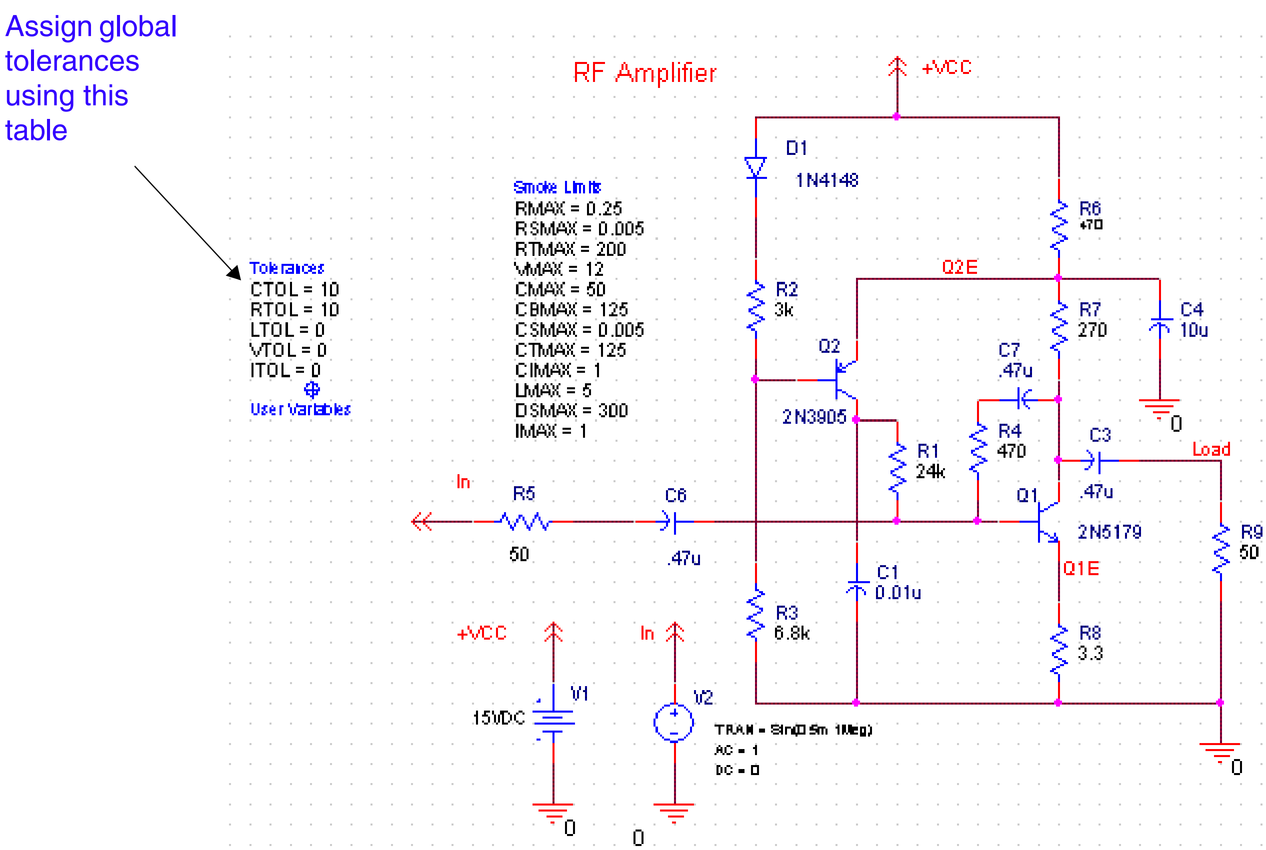
The circuit contains components with the tolerances of their parameters specified, so you can use the components without any modification.

Two PSpice simulation profiles have already been created and tested. Circuit measurements, entered in PSpice, have been set up and tested.

1 In your schematic editor, browse to the RFAmp tutorials directory.

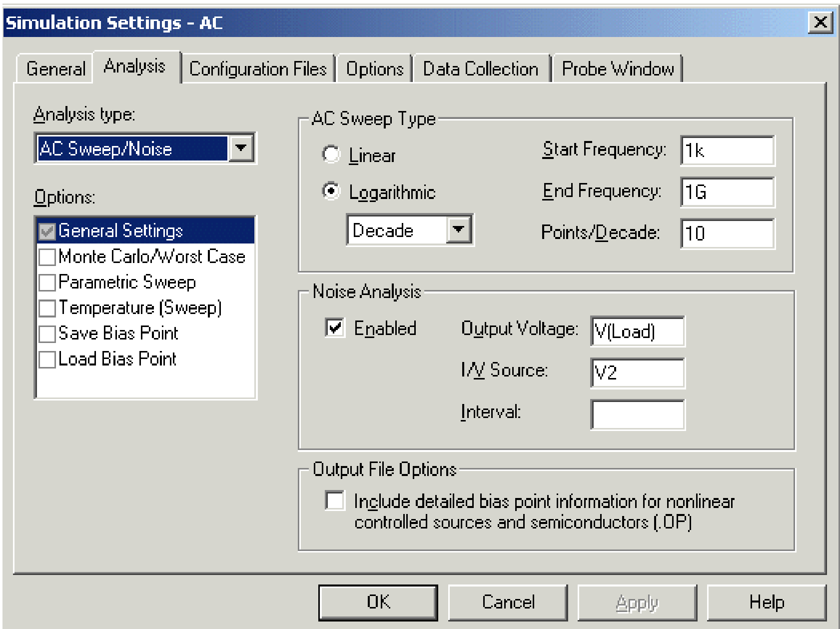
<target directory>
\PSpice\tutorial\Capture\pspiceaa\rfamp

2 Open the RFAmp project.



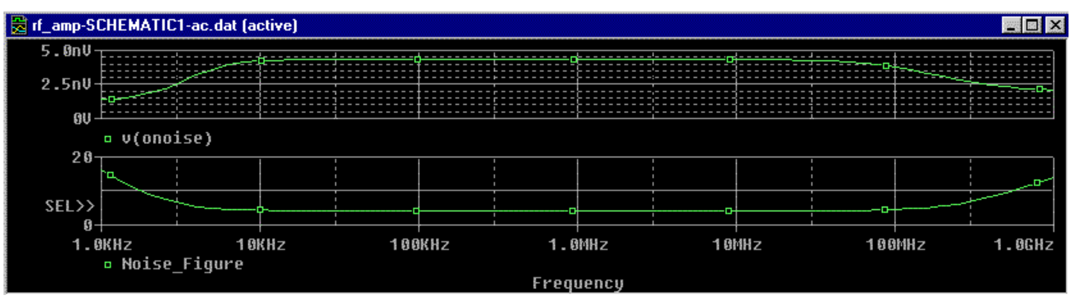
3 Select the SCHEMATIC1-AC simulation profile.

The AC simulation included with the RF example



1 Click ▶ to run the simulation.

2 Review the results.



The waveforms are what we expected.

In the simulator, view measurement results

			Measurement	
	Evaluate	Measurement	Value	
	☑	max(db(v(load)))	9.41807	
	☑	bandwidth(v(load),3)	150.57877meg	
	☑	min(10*log10(v(inoise)*v(inoise)/8.28...	4.14805	
	☑	max(v(onoise))	4.33832n	

The measurements in PSpice give the results we expected.

You can see the following for more information:

Components and tolerances	Preparing your design for Advanced Analysis
Creating measurements	Composing measurement expressions
Testing measurements	Viewing results of measurements

Setting up Sensitivity in Advanced Analysis

To set up sensitivity in Advanced Analysis:

1 From the **PSpice** menu in your schematic editor, choose **Advanced Analysis** - **Sensitivity**.

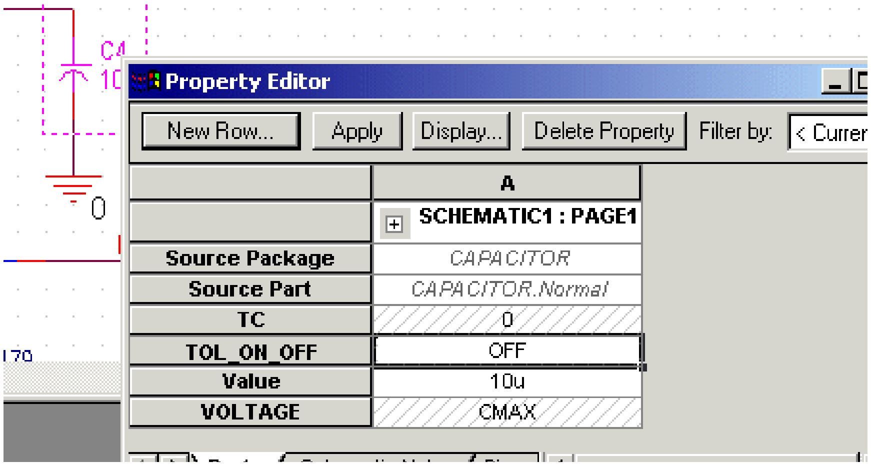
The Advanced Analysis Sensitivity tool opens.

Parameters Window

In the Parameters window, a list of component parameters appears with the parameter values listed in the Original column. Only the parameters for which tolerances are specified appear in the Parameters window.

Note: Sensitivity analysis can only be run if tolerances are specified for the component parameters.

In case you want to remove a parameter from the list, you can do so by using the TOL_ON_OFF property. In the schematic design, set the value of TOL_ON_OFF property attached to the instance as OFF. If there is no TOL_ON_OFF property attached to the instance of the device, attach the property and set its value to OFF. This is so, because if the tolerance value is specified for a parameter and TOL_ON_OFF property is not attached to the component, by default Advanced Analysis assumes that the value of TOL_ON_OFF property is set to ON.



In case of hierarchical designs, the value of the TOL_ON_OFF property attached to the hierarchical block has a higher priority over the property value attached to the individual components. For example, if the hierarchical block has the TOL_ON_OFF property value set to OFF, tolerance values of all the components within that hierarchical design will be ignored.

Specifications Window

In the Specifications window, add measurements for which you want to analyze the sensitivity of the parameters. You can either import the measurements created in PSpice or can create new measurements in Advanced Analysis.

To import measurements:

1 In the Specifications table, click on the row containing the text "Click here to import a measurement created within PSpice."

The **Import Measurement(s)** dialog box appears.

2 Select the measurements you want to include.

To create new measurements:

1 From the **Analysis** drop-down menu, choose **Sensitivity** - **Create New Measurements**.

The **New Measurement** dialog box appears.

2 Create the measurement expression to be evaluated and click OK.

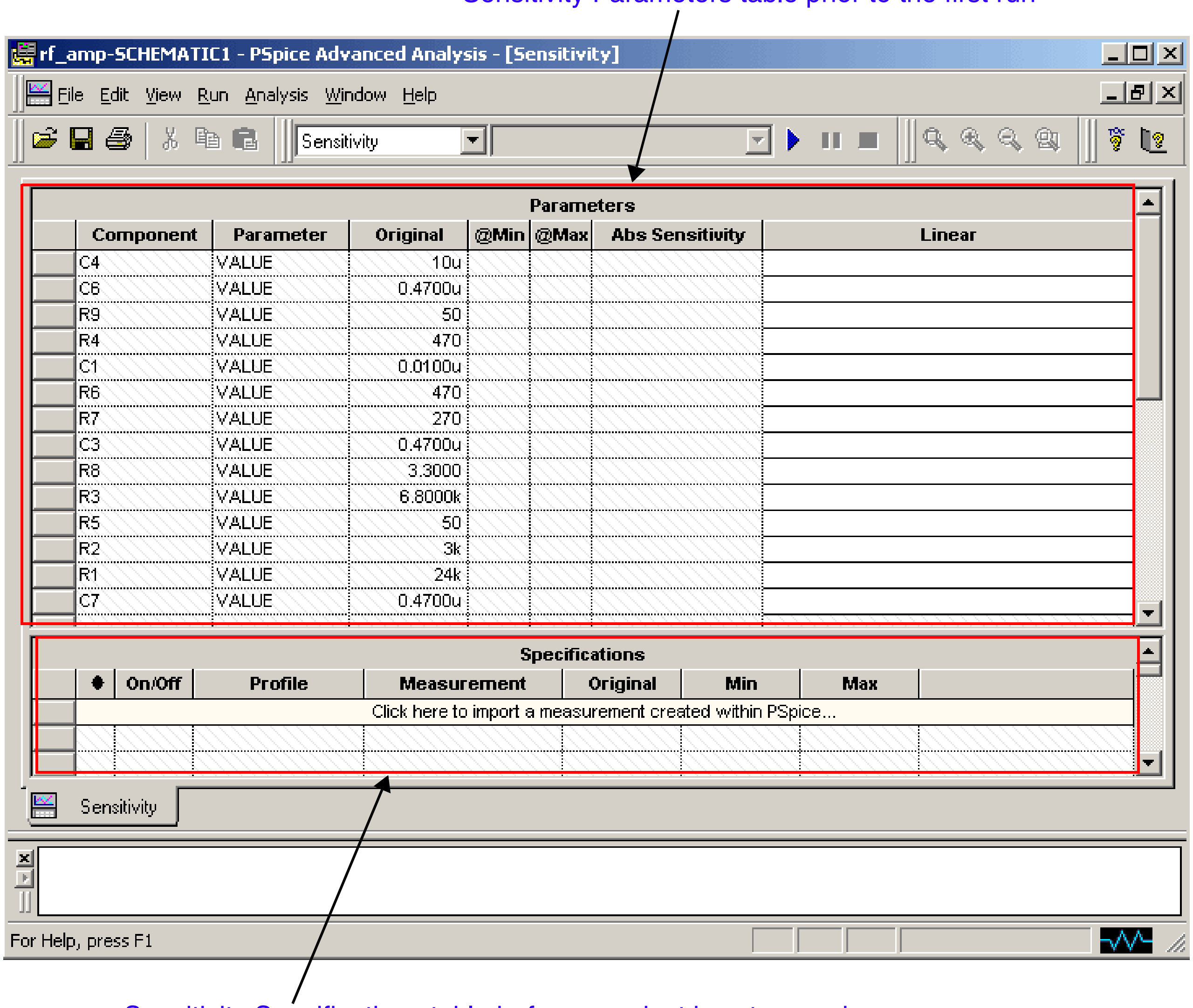
Example: Setting up Sensitivity in Advanced Analysis

Here is an example that shows how to set up Sensitivity in Advanced Analysis:

1 From the **PSpice** menu in your schematic editor, select **Advanced Analysis** / **Sensitivity**.

The Advanced Analysis window opens, and the Sensitivity tool is activated. Sensitivity automatically lists component parameters for which tolerances are specified and the component parameter original (nominal) values.

Sensitivity Parameters table prior to the first run

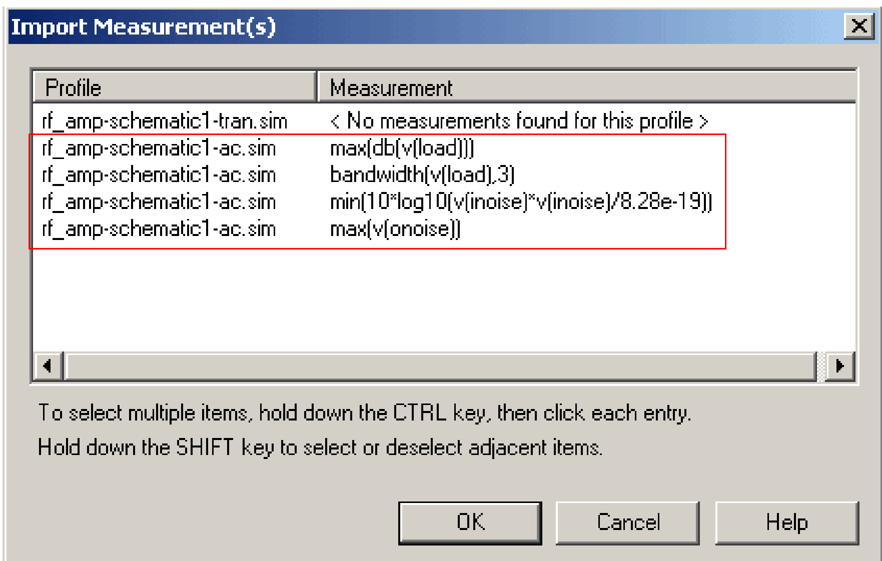


Sensitivity Specifications table before a project is set up and run

In case you want to remove some parameters from the Parameters list, you can do so by modifying the parameter properties in the schematic tool.

2 In the Specifications table, right click the row titled, “Click here to import a measurement created within PSpice.”

The **Import Measurement(s)** dialog box appears with measurements configured earlier in PSpice .



3 Select the four ac.sim measurements.

4 Click **OK**.

The Specifications table lists the measurements.

		On/Off	Profile	Measurement	Original	Min	Max
		✓	rf_amp-schematic1...	max(db(v(load)))			
		✓	rf_amp-schematic1...	bandwidth(v(load),3)			
		✓	rf_amp-schematic1...	min(10*log10(v(inoi...			
		✓	rf_amp-schematic1...	max(v(onoise))			
	Click here to import a measurement created within PSpice...						

You can see the following for more information:

Creating measurements | Composing measurement expressions
Testing measurements | Viewing results of measurements

Running sensitivity

The following section explains the important aspects and terms related to sensitivity runs, such as absolute and relative sensitivity, worst-case maximums and minimums, the runs that sensitivity performs, how to start a sensitivity run, and how to display Sensitivity data.

Absolute sensitivity

Absolute sensitivity is the ratio of change in a measurement value to a one unit positive change in the parameter value.

For example: There may be a 0.1V change in voltage for a 1 Ohm change in resistance.

The formula for absolute sensitivity is:

$$[(M_s - M_n) / (P_n * S_v * Tol)]$$

Where:

M_s = the measurement from the sensitivity run for that parameter

M_n = the measurement from the nominal run

Tol = relative tolerance of the parameter

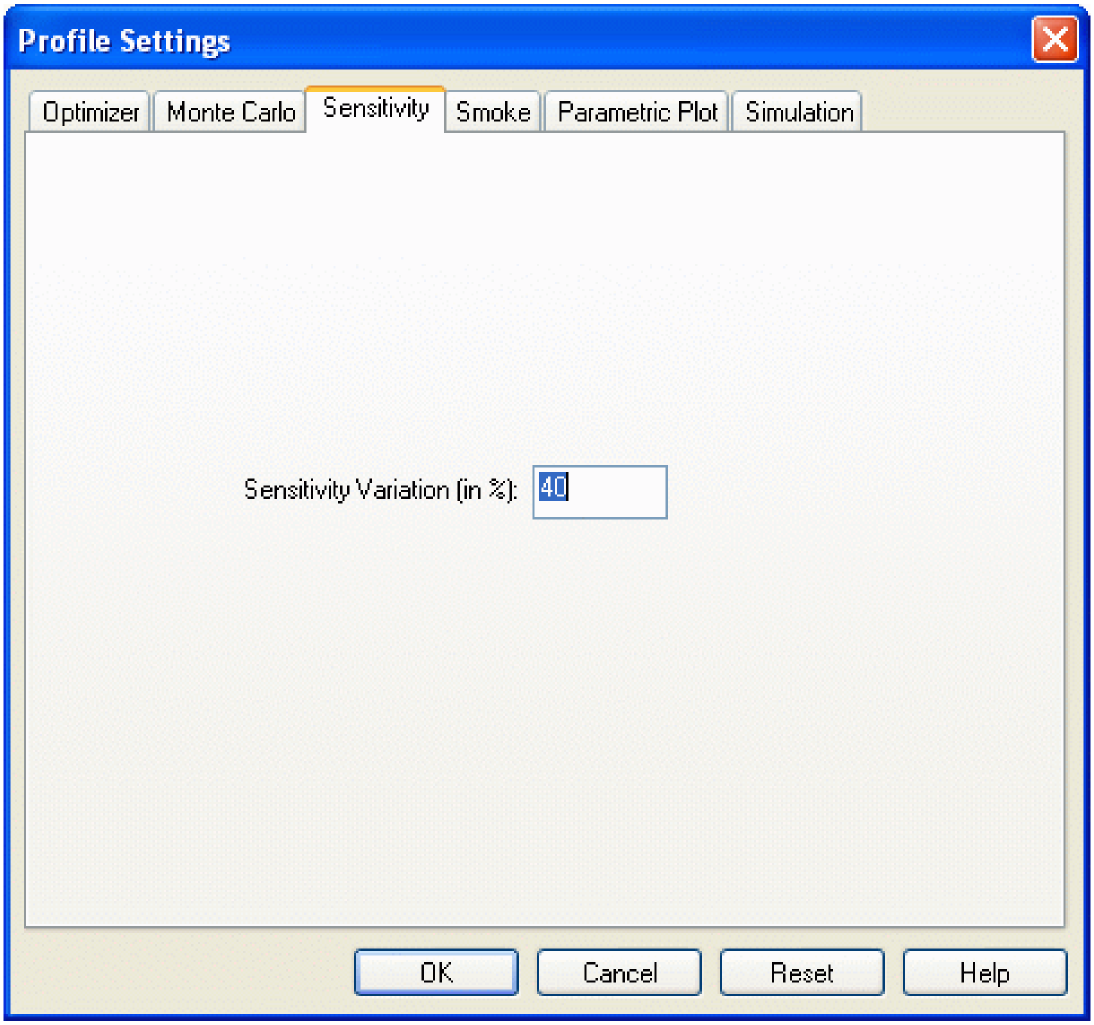
P_n = Nominal parameter value

S_v = Sensitivity Variation. (Default = 40%)

By default, the parameter value is varied within 40% of the set tolerance.

You can change this value to any desired percentage using the Profile settings dialog box.

1. From the Edit drop-down menu, choose *Profile Settings*.
2. In the Profile Settings dialog box, select the *Sensitivity* tab.



3 In the *Sensitivity Variation* box, specify the value by which you want to vary the parameter value.

4 Click OK to save your settings.

The values entered by you in the Profile Setting dialog box, are stored for the future use as well. Every time you load the project, old values are used for advanced analysis simulations.

Example

For example, if you specify the Sensitivity Variation as 10%, the parameter values will be varied within 10% of the tolerance value.

Consider that you want to test a resistor of 100k for sensitivity. The tolerance value attached to the resistor is 10%.

By default, for sensitivity calculations, the value of resistor will be varied from 96K to 104K. But if you change thedefault value of Sensitiviy Variation to 10%, the resistor values will be varied from 99K to 101K for sensitivity calculations.

Relative sensitivity

Relative sensitivity is the percentage of change in a measurement based on a one percent positive change of a component's parameter value.

For example: For each 1 percent change in resistance, there may be 2 percent change in voltage.

The formula for relative sensitivity is:

$$[(Ms - Mn) / (S_v * Tol)]$$

Where:

M_s = the measurement from the sensitivity run for that parameter

M_n = the measurement from the nominal run

Tol = relative tolerance of the parameter

S_v = Sensitivity Variation. (Default = 40%)

Relative sensitivity calculations determine the measurement change between simulations with the component parameter first set at its original value and then changed by *Sv* percent of its positive tolerance. Linearity is assumed. This approach reduces numerical calculation errors related to small differences.

For example, assume that an analysis is run on a 100-ohm resistor which has a tolerance of 10 percent. The maximum value for the resistor would be 110 ohms. Assuming the default value of *Sv*, which is 40%, the analysis is run with the value of the resistor set to 104 ohms (40 percent of the 10 ohm tolerance) and a measurement value is obtained. Using that value as a base, Sensitivity assumes that the resistance change from 100 to 104 ohms is linear and calculates (interpolates) the measured value at 1 percent tolerance (101 ohms).

Worst-case minimums and maximums

For each measurement, Sensitivity sets all parameters to their tolerance limits in the direction that will increase the measurement value, runs a simulation, and records the measurement value. Sensitivity then sets the parameters to the opposite tolerance limits and gets the resulting value.

If worst-case measurement values are within acceptable limits for the design, the measurements can in most cases be ignored for the purpose of optimization.

Sensitivity assumes that the measured quantity varies monotonically throughout the range of tolerances. If not (if there is an inflection point in the curve of output function values), the tool does not detect it. Symptoms of this include a maximum worst-case value that is less than the original value, or a minimum value greater than the original value.

Sensitivity analysis runs

Sensitivity performs the following runs:

- A nominal run with all parameters set at original values
- The next run with one parameter varied within tolerance

  Values are obtained for each measurement. View the Log File for parameter values used in each measurement calculation.
- Subsequent runs with one parameter varied within tolerance
- A minimum worst-case run for each measurement
- A maximum worst-case run for each measurement

For our example circuit with 4 measurements and 12 parameters with tolerances, Sensitivity performs 21 runs.

There is one worst-case minimum and one worst-case maximum run per measurement

1 + 12 + (2 x 4) = 21 runs

The nominal run using the original parameter values

There are four measurements used in this example

There is one run for each parameter varied within tolerance. We use 12 parameters

To see the details of parameter and measurement calculations: from the **View** menu select **Log File**.

Starting a Sensitivity run

Click to start

To start a sensitivity run:

- Click on the top toolbar.

  The Sensitivity analysis begins. The messages in the output window tell you the status of the analysis.

Displaying Sensitivity data

Sensitivity displays results in two tables for each selected measurement:

- Parameters table
  - Parameter values at minimum and maximum measurement values
  - Absolute / Relative sensitivities per parameter

- Linear / Log bar graphs per parameter
- Specifications table
  - Worst-case min and max measurement values

Sorting data

To sort data:

➤ Double click on column headers to sort data in ascending or descending order.

Reviewing measurement data

– Select a measurement on the Specifications table.

A black arrow appears in the left column on the Specifications table, the row is highlighted, and the **Min** and **Max** columns display the worst-case minimum and maximum measurement values.

The Parameters table will display the values for parameters and measurements using the selected measurement only.

Interpreting @min and @max

Values displayed in the **@min** and **@max** columns are the parameter values at the measurement's worst-case minimum and maximum values.

If a measurement value is insensitive to a component, the sensitivity displayed for that component will be zero. In such cases, values displayed in the @Min and @Max columns will be same and will be equal to the Original value of the component.

Negative and positive sensitivity

If the absolute or the relative sensitivity is negative it implies that for one unit positive increase in the parameter value, the measurement value increases in the negative direction.

For example, if for a unit increase in the parameter value, the measurement value decreases, the component exhibits negative sensitivity. It can also be that for a unit decrease in the parameter value, there is an increase in the measurement value.

On the other hand, positive sensitivity implies that for a unit increase in the component value, there is an increase in the measurement value.

Changing from Absolute to Relative sensitivity

1. Right click anywhere in the Parameters table.
2. Select **Display** / **Absolute Sensitivity** or **Relative Sensitivity** from the pop-up menu.

Changing bar graph style from linear to log

Most of the sensitivity values can be analyzed using the linear scale. Logarithmic scale is effective for analyzing the smaller but non-zero sensitivity values.

To change the bar graph style,

1. Right-click anywhere in the Parameters table.
2. Select **Bar Graph Style** / **Linear** or **Log** from the pop-up menu.

Important

If 'X' is the bar graph value on a linear scale, then the bar graph value on the logarithmic scale is not log (X). The logarithmic values are calculated separately.

Interpreting <MIN> results

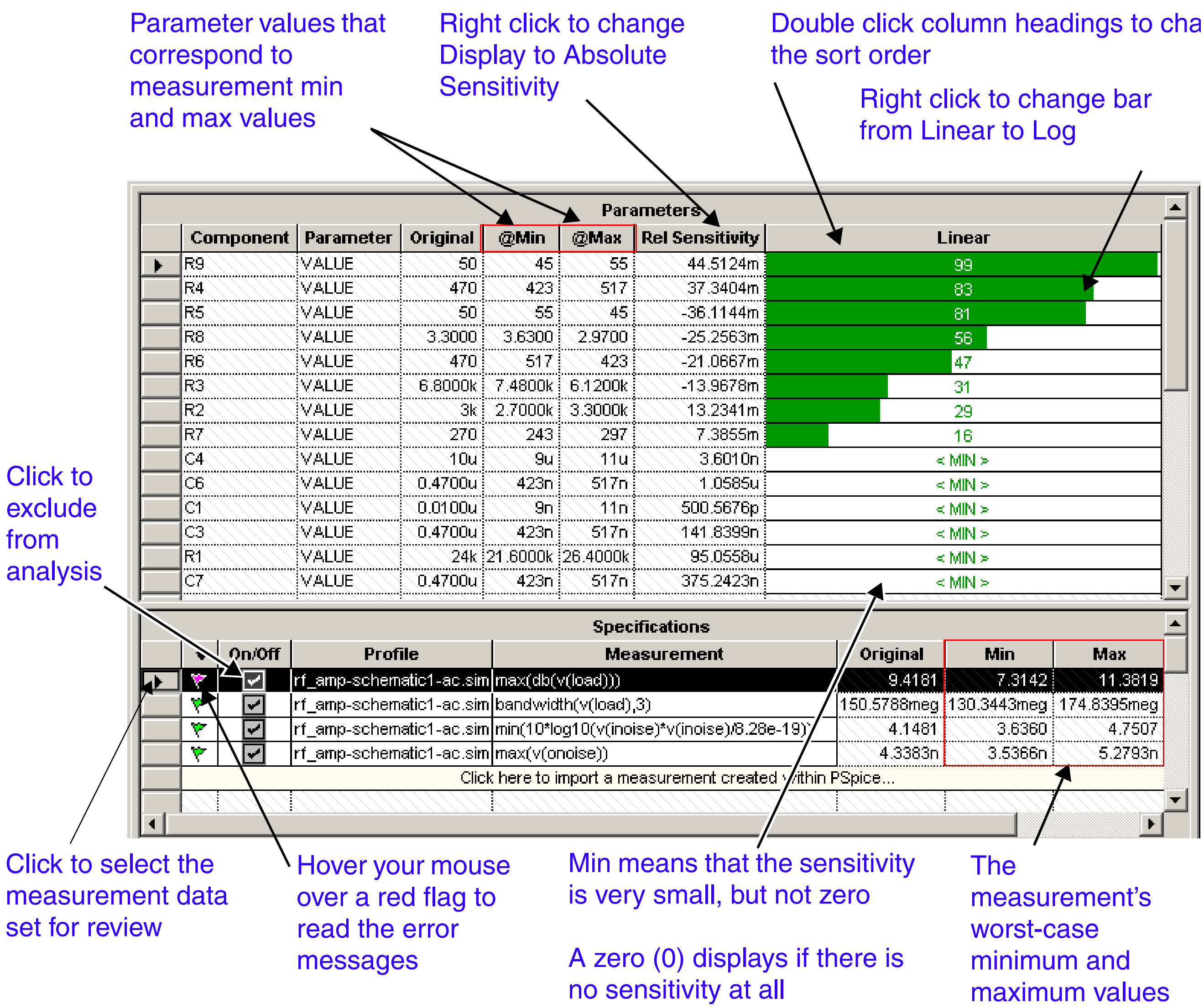
Sensitivity displays <MIN> on the bar graph when sensitivity values are very small but nonzero.

Interpreting zero results

Sensitivity displays zero in the absolute / relative sensitivity and bar graph columns if the selected measurement is not sensitive to the component parameter value.

Example: Displaying Sensitivity data

Results are displayed in the Parameters and Specifications tables according to the selected measurement.



To sort data:

– Double click on the **Linear** column header to sort the bar graph data in ascending order. Double click again to sort the data in descending order.

To select the measurement to view

– Select a measurement in the Specifications table.

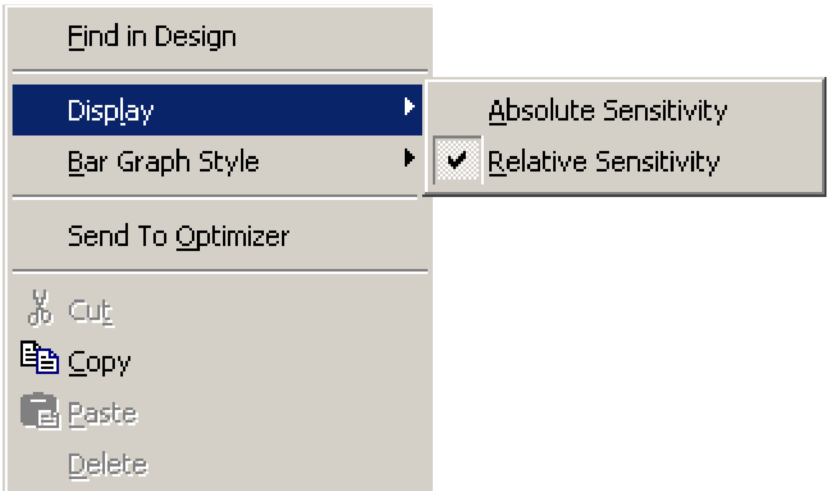
The data in the Parameters table relates to the measurement you selected.

Table...	Column heading...	Means...
Parameters	Original	The nominal component parameter values used to calculate nominal measurement.
	@Min	The parameter value used to calculate the worst-case minimum measurement.
	@Max	The parameter value used to calculate the worst-case maximum measurement.
	absolute sensitivity	The change in the measurement value divided by a unit of change in the parameter value.
	relative sensitivity	The percent of change in a measurement value based on a one percent change in the parameter value.
Specifications	Original	The nominal value of the measurement using original component parameter values.
	Min	The worst-case minimum value for the measurement.
	Max	The worst-case maximum value for the measurement.

Note: To see all the parameter and measurement values used in Sensitivity calculations: from the View menu, select Log File.

1 **Changing from Absolute to Relative sensitivity**Right click anywhere on the Parameters table.

A pop-up menu appears



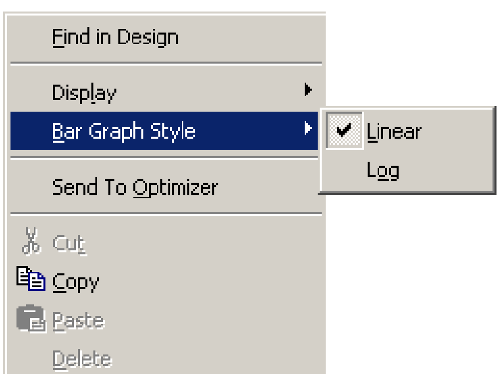
2 Select **Relative Sensitivity**.

Note: See

To change the bar graph to linear view

1 Right click anywhere on the Parameters table.

A pop-up menu appears.

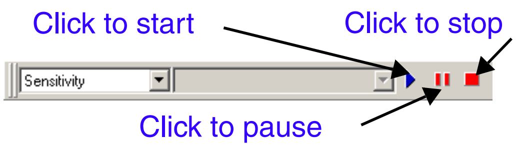


2 Select **Linear**.

Controlling Sensitivity

Data cells with cross-hatched backgrounds are read-only and cannot be edited. The graphs are also read-only.

Pausing, stopping, and starting Sensitivity



Pausing and resuming

1 Click on the top toolbar.

The analysis stops, available data is displayed, and the last completed run number appears in the output window.

2 Click the or to resume calculations.

Stopping

– Click on the top toolbar.

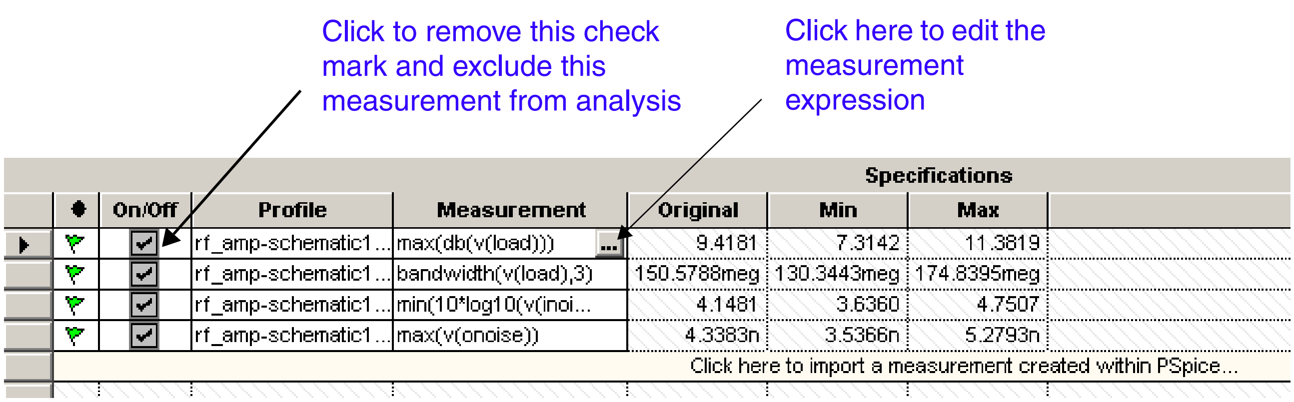
If a Sensitivity analysis has been stopped, you cannot resume the analysis.

Sensitivity does not save data from a stopped analysis.

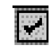
Starting

– Click to start or restart.

Controlling measurements in Sensitivity



	On/Off	Profile	Measurement	Original	Min	Max
				Specifications		
	✔	rf_amp-schematic1...	max(db(v(load)))	9.4181	7.3142	11.3819
	✔	rf_amp-schematic1...	bandwidth(v(load),3)	150.5788meg	130.3443meg	174.8395meg
	✔	rf_amp-schematic1...	min(10*log10(v(inoi...	4.1481	3.6360	4.7507
	✔	rf_amp-schematic1...	max(v(onoise))	4.3383n	3.5366n	5.2793n
			Click here to import a measurement created within PSpice...			

- To exclude a measurement specification from Sensitivity analysis: click  on the applicable measurement row in the Specifications table.

  This removes the check and excludes the measurement from the next Sensitivity analysis.

- To add a new measurement: click on the row containing the text “Click here to import a measurement created within PSpice.”

  The **Import Measurement(s)** dialog box appears.

  Or:

  Right click on the Specifications table and select **Create New Measurement**.

  The **New Measurement** dialog box appears.

- To export a new measurement to Optimizer or Monte Carlo, select the measurement and right click on the row containing the text “Click here to import a measurement created within PSpice.”

  Select **Send To** from the pop-up menu.

You can see the following for more information:

Creating measurement expressions in PSpice	Composing measurement expressions
Creating measurements in Advanced Analysis	Creating measurements in Advanced Analysis
Checking measurement expressions in PSpice	Viewing results of measurements

Adjusting component values

Use **Find in Design** from Advanced Analysis to quickly return to the schematic editor and change component information.

For example: You may want to tighten tolerances on component parameters that are highly sensitive or loosen tolerances on component parameters that are less sensitive.

1. Right click on the component’s critical parameter in the Sensitivity Parameters table and select **Find in Design** from the pop-up menu.
2. Change the parameter value in the schematic editor.
3. Rerun the simulation and check results.

4 Rerun Sensitivity.

Example: Adjusting component values

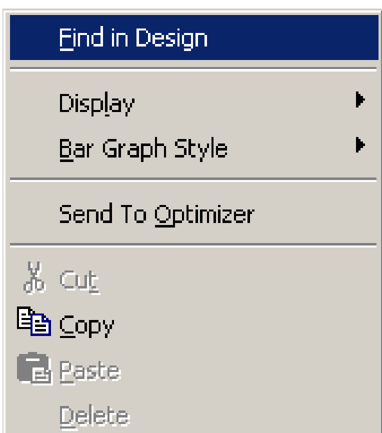
In the RF example, we will not change any component parameters.

With another example you may decide after reviewing sensitivity results that you want to change component values or tighten tolerances. You can use **Find in Design** from Advanced Analysis to return to your schematic editor and locate the components you would like to change.

1 In the Parameters table, highlight the components you want to change.

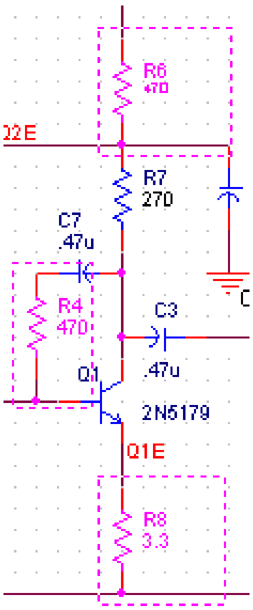
2 Right click the selected components.

A pop-up menu appears.



3 Left click on **Find in Design**.

The schematic editor appears with the components highlighted.



4 Change the parameter value in the schematic editor.

5 Rerun the PSpice simulation and check results.

6 Rerun Sensitivity.

Varying the tolerance range

During Sensitivity analysis, by default Advanced Analysis varies parameter values by 40% of the tolerance range. You can modify the default value and specify the percentage by which the parameter values should be varied within the tolerance range.

To specify the percentage variation:

1 From the **Edit** drop-down menu in Advanced Analysis, choose **Profile Settings**.

2 In the Profile Settings dialog box, select the **Sensitivity** tab.

3 In the Sensitivity Variation text box, specify the percentage by which you want the parameter values to be varied.

4 Click OK to save the modifications.

If you now run the Sensitivity analysis, the value specified by you would be used for calculating the absolute and relative sensitivity.

Sending parameters to Optimizer

Review the results of the Sensitivity calculations. We need to use engineering judgment to select the sensitive components to optimize:

- We won't change R5 or R9 because they control the input and output impedances.
- We won't change R2 or R3 because they control transistor biasing.

The linear bar graph at the Relative Sensitivity setting shows that R4, R6, and R8 are also critical parameters. We'll import these parameters and values to Optimizer.

To send parameters to Optimizer:

1. Select the critical parameters in Sensitivity.
2. Right click and select **Send to Optimizer** from the pop-up menu.
3. Select **Optimizer** from the drop-down list on the top toolbar.

   This switches the active window to the Optimizer view where you can double check that your critical parameters are listed in the Optimizer Parameters table.
4. Click the **Sensitivity** tab at the bottom of the Optimizer Specifications table.

   This switches the active window back to the Sensitivity tool.

– Click  .

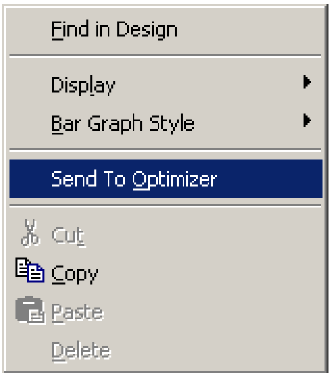
  Or:

  From the **File** menu, select **Print**.

Here is an example of sending parameters to the Optimizer:

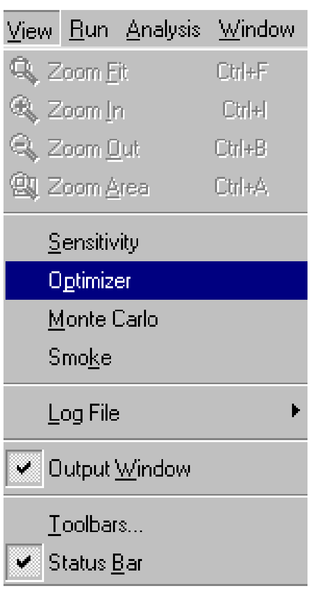
1. In the Parameters table, hold down the Ctrl key and select R4, R6, and R8.
2. Right click the selected components.

A pop-up menu appears.



3 Select **Send to Optimizer**.

4 From the **View** menu, select **Optimizer**.



Select Optimizer view to switch to the Optimizer window and see the parameters you sent over from Sensitivity

Optimizer becomes the active window and your critical parameters are listed in the Optimizer Parameters table.

– Click .

Or:

From the **File** menu, select **Print**.

The Optimizer Tool

Optimizer is a design tool for optimizing analog circuits and their behavior. It helps you modify and optimize analog designs to meet your performance goals.

Note: Advanced Analysis Optimizer is available with the following products:

- PSpice[1] Advanced Optimizer Option
- PSpice Advanced Analysis
- PSpice Optimizer

Optimizer fine tunes your designs faster and automatically than trial and error bench testing can. Use Optimizer to find the best component or system values for your specifications.

Advanced Analysis Optimizer can be used to optimize the designs that meet the following criteria:

- Design should simulate with PSpice.

  You can optimize a working circuit design that can be simulated using PSpice and the simulation results are as desired.

- Components in the design must have variable parameters, each of which relates to an intended performance goal.

Optimizer cannot be used to:

- Create a working design
- Optimize a digital design or a design in which the circuit has several states and small changes in the variable parameter values causes a change of state. For example, a flip-flop is on for some parameter value, and off for a slightly different value.

Setting up the circuit in the schematic editor

Start with a circuit in the schematic editor. The circuit simulations and measurements should be already defined.

The simulation can be a Time Domain (transient), a DC Sweep, or an AC Sweep/Noise analysis.

---

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

1From your schematic editor, open your circuit.

2Simulate the circuit.

3Check your key waveforms in PSpice and make sure they are what you expect.

Test your measurements and make sure they have the results you expect.

Example: Setting up the circuit in the schematic editor

This example uses the tutorial version of RFAmp located at:

`<target directory>\PSpice\tutorial\capture\pspiceaa\rfamp`

The circuit is an RF amplifier with 50-ohm source and load impedances. It includes the circuit schematic, PSpice simulation profiles, and measurements.

Note: For a completed example see:

<target directory>\PSpice\Capture_Samples\AdvAnls\RFAmp directory.

The example uses the goals and constraints features in the Modified LSQ engine. The engine strives to get as close as possible to the goals while ensuring that the constraints are met.

When designing an RF circuit, there is often a trade-off between the bandwidth response and the gain of the circuit. In this example we are willing to trade some gain and input and output noise to reach our bandwidth goal.

Optimizer goal:

- Increase bandwidth from 150 MHz to 200 MHz

Note: Enter meg or e6 for MHz when entering these values in the Specifications table.

Optimizer constraints:

- Gain of at least 5 dB (original value is 9.4 dB)
- Max noise figure of 5 (original value is 4.1)
- Max output noise of 3 nano volts per root Hz (original value is 4.3 nano volts per root Hz)

To set up the circuit:

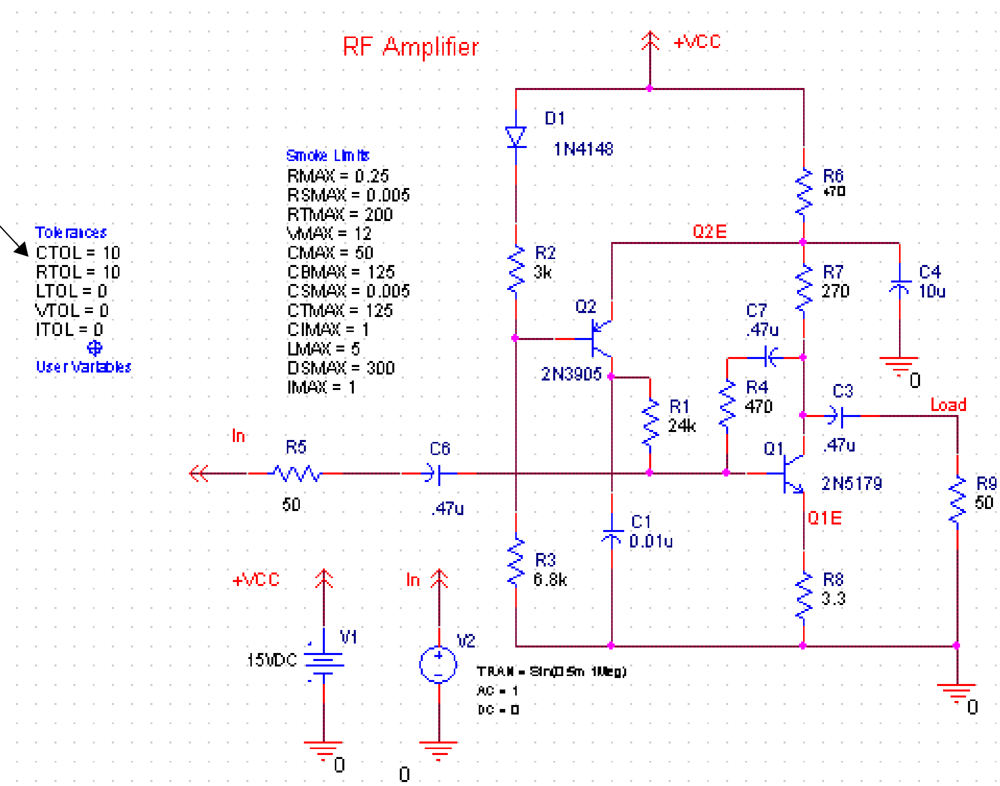
1In your schematic editor, browse to the RFAmp tutorials directory.

<target directory>
\PSpice\tutorial\Capture\pspiceaa\rfamp

2Open the RFAmp project.

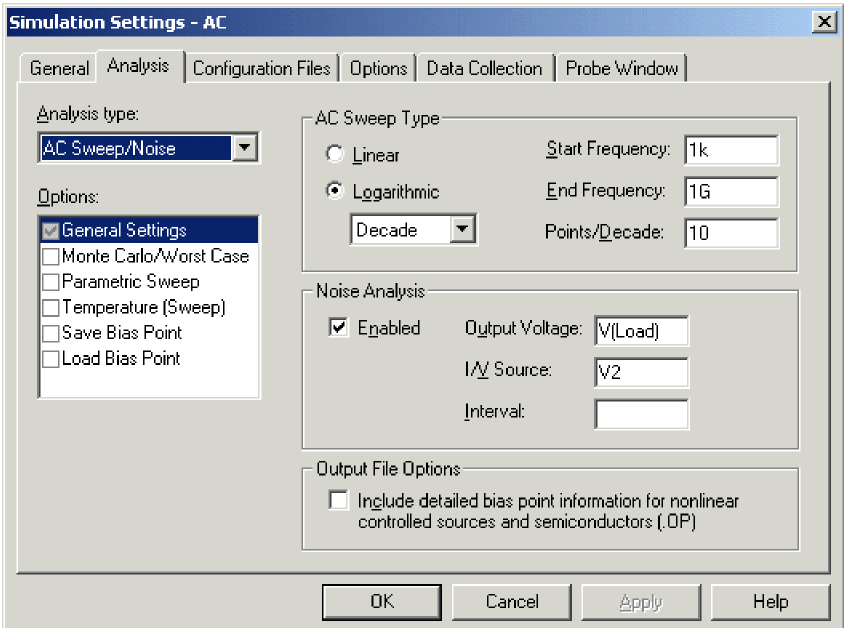
The RF amplifier circuit example

Assign global tolerances using this table



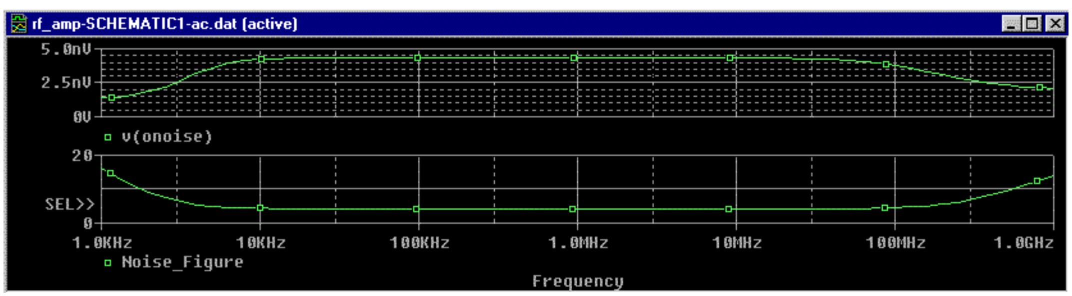
3 Select the SCHEMATIC1-AC simulation profile.

The AC simulation included in the RFAmp example



4Click to run the PSpice simulation.

5Review the results.



The waveforms in PSpice are what we expected.

In PSpice, view measurement results

	Evaluate	Measurement	Value	Measurement Results
	☑	max(db(v(load)))	9.41807	
	☑	bandwidth(v(load),3)	150.57877meg	
▸	☑	min(10*log10(v(inoise)*v(inoise)/8.28...	4.14805	
	☑	max(v(onoise))	4.33832n	

The measurements in PSpice give the results we expected.

You can see the following for more information:

Components and tolerances	Preparing your design for Advanced Analysis
Creating measurement expressions	Composing measurement expressions
Checking measurement expressions in PSpice	Viewing results of measurements

Setting up Optimizer in Advanced Analysis

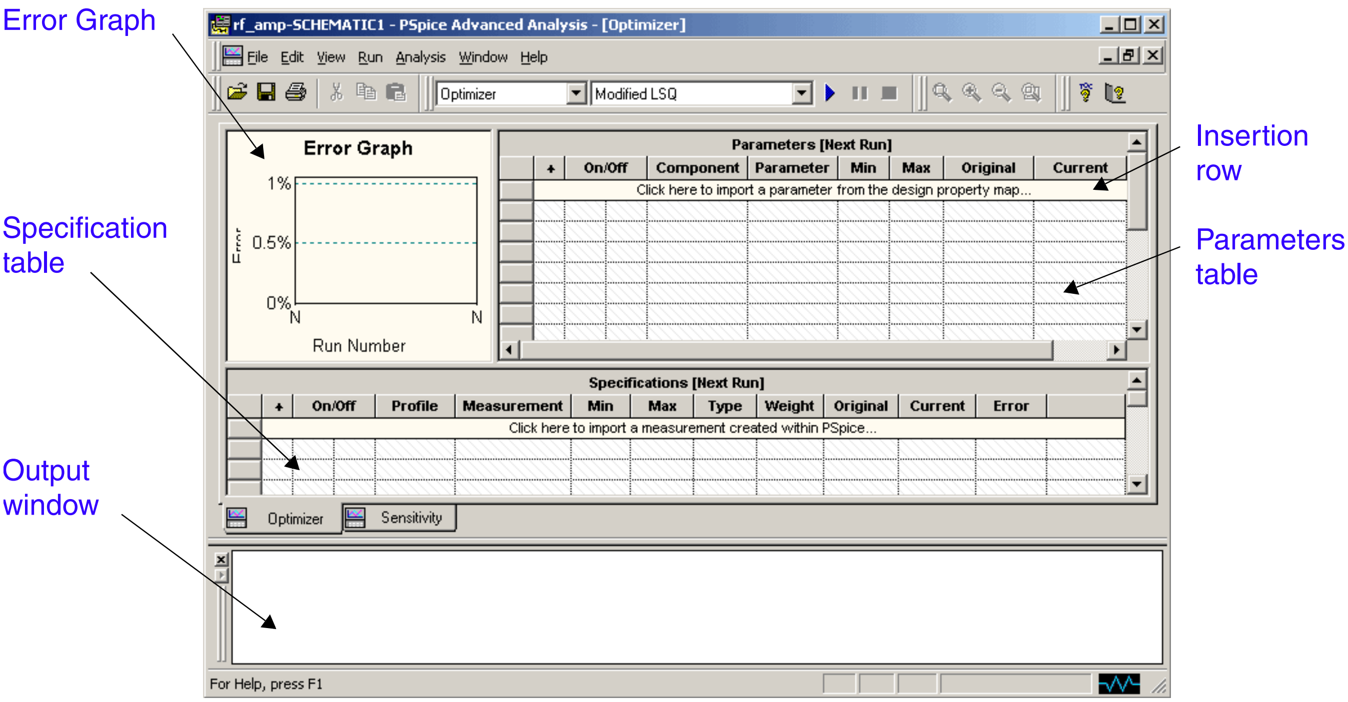
You can see the following for more information:

Parameterized components	Preparing your design for Advanced Analysis
Creating measurement expressions	Composing measurement expressions

Opening Optimizer in Advanced Analysis

– From the **PSpice** menu in your schematic editor, select **Advanced Analysis / Optimizer.**

  The Advanced Analysis Optimizer tool opens.



You can see the following for more information:

Circuit requirements for running Optimizer   Setting up the circuit

Selecting an engine

Optimizer in advanced analysis supports multiple engines. These are Modified LSQ (MLSQ), Random, and Discrete engines. In an optimization cycle, a combination of these engines is used.

Use these Optimizer engines for these reasons:

- Modified LSQ engine: to rapidly converge on an optimum solution.
- Random engine: to pick a starting point that avoids getting stuck in local minima when there is a problem converging.
- Discrete engine: to pick commercially available component values and run the simulation one more time with the selected commercial values.

The normal flow in which these engines are used is Random engine, followed by MLSQ engine, and finally the Discrete engine.

To know more about the Optimizer engines see Engine overview.

To select an optimizing engine:

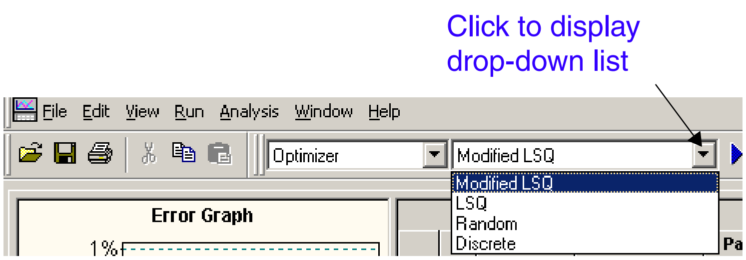
– From the top toolbar engine drop-down list, select one of the three optimizing engines.

Note: The Discrete engine is used at the end of the optimization cycle to round off component values to commercially available values.

For example, to select the Modified LSQ engine:

1Click on the drop-down list to the right of the Optimizer tool name.

A list of engines appears.



2Select the Modified LSQ engine.

You can see the following for more information:

Engine overview	Optimizer Engine Overview
Modified LSQ engine	The Modified LSQ engine
Discrete engine	The Discrete engine
Random engine	The Random engine

Setting up component parameters

In this step, you identify the components or the parts in the circuit, whose parameter values you need to vary. Though the Optimizer in Advanced Analysis can support any number of components, it is recommended that the number of components with the variable parameter values should be kept to minimum.

You can specify parameters using:

- Schematic Editor
- Optimizer
- Sensitivity

Schematic Editor

1In the schematic editor, select the component, whose parameter values you want to vary.

2Select **PSpice > Advanced Analysis > Export Parameters to Optimizer**.

The component gets added in the *Parameters* table.

Note: After you select the component, you can right-click and select **Export Parameters to Optimizer** from the pop-up menu. This command is enabled only if the selected component is based on PSpice-provided templates.

Optimizer

1In the Parameters table in Advanced Analysis, click on the row containing the text "Click here to import."

The **Parameters Selection** dialog box appears.

2Highlight the components you want to vary and click **OK**.

The components are now listed in the *Parameters* table.

Sensitivity

1After you run the sensitivity analysis, select the most sensitive components and right-click.

2From the pop-up menu, select **Send to Optimizer**.

Selected components are listed in the *Parameters* table.

When you add a component to the *Parameters* table, the parameter name, the original value of the parameter, and the minimum and maximum values of the parameter are also listed in the *Parameters* table. The **Min** and **Max** values sets the range the engine will vary the component's parameters. These values are calculated by the Optimizer based on the original value. By default, **Min** value is one-tenth of the **Original** value and **Max** value is ten times the **Original** value.

You can use your engineering judgment to edit the Parameters table **Min** and **Max** values for the Optimization.

Caution

If you reimport any of the parameter that is already present in the *Parameters* ***table, the entries in the*** **Original*****,*** **Min*****, and*** **Max** ***columns are overwritten by the new values.***

Guidelines for selecting components

Optimization parameters need to carefully selected to ensure quicker optimizations and the best results.

- Vary your specification's most sensitive components. Run a sensitivity analysis to find them.
- Use good engineering judgment. Don't vary components whose values need to stay the same for successful circuit operation.

  For example: if the input and output resistors need to be 50 ohms for impedance matching, do not choose those components to optimize.
- Vary just one component if varying other components can cause the same effect.

  For example: in an RC filter combination, both the resistor and capacitor affect the bandwidth. Selecting one parameter simplifies the problem. If your goal cannot be met with one parameter, you can add the second parameter.

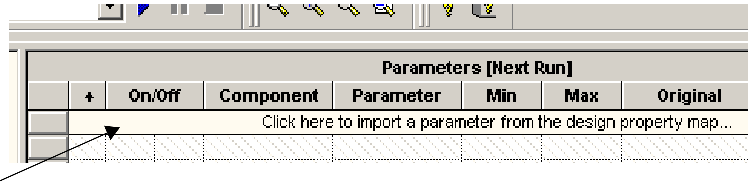
Guidelines for setting up Parameters

- Make sure that ranges you specify take into account power dissipation and component cost.

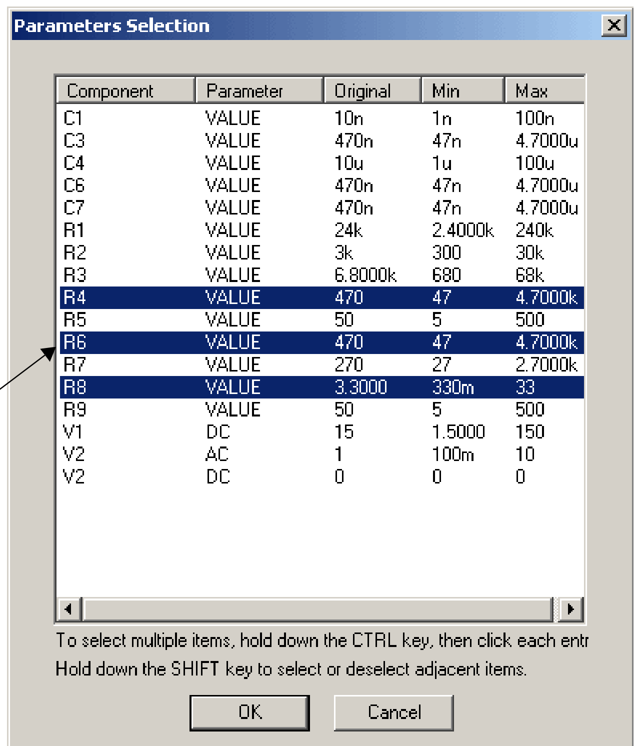
  For example: a resistor with a small value (low ohms) could require a larger, more expensive power rating.
- Start with a small set of parameters (three or four) and add to the list during your optimization process.
- Aim for parameters with initial values near the range midpoints. Optimizer has more trouble finding solutions if parameter values are close to the endpoint of the ranges.
- Keep optimization parameter ranges within 1 or 2 orders of magnitude.

Example: Setting up component parameters

1In the Parameters table, click on the row containing the text “Click here to import...”



The **Parameters Selection** dialog box appears.



2Highlight these components in the **Parameters Selection** dialog box:

- ❑ R6, the 470 ohm resistor
- ❑ R4, the 470 ohm resistor
- ❑ R8, the 3.3 ohm resistor

3Click **OK**.

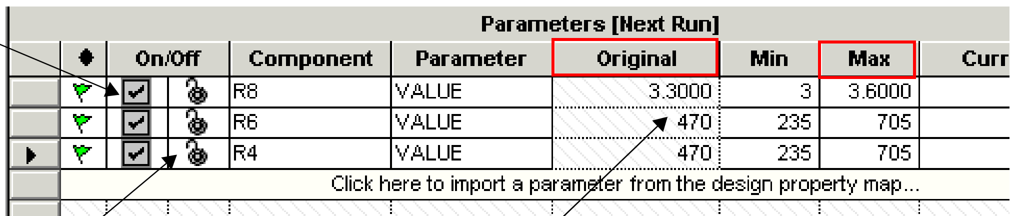
The components are now listed in the Parameters table

4In the Parameters table **Min** and **Max** columns, make these edits:

- ❑ R8: min value 3, max value 3.6
- ❑ R6: min value 235, max value 705
- ❑ R6: min value 235, max value 705

This tightens the range the engine will vary the resistance of each resistor, for more efficient optimization.

Click to remove the check mark, which tells Optimizer to use the Original value without variation during the next optimizing run.



Click to lock in the current value without variation during the next optimizing run.

Click a Min or Max value to type in a change.

Default component values are supplied.
For resistors, capacitors, and inductors the default range is one decade in either direction.

You can see the following for more information:

Choosing parameterized components	Preparing your design for Advanced Analysis
Syntax rules for entering numbers	Introducing the numerical conventions

Setting up specifications in Optimizer

Using the Advanced Analysis Optimizer you can set two types of specifications:

- Measurement specifications - should be used in cases where circuit performance is measurable in terms of variable parameter values, such as gain margin for the circuit.
- Curve-fit specifications - should be used in cases where circuit output is a waveform, such as in wave shaping circuits.

Setting measurement specifications

Measurements (set up earlier in PSpice) specify the circuit behavior we want to optimize. The measurement specifications set the min and max limits of acceptable behavior.

When using the Modified LSQ engine, you can also weigh the importance of the measurement specifications and mark them as constraints or goals.

The engine strives to get as close as possible to the goals while ensuring that the constraints are met.

When there is more than one measurement specification, change the number in the weight column if you want to emphasize the importance of one specification with respect to another.

In the Advanced Analysis Optimizer, you can specify the measurement specification in the Standard tab.

1. In the Specifications table, click on the row containing the text “Click here to import...”

   The **Import Measurements** dialog box appears with measurements configured earlier in PSpice.

2. Highlight the measurements you want to vary and click **OK**.

   The components are now listed in the Specifications table.

3. Specify the acceptable minimum and maximum measurement values in the Specifications table **Min** and **Max** columns.

4. If you are using the Modified LSQ engine, mark the measurement as a goal or constraint by clicking in the **Type** column.

   The engine strives to get as close as possible to the goals while ensuring that the constraints are met.

5. Weigh the importance of the specification using the **Weight** column.

   Change the number in the weight column if you want to emphasize the importance of one specification with respect to another. Use a positive integer greater than or equal to one.

Note: Trial and error experimenting is usually the best way to select an appropriate weight. Pick one weight and check the Optimizer results on the Error Graph. If the results do not emphasize the weighted trace more than the rest of the traces on the graph, pick a higher weight and rerun the Optimization. Repeat until you get the desired results.

Guidelines for setting up measurement specifications

- Determine your requirements first, then how to measure them.

- Don't set conflicting goals.

  For example: Vout > 5 and Vout < 2 when the input is 3V.

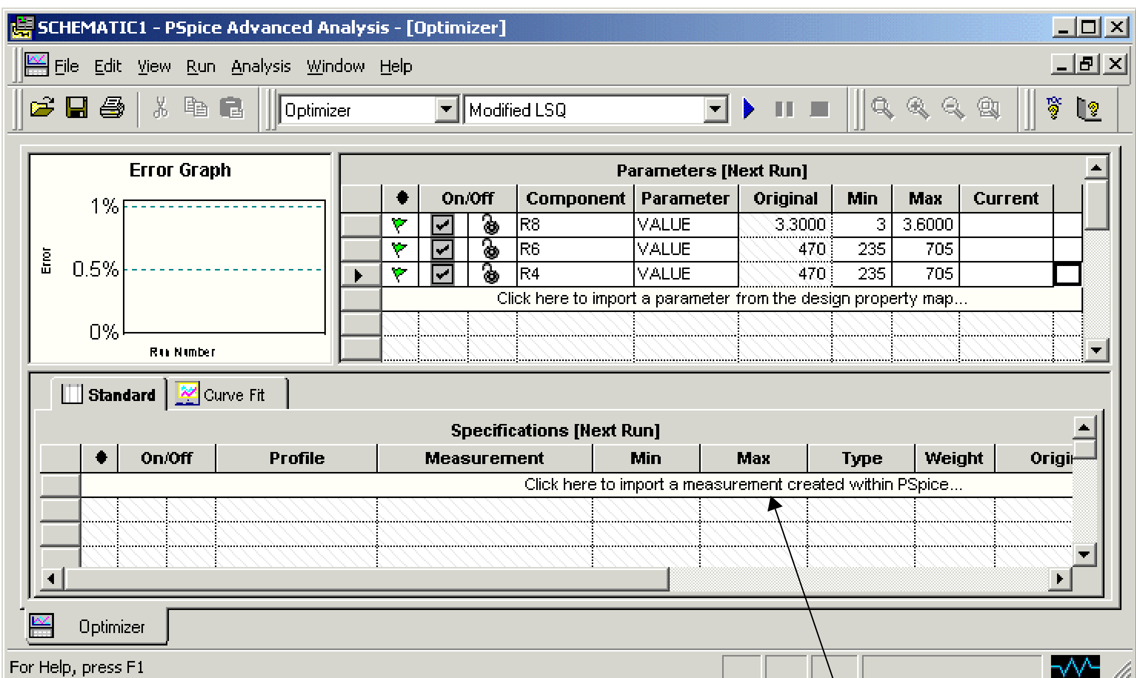
- Make sure enough data points are generated around the points of measurements. Good resolution is required for consistent and accurate measurements.

- Simulate only what's needed to measure your goal.

  For example: for a high frequency filter, start your frequency sweep at 100 kHz instead of 1 Hz.

The following illustrates the procedure with an example:

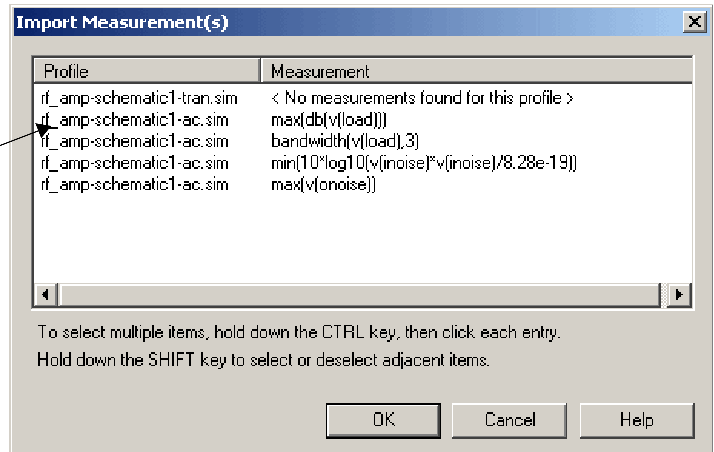
1In the Specifications table, click on the row containing the text "Click here to import...."



Click to import measurements

The **Import Measurements** dialog box appears with measurements configured earlier in PSpice.

Hold down the CTRL key and click to add multiple measurements



2Select all the AC sim measurements and click **OK**.

The measurements are now listed in the Specifications table.

	◆	On/Off		Profile	Measurement
		☑		rf_amp-schematic1-ac.sim	max(db(v(load)))
		☑		rf_amp-schematic1-ac.sim	bandwidth(v(load),3)
		☑		rf_amp-schematic1-ac.sim	min(10*log10(v(inoise)*v(inoise)/8.28e-19))
		☑		rf_amp-schematic1-ac.sim	max(v(onoise))
					Click here to import a measure

3In the **Max(DB(V(Load)))** row of the Specifications table:

- ❑ Min column: type in a minimum dB gain of **5**.
- ❑ Max column: type in a maximum dB gain of **5.5**.
- ❑ Type column: click in the cell and change to **Constraint**
- ❑ Weight column: type in a weight of **20**

4In the **Bandwidth(V(Load),3)** row:

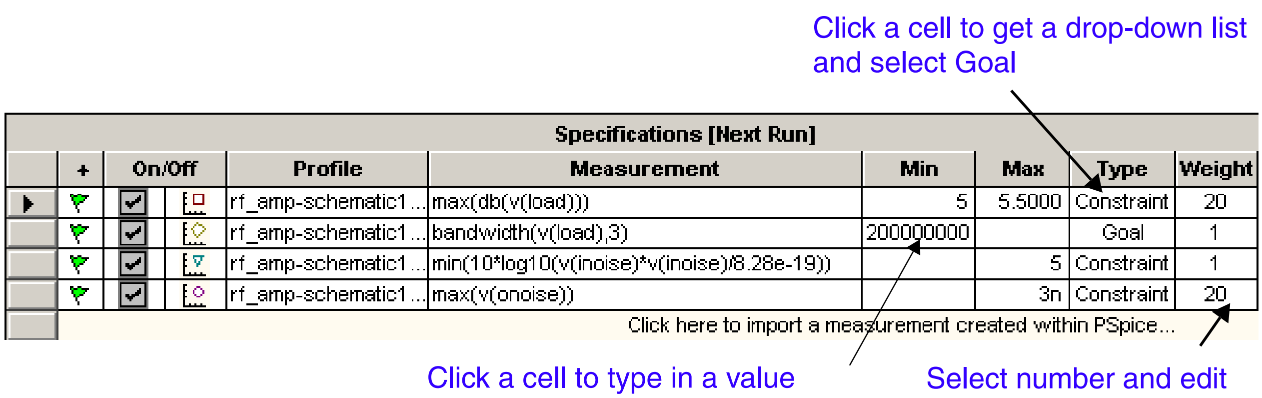
- ❑ Min column: type in a minimum bandwidth response of **200e6**
- ❑ Max column: leave empty (unlimited)
- ❑ Type column: leave as a **Goal**
- ❑ Weight column: leave the weight as **1**

5In the **Min (10*log10(v(in...** row:

- ❑ Min column: leave empty
- ❑ Max column: type in a maximum noise figure of **5**
- ❑ Type column: click in the cell and change to **Constraint**
- ❑ Weight column: leave the weight as **1**

6In the **Max(V(onoise))** row:

- ❑ Min column: leave empty
- ❑ Max column: type in a maximum noise gain of **3n**
- ❑ Type column: click in the cell and change to **Constraint**
- ❑ Weight column: type in a weight of **20**



Tip

It is recommended that you complete the steps for setting up component parameters and measurement specifications. In case you choose not to perform the steps, you can use the `SCHEMATIC1_complete.aap` file located at `..\tools\pspice\tutorial\capture\pspiceaa\rfamp\rf_amp-PSpiceFiles\SCHEMATIC1`. To use the aap file provided with the design example, rename `SCHEMATIC1_complete.aap to SCHEMATIC1.aap`.

You can see the following for more information:

Making a quick edit to a measurement expression	Editing a measurement within Advanced Analysis
Creating measurement expressions	Composing measurement expressions

Syntax rules for entering numbers Introducing the numerical conventions

Using Curve-Fit

Use curve fitting for following:

1To optimize a model to one or more sets of data points. Using curve fitting, you can optimize multiple model parameters to match the actual device characteristic represented either waveforms from data sheets or measured data.

2When the measurement expressions are specified as values at particular points, YatX().

3To optimize circuits that need a precise AC or impulse response. For example, you can use curve fitting for optimizing signal shaping circuits, where the circuit waveform must match the reference waveform.

To use curve fitting for optimizing a design, you need to specify the following in the Curve Fit tab of the Advanced Analysis Optimizer:

1A curve-fit specification

You can either import a specification from an existing `.opt` file or can create a new specification.

Creating a new specification includes specifying a trace expression, a reference file containing measured points and the corresponding measurement values, and a reference waveform.

2List of parameters to be changed

All the optimizable parameters in a circuit are listed in the property map file. This file is created when you netlist the design, and has information of each of the device used in the circuit design.

Performing curve fit

1Open a Capture project (*.opj) or Design Entry HDL project and simulate it.

Verify that circuit is complete and is working fine.

2Invoke Advanced Analysis Optimizer, select the Curve Fit tab.

3Create a curve-fit specification.
Specify the following:

a.Trace Expression

Select a simulation profile and add a trace expression.

b. Name and location of the Reference file

c. Reference waveform as specified in the reference file.

d. Tolerance

e. Weight

4 Select the optimizable parameters.

For each parameter, the original value, the min value (original value/10), and the max value (original value*10) displays automatically. You can change the min-max range as per the requirement.

5 Specify the method for error calculation.

a. From the *Edit* menu, choose *Profile Settings*.

b. From the *Curve-Fit Error* drop-down list in the *Optimizer* tab of the *Profile Settings* dialog box, select the method to be used for the error calculation.

6 Specify whether or not you want to store simulation data.

a. In the *Profile Settings* dialog box, select the Simulation tab.

b. From the Optimizer drop-down list, select *Save All Runs*, if you want the simulation data to be stored, and select *Save None* if you do not want the simulation data to be stored.

7 Select an engine and start the Advanced Analysis Optimizer.

Creating curve fit specification

To create curve fit specification:

1 Specify the Trace Expression.

c. In the Specifications area, click the row stating “Click here to enter a curve-fit specification”.

d. In the New Trace Expression dialog box, select the simulation profile from the Profile drop-down list, and also specify the trace expression or the measurement for which you want to optimize the design.

2 Specify the reference file.

3Specify the reference waveform. The Ref. Waveform drop-down list box lists all the reference waveforms present in the reference file that is specified in the previous step.

4Specify the Weight for the specification.

5Specify the relative tolerance.

You can see the following for more information:

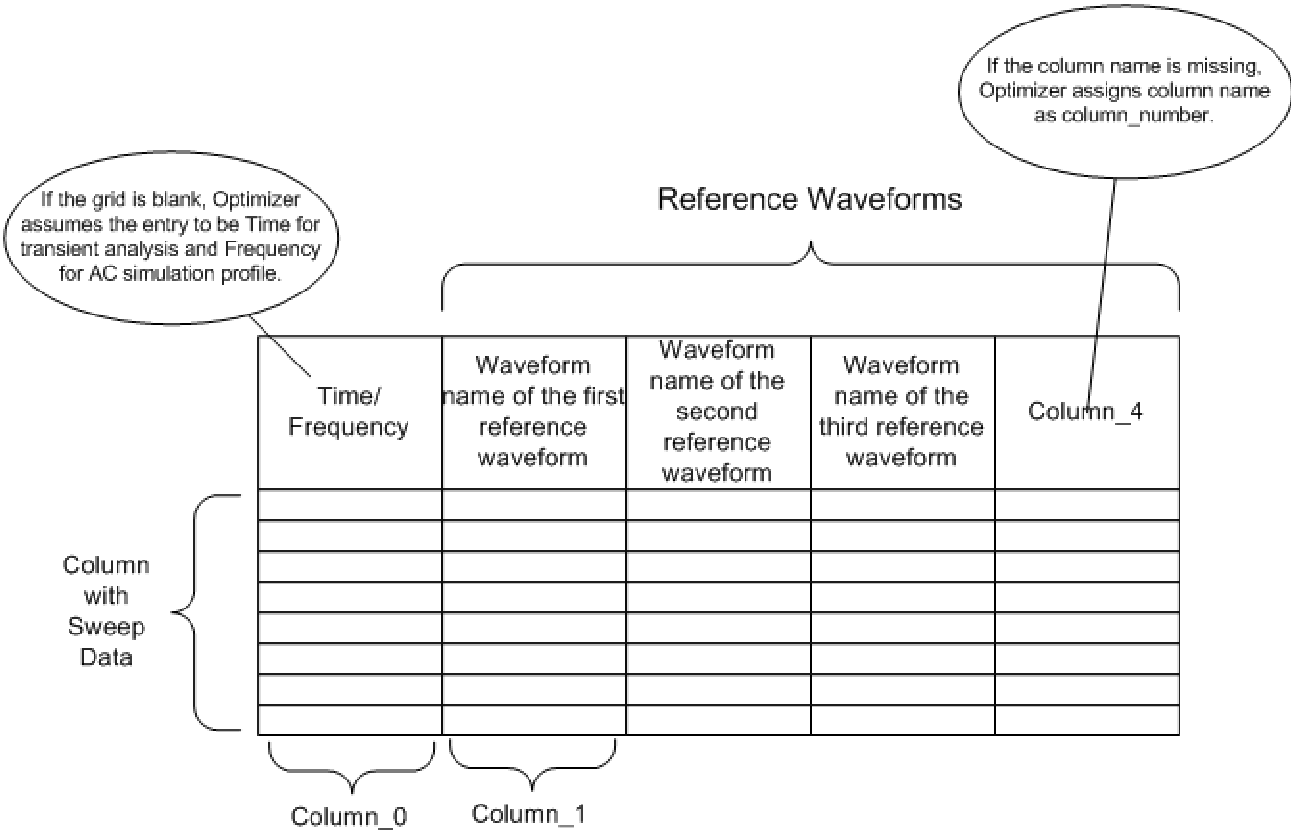
For information on...	see...
What are reference files?	Creating reference files
Creating reference files.	Creating reference files

Creating reference file

To be able to use curve fitting for optimizing your circuit, you must have a reference waveform. In Advanced Analysis Optimizer, the reference waveform is specified in form of multiple data points stored in a reference file. A reference file is a text file that contains the reference waveform with respect to a sweep in the tabular form with the data values separated by *white spaces*, *blanks*, *tabs* or *comma*.

An reference file has to have a minimum of two columns, one for the sweep data and one for the reference waveform. A reference file can have multiple columns. Each extra column represents a different reference waveform.

The format of a multiple column reference file is shown below:



A sample MDP file with one reference waveform is shown below.

```
  Time                V(D4:2)
                   0     1.35092732941686e-022
              2e-010    0.119616948068142
 2.17331331036985e-010    0.129942461848259
 2.51993993110955e-010    0.150499030947685
 3.21319317258894e-010     0.19108946621418
 4.59969965554774e-010    0.270239174365997
 7.37271262146533e-010    0.420916199684143
 1.14672723207623e-009    0.627191662788391
 1.52335408125073e-009    0.802674531936646
 2.27660777959973e-009     1.13146245479584
 3.77361568603665e-009     1.87895023822784
 6.76763149891049e-009      3.6644229888916
 1.27556631246582e-008     7.35082197189331
```

```
2.46214577833191e-008     14.6913433074951
 4.1200489727594e-008      24.834680557251
6.12008282819763e-008     36.7118606567383
8.12011668363586e-008     48.0069961547852
1.01201505390741e-007     58.5374412536621
1.21201843945123e-007     68.1351776123047
1.41202182499506e-007     76.6477890014648
1.61202521053888e-007     83.9403915405273
 1.8120285960827e-007     89.8975143432617
2.01203198162653e-007     94.4249801635742
2.21203536717035e-007     97.4511413574219
2.41203875271417e-007     98.9281539916992
2.61204213825799e-007      98.832633972168
2.81204552380182e-007     97.1660690307617
3.01204890934564e-007     93.9547653198242
```

First column of the reference file contains the sweep data, which is plotted on the X-axis. The first element in the header row indicates the type of analysis. For transient analysis the entry should be **Time**, for ac analysis it is **Freq** (frequency). For the DC-analysis there is no special entry. In case you leave the column header of the first column blank, the Advanced Analysis Optimizer assumes the entries in the sweep column to be time or frequency depending on whether the simulation profile is ac or transient, respectively.

The remaining entries in the header row indicate the names of the reference waveform in each column. These entries are displayed in the Reference Waveform drop-down list of the Curve Fit tab.

You can create a reference file using one of the following.

- Manually

  Write the x,y points of the reference waveform in a text file. Save the text file with either `.mdp`, `.csv`, or `.txt` extension.

- Using the Export command in the PSpice File menu.

  a. Load a `.dat` file in PSpice.

  b. In the PSpice File menu, choose Export. Select Text (.txt file).

  c. The Export Text Data dialog box appears.

  The Output Variable to Export list displays the list of existing traces. You can add or delete traces from this list.

d. In the File name field, specify the name of the reference file and the location where the reference file is to be saved.

e. Click OK to generate the reference file.

To know the details about the Export Text Data dialog box, see *PSpice online help*.

The reference file generated using the Export menu command, has data values separated by tab.

Error calculation

The error displayed in the Error column of the Curve Fit tab is influenced by the following factors:

- Relative Tolerance, specified by the user in the Tolerance column of the Curve Fit tab.
- Curve Fit Gear, specified by the user in the Optimizer tab of the Profile Settings dialog box. Curve fit gears are the methods used for error calculations.

  Note: The Profile Settings dialog box is displayed when you choose Profile settings from the Advanced Analysis Edit menu.

The error displayed is the difference between Root Mean Square Error (E_{rms}) and the tolerance specified by the user.

The Root Mean Square Error (E_{rms}) is calculated using the following formula:

$$E_{rms} = 100 \times \frac{\sqrt{\sum (R_i - S_i)^2}}{\sqrt{\sum (R_i)^2}}$$

Where

$$R_i = Y_{at} X(R, X_i)$$

V_i represents the reference value at the same sweep point.

and

$$S_i = Y_{at} X(S, X_i)$$

Y_i is the simulated data value.

X_i indicates the set of sweep values considered for the error calculation. The value of X_i depends on the gear type selected by the user.

Legacy gear

In this case, each point in the reference waveform is treated as an individual specification (goal) by the Optimizer. In this method, every data point is optimized. Therefore, the error at each data point should be zero. The Optimizer calculates error at each of the reference point and the final error is the RMS of the error at all reference points.

> **Note:** The legacy gear works only if the number of data points to be optimized is less than 250. If the number of data points is more than 250, next gear selected automatically.

Weighted reference gear

In this case, the Advanced Analysis Optimizer considers a union of the reference data points as well as simulation data points in the common interval of time or frequency values. A weight factor is multiplied to the error at each X_i. In this case, Xi will contain both, the reference file points and the simulation sweep points, but the error is calculated by multiplying the weight factor to the error at each point. Therefore, the error is:

$$E_{rms} = 100 \times \frac{\sqrt{\sum W_i \times (R_i - S_i)^2}}{\sqrt{\sum W_i \times (R_i)^2}}$$

Where W_i is the weight that is calculated using the following formula.

- For data points appearing only in the simulation data.

  $$W_i = 1$$

- For data points appearing in the reference waveform.

  $$W_i = \left\lceil \frac{b}{a} \right\rceil^2$$

  Where

  $$b = sizeof\{X_{ref + sim}\}$$

  and

  $$a = sizeof\{X_{ref}\}$$

The `sizeof` function returns the size of the vector.

$X_{ref + sim}$ indicate the union of the reference data points as well as simulation data points in a common interval.

Note: The weighted reference gear is same as Reference data points only gear for cases where $\frac{b}{a} \to \infty$.

Reference only gear

In this case, the Advanced Analysis Optimizer tries to fit in the simulation curve to the curve specified by the reference waveform, and the goal is to minimize the (RMS_{error}/RMS_{ref}) below the tolerance level specified by the user. The error is calculated only at the reference data points. Therefore, X_i will only contain the points on the reference waveform.

The error calculation formula is same as used in the Weighted reference gear, except that W_i is zero for all data points that are not on the reference waveform.

Simulation also gear

In this case, the Advanced Analysis Optimizer considers a union of the reference data points as well as simulation data points in the common interval of Time or frequency values.

Therefore, the error is calculated using the following formula:

$$E_{rms} = 100 \times \frac{\sqrt{\sum (R_i - S_i)^2}}{\sqrt{\sum (R_i)^2}}$$

Note: Notice that if W_i is equal to 1 for all X_i, then the Weighted reference gear is same as the Simulation and reference data points alike gear.

Example

Consider a situation in which the reference sweep or the value of X for the reference waveform, ranges from 30u to 110u. The value of X for the simulation waveform ranges from 0u to 100u. In this case, sweep value for error calculation (Xi) will range from 30u to 100u. This is so because the common interval between ranges 0-100u and 30u-110u is 30u to 100u. Lets assume that in the above-mentioned range, there are 100 reference data points and a total of 400 data points (simulation plus reference) on which error is being calculated. The Erms will be calculated for all the 400 data points.

For each value of Xi, Si, which is the simulated value at Xi, can either be an exact value specified in the simulation data (.dat) file, or it can be the interpolated value at Xi. Similarly, Ri, which is the reference value at Xi, can either be an exact value specified in the reference file, or it can be the interpolated value at Xi.

Thus, for the simulation also curve-fit error gear, Xi contains both the reference file points and the simulation sweep points (a total of 400 data points). The error between the Ri and Si is calculated at each of the 400 points and the RMS of this error waveform is calculated. The ratio of RMS of the error waveform and the RMS of the reference waveform R is calculated and normalized to the equivalent percentage.

For the weighted reference curve-fit error gear, the weighted RMS error is calculated at each of the 400 points (Xi). In this case there is one reference point for every four simulation data points (assuming linear distribution of reference and simulated data points). So each of the reference points is weighted by a scale factor of four (400/100).

Note: In all gears except the legacy gear, error is calculated for all the sweep points that are overlapping between the output wave form and the reference waveform.

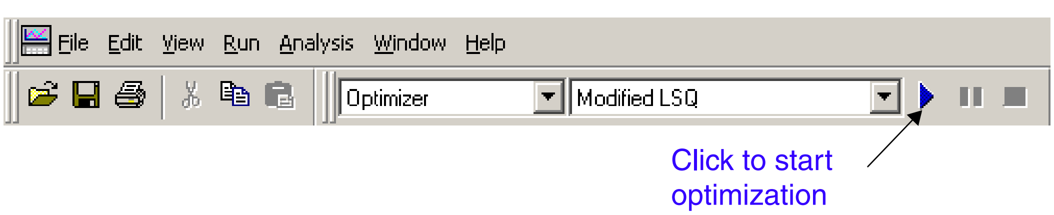
Running Optimizer

Starting a run

To start a run:

– Click on the top toolbar.

The optimization analysis begins. The messages in the output window tell you the status of the analysis.



As the optimization proceeds, the Error Graph shows a plot with an error trace for each measurement.

Data in the Parameters and Specifications tables is updated.

Optimizer finds a solution after five runs.

Displaying run data

To display run data:

– Place your cursor anywhere in the Error Graph to navigate the historical run data.

The Parameters and Specifications tables display the corresponding data calculated during that run. The optimization engine used for each run is displayed in the Optimization Engine drop-down list box. Though the engine name is displayed, the list box is disabled indicating that you can only view the engine used for the optimizer run selected in the Error Graph.

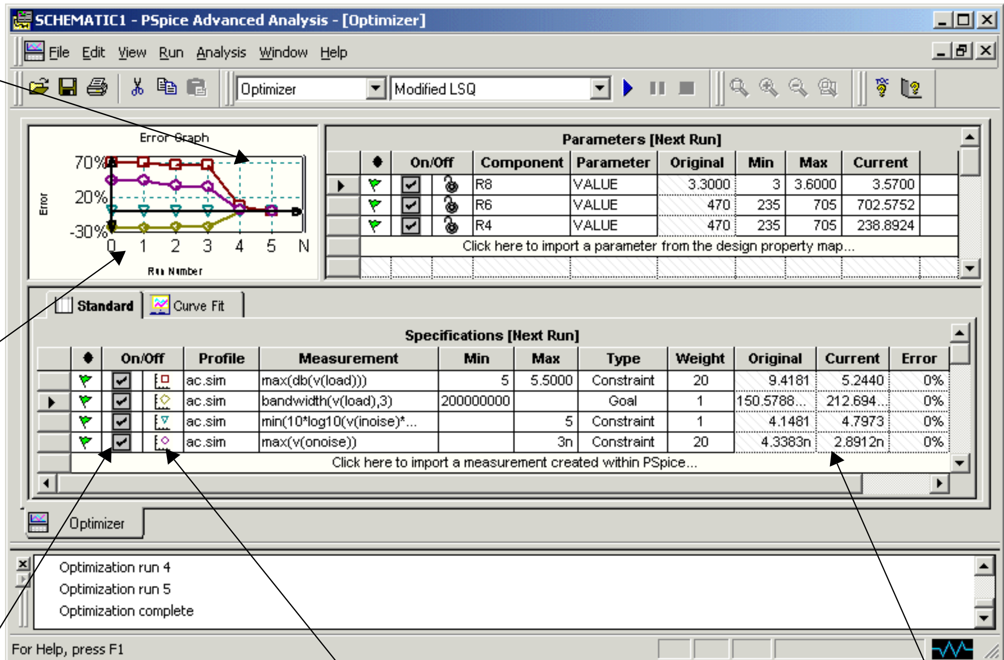
Note: The Advanced Analysis Optimizer saves only the engine name associated with the simulation run. Engine settings are not saved.

Note: Historical run data cannot be edited. It is read-only, as indicated by the cross-hatched background.

Click a run line to see data for that run The data in the Parameters and Specifications tables will change to reflect the values of that run

To clear the Error Graph and remove all historical data: right click on the Error Graph and select Clear History from the pop-up menu.

Click to remove the check mark, which excludes the measurement from the next optimization run



Click the graph symbol to toggle the symbol off, which hides the measurement's trace on the Error Graph

Cells with cross-hatched backgrounds are read-only and cannot be edited.

Clearing the Error Graph history

Selecting the Clear error graph history, retains the value of parameters at the last run. Simulation information for all previous simulation runs is deleted.

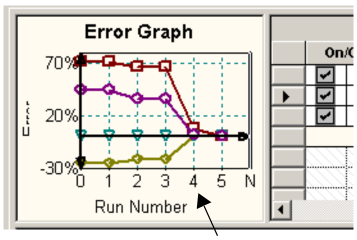
For example, if the Optimizer has information stored for N number of simulation runs then select Clear Error graph history will delete all the simulation information from 0 to N-1 runs. The values in the current column of the Parameters window are used as the starting point for the next simulation run.

To get back the original parameter values, you need to delete all parameters and import again.

To clear error graph history:

– Right click on the Error Graph and select **Clear History** from the pop-up menu.

This removes all historical data and restores the current parameter values to last parameter value.



Right click on the Error Graph and select Clear History.

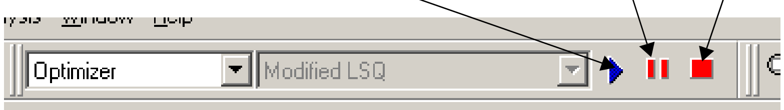
Controlling Optimization

Pausing, stopping and starting

Click to start optimization

Click to pause optimization

Click to stop optimization



You can stop an analysis to explore optimization trends in the Error Graph, reset goals when results are not what you expected, or change engines.

- To start or continue, click ▶ on the top toolbar.
- To pause, click ❚❚ on the top toolbar.

  The analysis pauses at an interruptible point and displays the current data.
- To stop, click ■ on the top toolbar.

Note: Starting after pause or stop resumes the analysis from where you left off.

Controlling component parameters

The range that Optimizer varies a component's parameter is controlled by the Max and Min values.

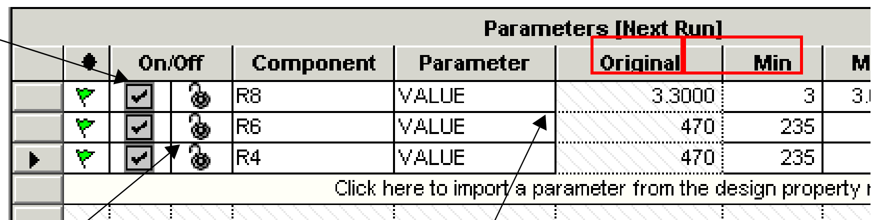
Default component values are supplied. For resistors, capacitors, and inductors the default range is one decade in either direction.

For more efficient optimization, tighten up the range between the Min and Max values.

- To change the minimum or maximum value a parameter is varied: click in the **Min** or **Max** column in the Parameters table and type in the change.
- To use the original parameter value (with no change) during the next optimizing run: click in the Parameters table to toggle the check mark off.
- To lock in the current value (with no change) of a parameter for the next optimizing run: click on the lock icon in the Parameters table to toggle the lock closed .

  Note: If you cannot edit a value, and this is not the first run, you may be viewing historical data. To return to current data, click to the right of the horizontal arrow in the Error Graph.

Click to remove the check mark, which tells Optimizer to use the Original value without variation during the next optimizing run.



Parameters [Next Run]

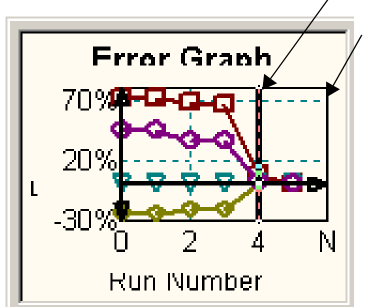
		On/Off		Component	Parameter	Original	Min	M
				R8	VALUE	3.3000	3	3.
				R6	VALUE	470	235	
				R4	VALUE	470	235	
	Click here to import a parameter from the design property							

Click to lock in the current value without variation during the next optimizing run.

Click a Min or Max value to type in a change.

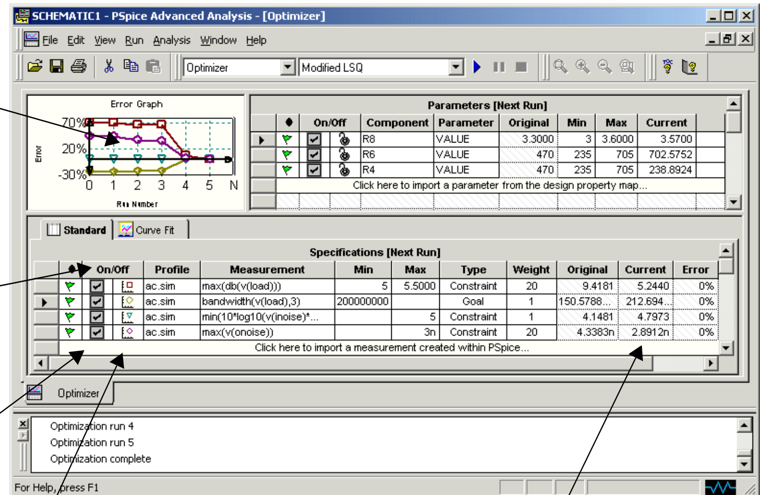
Default component values are supplied.
For resistors, capacitors, and inductors the default range is one deca
in either direction.

Note:
If you can’t edit a value, you might be viewing the historical data (if y
have already run an optimization).



Click here to make changes which will affect the next run.

Controlling measurement specifications



- To exclude a measurement from the next optimization run, click the in the Specifications table, which removes the check mark.
- To hide a measurement's trace on the Error Graph, click the graph symbol icon ( ) in the Specifications table, which toggles the symbol off.
- To edit a measurement, click on the measurement you want to edit, then click on .
- To add a new measurement, click on the row that reads "Click here to import a measurement..."
- To export a new measurement to Optimizer or Monte Carlo, select the measurement and right click on the row containing the text "Click here to import a measurement created within PSpice."

  Select **Send To** from the pop-up menu.

The example for this topic comes with measurements already set up in PSpice.

You can see the following for more information:

Editing measurement expressions within Advanced Analysis	Editing a measurement within Advanced Analysis
Setting up measurement specifications in the Optimizer Specifications table	Setting up measurement specifications in Optimizer
Checking measurement expressions in PSpice	Viewing results of measurements

Copying History to Next Run

During optimization, you might want to modify an Optimizer run by copying parameter values from a previous optimization run into the current run database. You can then modify optimization specifications or engine settings, and run the Optimizer again to see the effects of varying certain parameters.

The Copy History to Next Run command allows you to copy the parameter values of the selected run to the last run which is also the starting point for the next simulation run.

Caution

Using Copy History To Next Run, you can only copy the parameter values of the selected run. The specifications, engine, and engine settings are not copied.

Use the following procedure to copy history.

1In the Error Graph, select a run that you want to copy.

The history marker appears positioned on the selected run.

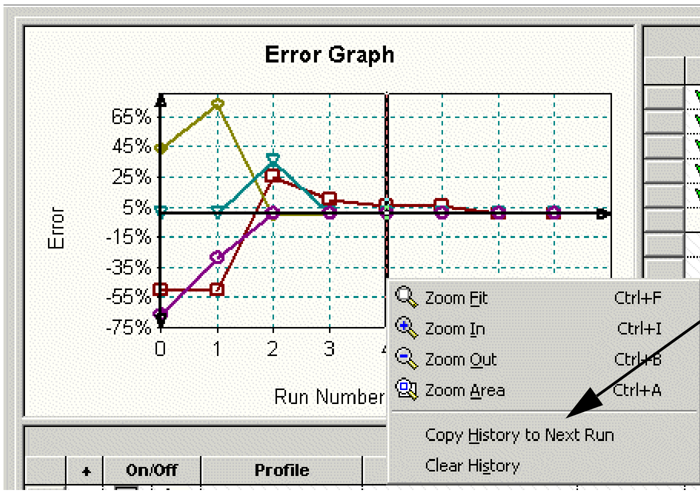
2Right-click on the Error Graph.

3Select **Copy History To Next Run** from the pop-up menu.

The parameters values are copied from the current marker run, for example, Run 1 to the end run.

Note: The **Copy History To Next Run** command is available only when you stop the Optimizer. Selecting **Pause** does not enable this menu command.

Consider a case where during optimization, parameter values do not converge after a particular point. In such cases, you can stop the Optimizer, copy the parameter values to the last run, select a different Optimizer engine and run the optimizer again.



Select a run in the Error Graph, right-click on the error graph and select Copy History To Next Run.

Assigning available values with the Discrete engine

The Discrete engine is used at the end of the optimization cycle to round off components to commercially available values.

Perform the following steps to assign values:

1From the top toolbar engine field, select **Discrete** from the drop-down list.

A new column named **Discrete Table** appears in the Parameters table.

2For each row in the Parameters table that contains an RLC component, click in the **Discrete Table** column cell.

An arrow appears, indicating a drop-down list of discrete values tables.

3Select from the list of discrete values tables.

A discrete values table is a list of components with commercially available numerical values. These tables are available from manufacturers, and several tables are provided with Advanced Analysis.

4Click .

The Discrete engine runs.

The Discrete engine first finds the nearest commercially available component value in the selected discrete values table.

Next, the engine reruns the simulation with the new parameter values and displays the measurement results.

At completion, the **Current** column in the Parameters table is filled with the new values.

5Return to your schematic editor and put in the new values.

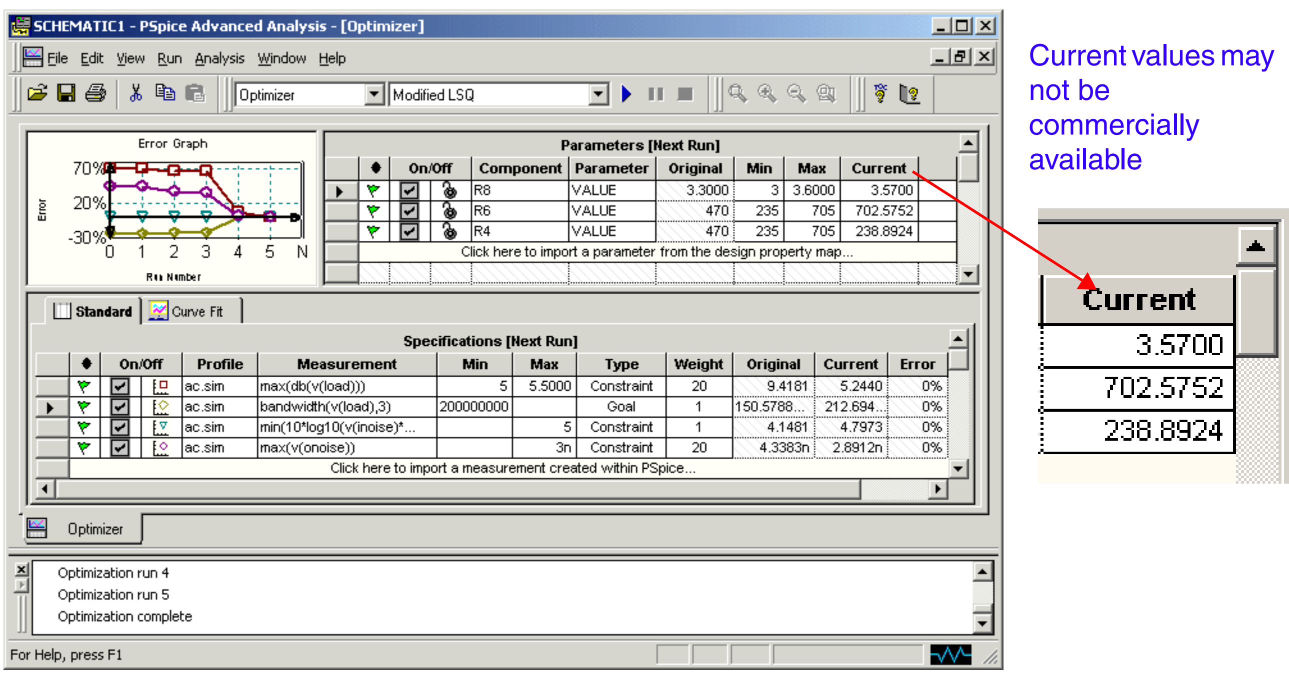
Note: You can use Find in Design to locate components in you schematic editor. See Finding components in your schematic editor.

6While you are still in your schematic editor, rerun the simulation.

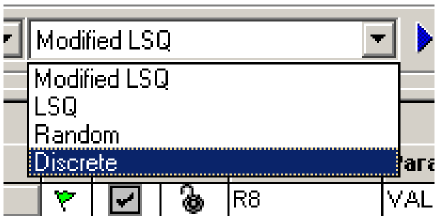
Check your waveforms and measurements in PSpice and make sure they are what you expect.

Example: Assigning available values with the Dscrete engine

At the end of the example run, Optimization was successful for all the measurement goals and constraints. However, the new resistor values may not be commercially available values. You can find available values using the Discrete engine.



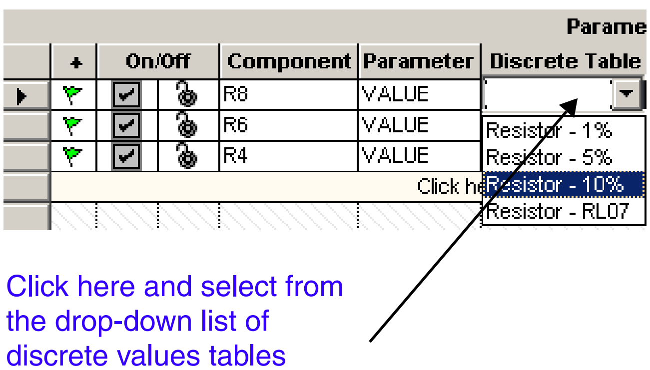
1From the top toolbar engine text box, select Discrete from the drop-down list.



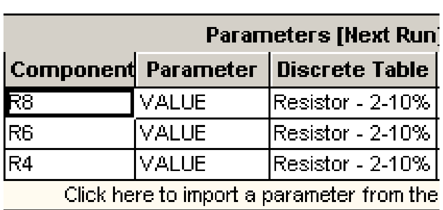
A new column named **Discrete Table** appears in the Parameters table. Discrete values tables for RLC components are provided with Advanced Analysis.

2To select a discrete values table, click on any RLC component's **Discrete Table** column.

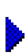
You will get a drop-down list of commercially available values (discrete values tables) for that component.



3Select the 10% discrete values table for resistor R8. Repeat these steps to select the same table for resistors R6 and R4.



Parameters [Next Run]		
Component	Parameter	Discrete Table
R8	VALUE	Resistor - 2-10%
R6	VALUE	Resistor - 2-10%
R4	VALUE	Resistor - 2-10%
Click here to import a parameter from the		

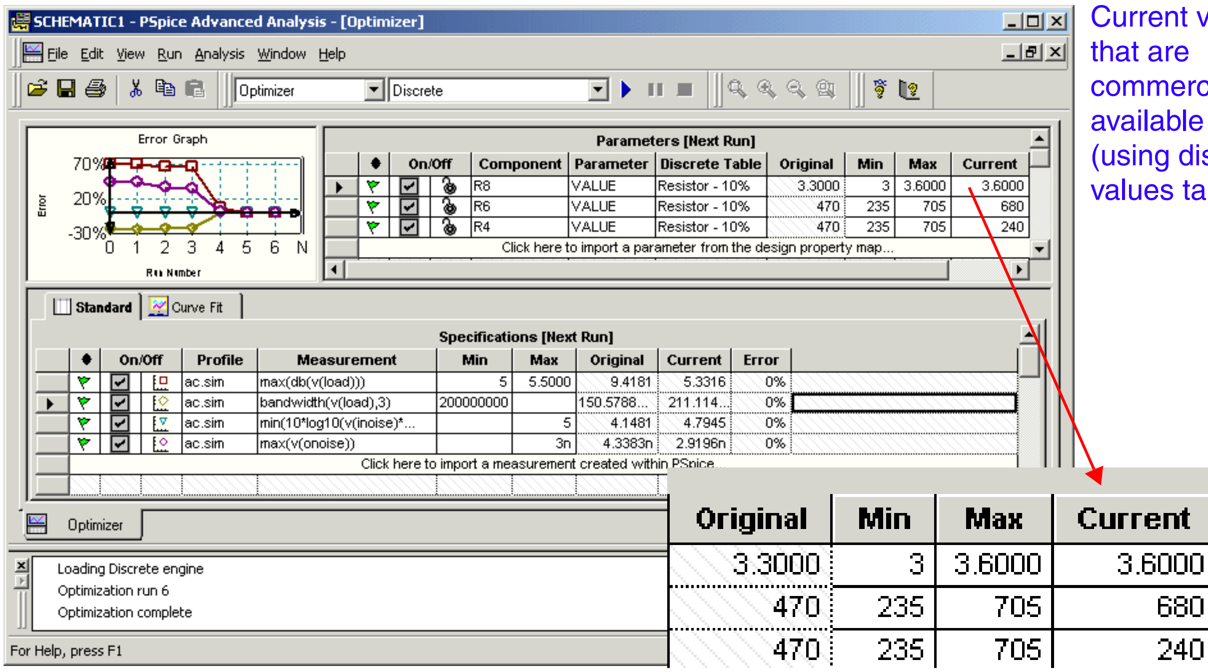
4Click .

The Discrete engine runs.

First, the Discrete engine finds the nearest commercially available component.

Next, the engine reruns the simulation with the new parameter values and displays the measurement results.

At completion, the **Current** column in the **Parameters** table is filled with the new values.



Current values that are commercially available (using discrete values tables)

5Return to your schematic editor and change:

- R8 to 3.6 ohms
- R6 to 680 ohms
- R4 to 240 ohms

6While you are still in your schematic editor, rerun the simulation titled AC.

Check your waveforms and measurements in PSpice and make sure they are what you expect.

Finding components in your schematic editor

You can use the **Find in Design** feature to return to your schematic editor and locate the components you would like to change.

1In the Parameters table, highlight the components you want to change.

2With the components selected, right click the mouse button.

A pop-up menu appears.

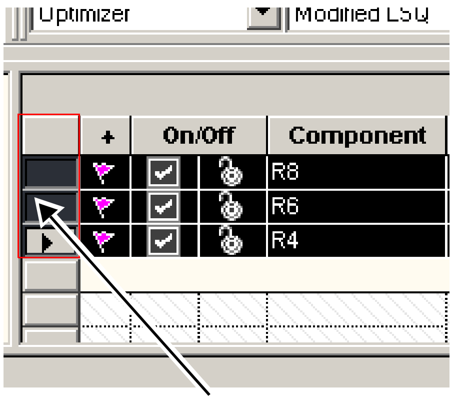
3Select **Find in Design.**

The schematic editor appears with the components highlighted.

Example: Finding components in schematic editor

You can use **Find in Design** from Advanced Analysis to return to your schematic editor and locate the components you would like to change.

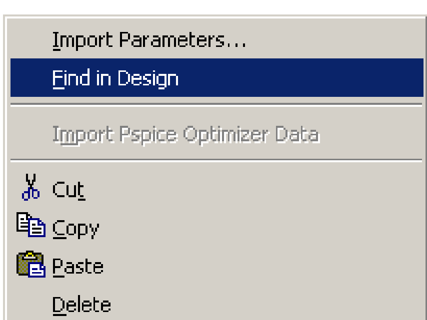
1In the Parameters table, highlight the components you want to change.



Click here to select components (hold down shift key to select several)

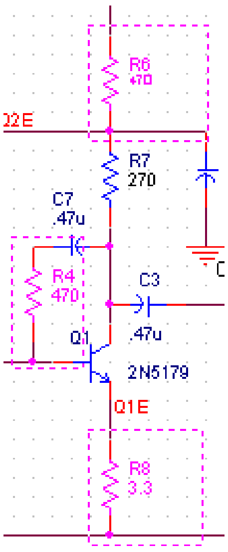
2With the components selected, right click the mouse button.

A pop-up menu appears.



3Left click on **Find in Design.**

The schematic editor appears with the components highlighted.



Examining a run in PSpice

During the optimization process, one or more optimizer runs can fail. To investigate optimization failures,

- Select **Analysis > Optimizer > Troubleshoot in PSpice**.

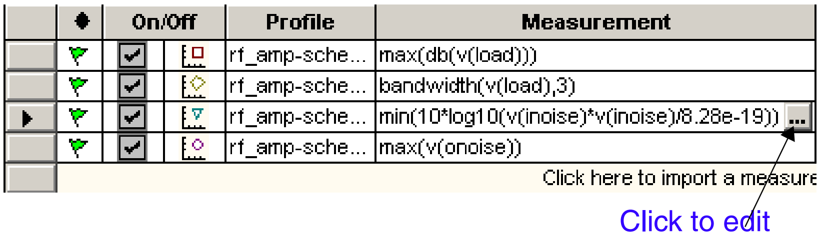
  The simulation profile associated with the selected measurement opens in PSpice. PSpice then automatically opens the waveform viewer and shows a comparison of the last Optimizer simulation to a nominal PSpice simulation. PSpice lists results for both runs in the Measurement spreadsheet for easy comparison.

Editing a measurement within Advanced Analysis

At some point you may want edit a measurement. You can edit from the Specifications table, but any changes you make will not appear in measurements in the other Advanced Analysis tools or in PSpice.

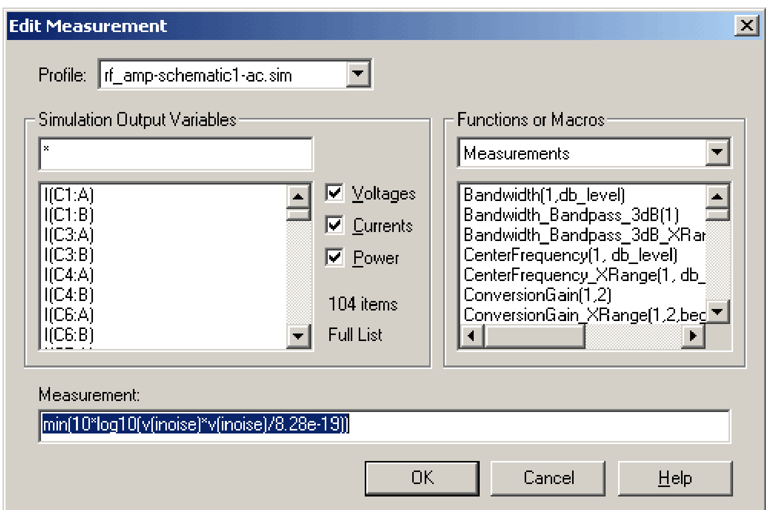
1Click on the measurement you want to edit.

A tiny box containing dots appears.



2Click .

The Edit Measurement dialog box appears.



3Make your edits.

It’s a good idea to edit and run your measurement in PSpice and check its performance before running Optimizer.

4Click **OK**.

– Click .

Or:

From the **File** menu, select **Print**.

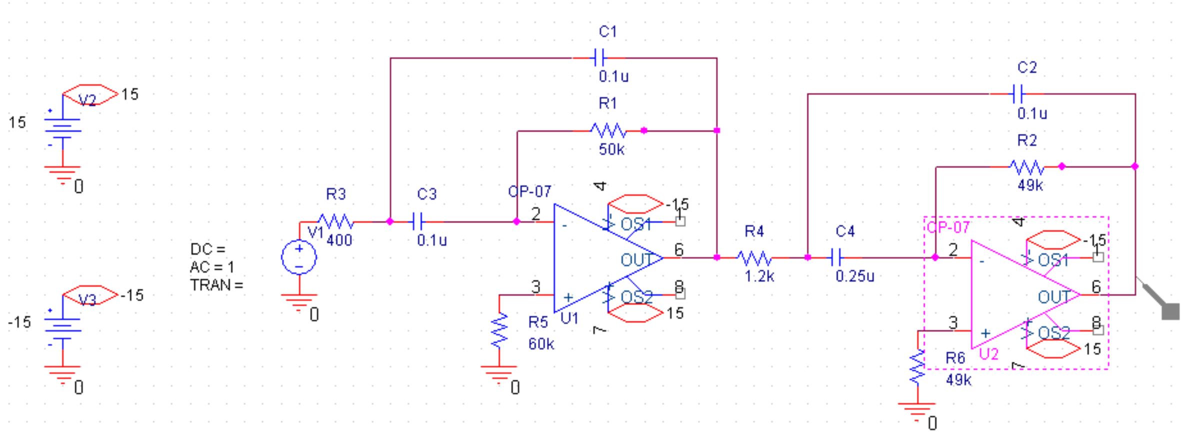
Using Curve-Fit

The design example covered in this section, explains how you can use curve fitting to achieve desired response from a multiple feedback two pole active bandpass filter.

This bandpass filter uses two, 7-pin operational amplifiers. A plot window template marker, Bode Plot dB - dual Y axes is added at the output of the second operational amplifier (before R7). This marker is used to plot the magnitude and the phase gain of the output voltage.

The design example is available at
`..\tools\pspice\tutorial\capture\pspiceaa\bandpass`.

Figure 12-1 Bandpass Filter

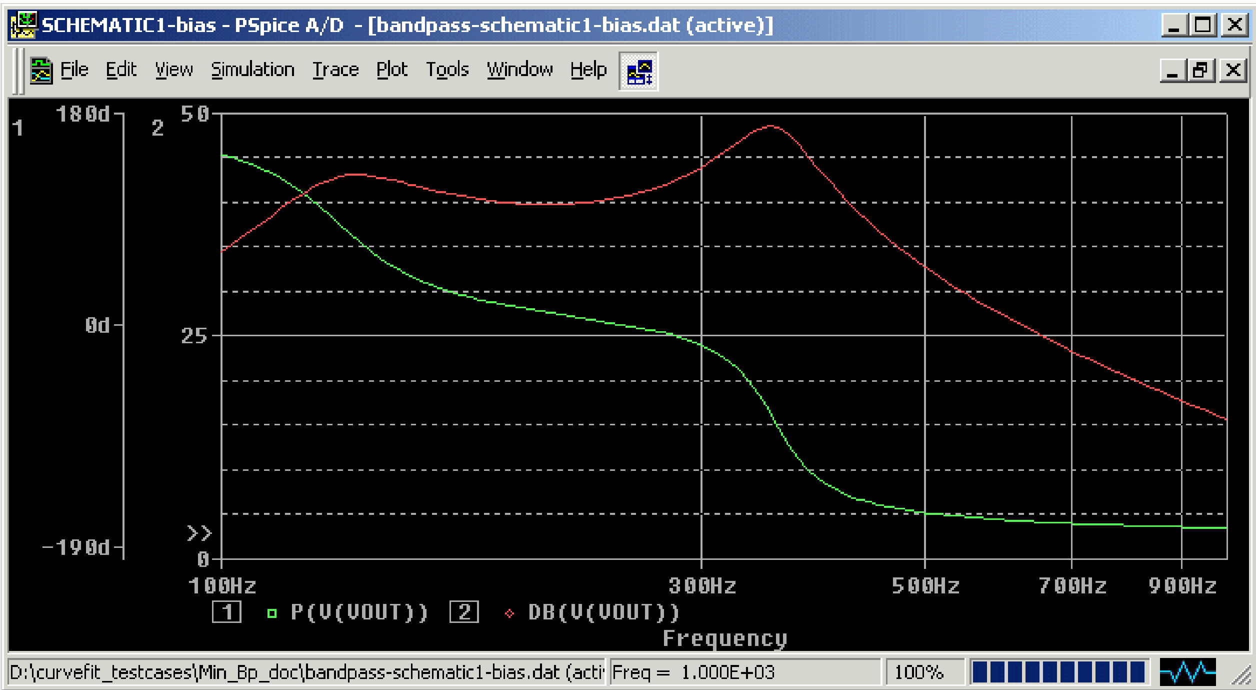


1 Draw the circuit as shown in Figure 12-1.

2 Simulate the circuit.

From the PSpice menu, choose Run.

3The PSpice probe widow appears displaying the simulation results. Two traces, one for phase gain of the output voltage and another for the voltage gain(dB) of the output voltage are displayed.



We will now optimize the values of the component parameters in the circuit, such that the output waveform matches the waveform described in the reference file. For this design example, we will use `reference.txt` for specifying the reference waveform for `DB(V(`V_{out}`)` and `P(V(`V_{out}`)`.

Note: In a real life scenario, you will have to create a reference file containing the reference waveform, before you can use the curve fitting in Advanced Analysis Optimizer.

Opening Optimizer in Advanced Analysis

From the PSpice menu, choose Advanced Analysis Optimizer.

Selecting an engine

1Click on the drop-down list to the right of the Optimizer tool name.

2From the drop-down list, select the Modified LSQ engine.

Setting up component parameters

1In the Parameters window, add the parameters that you want to optimize to obtain the desired output.

Select the *Click here to import a parameter from the design property file* row.

2In the Parameter Selection dialog box, select `C1`,`C2`,`C3`,`C4`,`R1`,`R2`,`R3`, and `R4`, and click OK.

The selected components, their original values, and the min and max values that are calculated using the original values, appear in the Parameters window.

For example, in the circuit, value of R4 is 1.2K. Therefore, the value displayed in the Original column against R4 is 1200. The min value displayed is 120 (1200/10) and the max value displayed is 12000 (1200*10).

3In the Parameters tab, if you do not want the value of a particular parameter to change, you can do so by locking the parameter value. Lock the parameter values for R6 and R5.

4You can also ignore some of the parameter values.

Though we added the parameter R3, we will ignore it for this optimizer session. To do this, clear the check mark next to the message flag.

Setting up curve-fit specification

1Select the *Curve Fit* tab in the Optimizer window.

2In the *Curve Fit* tab, add specifications. Select the *Click here to enter a curve-fit specification* row.

3In the *New Trace Expression* dialog box, first select P() from the list of Analog operators and Functions, and then select V(out) from the list of Simulation Output Variables.

The Measurement text box should read *P(V(out))*.

4Click *OK* to save the new trace expression.

5In the Reference File text box, specify the location of `reference.txt`.

6Click the *Ref.Waveform* list box. From the drop-down list that appears, select *PHASE*.

Note: The entries in the drop-down list are the column headings in the reference file. If you open the reference file, `reference.txt,` you will see that *PHASE* is the heading of the second column and the third column has no heading. When the column headers are blank in the reference file, the reference waveform drop-down list displays entries such as, Column_2 and Column_3, instead of a name.

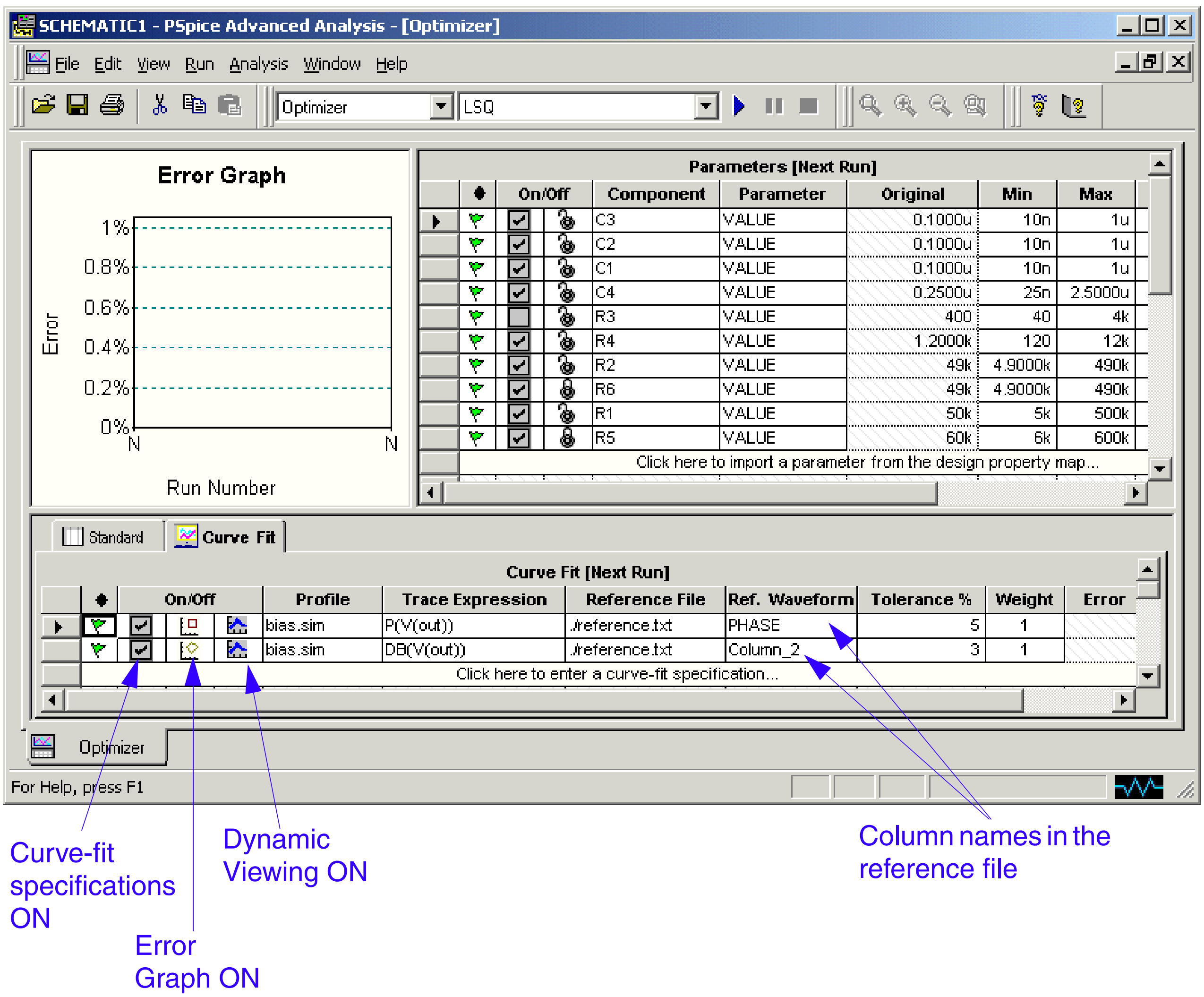
7Specify the tolerance and weight at 5 and 1, respectively.

This completes the process of creating a new curve-fit specification. In case you want to enable dynamic viewing of the output waveform, select the third field in the On/Off column.

8Similarly, add another specification. Specify the trace expression as *DB(V(out))*, reference file as `reference.txt`, reference waveform as *Column_2*, tolerance as 3, and weight as 1.

9Turn the dynamic viewing on.

The snapshot of the Optimizer, after you have modified the settings, is shown below:



10In case you want that the simulation data should be available to you even after the optimization session is complete, you need to modify the Optimizer settings. From Advanced Analysis the Edit menu, choose Profile settings.

Tip

It is recommended that you complete the steps for setting up component parameters and curve-fit specifications. In case you choose not to perform the steps, you can use the `SCHEMATIC1_complete.aap` file located at `..\tools\pspice\tutorial\capture\pspiceaa\bandpass\bandpass-PSpiceFiles\SCHEMATIC1`. To use the aap file provided with the design example, rename `SCHEMATIC1_complete.aap to SCHEMATIC1.aap`.

11Select the *Simulation* tab in the *Profile Settings* dialog box, and ensure that Optimizer data collection is set to *Save All Runs*.

12Run the Optimizer.

The PSpice UI comes up displayed the changes in the output waveform for each Optimizer run. The PSpice UI comes up only if you have turned the dynamic viewing on.

After the optimization is complete, you can view any of the Optimizer runs, provided you had selected the Save All Runs option in the Profile Settings dialog box.

Viewing an Optimizer run

1Select run 4 in the Error Graph section.

2Select the curve-fit specification for which you want to view the run. Select the first specification.

3Right-click and select View[Run #4] in PSpice.

The trace for the selected run opens in PSpice.

4You can create custom derate files at any location and then associate these with the Advanced Analysis discrete engine Select **PSpice > Advanced Analyses > Import Optimizable Parameters**.

5Select **PSpice > Advanced Analyses > Export Parameters to Optimizer**.

Creating curve fit specification

The curve fit tab contains following fields:

1Message Flag

A message flag can have three values: red, green, and yellow. The red flag indicates an error in the specification. A yellow flag indicates that the optimizer progress has stopped for some reason, and the green flag indicates that the optimization process is going fine.

2Curve Fit Specification On/Off

The check mark indicates that this specification will be included in the current Optimizer run. If this check box is clear, all other columns in the row are also ignored, indicating that the specification will not be considered for the next Optimizer runs.

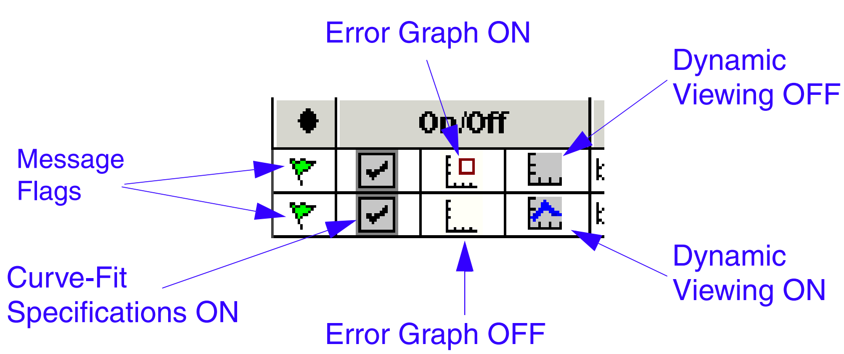
3Error Graph On/Off

Error Graph should be on whenever you want to view the the trace for the specification on the Error Graph. The error graph plots the error variation for trace expression with each Optimizer run.

For example, consider a case where the circuit has multiple specifications and you want to see the error graph only for a particular specification. To do this, disable the trace for other specifications by setting the Error Graph of all other specification to Off.

4Dynamic Viewing On/Off

Dynamic Viewing should be on whenever you want to launch the probe to dynamically display the status of the curve to be optimized with respect to the reference waveform. Thus both the reference waveform and the curve to be optimized are displayed in the same view.



5Profile name

This drop-down list box lists all the profiles available in the design.

You can also specify the simulation profile in the New Trace expression dialog box.

6Trace Expression

This column lists the trace expression to be optimized. You can create new trace expression, by selecting the row stating *Click here to enter a new curve-fit specification*.

7Reference File

This text box is used to specify the file that contains the reference waveform in a tabular format (in terms of X-Y value pairs). You can use the browse button to select the file.

8Reference Waveform

This drop-down list box lists all the waveforms in the reference file. The optimizer tries to fit the model parameters to the waveform specified in the reference waveform list box.

The reference waveforms are listed as columns in the reference files. These columns are referenced by column names specified in the header row. If there's no header row, column ordinal number from left-to-right is used. For example, col_1, col_2, and so on.

9Tolerance

The value displayed in this column influences the acceptance criteria for a specification. This column displays the relative tolerance value specified by the user. Relative tolerance is the allowable ratio of the RMS of the difference of the two traces to the RMS of the reference trace.

10Weight

This field is valid only for the MLSQ Optimizer engine. This field is useful in cases there are more than one curve-fit specifications. Weight values are used for prioritizing the curve-fit specifications. The higher weight values have higher priorities.

For example, a specification with weight 10 has a higher priority over a specification with weight 4.

11Error value

This field is not editable and indicates the excess of E_{rms} (RMS Error) above the specified relative tolerance. This value will be zero if the E_{rms} is less than tolerance.

For example, if after any run the E_{rms} is 7% and the specified tolerance is 3%, then the value displayed in the Error column will be 4%, which is E_{rms} (7%) - tolerance(3%).

Optimizer Engine Overview

Optimizer includes three engines:

- Modified LSQ engine

  The Modified LSQ engine uses both constrained and unconstrained minimization algorithms, which allow it to optimize goals subject to nonlinear constraints.

When using the Modified LSQ engine, you can set your measurement specifications as goals or constraints.The engine strives to get as close to the goals as possible while ensuring that the constraints are met.

- Random engine

  The Random engine randomly picks values within the specified range and displays misfit error and parameter history.

- Discrete engine

  The Discrete engine is used at the end of the optimization cycle to round off component values to the closest values available commercially. Typically, once you have optimized your circuit, you will most likely want to convert your component values into “real-world” parts.

  For example, the Optimizer determines that the 3K resistor in the RF amplifier circuit should be 2.18113K, but you cannot use this value in your manufactured design. You can then specify a discrete table and switch to the Discrete Engine. The Discrete engine determines a new value for this resistor depending on the table used. For a one percent table, the new value is 2.21K.

  The Optimizer in Advanced Analysis provides discrete value defaults for resistors, capacitors, and inductors.

The Smoke Analysis Tool

Smoke analysis is available with the following products:

- PSpice[1] Smoke Option
- PSpice Advanced Analysis

Long-term circuit reliability

Smoke warns of component stress due to power dissipation, increase in junction temperature, secondary breakdowns, or violations of voltage / current limits. Over time, these stressed components could cause circuit failure.

Smoke uses Maximum Operating Conditions (MOCs), supplied by vendors and derating factors supplied by designers to calculate the Safe Operating Limits (SOLs) of a component's parameters.

Smoke then compares circuit simulation results to the component's safe operating limits. If the circuit simulation exceeds the safe operating limits, Smoke identifies the problem parameters.

Use Smoke for Displaying Average, RMS, or Peak values from simulation results and comparing these values against corresponding safe operating limits

Safe operating limits

Smoke will help you determine:

- Breakdown voltage across device terminals
- Maximum current limits
- Power dissipation for each component
- Secondary breakdown limits
- Junction temperatures

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

Smoke strategy

Smoke is useful as a final design check after running Sensitivity, Optimizer, and Monte Carlo, or you can use it on its own for a quick power check on a new circuit.

Plan ahead

Smoke requires:

- Components that are Advanced Analysis-ready
- A working circuit schematic and transient simulation
- Derating factors

  Smoke uses “no derating” as the default.

Note: See the online *Advanced Analysis library list* and the *PSpice library list* for components containing smoke parameter data.

Setting up in the schematic editor

Advanced Analysis requires:

- A circuit schematic and working PSpice simulation
- Measurements set up in PSpice
- Performance goals for evaluating measurements
- Performance goals

Smoke analysis also requires:

- Any components included in a Smoke analysis must have smoke parameters specified.
- Time Domain (transient) analysis as a simulation

  Smoke does not work on other types of analyses, such as DC Sweep or AC Sweep/Noise analyses.

To set up smoke analysis:

1 From your schematic editor, open your circuit.

2 Run a PSpice simulation.

3 Check your key waveforms in PSpice and make sure they are what you expect.

Note: For information on circuit layout and simulation setup, see your schematic editor and PSpice user guides.

You can see the following for more information:

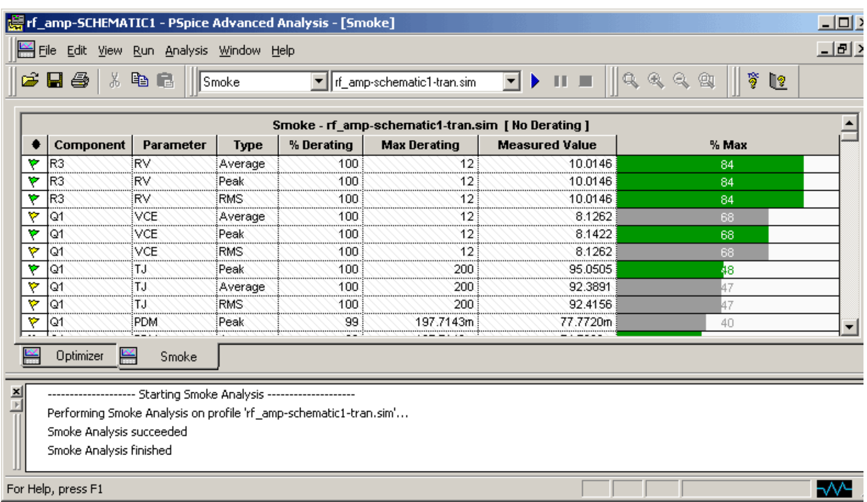
Components and tolerances	Preparing your design for Advanced Analysis
Creating measurement expressions	Composing measurement expressions
Checking measurement expressions in PSpice	Viewing results of measurements

Running Smoke

Starting a run in Smoke

– From the **PSpice** menu in your schematic editor, select **Advanced Analysis** / **Smoke**.

The Smoke tool opens and automatically runs on the active transient profile.

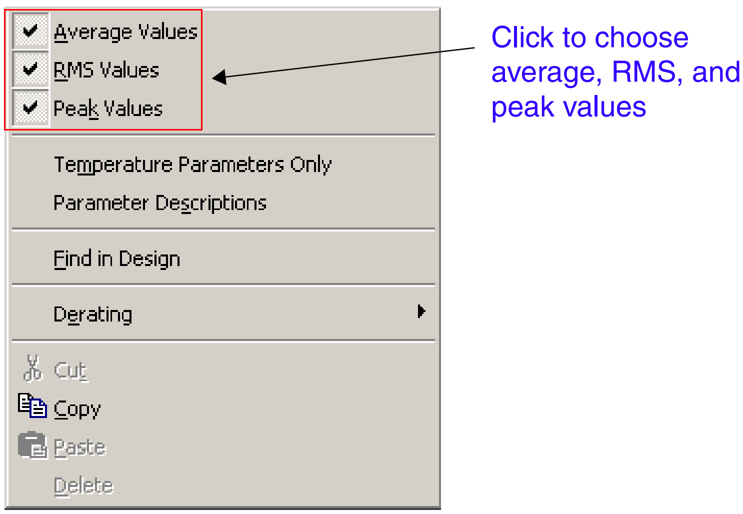


Smoke calculates safe operating limits using component parameter maximum operating conditions and derating factors.

The output window displays status messages.

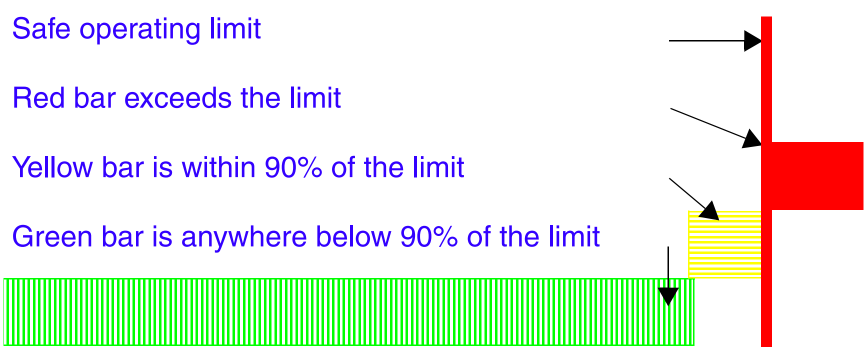
Viewing Smoke results

1 Right click and from the pop-up menu select **Average**, **RMS**, and **Peak Values**.



In the %Max column, check the bar graphs.

- Red bars show values that exceed safe operating limits.
- Yellow bars show values getting close to the safe operating limits: between 90 and 100 percent of the safe operating limits.
- Green bars show values well within the safe operating limits: less than 90 percent of the safe operating limits.
- Grey bars indicate that limits are not valid for the parameters.



The value in the % Max column is calculated using the following formula:

(12-1) %Max=Actual operating Value/Safe operating limit *100

Where:

Actual operating value	■ is displayed in the *Measured Value* column. ■ is calculated by the simulation controller.
Safe operating limit	■ is displayed in the *Max Derating* column. ■ is `MOC*derating_factor`. ■ MOC or the Maximum Operating Condition is specified is the vendor supplied data sheet ■ derating factor, is specified by the users in the *% Derating* column.

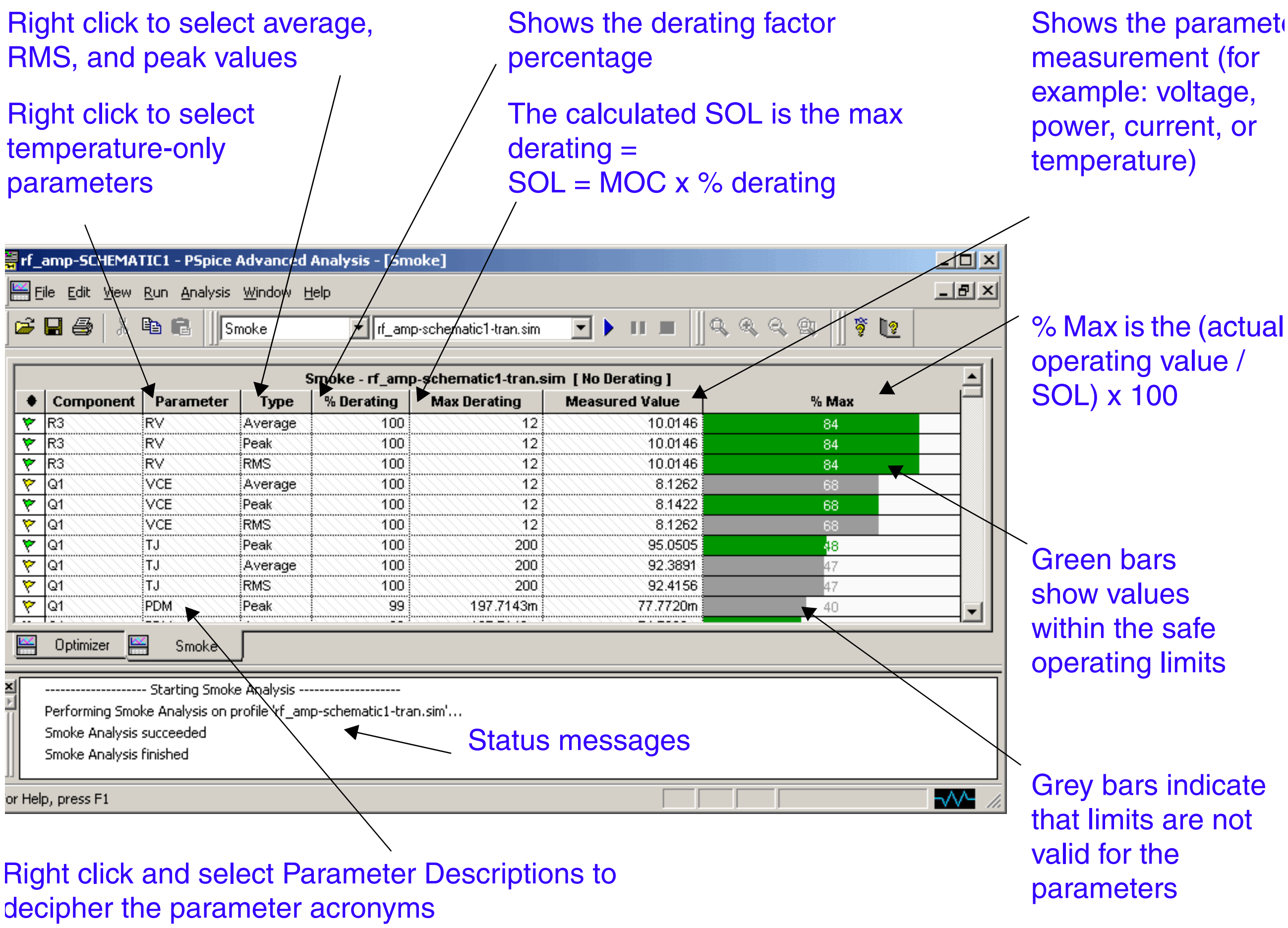
The value calculated using the Equation 12-1 on page 101 is rounded off to the nearest integer, larger than the calculated value, and then displayed in the %Max column.

For example, if the calculated value of %Max is 57.06, the value displayed in the %Max column will be 58.

2 Right click on the table and select **Temperature Parameters Only** from the pop-up menu.

Only maximum resistor or capacitor temperature (TB) and maximum junction temperature (TJ) parameters are displayed. When reviewing these results, only average and peak values are meaningful.

In this example, none of the parameters are stressed, as indicated by the green bars.



– Click  .

Or:

From the **File** menu, select **Print**.

Configuring Smoke Analysis

Changing components or parameters

Smoke results are read-only. To modify the circuit:

1 Make your changes in your schematic editor.

2 Rerun the PSpice simulation.

Follow the steps in Setting up in the schematic editor and Running Smoke.

Controlling smoke on individual design components

You can use the SMOKE_ON_OFF property to control whether or not you want to run smoke analysis on individual devices or blocks in a schematic.

If you attach the SMOKE_ON_OFF property to the device instance for which you do not want to perform the smoke analysis, and set the value to OFF, the smoke analysis would not run for this device.

This property can also be used on hierarchical blocks. The value of the SMOKE_ON_OFF property attached to the parent block has a higher priority over the property value attached to the individual components.

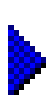
Selecting other deratings

To select other deratings:

1 Right click and from the pop-up menu select **Derating**.

2 Select one of the three derating options on the pull-right menu:

- No Derating
- Standard Derating
- Custom Derating Files

3 Click  on the top toolbar to run a new Smoke analysis with the revised derating factors.

New results appear.

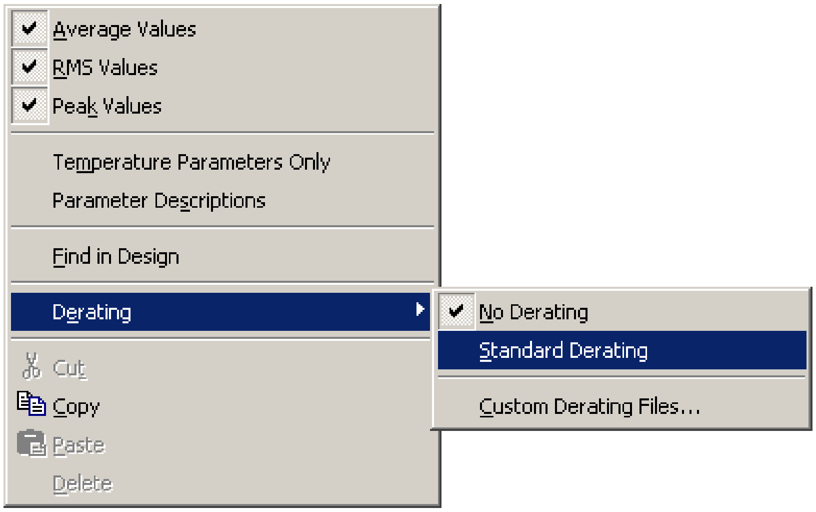
The default derating option uses 100% derating factors, also called No Derating.

In the following example, you will run the circuit with standard derating and examine the results.

Selecting standard derating

1 Right click and from the pop-up menu select **Derating**.

2 Select **Standard Derating** from the pull-right menu.



3 Click on the top toolbar to run a new Smoke analysis.

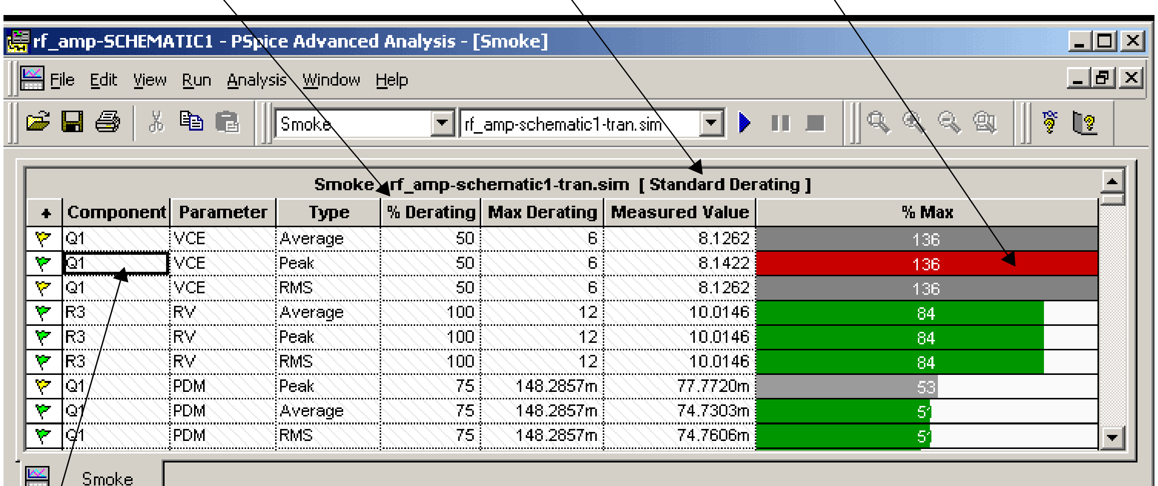
New results appear.

The red bar indicates that Q1's VCE parameter is stressed.

Standard derating factors used in the calculations

Standard Derating appears in the title

Component Q1's VCE parameter is stressed to 136 percent of its safe operating limit



Right click on Q1 and from the pop-up menu select Find in Design. This takes you to the schematic where the component parameter can be changed.

4 Resolve the component stress:

- Right click on Q1 VCE and from the pop-up menu select **Find in Design** to go to the schematic and adjust Q1's VCE value.

Or:

- Right click and from the pop-up menu select **Deratings** \ **No Derating** to change the derating option back to **No Derating**.

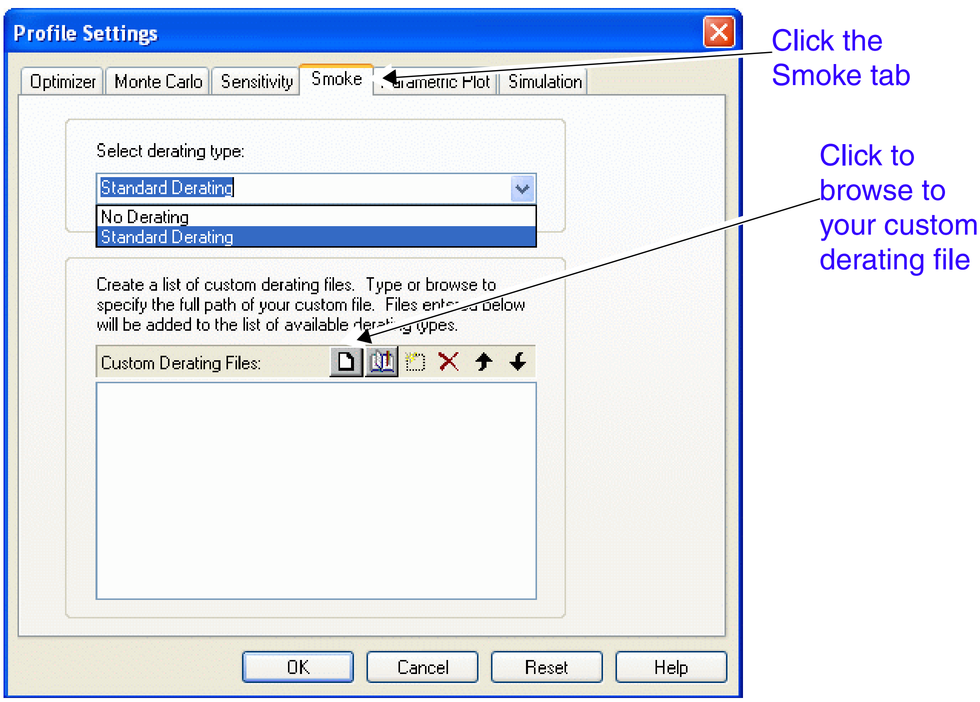
5 Click  on the top toolbar to rerun Smoke analysis after making any adjustments.

6 Check the results.

Selecting custom derating

1 If you have your own custom derating factors, you can browse to your own file and select it for use in Smoke. For information on creating a custom derating file, see our technical note posted on our web site at Once you have your custom derating file in place, right click and from the pop-up menu select **Derating**.

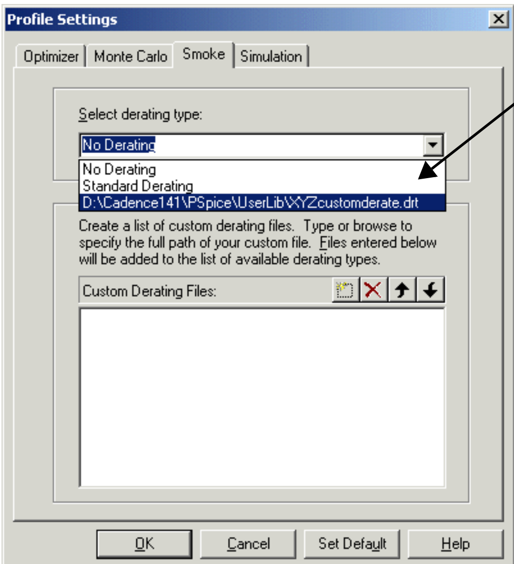
2 Select **Custom Derating Files** from the pull-right menu.



3 Click the browse icon.

4 Browse and select your file.

The file name is added to the list in the Custom Derating Files text box and the drop-down list.



Select the custom derating file in the drop-down list after finding the file using the browse text box below.

5 Select the custom derating file from the drop-down list.

6 Click **OK**.

7 Click  on the top toolbar to run a new Smoke analysis.

New results appear.

8 Check the results.

To make changes, follow the steps for changing derating options or schematic component values.

Smoke parameter names

The following tables summarize smoke parameter names you will see in the Smoke results. The tables are sorted by user interface parameter names and include:

- Passive component parameters
- Semiconductor component parameters
- OpAmp component parameters

For passive components, three names are used in Smoke analysis: symbol property names, symbol parameter names, and parameter names used in the Smoke user interface. This table is sorted in alphabetical order by parameter names that display in the Smoke user interface.

Smoke User Interface Parameter Name	Passive Component	Maximum Operating Condition	Symbol Property Name	Symbol Smoke Parameter Name	Variable Table Default Value
CI	Capacitor	Maximum ripple	CURRENT	CIMAX	1 A
CV	Capacitor	Voltage rating	VOLTAGE	CMAX	50 V
IV	Current Supply	Max. voltage current source can withstand	VOLTAGE	VMAX	12 V
LI	Inductor	Current rating	CURRENT	LMAX	5 A
LIDC	Inductor	DC current value	CURRENT	DC_CURRENT	
LV	Inductor	Dielectric strength	DIELECTRIC	DSMAX	300 V
PDM	Resistor	Maximum power dissipation of resistor	POWER	RMAX	0.25 W
RBA* (=1/SLOPE)	Resistor	Slope of power dissipation vs. temperature	SLOPE	RSMAX	0.005W/degC
RV	Resistor	Voltage Rating	VOLTAGE	RVMAX	--
SLP*	Capacitor	Temperature derating slope	SLOPE of volt temperature curve	CSMAX	0.005 V/degC
TBRK*	Capacitor	Breakpoint temperature	KNEE	CBMAX	125 degC
TMAX*	Capacitor	Maximum temperature	MAX_TEMP	CTMAX	125 degC

Smoke User Interface Parameter Name	Passive Component	Maximum Operating Condition	Symbol Property Name	Symbol Smoke Parameter Name	Variable Table Default Value
TMAX, TB	Resistor	Maximum temperature resistor can withstand	MAX_TEMP	RTMAX	200 degC
VI	Voltage Supply	Max. current voltage source can withstand	CURRENT	IMAX	1 A

* Internal parameters not shown in user interface

The following table lists smoke parameter names for semiconductor components. The table is sorted in alphabetical order according to parameter names that will display in the Smoke results.

Smoke Parameter Name and Symbol Property Name	Semiconductor Component	Maximum Operating Condition
IB	BJT	Maximum base current (A)
IC	BJT	Maximum collector current (A)
PDM	BJT	Maximum power dissipation (W)
RCA	BJT	Thermal resistance, Case-to-Ambient (degC/W)
RJC	BJT	Thermal resistance, Junction-to-Case (degC/W)
SBINT	BJT	Secondary breakdown intercept (A)
SBMIN	BJT	Derated percent at TJ (secondary breakdown)
SBSLP	BJT	Secondary breakdown slope
SBTSLP	BJT	Temperature derating slope (secondary breakdown)
TJ	BJT	Maximum junction temperature (degC)
VCB	BJT	Maximum collector-base voltage (V)

Smoke Parameter Name and Symbol Property Name	Semiconductor Component	Maximum Operating Condition
VCE	BJT	Maximum collector-emitter voltage (V)
VEB	BJT	Maximum emitter-base voltage (V)
IF	Diode	Maximum forward current (A)
PDM	Diode	Maximum power dissipation (W)
RCA	Diode	Thermal resistance, Case-to-Ambient (degC/W)
RJC	Diode	Thermal resistance, Junction-to-Case (degC/W)
TJ	Diode	Maximum junction temperature (degC)
VR	Diode	Maximum reverse voltage (V)
IC	IGBT	Maximum collector current (A)
IG	IGBT	Maximum gate current (A)
PDM	IGBT	Maximum Power dissipation (W)
RCA	IGBT	Thermal resistance, Case-to-Ambient (degC/W)
RJC	IGBT	Thermal resistance, Junction-to-Case (degC/W)
TJ	IGBT	Maximum junction temperature (degC)
VCE	IGBT	Maximum collector-emitter (V)
VCG	IGBT	Maximum collector-gate voltage (V)
VGEF	IGBT	Maximum forward gate-emitter voltage (V)
VGER	IGBT	Maximum reverse gate-emitter (V)
ID	JFET or MESFET	Maximum drain current (A)
IG	JFET or MESFET	Maximum forward gate current (A)
PDM	JFET or MESFET	Maximum power dissipation (W)
RCA	JFET or MESFET	Thermal resistance, Case-to-Ambient (degC/W)

Smoke Parameter Name and Symbol Property Name	Semiconductor Component	Maximum Operating Condition
RJC	JFET or MESFET	Thermal resistance, Junction-to-Case (degC/W)
TJ	JFET or MESFET	Maximum junction temperature (degC)
VDG	JFET or MESFET	Maximum drain-gate voltage (V)
VDS	JFET or MESFET	Maximum drain-source voltage (V)
VGS	JFET or MESFET	Maximum gate-source voltage (V)
ID	MOSFET or Power MOSFET	Maximum drain current (A)
IG	MOSFET or Power MOSFET	Maximum forward gate current (A)
PDM	MOSFET or Power MOSFET	Maximum power dissipation (W)
RCA	MOSFET or Power MOSFET	Thermal resistance, Case-to-Ambient (degC/W)
RJC	MOSFET or Power MOSFET	Thermal resistance, Junction-to-Case (degC/W)
TJ	MOSFET or Power MOSFET	Maximum junction temperature (degC)
VDG	MOSFET or Power MOSFET	Maximum drain-gate voltage (V)
VDS	MOSFET or Power MOSFET	Maximum drain-source voltage (V)
VGSF	MOSFET or Power MOSFET	Maximum forward gate-source voltage (V)
VGSR	MOSFET or Power MOSFET	Maximum reverse gate-source voltage (V)
ITM	Varistor	Peak current (A)
RCA	Varistor	Thermal resistance, Case-to-Ambient (degC/W)

Smoke Parameter Name and Symbol Property Name	Semiconductor Component	Maximum Operating Condition
RJC	Varistor	Thermal resistance, Junction-to-Case (degC/W)
TJ	Varistor	Maximum junction temperature (degC)
IFS	Zener Diode	Maximum forward current (A)
IRMX	Zener Diode	Maximum reverse current (A)
PDM	Zener Diode	Maximum power dissipation (W)
RCA	Zener Diode	Thermal resistance, Case-to-Ambient (degC/W)
RJC	Zener Diode	Thermal resistance, Junction-to-Case (degC/W)
TJ	Zener Diode	Maximum junction temperature (degC)

The following table lists smoke parameter names for Op Amp components. The table is sorted in alphabetical order according to parameter names that will display in the Smoke results.

Smoke Parameter Name	Op Amp Component	Maximum Operating Condition
IPLUS	OpAmp	Non-inverting input current
IMINUS	OpAmp	Inverting input current
IOUT	OpAmp	Output current
VDIFF	OpAmp	Differential input voltage
VSMAX	OpAmp	Supply voltage
VSMIN	OpAmp	Minimum supply voltage
VPMAX	OpAmp	Maximum input voltage (non-inverting)
VPMIN	OpAmp	Minimum input voltage (non-inverting)
VMMAX	OpAmp	Maximum input voltage (inverting)
VMMIN	OpAmp	Minimum input voltage (inverting)

Adding Custom Derate file

Why use derating factors?

If you want a margin of safety in your design, apply a derating factor to your maximum operating conditions (MOCs). If a manufacturer lists 5W as the maximum operating condition for a resistor, you can insert a margin of safety in your design if you lower that value to 4.5W and run your simulation with 4.5W as the safe operating limit (SOL).

`As an equation: MOC x derating factor = SOL.`

In the example `5W x 0.9 = 4.5W`, the derating factor is 0.9. Also, 4.5W is 90% of 5W, so the derating factor is 90%. A derating factor can be expressed as a percent or a decimal fraction, depending on how it's used in calculations.

What is a custom derate file?

A custom derating file is an ASCII text file with a `.drt` extension that contains smoke parameters and derating factors specific to your project. If the "no derating" and "standard derating" factors provided with Advanced Analysis do not have the values you need for your project, you can create a custom derating file and type in the specific derating factors that meet your design specifications.

Figure 2 shows a portion of a custom derating file. The file lists resistor smoke parameters and derating factors. In your custom derating file, enter the derating factors as decimal percents in double quotes.

For the example below, if the resistor had a power dissipation (PDM) maximum operating condition of 5W, the .9 derating factor tells Advanced Analysis to use 0.9 x 5 = 4.5W as this resistor's safe operating limit.

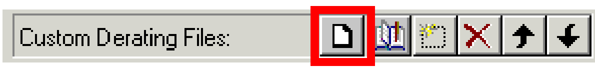
```
("RES"
("PDM" "1")
("TMAX" "1")
("TB" "1")
)
```

Figure 12-1 Resistor smoke parameters and derating factors in a portion of a custom derating file

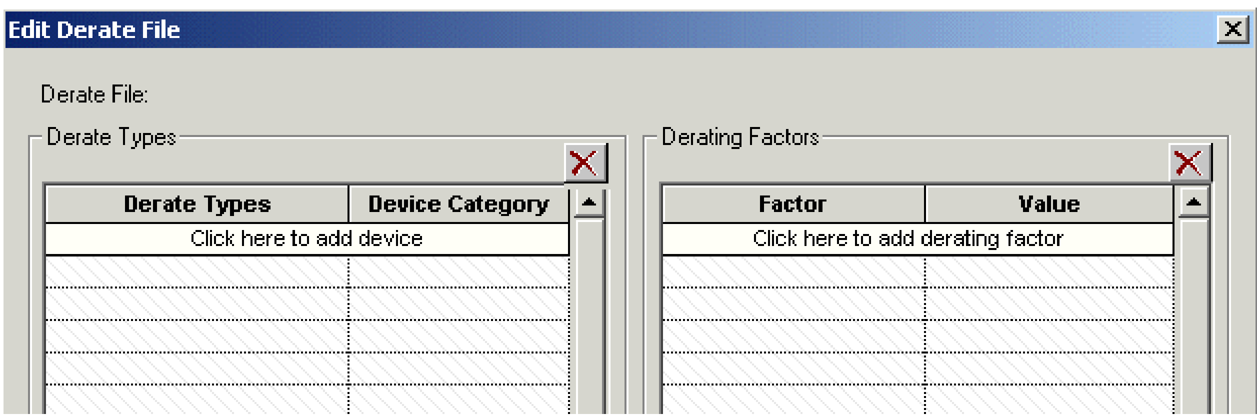
Creating a new custom derate file

Advanced Analysisprovides you the capability to create and edit derate files. You can perform this operation by using the Edit Derate File dialog box. To open the Edit Derate File dialog box, click the Create Derate File button in the Profile Settings dialog box.

1 To create a new derate file from scratch, click the *Create Derate File* button in the Profile Settings dialog box.



The Edit Derate File dialog box appears.

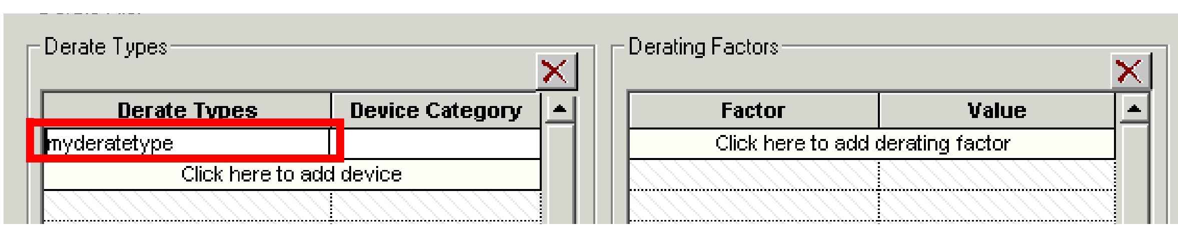


In the Edit Derate Type dialog box, type the derate type and select the device category. The derate type can be any user defined value.

2 To add a new derate type, click the *Click here to add a device* row.

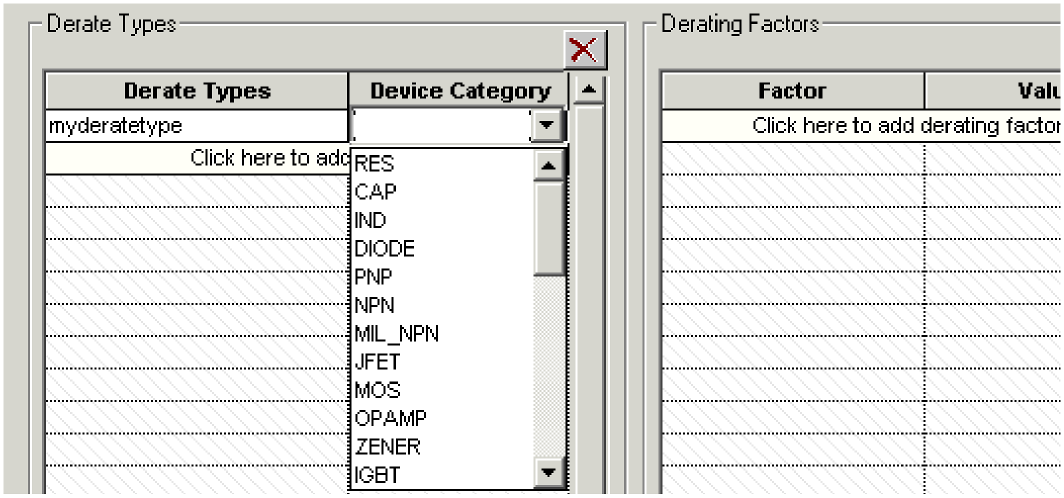
A blank row gets added in the Derate Types pane.

3 In the Derate Types text box, enter the name such as `myderatetype`



4 Click the Device Category grid.

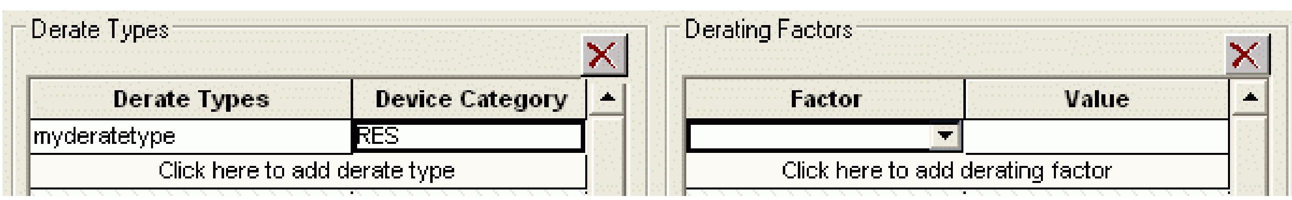
5 From the drop-down list box select derate category, for example `RES`.



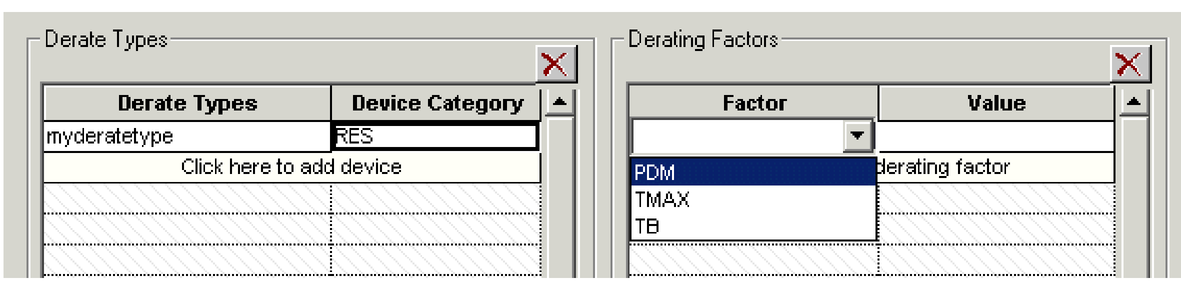
`myderatetype` is the derate type for a resistor of type 'RES'.

6 To specify the derate values for various resistor parameters, click the *Click here to add derating factor* row in the Derating Factors window.

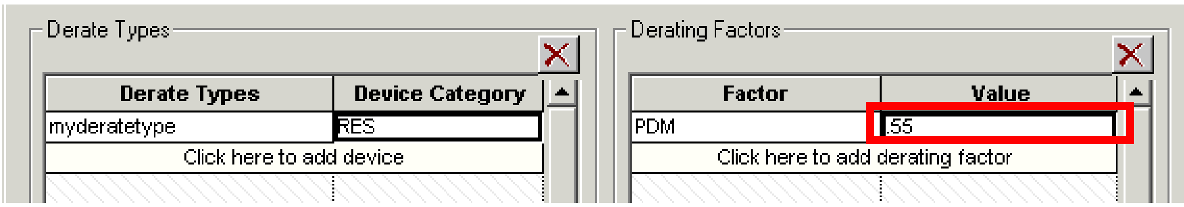
A blank row gets added.



7 Select the derate factor from the Factor drop down list.



The corresponding default value for the derate factor is automatically filled in.



8 Modify the value of the derate factor as per the requirement.

9 Similarly, specify additional derate types and their corresponding categories, factors, and values.

Note: Derate factors are populated based on the selected device category

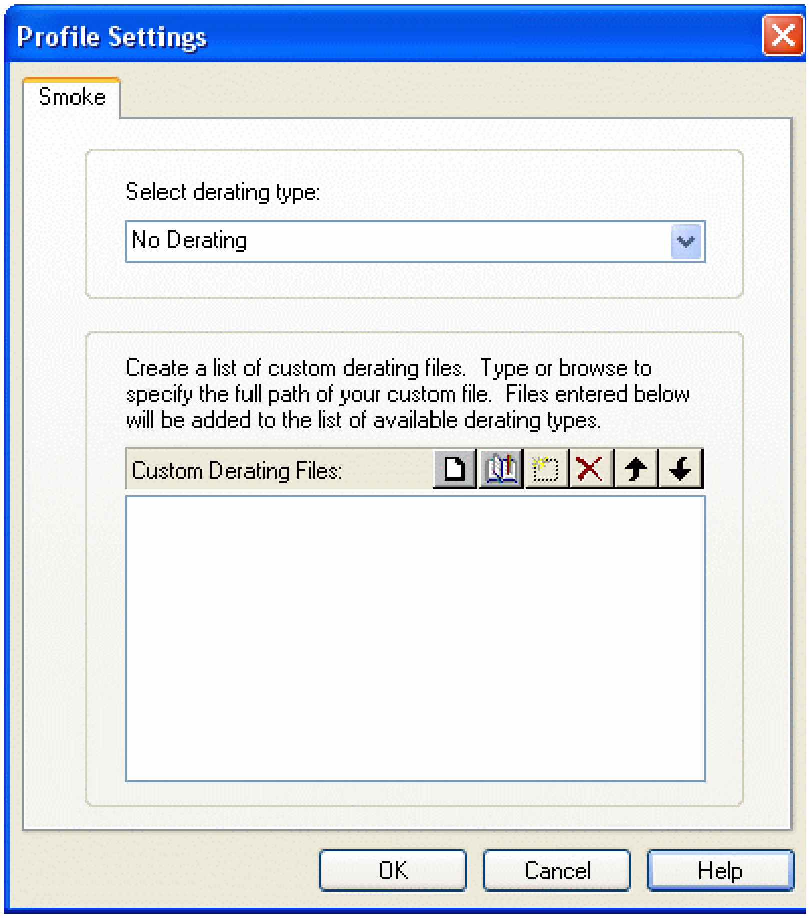
10 Save the derate file.

Note: To use the custom derate file, in the Property Editor, add a new property for the component with the name `DERATE_TYPE` and value same as the Derarte Type specified, such as *myderatetype*. Select the corresponding derate file and run smoke.

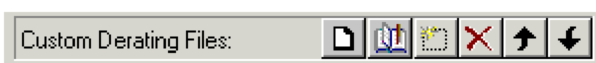
Modifying existing derate file

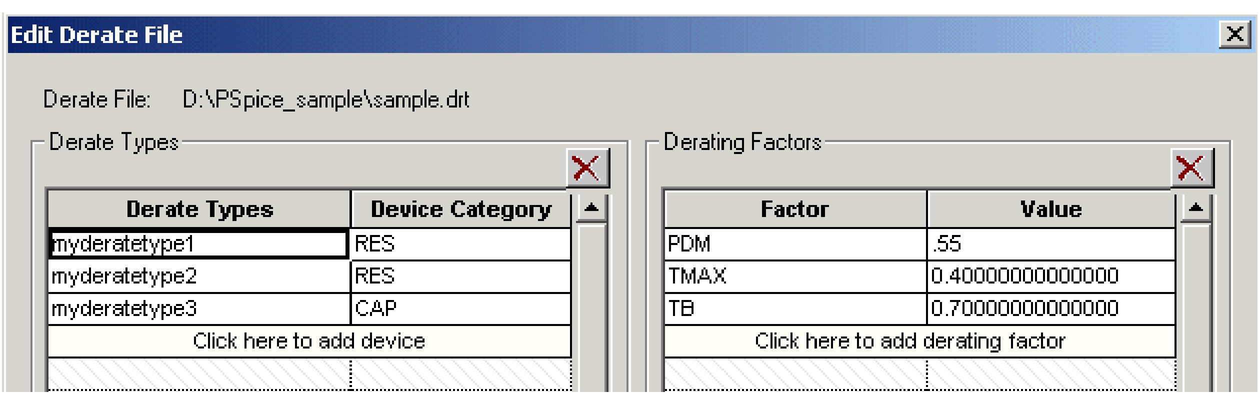
You can also use this dialog box to modify the device type, device category, and the associated derating factor in an existing derate file.

1 Type the full path or browse to select an exisiting derate file.



2 Click the Edit Derate File button to display the Edit Derate File dialog box.



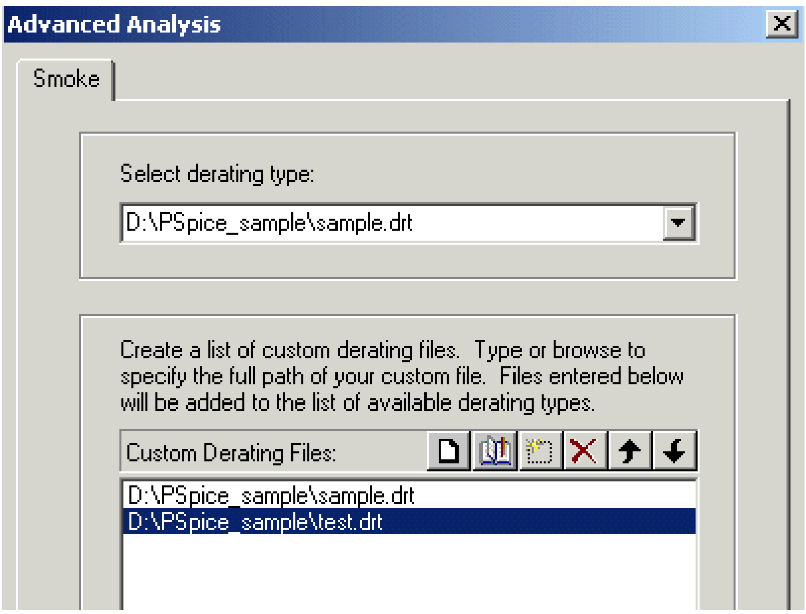


Adding the custom derating file to your design

To choose your custom derating file and apply the custom derating factors:

1. Right click on the Smoke display.
2. From the pop-up menu, select Derating > Custom Derating Files.

   The Advanced Analysis Smoke tab dialog appears.



3. To add one or more files to the Custom Derating Files list box, click the New(Insert) button.
4. Browse and select the custom derating file.

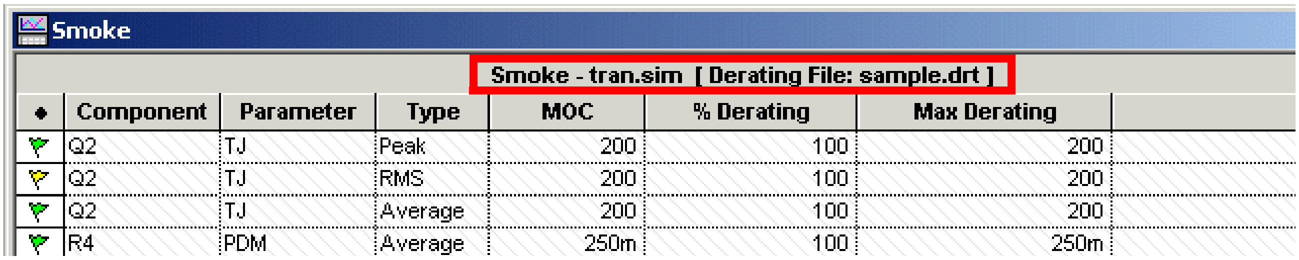
   The custom derating filename gets added in the Custom Derating Files list box.
5. In the Select derating type drop-down list, select the name of the derate file that you want to use during the smoke analysis.
6. Click OK.
7. Click the Run button (blue triangle).

   The Smoke data display title changes to "`Smoke - <profile name> [custom derate file name]`."

   Smoke results appear after the analysis in complete. The value of derate factors specified by you appear in the %Derating column.

Note: If the active derate file is different from the derate file used for the smoke results displayed, an asterix (*) symbol will be displayed along with the derate file name.

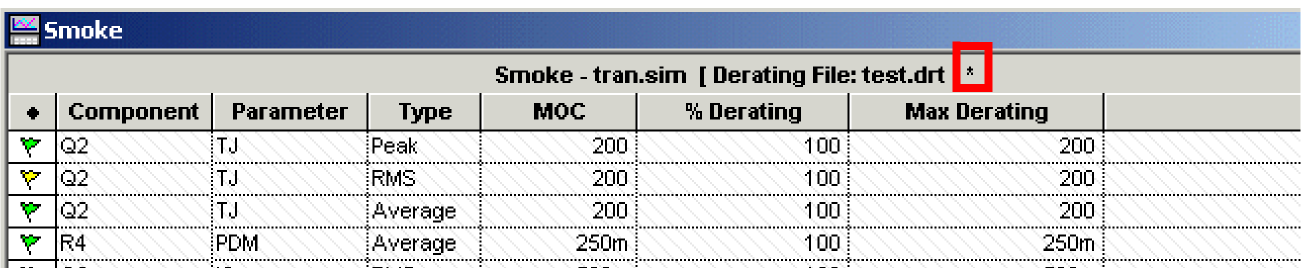
Consider an example where sample.drt was used to acheive the displayed smoke results.


Smoke

Smoke - tran.sim [Derating File: sample.drt]

◆	Component	Parameter	Type	MOC	% Derating	Max Derating
	Q2	TJ	Peak	200	100	200
	Q2	TJ	RMS	200	100	200
	Q2	TJ	Average	200	100	200
	R4	PDM	Average	250m	100	250m

In this case, if you change the active derate file to test.drt or if you edit the existing sample.drt, an asterix (*) symbol will be displayed along with the derate file name.


Smoke

Smoke - tran.sim [Derating File: test.drt *]

◆	Component	Parameter	Type	MOC	% Derating	Max Derating
	Q2	TJ	Peak	200	100	200
	Q2	TJ	RMS	200	100	200
	Q2	TJ	Average	200	100	200
	R4	PDM	Average	250m	100	250m

Caution

When you select a new derate file to be used for the smoke analysis, the contents of the %Derating column are updated with the new values only when you rerun the smoke analysis. Till you run the smoke analysis again, the values displayed in the %Derating column will be from the derate file used in the previous run.

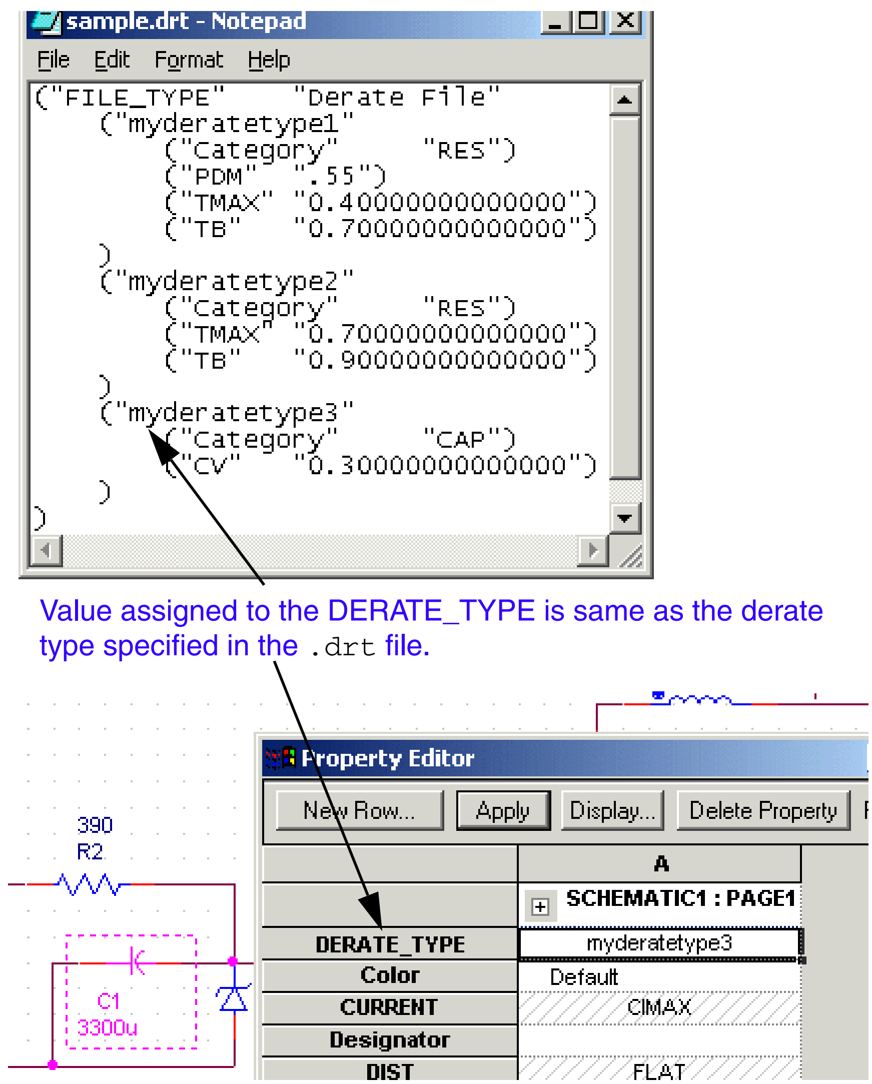
Reading values from the derate file

To be able to use the custom derate file, add the DERATE_TYPE property on the design instance. The value assigned to the DERATE_TYPE property should match the Derate Type specified by you in the derate file.

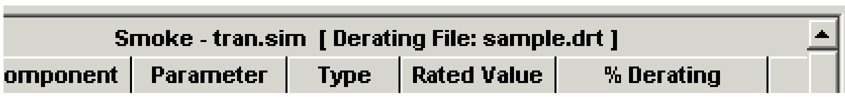
Consider a sample derate file, `sample.drt`. This derate file has two derate types for RES category, and one for capacitor. To use this derate file during the smoke analysis, load this file in Advanced Analysis. See Adding the custom derating file to your design on page 117.

Before you can use the derate file successfully, you need to complete the following steps in the schematic editor.

1. Select the capacitor C1 and right-click.
2. From the pop-up menu, in Capture select Edit properties. In Design Entry HDL, select Attributes.
3. In the Property Editor window, click the New Row button. In Design Entry HDL, click Add in the Attributes dialog box.
4. In the Add New Row dialog box, specify the name of the new property as DERATE_TYPE.
5. Specify the property value as `myderatetype3`, which is same as the derate type specified by you in the `sample.drt` file, and click OK.



6. Regenerate the PSpice netlist. From the **PSpice** drop-down menu select **Create Netlist**.
7. Run the smoke analysis. From the **PSpice** drop-down menu, select **Advanced Analysis** and then choose **Smoke**.
8. In Advanced Analysis, ensure that the `sample .drt` file is loaded and active. Then run the smoke analysis.



To know more about loading a customized derate file to your design, see Adding the custom derating file to your design on page 117

The Monte Carlo Tool

Monte Carlo analysis is available with the following products:

- PSpice[1] Advanced Optimizer Option
- PSpice Advanced Analysis

Monte Carlo predicts the behavior of a circuit statistically when part values are varied within their tolerance range. Monte Carlo also calculates yield, which can be used for mass manufacturing predictions.

Use Monte Carlo for:

- Calculating yield based on your specs
- Integrating measurements with graphical displays
- Displaying results in a probability distribution function (PDF) graph
- Displaying results in a cumulative distribution function (CDF) graph
- Calculating statistical data
- Displaying measurement values for every Monte Carlo run

Monte Carlo strategy

Monte Carlo requires:

- Circuit components that are Advanced Analysis-ready
- A circuit schematic and working PSpice simulation
- Measurements set up in PSpice

Plan Ahead

Setting options

- Start with enough runs to provide statistically meaningful results.

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

- Specify a larger number of runs for a more accurate graph of performance distribution. This more closely simulates the effects of mass production.
- Start with a different random seed value if you want different results.
- Set the graph bin number to show the level of detail you want. Higher bin numbers show more detail, but need more runs to be useful.
- If you are planning an analysis of thousands of runs on a complex circuit, you can turn off the simulation data storage option to conserve disk space. However, at this setting, the simulation will run slower.
  To turn off data storage:
  - From the Advance Analysis menu select
    Edit / Profile Settings/ Simulation
  - From the Monte Carlo field, select **Save None**.

    The simulation data will be overwritten by each new run. Only the last run's data will be saved.

Importing measurements

- Find the most sensitive measurements in Sensitivity and perform Monte Carlo analysis on those measurements only. Limiting Monte Carlo to only important measurements saves run time.

You can see the following for more information:

Parameterized components	Preparing your design for Advanced Analysis
Creating measurement expressions	Composing measurement expressions

Setting up Monte Carlo in your schematic editor

Starting out:

- Have a working circuit in schematic editor[1].

1. Schematic editor in this manual refers to either OrCAD Capture or Design Entry HDL.

The simulations can be Time Domain (transient), DC Sweep, and AC Sweep/Noise analyses.

- The circuit components you want to include in the data need to be Advanced Analysis-ready, with the tolerances of the circuit components specified.

To set up Monte Carlo:

1 From your schematic editor, open your circuit.

2 Run a PSpice simulation.

Note: Advanced Analysis Monte Carlo does not use PSpice Monte Carlo settings.

Note: You can run Advanced Analysis Monte Carlo on more than one simulation profile at once. However, if you have a multi-run analysis set up in PSpice (for example, a parametric sweep or a temperature sweep), Advanced Analysis Monte Carlo will reduce the simulation profile to one run before starting the Advanced Analysis Monte Carlo calculations. For temperature sweeps, the first temperature value in the list will be used for the Advanced Analysis Monte Carlo calculations.

3 Check your key waveforms in PSpice and make sure they are what you expect.

4 Test your measurements and make sure they have the results you expect.

For information on circuit layout and simulation setup, see your schematic editor and PSpice user guides.

You can see the following for more information:

Components and tolerances	Preparing your design for Advanced Analysis
Creating measurement expressions	Composing measurement expressions
Checking measurement expressions in PSpice	Viewing results of measurements

Setting up Monte Carlo in Advanced Analysis

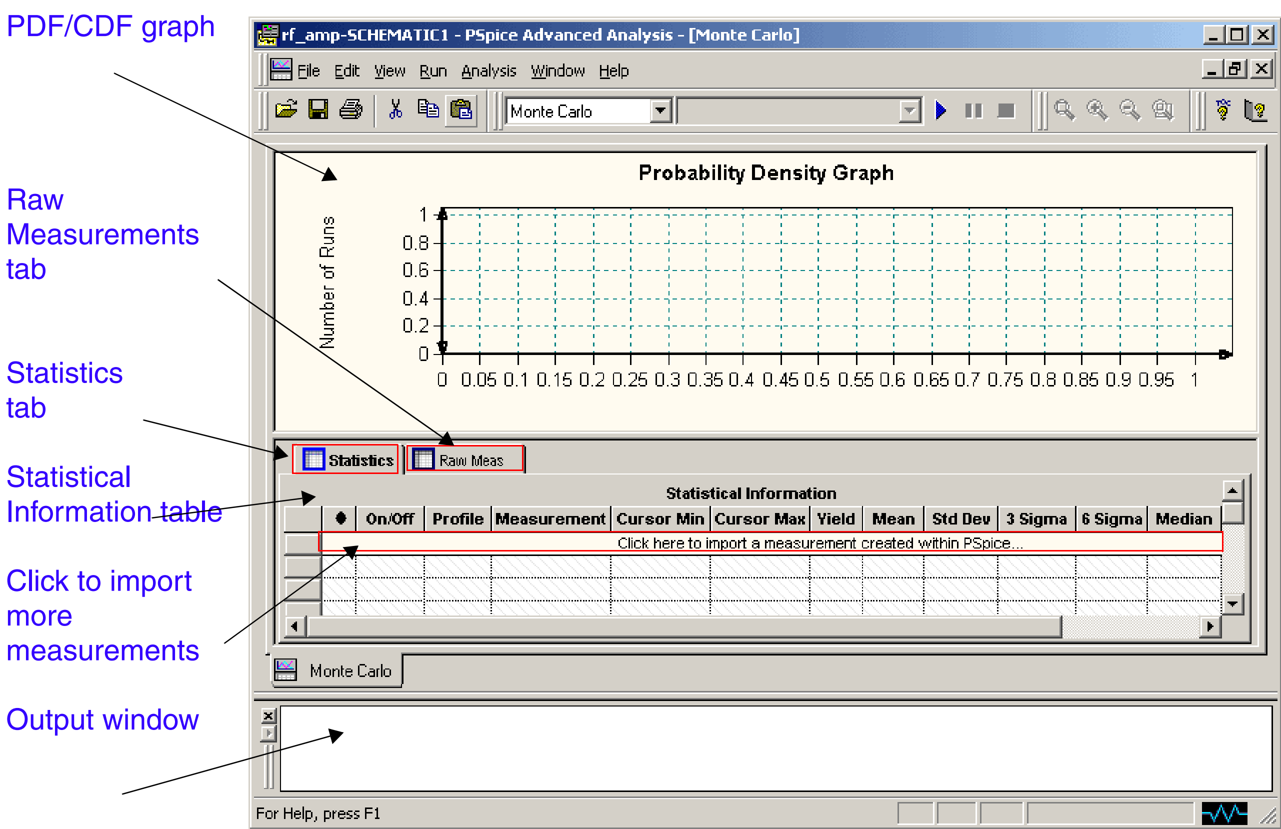
To set Monte Carlo in Advanced Analysis:

1. Open Monte Carlo
2. Import measurements into Monte Carlo
3. Set Monte Carlo options

Opening Monte Carlo

– From the schematic editor **PSpice** menu, select **Advanced Analysis** / **Monte Carlo**.

The Advanced Analysis Monte Carlo tool opens.



Importing measurements into Monte Carlo

To import measurements:

1 In the Statistical Information table, click on the row containing the text “Click here to import a measurement created within PSpice.”

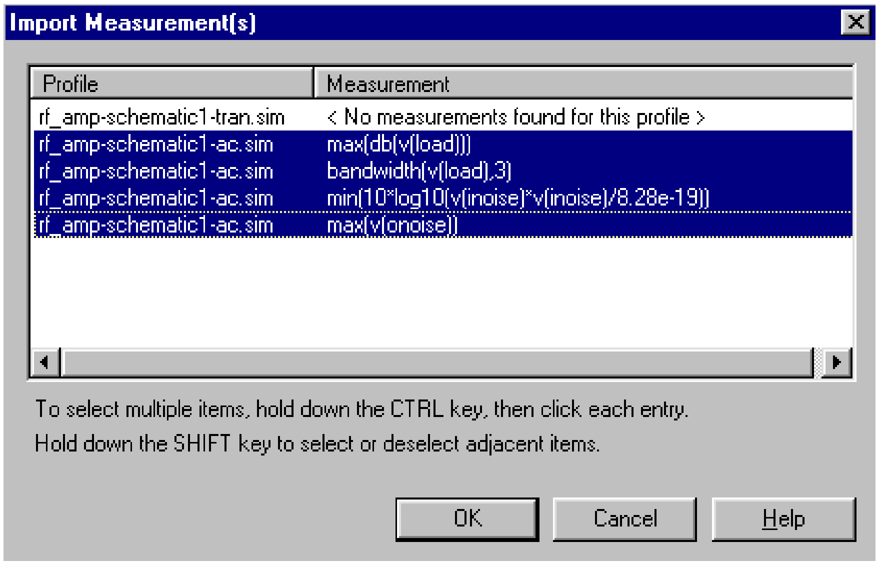
The **Import Measurement(s)** dialog box appears.

2 Select the measurements you want to include.

Here is an example:

1 In the Statistical Information table, click on the row containing the text “Click here to import a measurement created within PSpice.”

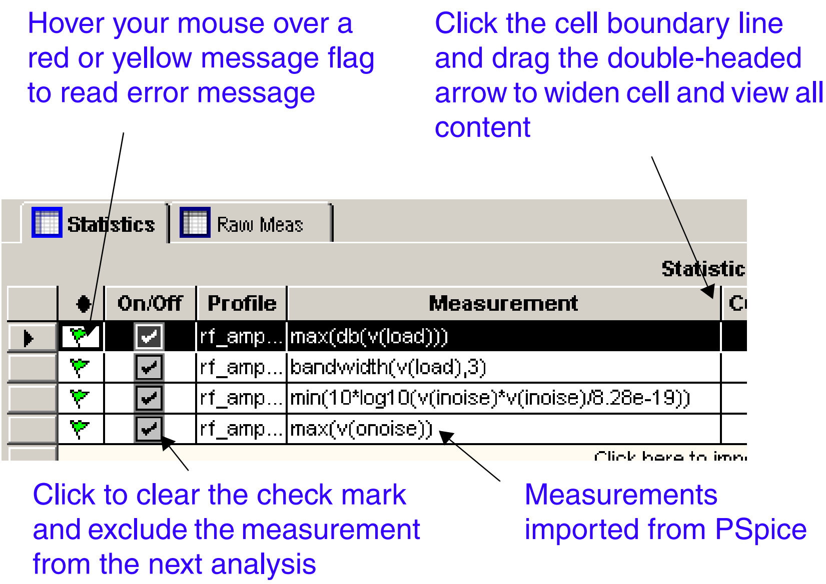
The **Import Measurement(s)** dialog box appears.



2 Select the four measurements:

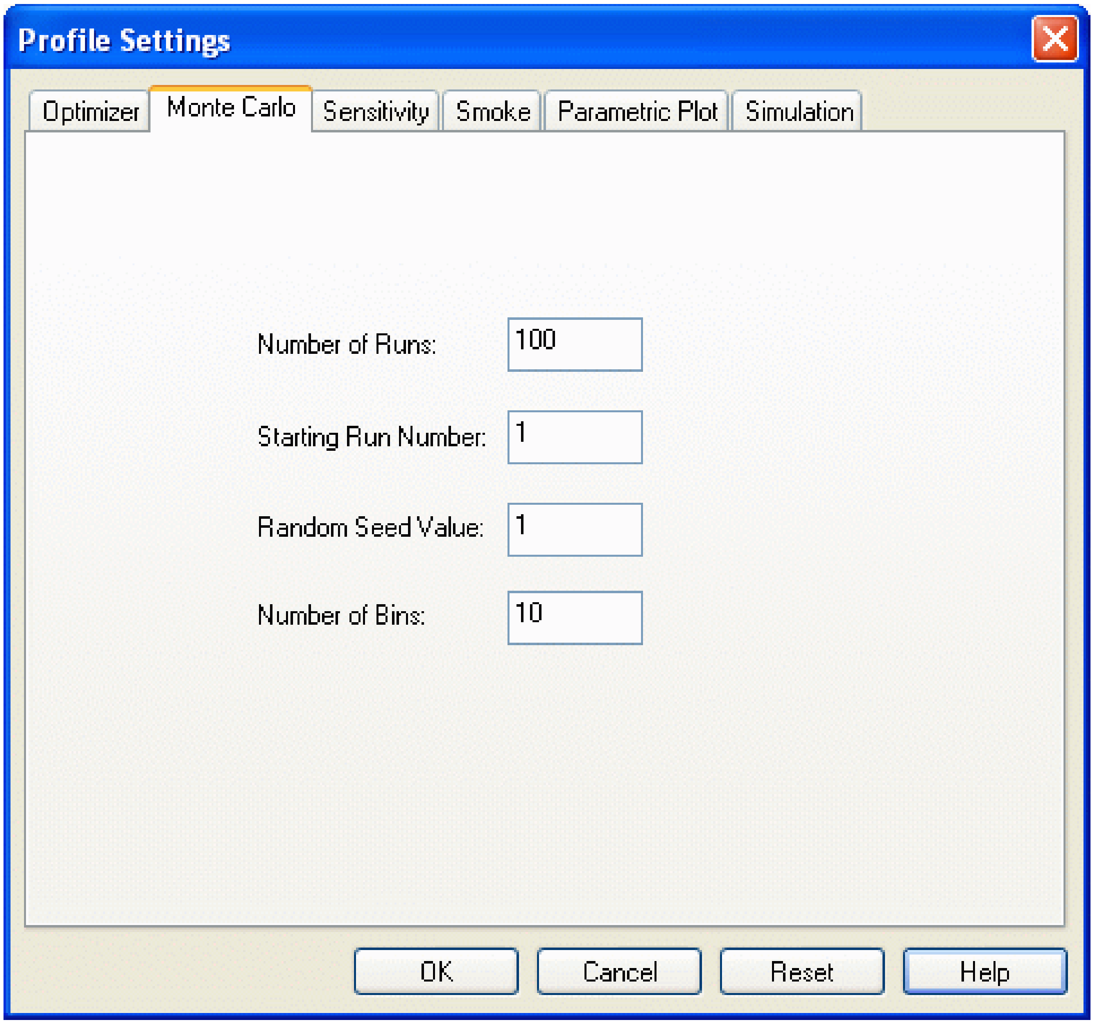
- Max(DB(V(Load)))
- Bandwidth(V(Load),3)
- Min(10*Log10(V(inoise)*V(inoise)/8.28e-19))
- Max(V(onoise))

3 Click **OK**.



Setting Monte Carlo options

From the Advanced Analysis **Edit** menu, select **Profile Settings**, click the **Monte Carlo** tab, and enter the following Monte Carlo options:



- Number of runs

  This is the number of times the selected simulation profiles will be run. For each run, component parameters with tolerances will be randomly varied. Run number one uses nominal component parameter values. The maximum number of runs is primarily limited by the amount of available memory.

- Starting run number

  The default starting run number is one. This is the nominal run. If the random seed value is kept constant, then you can change the starting run number in order to duplicate a partial Monte Carlo simulation. You can use this to isolate specific random results which are of particular interest, without having to run an entire Monte Carlo simulation again.

- Random seed value

The random number generator uses this value to produce a sequence of random numbers. Change the seed in order to produce a unique random sequence for each Monte Carlo simulation. If the seed and device properties are not changed, then the same sequence of random numbers will be generated each time a Monte Carlo analysis is done. You can use this procedure to reproduce a random simulation.

- Number of bins

  This value determines the number of divisions in the histogram. A typical value is one tenth of the number of runs. The minimum value is one and the maximum value is determined by the amount of available memory. It is recommended that this value be less than 10,000.

Here is an example:

1. From the Advanced Analysis *Edit* menu, choose *Profile Settings*, click the *Monte Carlo* tab, and enter the values in the dialog box.
2. Click *OK*.

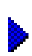
Starting a Monte Carlo run

Monte Carlo calculates a nominal value for each measurement using the original parameter values.

After the nominal runs, Monte Carlo randomly calculates the value of each variable parameter based on its tolerance and a flat (uniform) distribution function. For each profile, Monte Carlo uses the calculated parameter values, evaluates the measurements, and saves the measurement values.

Monte Carlo repeats the calculations for the specified number of runs, then calculates and displays statistical data for each measurement.

To start a Monte Carlo run:

– Click  on the top toolbar.

  The Monte Carlo analysis begins. The messages in the output window tell you the status of the analysis.

Here is an example:

1 Click .

The Monte Carlo analysis begins. The messages in the output window give you the status.

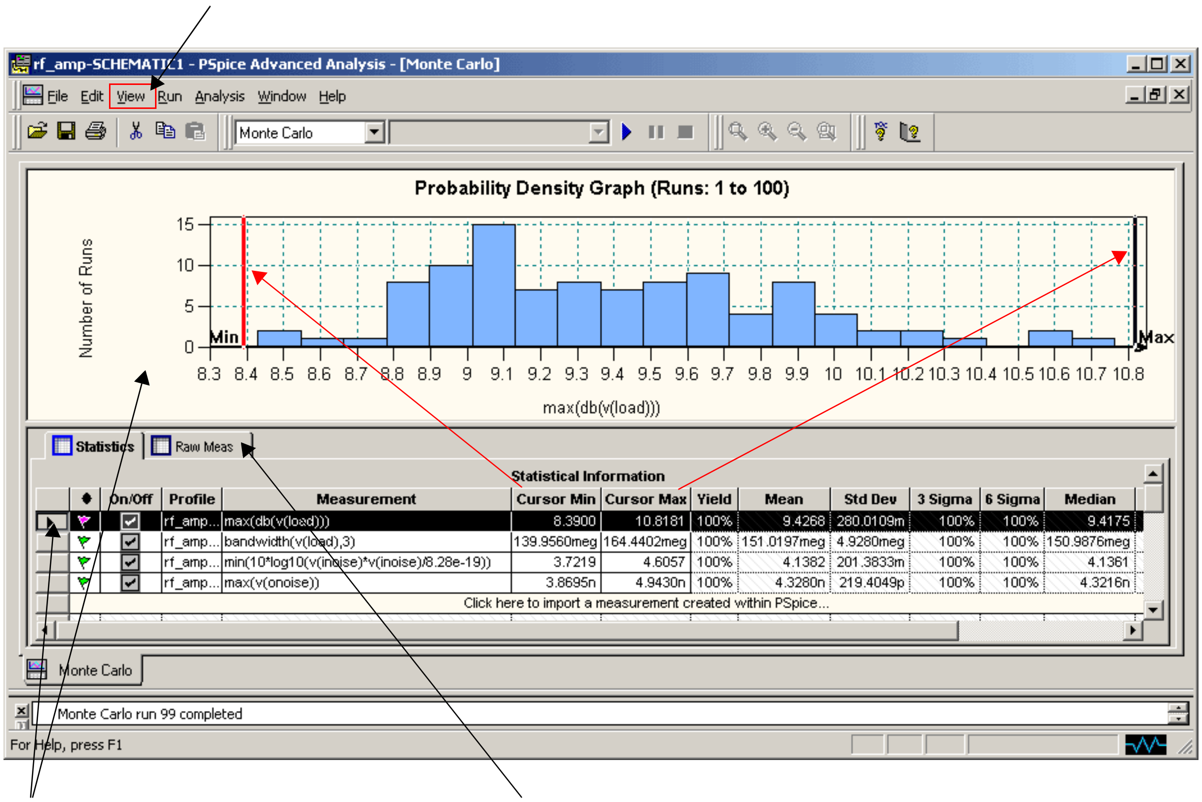
Monte Carlo calculates a nominal value for each measurement using the original parameter values.

After the nominal runs, Monte Carlo randomly calculates the value of each variable parameter based on its tolerance and a flat (uniform) distribution function. For each profile, Monte Carlo uses the calculated parameter values, evaluates the measurements, and saves the measurement values.

Monte Carlo repeats the above calculations for the specified number of runs, then calculates and displays statistical data for each measurement.

Ten bins of measurement data are displayed on the graph.

From the View menu, select Log File / Monte Carlo to see parameter values and other details



The selected measurement's min, max, and other run results are plotted on the PDF graph

Click Raw Meas tab for 100 run results

Controlling Monte Carlo run

The Monte Carlo analysis can only be run if tolerances are specified for the component parameters. In case you want to prevent running these analysis on a component, you can do so by using the TOL_ON_OFF property.

In the schematic design, attach the TOL_ON_OFF property to the device instance for which you do not want to perform the Sensitivity and MonteCarlo analysis. Set the value of the TOL_ON_OFF property to OFF. When you set the property value as OFF, the tolerances attached to the component parameters will be ignored and therefore, the component parameters will not be available for analysis.

Reviewing Monte Carlo data

You can review Monte Carlo results on two graphs and two tables:

- Probability density function (PDF) graph
- Cumulative distribution function (CDF) graph
- Statistical Information table, in the **Statistics** tab
- Raw Measurements table, in the **Raw Meas** tab

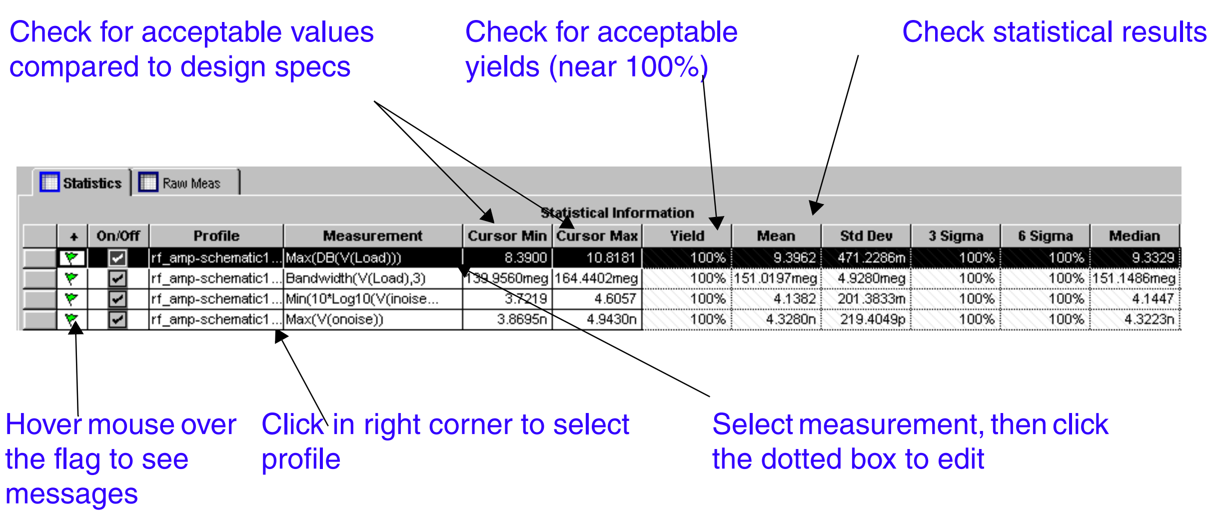
Reviewing the Statistical Information Table

For each run, Monte Carlo randomly varies parameter values within tolerance and calculates a single measurement value. After all the runs are done, Monte Carlo uses the run results to perform statistical analyses.

1 Click the **Statistics** tab to bring the table to the foreground.

2 Select a measurement row in the Statistical Information table.

A black arrow appears in the left column and the row is highlighted. The data in the graph corresponds to the selected measurement only.



You can review results reported for each measurement:

Column heading...	Means...
Cursor Min	Measurement value at the cursor minimum location.
Cursor Max	Measurement value at the cursor maximum location.
Yield (in percent)	The number of runs that meet measurement specifications (represented by the cursors) versus the total number of runs in the analysis. Used to estimate mass manufacturing production efficiency.
Mean	The average measurement value based on all run values. See Raw Measurement table for run values.
Std Dev	Standard deviation. The statistically accepted meaning for standard deviation.
3 Sigma (in percent)	The number of measurement run values that fall within the range of plus or minus 3 Sigma from the mean
6 Sigma (in percent)	The number of measurement run values that fall within the range of plus or minus 6 Sigma from the mean
Median	The measurement value that occurs in the middle of the sorted list of run values. See Raw Measurement table for run values

Reviewing the pdf graph

A PDF graph is a way to display a probability distribution. It displays the range of measurement values along the x-axis and the number of runs with those measurement values along the y-axis.

To review a PDF graph:

1. Select a measurement row in the Statistical Information table.
2. If the PDF graph is not already displayed, right click the graph and select **PDF Graph** from the pop-up menu.

   The corresponding PDF graph will display all measurement values based on the Monte Carlo runs.
3. Right click the graph to select zoom setting, another graph type, and y-axis units.

   A pop-up menu appears.

- Select **Zoom In** to focus on a small range of values.
- Select **CDF Graph** to toggle from the default PDF graph to the CDF graph.
- Select **Percent Y-axis** to toggle from the default absolute y-axis Number of Runs to **Percent of Runs**.

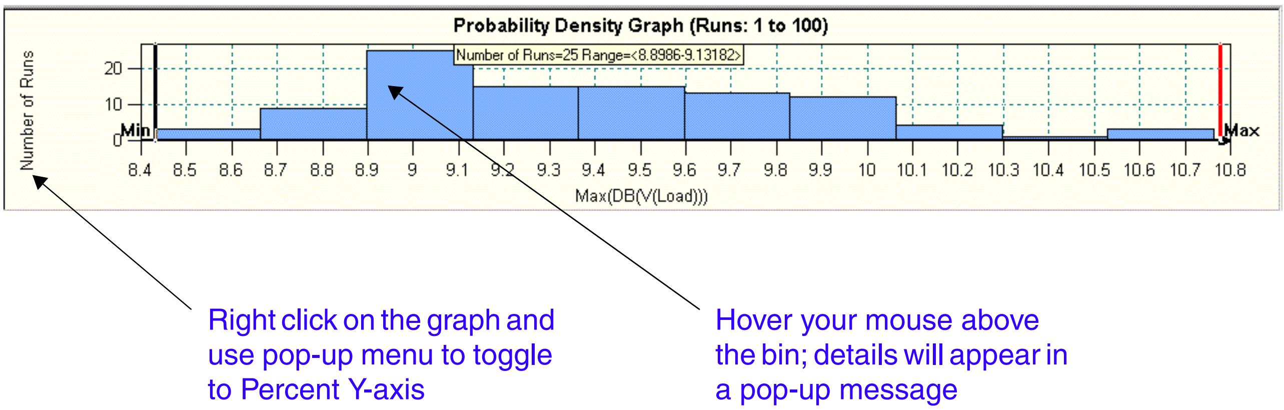
4 To change the number of bins on the x-axis:

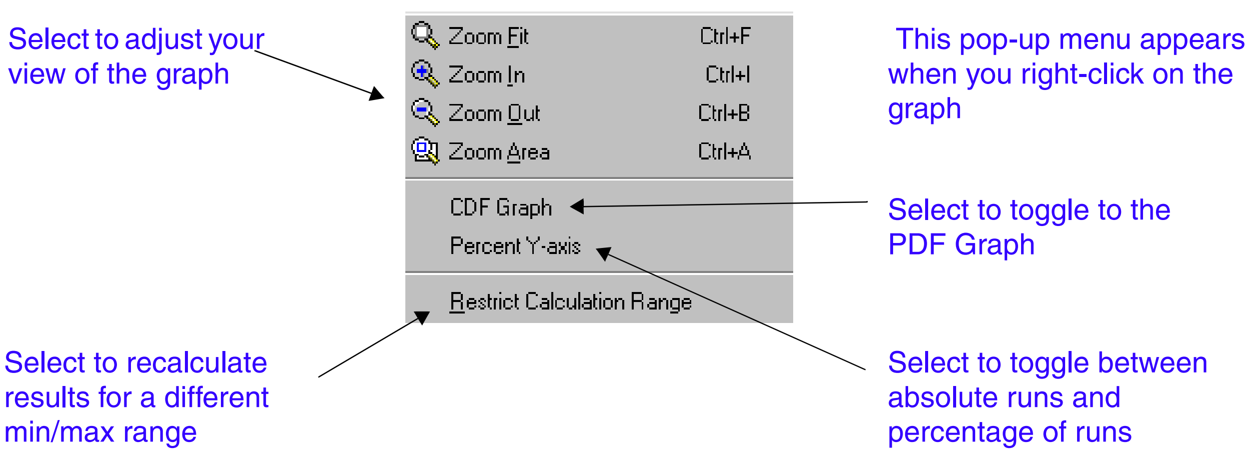
From the **Edit** menu, select **Profile Settings**, click the **Monte Carlo** tab, and typing a new number in the **Number of Bins** text box.

If you want more bars on the graph, specify more bins—up to a maximum of the total number of runs. Higher bin numbers show more detail, but require more runs to be useful.

The PDF graph is a bar chart. The x-axis shows the measurement values calculated for all the Monte Carlo runs.

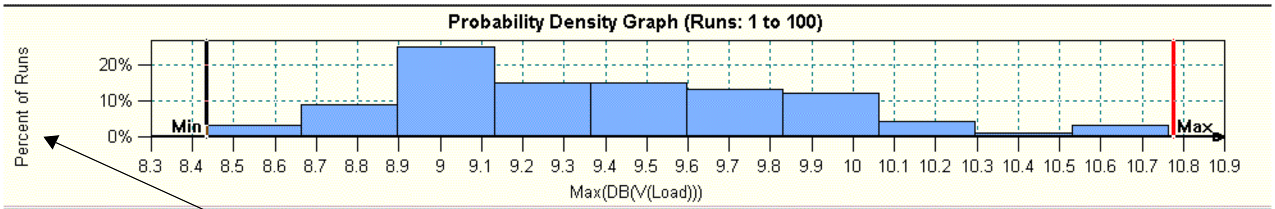
The y-axis shows the number of runs with measurement results between the x-axis bin ranges. The statistical display for this measurement's probability density function is shown on the PDF graph.





1 Right click on the graph and select **Percent Y-axis** from the pop-up menu.

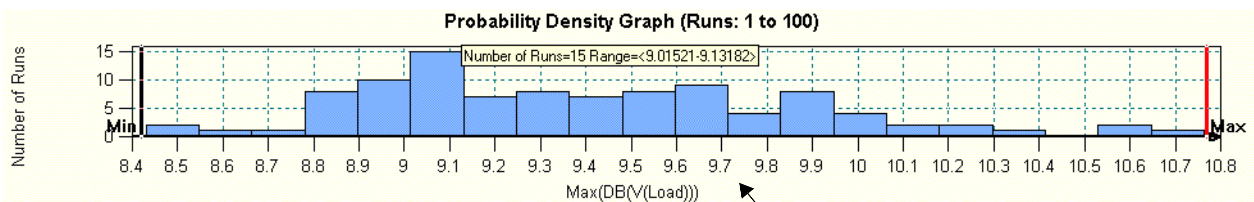
The Y-axis units changes from **Number of Runs** to **Percent of Runs**.



Y-axis changed to Percent of Runs

2 From the **Edit** menu, select **Profile Settings**, click the **Monte Carlo** tab, select the **Number of Bins** text box and type the number 20 in place of 10.

Notice the higher level of detail on the PDF graph.



Same statistical results, but 20 bins are specified: twice as many as first PDF graph

3 Right click on the graph and from the pop-up menu select **Zoom In** to view a specific range.

4 Select **Zoom Fit** to show the entire graph with cursors.

5 Click the **Max** cursor to select it (it turns red when selected), then click the mouse in a new location on the x-axis.

The cursor's location changes and the max value and yield numbers are updated in the Statistical Information table.

Note: Moving the cursor does not update the rest of the statistical results for this new min / max range. Use **Restrict Calculation Range** to recalculate the rest of the statistical results for this min / max range.

Reviewing the cdf graph

The CDF graph is another way to display a probability distribution. In mathematical terms, the CDF is the integral of the PDF.

To review a CDF graph:

1. Select a measurement row in the Statistical Information table.
2. If the CDF graph is not already displayed, right click on the PDF graph and select **CDF Graph** from the pop-up menu.

   The statistical display for the cumulative distribution function is shown on the CDF graph.
3. Right click the graph to select zoom setting and y-axis units.

   A pop-up menu will appear.

   - Select **Zoom In** to focus on a small range of values.
   - Select **PDF Graph** to toggle from the current CDF graph to the default PDF graph.
   - Select **Percent Y-axis** to toggle from the default absolute y-axis Number of Runs to **Percent of Runs**.
4. Change the number of bins on the x-axis by going to the **Edit** menu, selecting **Profile Settings**, clicking the **Monte Carlo** tab, and typing a new number in the **Number of Bins** text box.

- If you want more bars on the graph, specify more bins, up to a maximum of the total number of runs. Higher bin numbers show more detail, but require more runs to be useful.

Working with cursors

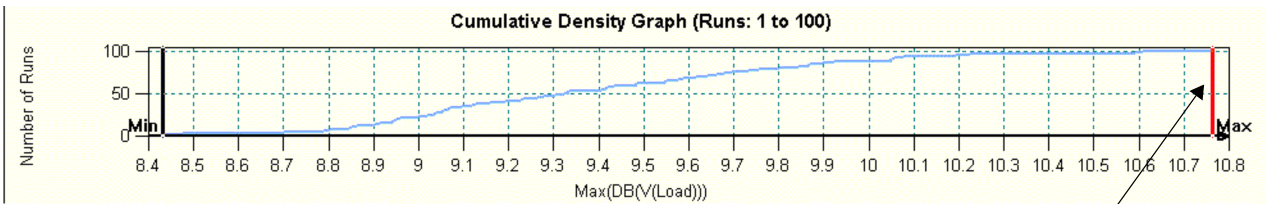
– To change a cursor location on the graph, click the cursor to select it and click the mouse in a new spot on the graph. A selected cursor appears red.

  The cursor's location on the graph changes, and the measurement min or max values in the Statistical Information table are updated. A new calculated yield displays.

The CDF graph is a cumulative stair-step plot.

1. Select the **Max(DB(V(Load)))** measurement in the Statistical Information table.

2 Right click on the PDF graph and select **CDF Graph** from the pop-up menu.



CDF graph with max cursor selected; before cursor is moved for restricted range calculation

3 Right click on the graph and select **Zoom In** to view a specific range.

4 Click the **Max** cursor to select the cursor.

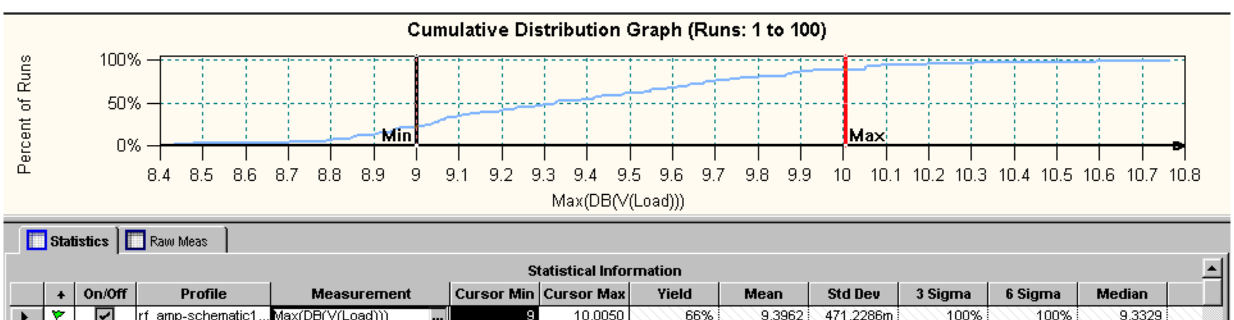
The Max cursor turns red.

5 Click the mouse at 10 on the x-axis.

The cursor moves to the new position on the x-axis.

6 Click the **Min** cursor and click the mouse at 9 on the x-axis.

When you change the cursor location the min, max, and yield values are updated on the Statistical Information table.



Cursor Min and Cursor Max data change to reflect new cursor positions

Yield value changes to reflect new min / max data

Restricting the calculation range

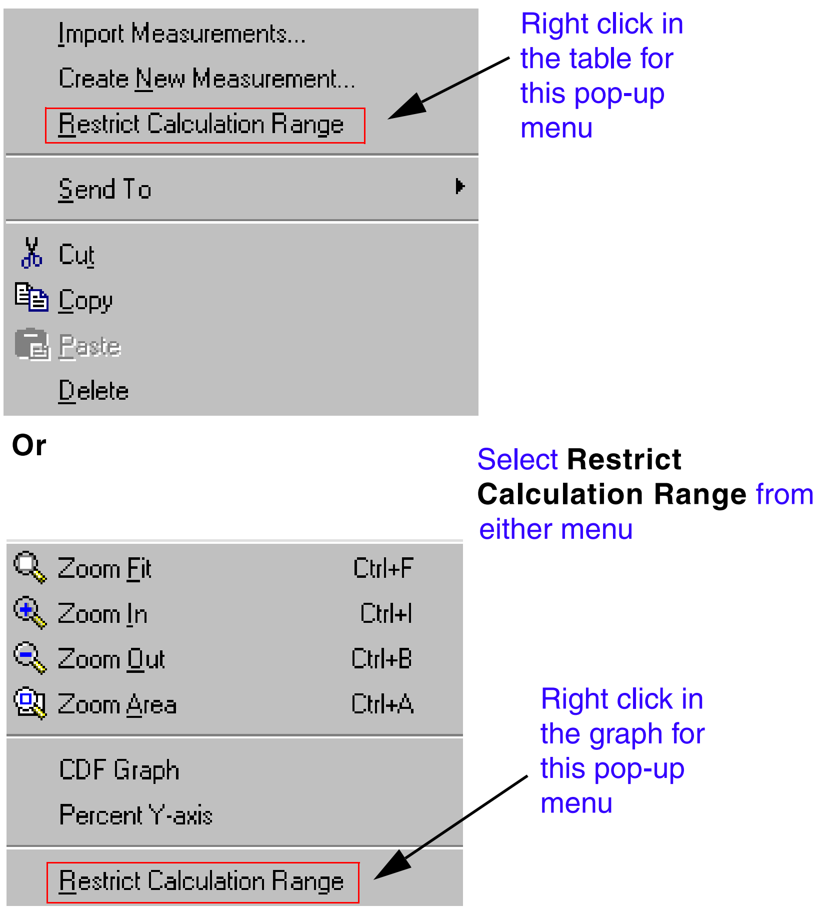
To quickly view statistical results for a different min / max range, use the **Restrict Calculation Range** command.

1 Set the graph cursors at **Min** = 9 and **Max** = 10.

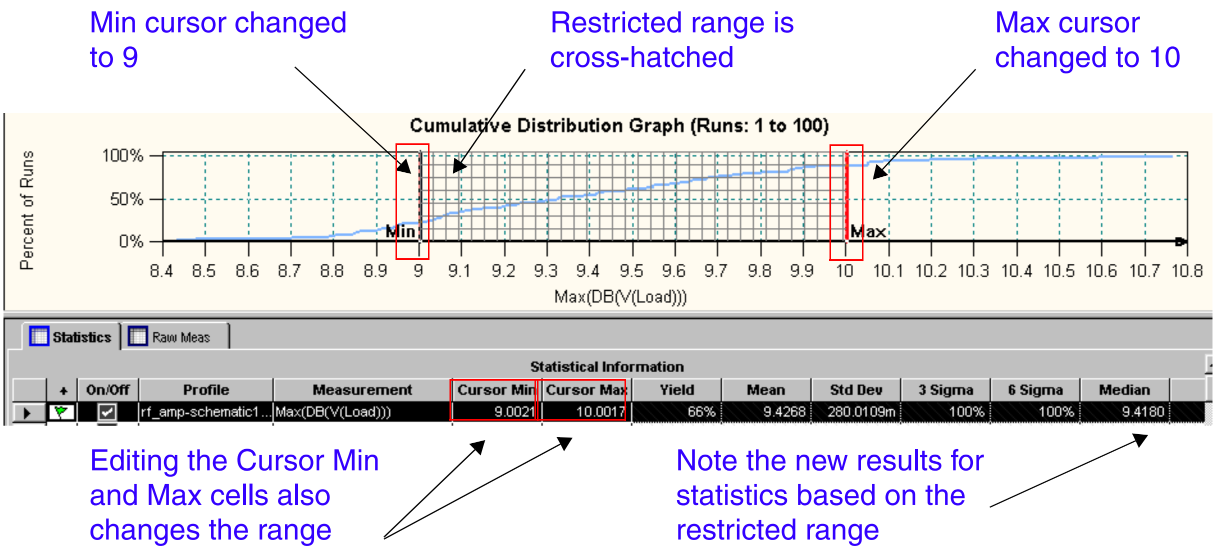
Or:

Edit the min or max values in the Statistical Information table.

2 Right click in the table or on the graph and select **Restrict Calculation Range** from the pop-up menu.



Monte Carlo recalculates the statistics and only includes the restricted range of values.



Restricting calculation range

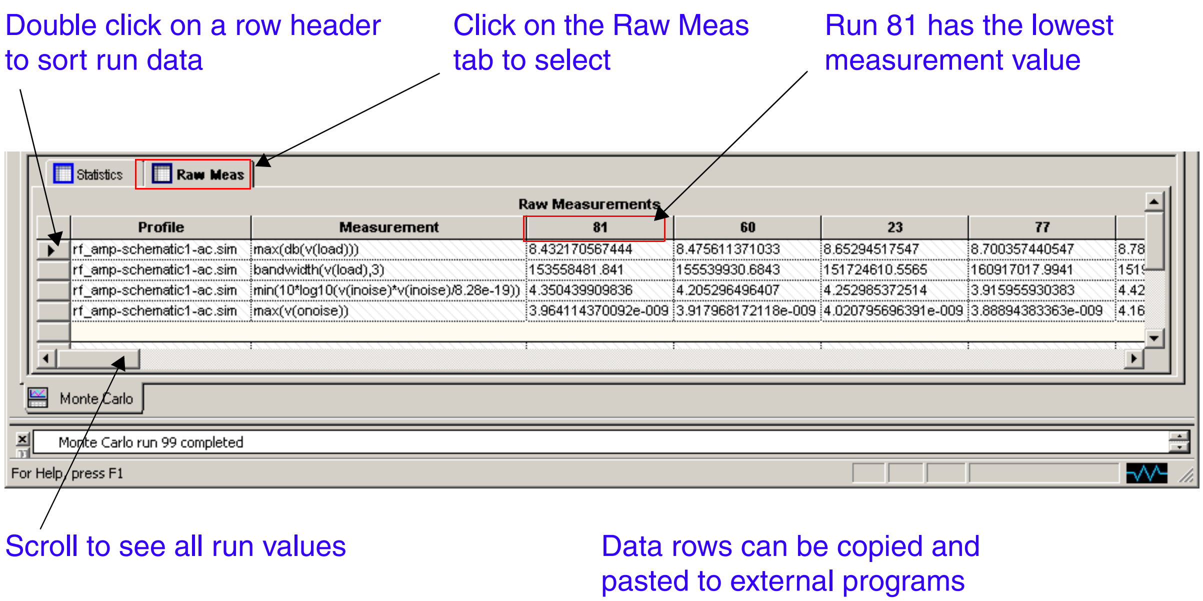
To restrict the statistical calculations displayed in the Statistical Information table to the range of samples within the cursor minimum and maximum range, set the cursors in their new locations and select the restrict calculation range command from the right click pop-up menus.

1. Change cursors to new locations.
2. Right click in the graph or in the Statistical Information table and select **Restrict Calculation Range** from the pop-up menu.

   The cross-hatched range of values that appears on the graph is the restricted range.

Reviewing the Raw Measurements table

The Raw Measurements table is a read-only table that has a one-to-one relationship with the Statistical Information table. For every measurement row on the Raw Measurements table, there is a corresponding measurement row on the Statistical Information table. The run values in the Raw Measurements table are used to calculate the yield and statistical values in the Statistical Information table.



To review a Raw Measurements table:

1 Click the **Raw Meas** tab.

The Raw Measurements table appears.

2 Select a row and double click the far left row header.

The row of data is sorted in ascending or descending order.

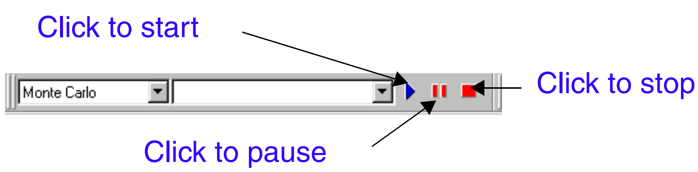
Note: Copy and paste the row of data to an external program if you want to further manipulate the data. Use the **Edit** menu or the right click pop-up menu copy and paste commands.

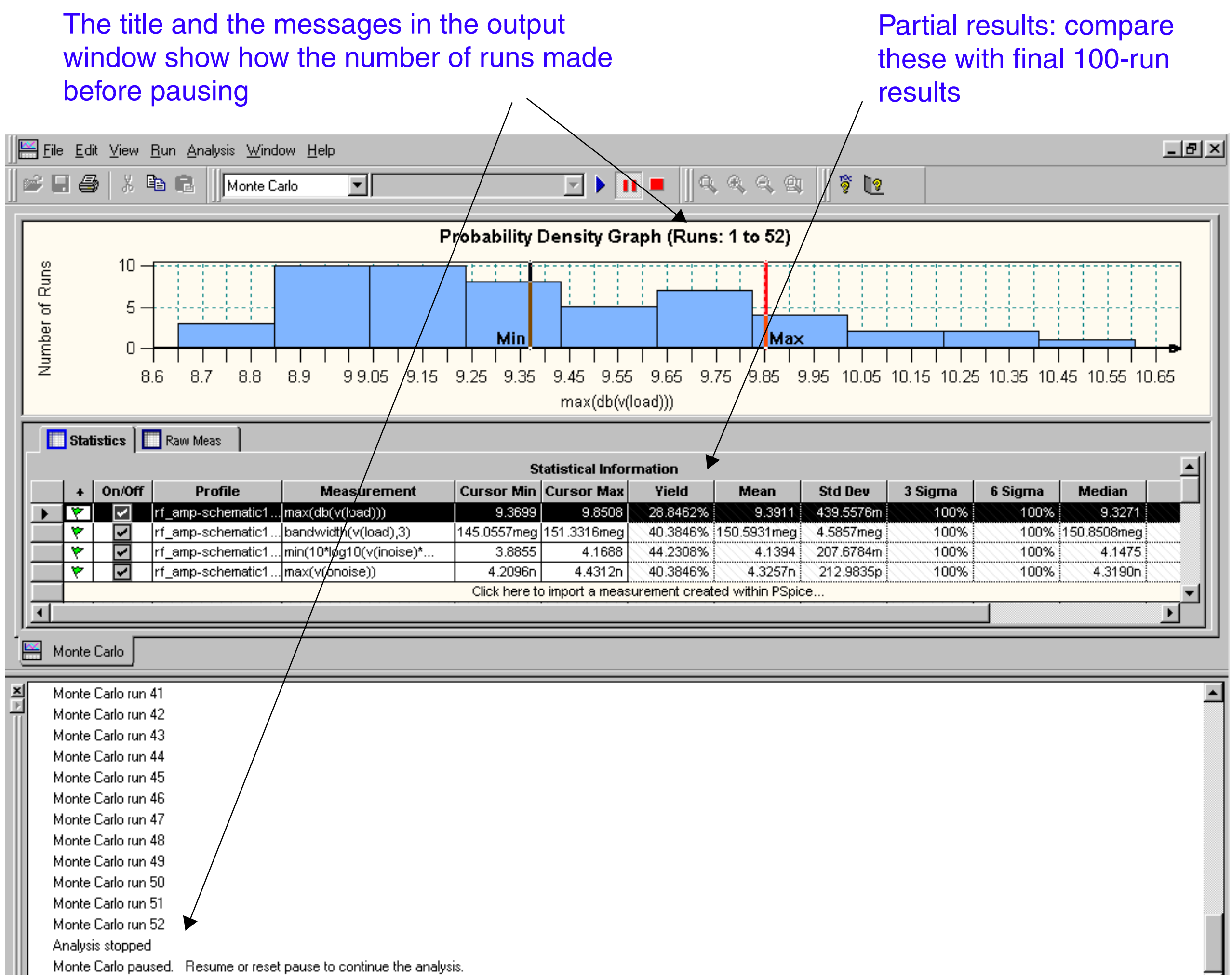
3 From the **View** menu, select **Log File** / **Monte Carlo** to view the component parameter values for each run.

Controlling Monte Carlo

The following sections explain how to fine-tune the process if you do not achieve your goals in the first Monte Carlo analysis.

Pausing, stopping, and starting Monte Carlo





Pausing and resuming

To review preliminary results on a large number of runs:

– Click ❚❚ on the top toolbar when the output window indicates approximately Monte Carlo run 50.

The analysis stops at the next interruptible point, available data is displayed and the last completed run number appears in the output window.

1 Click the depressed or to resume calculations.

Stopping

– Click on the top toolbar.

If a Monte Carlo analysis has been stopped, you cannot resume the analysis.

Starting

– Click to start or restart.

Changing components or parameters in Monte Carlo

If you do not get the results you want, you can return to the schematic editor and change circuit parameters.

1 Try a different component for the circuit or change the tolerance parameter on an existing component.

2 Rerun the PSpice simulation and check the results.

3 Rerun Monte Carlo using the settings saved from the prior analysis.

4 Review the results.

Controlling measurements in Monte Carlo

If you do not get the results you want and your design specifications are flexible, you can add, edit, delete or disable a measurement and rerun Monte Carlo analysis:

– To exclude a measurement from the next optimization run, click the in the Statistical Information table, which removes the check mark.

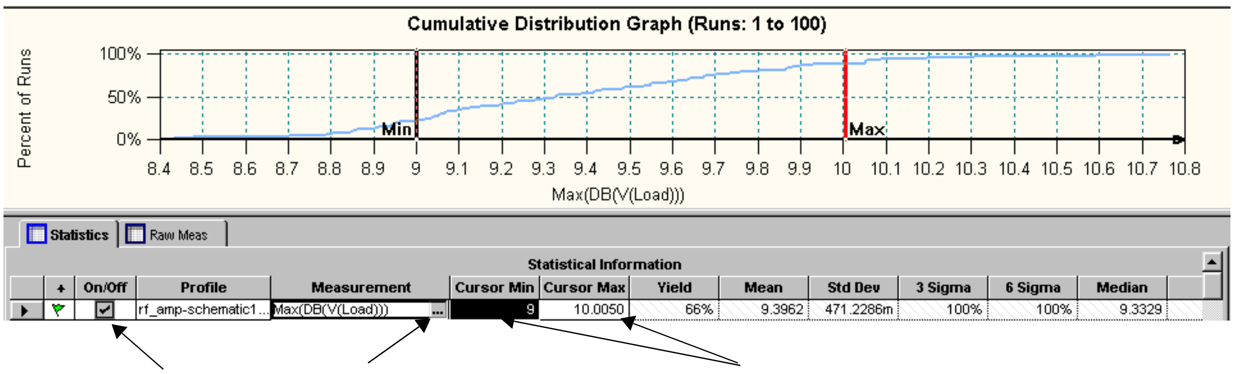
– To edit a measurement, click on the measurement you want to edit, then click .

– To edit a measurement specification Min or Max, click the minimum or maximum cursor on the graph (the selected cursor turns red), then click the mouse in the spot you want.

The new value will display in the **Cursor Min** or **Cursor Max** column in the Statistical Information table.

- To add a new measurement, click on the row that reads “Click here to import a measurement...”
- To export a new measurement to Optimizer or Monte Carlo, select the measurement and right click on the row containing the text “Click here to import a measurement created within PSpice.”

  Select **Send To** from the pop-up menu.



Click here to remove the check mark and exclude the measurement from further analysis

Click on the dotted box and edit the measurement expression

Edit Cursor Min and Cursor Max values on the table; rerun Monte Carlo; observe new results.

You can see the following for more information:

How to make your own measurement expressions in PSpice	Creating measurement expressions
Checking measurement expressions in PSpice	Viewing results of measurements
Creating measurements in Advanced Analysis	Creating measurements in Advanced Analysis

Storing simulation data

The simulation data will be overwritten by each new run. Only the last run’s data will be saved. If you are planning an analysis of thousands of runs on a complex circuit, you can turn off the simulation data storage option to conserve disk space.

To turn off data storage:

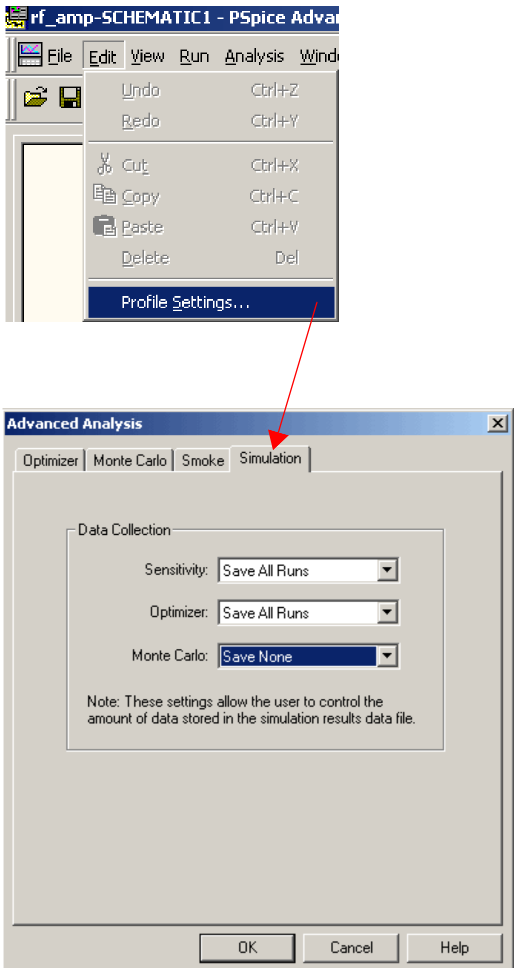
1 From the Advance Analysis menu select *Edit / Profile Settings/ Simulation*.

2 From the Monte Carlo field, select **Save None**.

The simulation data will be overwritten by each new run. Only the last run's data will be saved.

Here is an example:

1 From the Advance Analysis menu select *Edit / Profile Settings/ Simulation*.



2 From the Monte Carlo field, select **Save None**.

Parametric Plotter

The Parametric Plotter added to Advanced Analysis provides you with the functionality of sweeping multiple parameters. Once you have created and simulated a circuit, you can use the Parametric Plotter to perform this analysis.

Note: Parametric Plotter is available only if you have PSpice[1] Advanced Analysis license.

The Parametric Plotter gives users the flexibility of sweeping multiple parameters. It also provides a nice and an efficient way to analyze sweep results. Using Parametric Plotter, you can sweep any number of design and model parameters (in any combinations) and view results in PPlot/Probe in tabular or plot form.

Using the Parametric Plotter, you can:

- Sweep multiple parameters.
- Allow device/model parameters to be swept.
- Display sweep results in spreadsheet format.
- Plot measurement results in Probe UI.
- Post analysis measurement evaluation

Launching Parametric Plotter

From Schematic Editor

– From the PSpice menu in schematic editor, select Advanced Analysis > Parametric Plot.

  The Parametric Plotter window appears.

Stand Alone

1 From the Start menu, choose Programs > OrCAD 10.X > Advanced Analysis.

2 Open the `.aap` file.

3 From the Analysis drop-down list, select Parametric Plotter.

  The Parametric Plotter window appears.

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

You can now use the Parametric Plotter to analyze your circuit. Using Parametric Plotter is a two steps process.

1 In the first step, you select the parameters to be swept and also specify the sweep type Sweep Types. .

In the second step, you specify the measurements to evaluated at each sweep. See Adding measurements After you have identified the sweep parameters and specified measurements, run the sweep analysis and view the results in the Results tab or the Sweep Types of the Measurements window.

Sweep Types

During the sweep analysis, the parameters values are varied as per the user specifications. There are four possible ways in which you can vary the parameter values. These are:

- Discrete Sweep
- Linear Sweep
- Logarithmic octave sweep
- Logarithmic decade sweep

Discrete Sweep

For discrete sweep, you need to specify the actual parameter values to be used during the simulation runs. The parameter values are used in the order they are specified.

Example: Discrete Sweep

You can specify the values of variable parameters as 10, 100, 340, and so on.

Linear Sweep

For Linear sweep, specify the Start, End, and Step values. For each run of the parametric plotter, the parameter value is increased by the step value. In other words, the parameter values used during the simulation runs is calculated as `Start Value + Step Value`. This cycle continues till the parameter value is either greater than or equal to the `End Value`.

Example: Linear Sweep

If for a parameter you specify the start value as 1, End value as 2.5, and the step value as 0.5, the parameter values used by the Parametric Plotter are 1, 1.5, 2, and 2.5.

Logarithmic octave sweep

In the logarithmic octave sweep, the parameters are varied as a function of `ln(2)`.

For Logarithmic Octave sweep, you need to specify the Start Value, End Value, and number of points per Octave.

Number of points per Octave is number of points between the start value and two times start value. For example, if the start value is 10, number of points per Octave is 5, this implies that for sweep analysis, the Parametric Plotter will pick up 5 value between 10 and 20, with 20 being the fifth value.

During the analysis the parameter value in increased by a factor that is calculated using the following equation:

```
factor = exp[(ln(2)/N]
```

Where

N Number of points per octave

Example: Logarithmic octave sweep

Consider that the sweep type for a parameter is LogarithmicOct. The start value, end value and the number of points per Octave are specified as 10, 30, and 2, respectively.

The values used by the Parametric Plotter for LogarithmicOct sweep type will be `10`, `14.142`, `20`, `28.284`, and `40`.

In this example, the difference between start and end values is more than an octave, therefore, the actual number of values used by the Parametric Plotter is more than 2.

Logarithmic decade sweep

If the sweep type is LogarithmicDec, the parameter values are varied as a function of `ln(10)`. For Logarithmic decimal sweep, you need to specify the Start Value, End Value, and number of points per decade.

Number of points per decade is number of points between the start value and 10 times start value. For example, if the start value is 10, number of points per decade is 5, this implies that for sweep analysis, the Parametric Plotter will pick up 5 value between 10 and 100, with 100 being the fifth value.

During the analysis the parameter value in increased by a factor, which is calculated using the following equation:

```
factor = exp[(ln(10)/N]
```

Where

N Number of points per decade

Example: Logarithmic decade sweep

If you specify the start value as 10, end value as 100, and number of points per decade as 5, the parameter values used for sweep analysis will be 10, 15.8489, 25.1189, 39.8107, 63.0957, and 100.

Adding Sweep Parameters

In the Sweep Parameters window, add the parameters values that you want to vary during the sweep analysis.

1 In the Sweep Parameters window, click the *Click here to import a parameter from the design property map* row.

   The Parameter Selection dialog box appears with a list of components and the parameters for which you can sweep the parameter values.

   Only the component parameters that have been defined in the schematic, appear in the Parameter Selection dialog box.

2 For the parameter that you want to vary, specify the Sweep Type.

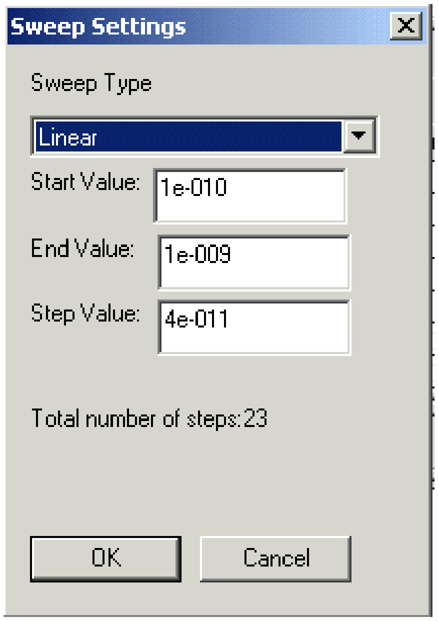
   a. In the Parameter Selection dialog box, click the *Sweep Type* grid.

   b. From the drop-down list, select the sweep type as `Discrete`, `Linear`, `LogarithmicDec`, or `LogarithmicOct`.

   Note: Sweep type defines the method used by the Parametric Plotter to calculate variable parameter values. To know more about the sweep types, see Sweep Types .

3 To specify the sweep values for the selected parameter, click the Sweep Values grid.

The Sweep Settings dialog box appears.



4 In the Sweep Settings dialog box, the sweep type you selected in the previous step appears in the *Sweep Type* drop-down list box. Specify the parameter values that would be used for each parameter during sweep analysis.

To know more about the sweep types and sweep values to be specified, see <u>Sweep Types</u>.

5 Click OK to save your specifications.

The selected parameters get added in the sweep parameter window. When you add the parameters, a Sweep Variable is automatically assigned to each of the parameters.

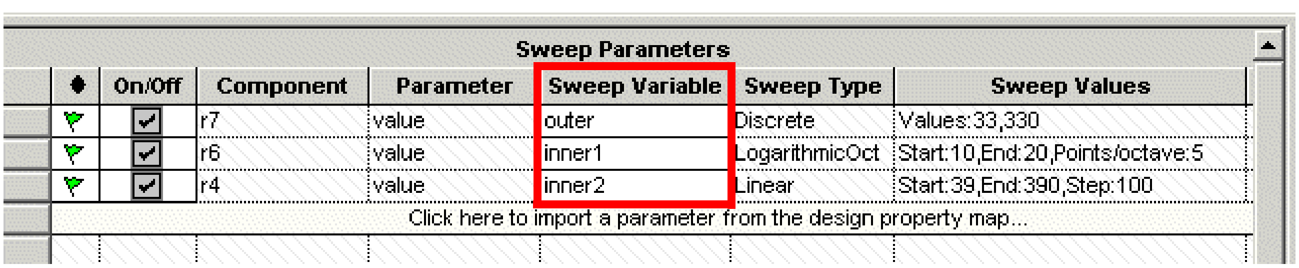


Figure 12-1 Setting sweep parameters

The value of the sweep variable is an indication of how parameters will be varied during sweep analysis. Sweep Variables values are assigned in the order in which sweep parameters are defined. If required, you can change these values. While modifying the values of Sweep Variable, ensure that each parameter has a unique value of sweep variable attached to it. Also the values should follow the sequence. For example, if you select three parameters to be varied during the sweep analysis, the sweep variables should have values as `outer`, `inner1`, and `inner2`. You cannot have random values such as `inner1`, `inner2`, and `inner4`.

For the sweep analysis, the values of parameters is varied in nested loops. For example, if you select two variables, the outer variable is fixed for the analysis, while the inner variable goes through all of its possible values. The outer variable is then incremented to its next value, and the inner variable again cycles through all of its possible values. This process is continued for all possible values of the outer variable.

The result for each run of the analyzer appears in the Results pane. By default, the results are displayed in the order described above.

Tip

Similar process is followed in case multiple (more than two) parameter values need to be varied.

For example, in Figure 12-1 on page 149, for constant values of r7 and r6, the value of r4 will be varied. The values of r7 and r6 will not change till r4 has been assigned all possible values within the range specified by the user. After r4 completes a cycle, the value of r6 will be increased, and r4 will again be varied for all possible values.

Adding measurements

Parametric Plotter is used for evaluating the influence of changing parameter values on an expression and on a trace. A measurement can be defined as an expression that evaluates to a single value, where a trace is an expression that evaluates to a curve.

Adding measurement expressions

You can either add a measurement expression that was created in PSpice or can even create a new measurement in PSpice Advanced Analysis.

Adding measurements created in PSpice

1 In the Measurements tab, click the *Click here to import a measurement created in PSpice* row.

The Import Measurements dialog box appears. This dialog box lists only the measurements that you created in PSpice.

2 Select the measurement that you want to be evaluated and click OK.

Selected measurement gets added in the Measurements tab.

Tip

Only the measurements that are listed in the Measurements Results window of PSpice are available in the Import Measurements dialog box.

Adding new measurements

1 In the Measurements tab, right-click and select Create New Measurements.

The New Measurement dialog box appears.

2 From the Profile drop-down list, select the simulation profile for which you want to create the measurement.

3 From the Measurements drop-down list, select the Measurement that you want to evaluate.

4 From the Simulation Output Variables list specify the variable on which the measurement is to be performed and click OK

The new measurement gets added to the Measurements tab.

Important

Using the New Measurements dialog box, you can only add the already defined measurements to the Parametric Plotter window. To define new measurements in PSpice use the *Trace > Measurements* command in PSpice.

Adding trace

Using the Parametric Plotter, you can evaluate the influence of changing parameter values on a trace. To be able to do this, you need to add a trace in the Measurements tab.

1 From the Analysis drop-down menu, select *Parametric Plotter > Create New Trace*.

Alternatively, right-click on the Measurements tab and select *Create New Trace*.

The New Trace Expression dialog box appears.

2 Create an expression to define the new trace and click OK.

The trace expression gets added in the Measurement window, with type as Trace.

Running Parametric Plotter

After you have specified the measurements and the list of variable parameters, run the Parametric Plotter.

– From the Run drop-down menu choose Start Parametric Plotter.

  Note: Alternatively, click the Run button on the toolbar or press `<CTRL>+<R>` keys.

For optimized performance of Parametric Plotter, maximum number of parametric sweeps supported in one session is 1000. If for your selection of parameters and measurements, the total number of sweeps required is greater than 1000, an error message is displayed in the Output Window, and analysis stops. As the simulation progresses, the Output Window also shows the profile selected and the number of sweep run being executed.

Note: You can set the maximum number of parametric sweeps in the *Parametric Plot* tab of the Profile Settings dialog box. Select *Reset to Default Value* to set the maximum number of sweeps to the default value.

Important

The Number of parametric sweeps required, which is displayed in the Output window, should be interpretted as the number of sweeps required per profile. The total number of sweeps required is calculated separately for each profile.

Viewing results

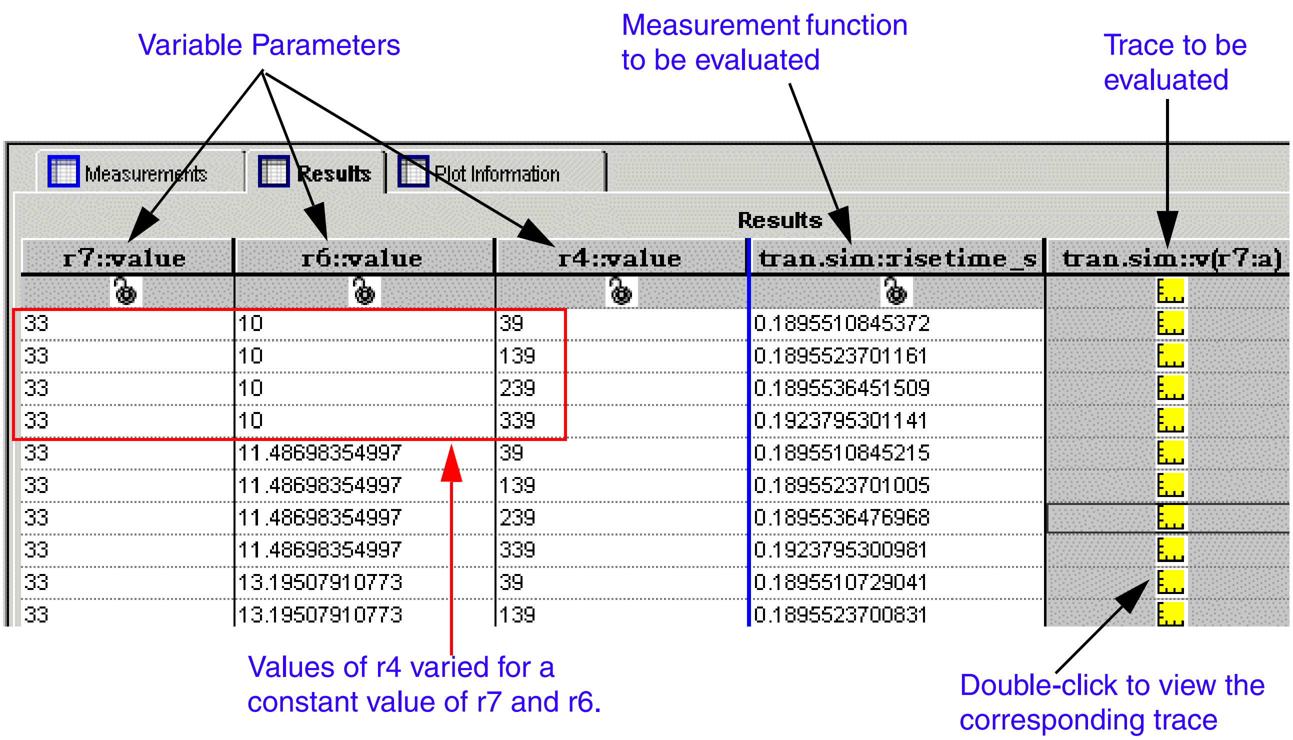
The results of the parametric sweep analysis are displayed in form of a spread sheet in the Results tab of the Measurement window. For the same results, you can define plot information using the Plot Information tab. The plot information is displayed in the PSpice Probe window.

Results tab

The results tab displays the simulation result for each run of the Parametric Plotter. Each run of the parametric plotter is indicated by a row in the Results tab. Therefore, if for the complete analysis Parametric plotter completes 100 runs, there will be 100 rows in the results tab.

The number of columns in the results tab is equal to the number of variable parameters and the number of measurements or the traces to be evaluated. There is one column each for a variable parameter and measurement expression to be evaluated.

In case of traces, instead of the measurement value, a trace is generated for each run of Parametric Plotter. As traces cannot displayed on the Results tab, therefore, instead of each trace a yellow colored bitmap is visible. To view the complete trace, double-click the yellow colored bitmap in the Results pane. The trace gets displayed in the PSpice Probe window.



Analyzing results

You can set up the Parametric Plotter to display data in a number of ways.

Sorting values

You can sort the results of the sweep analysis according to the values in any column.

For example, if you want to view the result of keep r4 to a constant value of 39, sort the values in the third column and view the results.

To sort the values displayed in a column, double-click on the column name. Once the contents of the column are sorted, subsequent click on the column name with toggle the order of sorting.

For example, after the Results pane is populated, double-clicking the column name arranges the values in ascending order. Now if you again double-click on the column name, the column contents willl get arranged in descending order.

Locking Values

While analyzing the simulation results, you can lock the values displayed in one column. Once you have locked the values of a column, the order in which the values are displayed in that column do not change. You can then sort the values in other columns.

For example, you can sort the values of r7 and lock the column. If you now sort the values of r6, the values will be sorted for fixed value of r7.

To lock the values displayed in a column, click the lock icon at the top of the column.

Plot Information

The Plot Information tab can be used to specify a plot that you want to view in the Probe window. Using the Plot Information tab, you can view multiple traces in one window. This is useful when you want to view the result of varying a parameter on the output.

At any given point of time, you can add a maximum of four plots.

Column Name	Description...
Message Flag 	A message flag can have three values: red, green, and yellow. The red flag indicates an error in the Plot. The green flag indicates that the plot added by the user is correct and can be viewed in the PSpice Probe window.
Plot Name	Name of the plot added. This field is populated by the Parametric Plotter itself and has values such as PLlot1, Plot2, and so on.
X-Axis	Lists the variable that will be plotted on the X-axis for the selected plot.
Y-Axis	Lists the variable that will be plotted on the Y-axis for the selected plot.

Column Name	Description...
Parameter	Lists the parameter that is unique for each curve. For example,
Constant	Lists the constant parameter value.

Adding Plot

1 From the *Analysis* menu select *Parametric Plotter > Add New Plot*.

The Plot Wizard appears.

Note: Alternatively, right-click on the Plot Information tab and select Add Plot.

2 In the Select Profile page of the Plot Wizard, specify the simulation profile for which you want the profile to be created and click Next.

3 In the select X-Axis Variable page of the wizard, specify the variable parameter that you want to plot on the X-axis of the plot.

From the variables drop-down list you can select any of the sweep parameter or the measurements that you specified in the Measurements tab.

Besides the variable parameter and the measurements, the drop-down list has an extra entry, which is time or frequency.

When you select a transient profile, you can select Time as the X-Axis variable and plot out results against time. When you select a AC profile, you can select Frequency as the X-Axis variable.

4 Click Next.

5 In the Select Y-Axis Variable page, select the variable to be plotted in the Y-axis and click Next.

Depending on your selection in the previous page of the Plot wizard, either the measurement expressions or traces appears in the Variables drop-down list.

When you select time or frequency as X-Axis Variable, all the traces added by you in the Measurements tab appear in the drop-down list. For all other selections of X-Axis Variables, the measurements added by you in the Measurements tab, are listed in the drop-down list.

6 In the Select Parameter page of the Plot Wizard, specify the parameter that will be varied for each trace to be plotted and click Next.

7 In cases where there are more than two variable parameters, you need to specify a constant value for the variable parameters that are not covered in step 3 or step 6.

Right-click on the parameter value and choose Lock.

8 Click Finish.

The complete plot information gets added in the Plot Information tab.

Viewing the plot

1 Select the plot to be displayed in the PSpice probe window.

2 From the Analysis drop-down menu, choose Parametric Plotter > Display Plot.

Alternatively, right-click on the selected row and choose Display Plot.

The PSpice probe window appears with multiple traces.

Measurements tab

The Measurements tab is used for specifying the measurements or the trace that are to be evaluated. This tab has multiple columns. The column name and the field description is listed in the table shown below.

Column Name	Description...
Message Flag 	A message flag can have three values: red, green, and yellow. The red flag indicates an error in the specification. A yellow flag indicates that the progress has stopped for some reason, and the green flag indicates that the analysis is going fine.

Column Name	Description...
On/Off	A check mark indicates that the measurement specification will be included in the current run of Parametric Plotter. If this check box is clear, all other columns in the row are also ignored, indicating that the specification will not be considered for the Parametric Plotter runs.
Profile	Lists all the profiles available in the design.
Measurement	Lists the measurement expressions or the measurement traces to be evaluated.
Type	Specifies whether the entry in the Measurement column is a Trace or a Measurement expression.
Min Value	Lists the minimum value of the measurement expression, obtained after the sweep analysis is complete.
Max Value	This column is populated after the sweep analysis is complete. Lists the maximum value of the measurement expression.

Parameteric Plotter Example

In this section, you will use Parametric Plotter to evaluate a simple test circuit for inductive switching. This circuit is created using a power mosfet from the `PWRMFET.OLB`.

The design example is available at `..\tools\pspice\tutorial\capture\pspiceaa\snubber`.

Add two voltage markers added to the circuit as shown in Figure 12-3 on page 159, are used to plot the input and the output voltages.

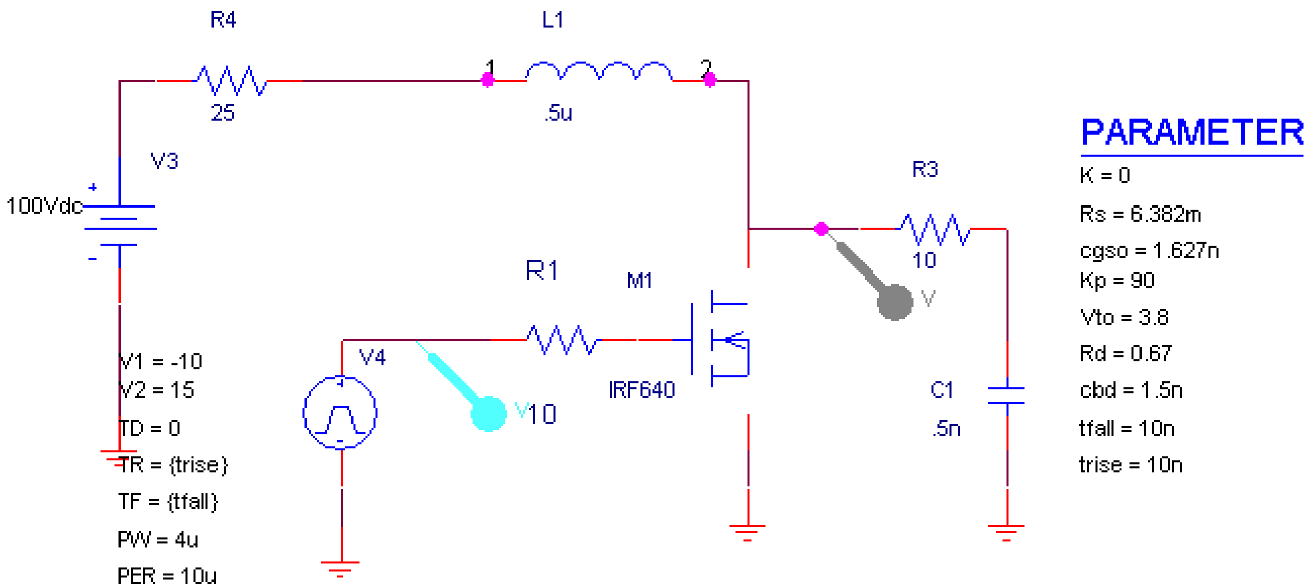


Figure 12-2 Inductive switching circuit

To view the input and the output voltages, you first need to simulate the circuit.

Simulating the circuit

– From the PSpice menu in schematic editor, select Run.

The input and the output waveforms are displayed in Figure 12-3. The output waveform displays a spike at every falling edge of the input waveform.

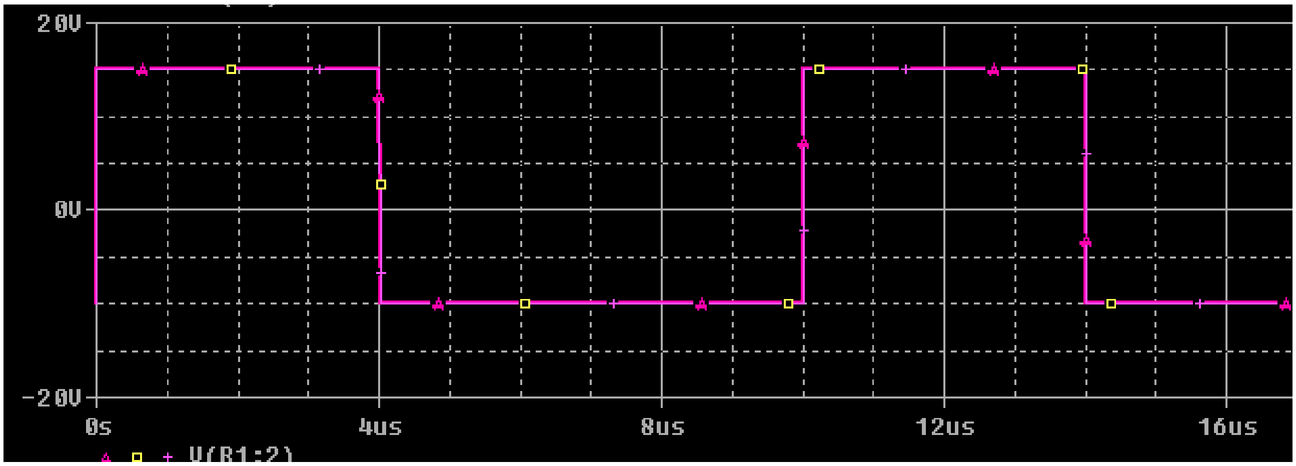


Figure 12-3 Input waveform

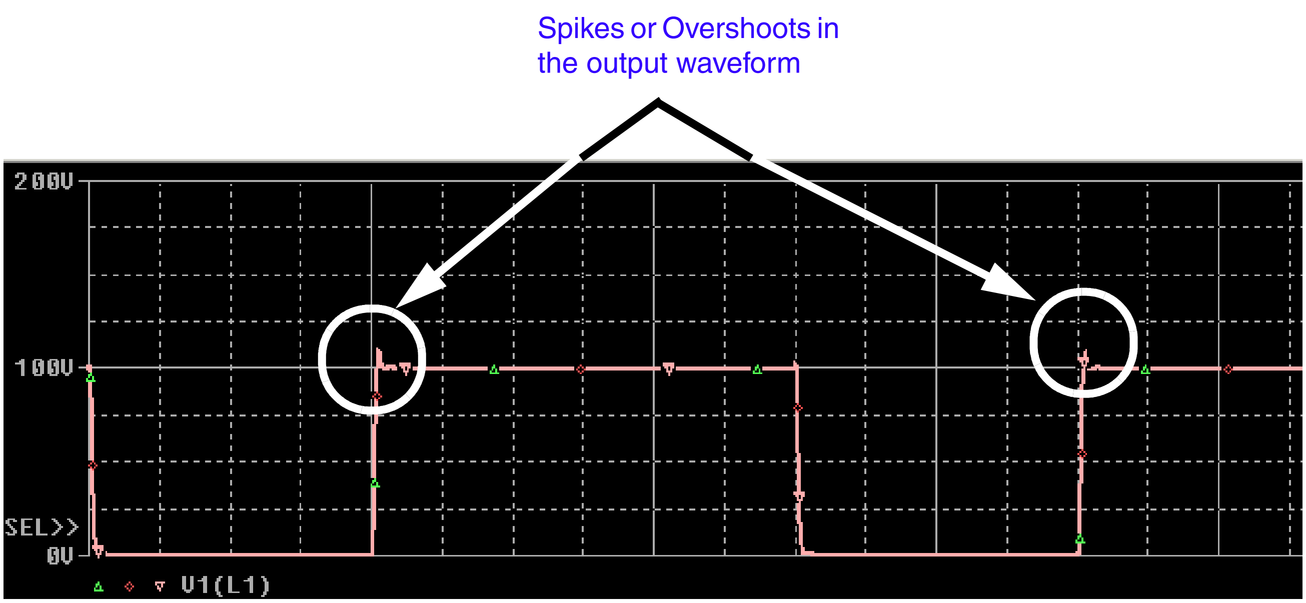


Figure 12-4 Output Waveform

Before users can use the output waveform, they need to adjust the circuit components so as to reduce the overshoot within the limit acceptable to the user. This can easily be done by increasing the values of resistor R3 and capacitor C1. But this results in increasing power dissipation across resistor R3.

Therefore, the design challange here is to balance the power dissipation and the voltage overshooot.

To find an acceptable soultion to the problem, we will vary the values of resistance R3, capacitor C1, and rise time of the input pulse and monitor the effect of varying the parameter values on the overshoot and the power dissipation across resistor R3.

To achieve this, use Parametric Plotter to run the sweep analysis. Before you can run the sweep analysis, complete the following sequence of steps.

1. Launch Parametric Plotter
2. Add sweep parameters
3. Add measurements
4. Run sweep analysis

Launch Parametric Plotter

From the PSpice menu in the schematic editor, select Advanced Analysis > Parametric Plot.

Add sweep parameters

For the switching circuit design, we will vary `trise` linearly, specify discrete values for `R3`, and vary `C1` logarithmically.

1. In the Sweep Parameters window, click the *Click here to import a parameter from the design property map* row.
2. In the Sweep Parameters window, select the parameter named `trise` and click inside the corresponding Sweep Type grid.
3. From the drop-down list, select Linear.
4. To specify the range within which the parameter values should be varied, click corresponding Sweep Values grid.
5. In the Sweep Settings dialog box, specify start value as `5n`, stop value as `12n` and the step value as `1n`.
   This implies that the rise time of the pulse will ve varied from 5 nano seconds to 12 nano seconds.
6. To add resistor R3 as the next sweep parameter, click the sweep type grid corresponding to the component named R3.
7. From the drop-down list, select Discrete.
8. To specify the values of resitor R3, click corresponding Sweep Values grid.
9. To specify a discrete value for resistor R3, click the New button and enter 5.

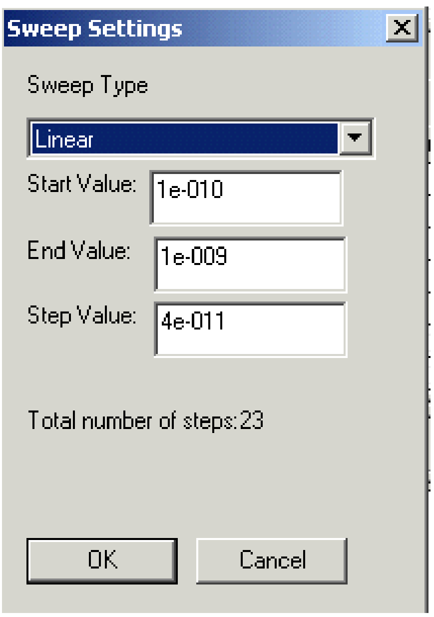
10 Similarly, specify other values as 15 and 20.

11 Click OK to close the Sweep Settings dialog box.

12 Finally, to add capacitor C1 as a sweep parameter and vary the capacitance value, click the sweep type grid corresponding to capacitor C1 and select Linear from the drop-down list.

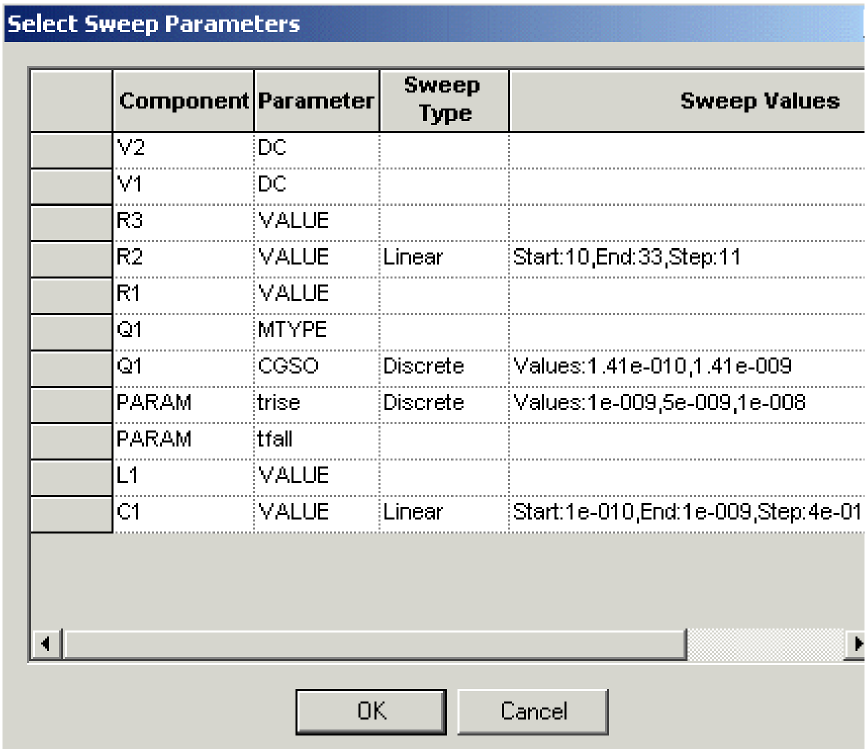
13 Click the Sweep Values grid.

14 In the Sweep Settings dialog box, specify theStart Value as .1n, End value as 1n, and number of points as 10, and click OK.

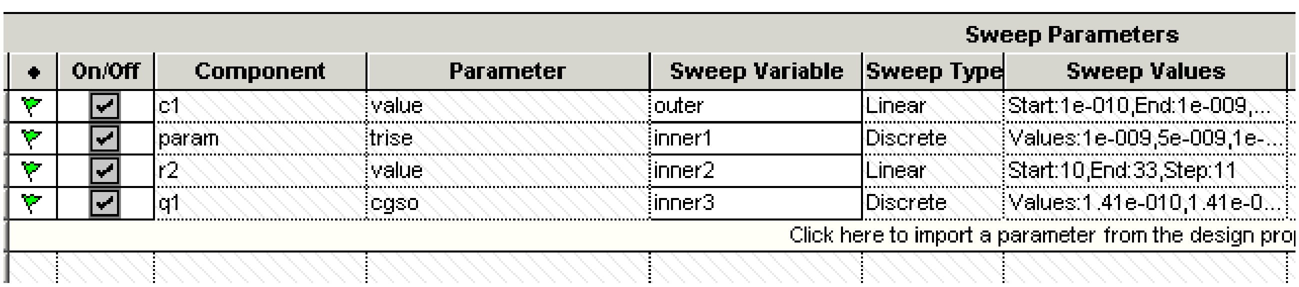


This implies that the sweep analysis will be performed for 10 values of capacitance between .1 nano farads to 1 nano farads.

15 In the Select Sweep Parameters dialog box, click OK to save your changes.



The changes are reflected in the Sweep Parameters window.



Besides the values entered by you in the Select Sweep Parameters dialog box, the Sweep Variable column also gets populated. Parametric Plotter assigns variables to the parameters depending on the order in which they are added. If required you can change this order.

Add measurements

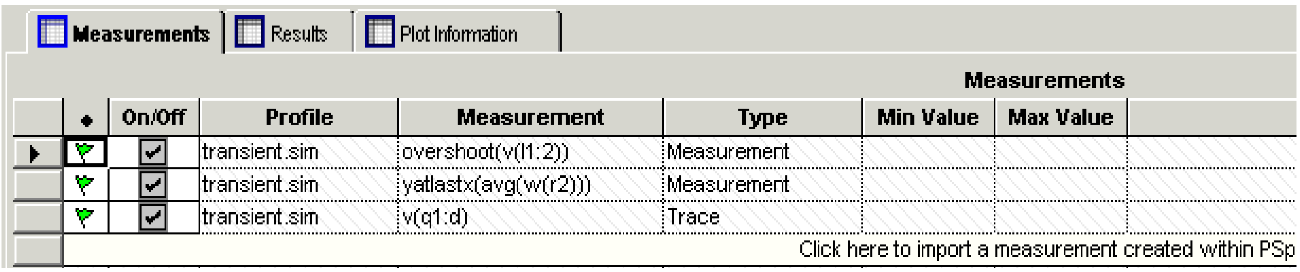
To evalute the influence of varying parameter values on the overshoot and power disspipation across resistor R3, and to include a trace, add these three as the measurement expressions to be evaluated.

1 In the Measurements tab, select Click here to add a measurement created in PSpice row.

2 In the Import Measurement(s) dialog box, select `Overshoot(V(l1:2)), yatlastX(AVG(W(R2))), and v(q1:d)` from the `transient.sim` profile.

3 Click OK.

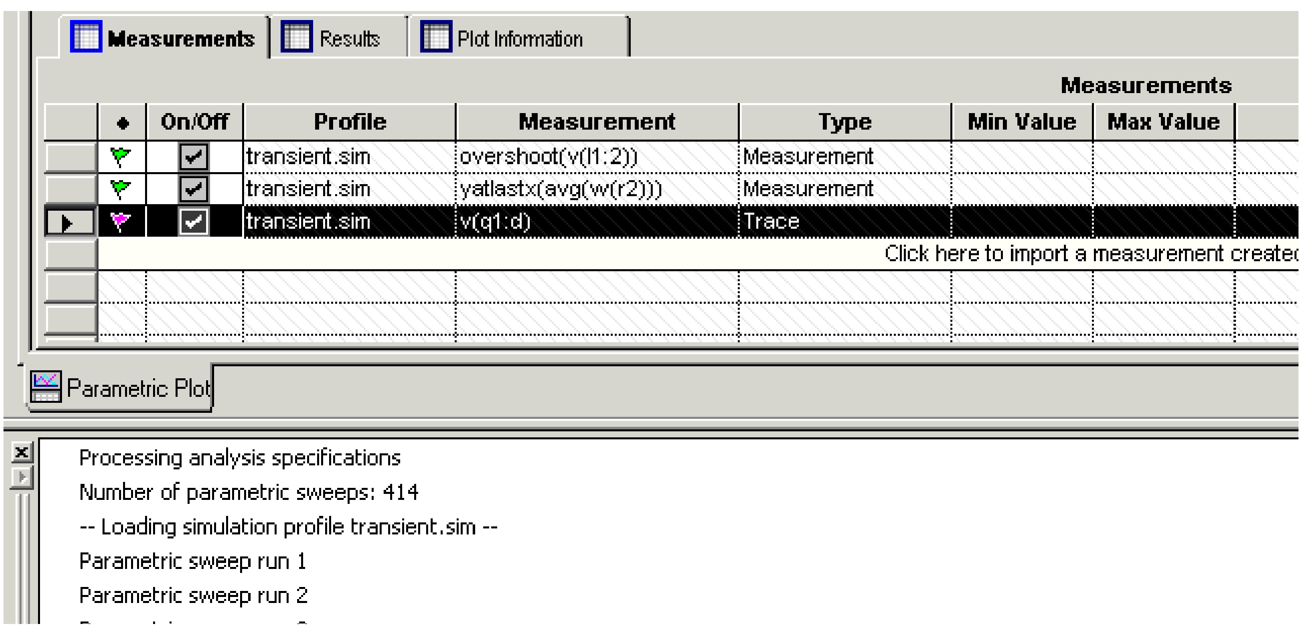
The measurements get added to the Measurements tab.



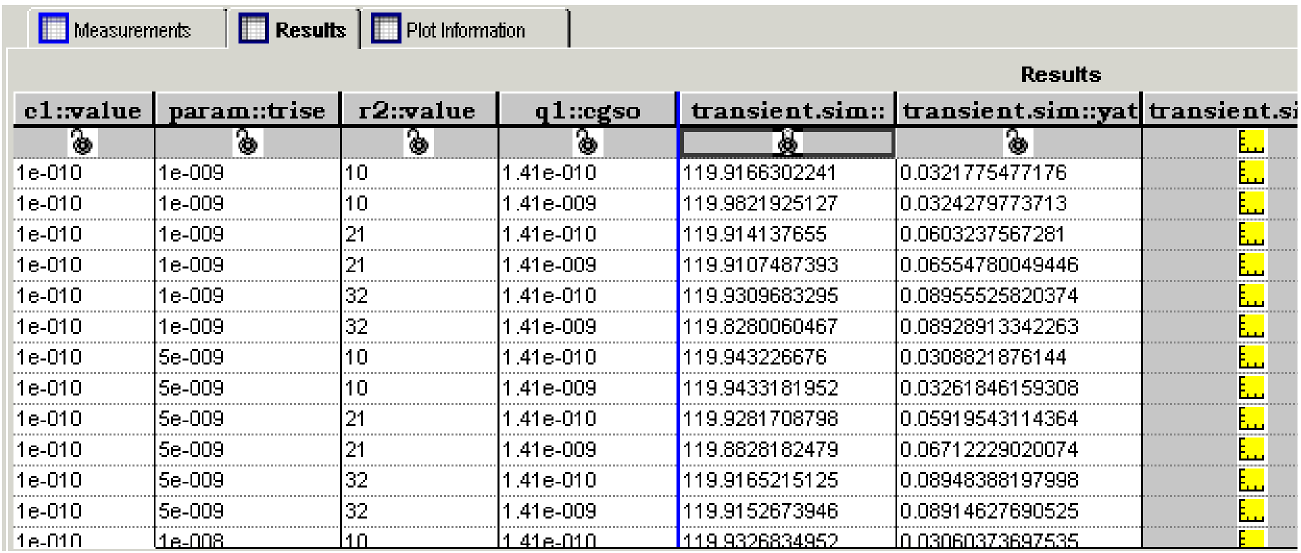
Run sweep analysis

– To run the sweep analysis, click the Start  button on the toolbar.

As Parametric Plotter starts running the Output window is populated with the total number of sweeps required to complete the analysis.



Once the analysis is over, the Min value and the Max Value columns are populated for each measurement specified in the Measurements tab. Besides this, results of each run of Parametric Plotter are displayed in the Results tab.



				Results		
c1::value	param::trise	r2::value	q1::cgso	transient.sim::	transient.sim::yat	transient.si
1e-010	1e-009	10	1.41e-010	119.9166302241	0.0321775477176	
1e-010	1e-009	10	1.41e-009	119.9821925127	0.0324279773713	
1e-010	1e-009	21	1.41e-010	119.914137655	0.0603237567281	
1e-010	1e-009	21	1.41e-009	119.9107487393	0.06554780049446	
1e-010	1e-009	32	1.41e-010	119.9309683295	0.08955525820374	
1e-010	1e-009	32	1.41e-009	119.8280060467	0.08928913342263	
1e-010	5e-009	10	1.41e-010	119.943226676	0.0308821876144	
1e-010	5e-009	10	1.41e-009	119.9433181952	0.03261846159308	
1e-010	5e-009	21	1.41e-010	119.9281708798	0.05919543114364	
1e-010	5e-009	21	1.41e-009	119.8828182479	0.06712229020074	
1e-010	5e-009	32	1.41e-010	119.9165215125	0.08948388197998	
1e-010	5e-009	32	1.41e-009	119.9152673946	0.08914627690525	
1e-010	1e-008	10	1.41e-010	119.9326834952	0.03060373697535	

Figure 12-5 Results tab in Parametric Plotter

In the Results tab, you can sort and lock the results displayed in various columns. For example, consider that in case of the inductive switching circuit, your primary goal is to restrict the power loss, which is measured by `yatlastx(avg(w(r2))`, to less than `0.006`, and then minimize the overshoot.

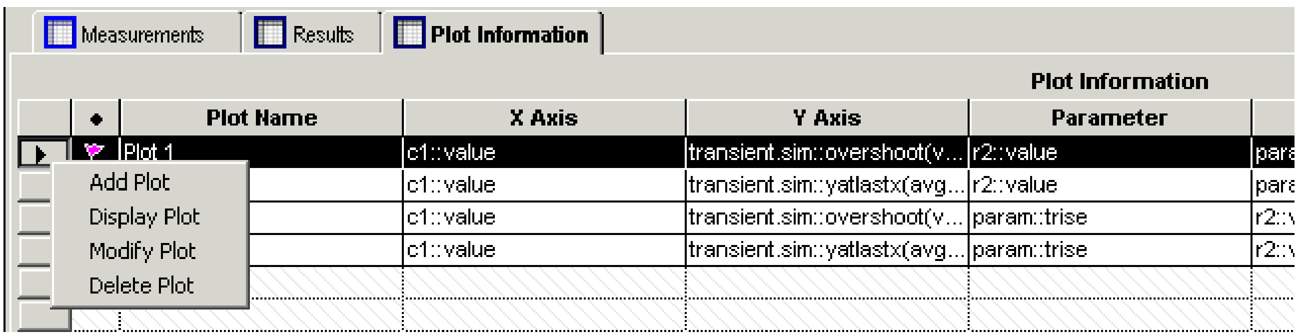
To achieve your goal, first sort the values displayed in the sixth column of Figure 12-5 on page 164. To sort the values, double-click on the column heading. The values get assorted in the ascending order. Next you lock the sorted values. To lock the values, click the lock icon on the top of the column.

After sorting the power loss values, sort the values displayed in the fifth column of Figure 12-5 on page 164. As a result of this sorting the values in the last column do not get disturbed. As a result, for all values of `atlastx(avg(w(r2))`, to less than `0.006`, the overshoot values get sorted. Thus you can view the combination(s) of the parameter values for which both the outputs are in the desired range.

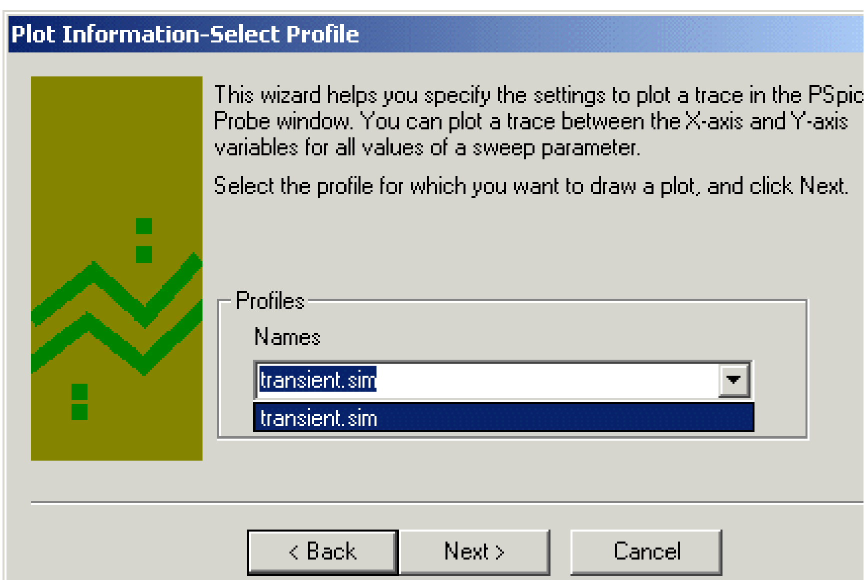
Add Plot

You can plot a trace between the X-axis and Y-axis variables for all values of a sweep parameter by using the Plot wizard. This wizard helps you specify the settings to plot a trace in the PSpice Probe window.

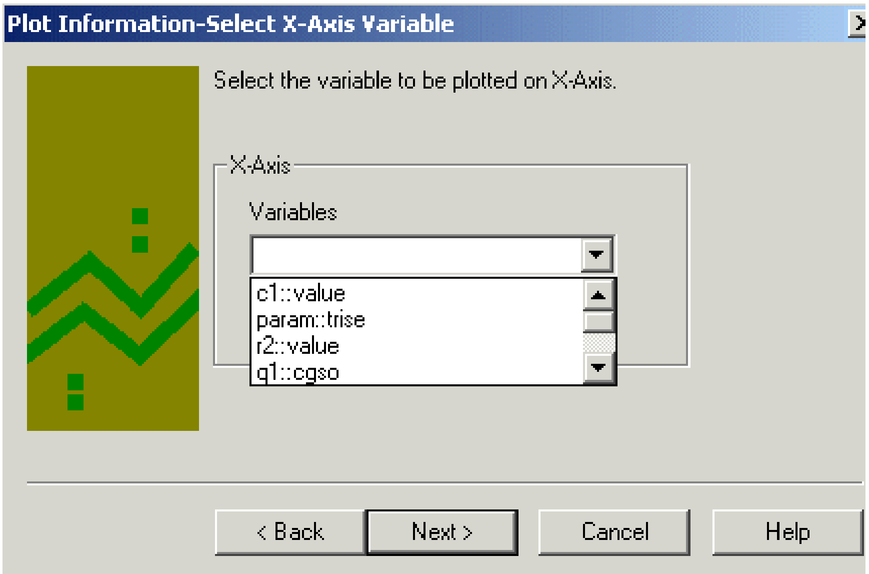
1 In the Plot Information tab, right-click in the plot information row and then click Add Plot. This displays the Plot wizard.



2 Select the `transient.sim` profile, and click Next.

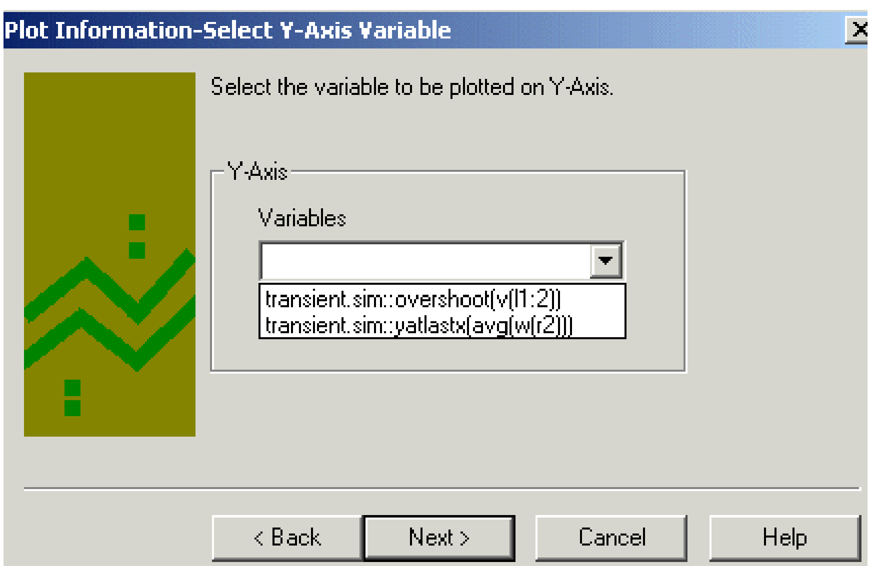


3 Select `r2::value` as the variable to be plotted on the X-axis, and click Next.

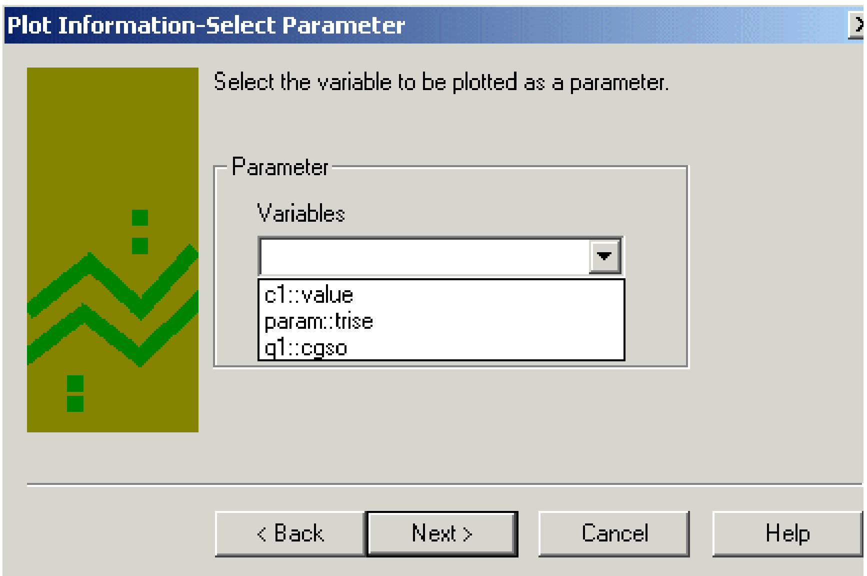


Note: If you select a Parameter or Measurement variable to be plotted on the X-axis, you will only be allowed to select a “Measurement” variable to be plotted on the Y-axis. If you select Time/Frequency variable, the wizard will only display a list of available traces that can be plotted on the Y-axis.

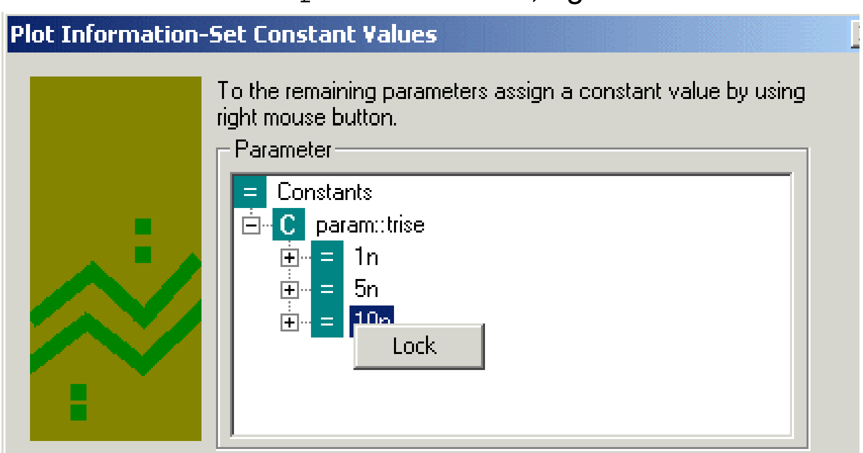
4 Select `transient.sim::overshoot(v[l1:2])` as the variable to be plotted on the Y-axis, and click Next.



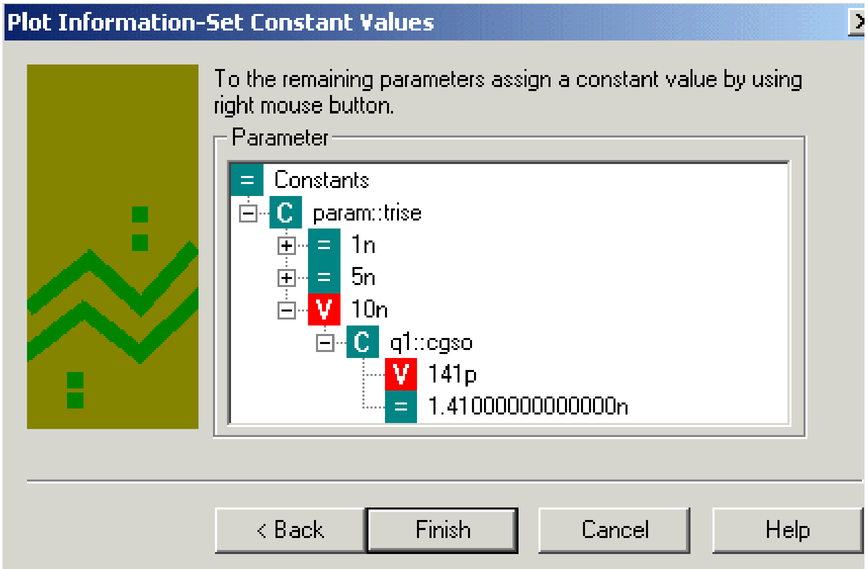
5 Select `c1::value` as the parameter to be varied, such that for each possible value of this parameter, you have a unique x-y trace, and click Next.



6 The remaining sweep parameters and their possible values are listed. For each parameter, select a constant value to be used for drawing the trace(s). To assign a constant valueto `param::trise`, right-click on `10n` and lock it.

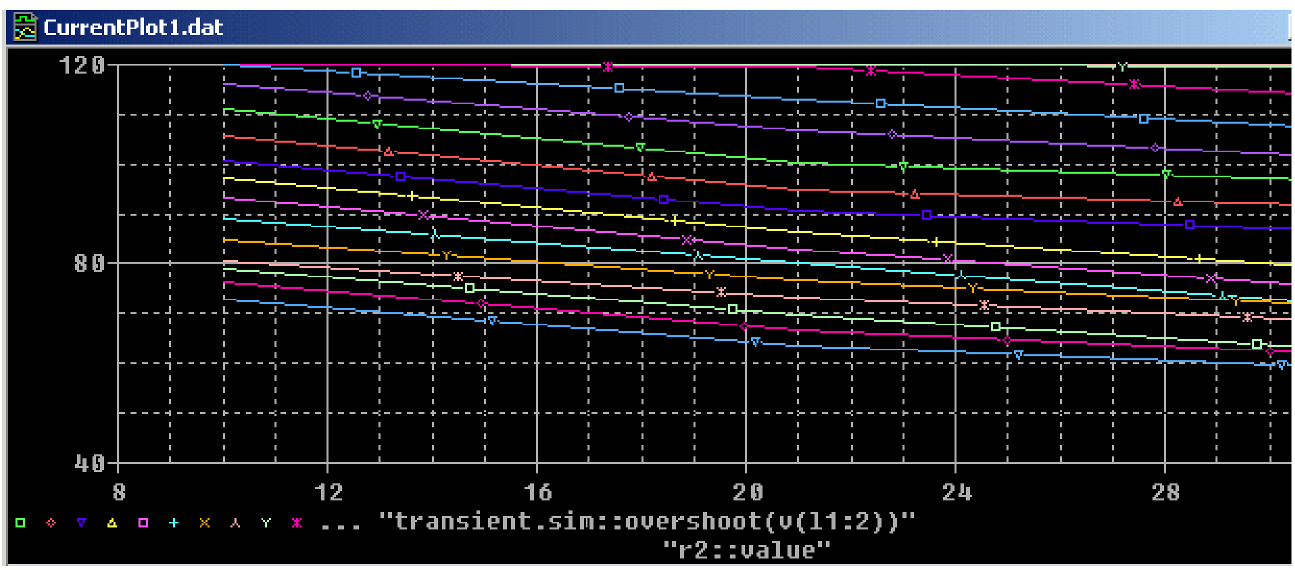


7 Similarly assign a constant value to q1::cgso. Click Finish.



8 In the Plot Information tab, right-click in the plot information row and then click Display Plot. This displays the trace that you plotted.

.



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(v(out))

Measurement strategy

- Start with a circuit created in the schematic editor and a working PSpice[1] 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.

You can see the following for more information:

List of measurement definitions included in PSpice	Measurement definitions included in PSpice
How to create custom measurement expressions	Composing custom measurement expressions
How to create custom measurement definitions	Creating custom measurement definitions

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

Creating measurement expressions

Before you create a measurement expression to use in Advanced Analysis:

1 Design a circuit in the schematic editor.

2 Set up a PSpice simulation.

The Advanced Analysis tools use these simulations:

- Time Domain (transient)
- DC Sweep
- AC Sweep/Noise

3 Run the circuit in PSpice.

Make sure the circuit is valid and you have the results you expect.

Composing a measurement expression

These steps show you how to create a measurement expression in PSpice. Measurement expressions created in PSpice can be imported into Sensitivity, Optimizer, and Monte Carlo.

You can also create measurements while in Sensitivity, Optimizer, and Monte Carlo, but those measurements cannot be imported into PSpice for testing.

First select a measurement definition, and then select output variables to measure. The two combined become a measurement expression.

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

1 From the **Trace** menu in PSpice, select **Measurements**.

The **Measurements** dialog box appears.

2 Select the measurement definition you want to evaluate.

3 Click **Eval** (evaluate).

The **Arguments for Measurement Evaluation** dialog box appears.

4 Click the **Name of trace to search** button.

The **Traces for Measurement Arguments** dialog box appears.

Note: You will only be using the Simulation Output Variables list on the left side. Ignore the Functions or Macros list.

5 Uncheck the output types you don’t need (if you want to simplify the list).

6 Click on the output variable you want to evaluate.

The output variable appears in the **Trace Expression** field.

7 Click **OK**.

The **Arguments for Measurement Evaluation** dialog box reappears with the output variable you chose in the **Name of trace to search** field.

8 Click **OK**.

Your new measurement expression is evaluated and displayed 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 a numerical evaluation by following the steps in the topic, Viewing the results of measurement evaluations.

Example: Composing a measurement expression

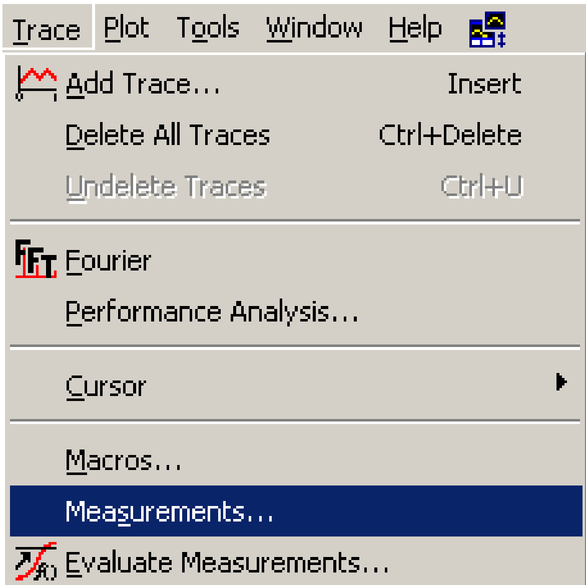
First you select a measurement definition, and then you select an output variable to measure. The two combined become a measurement expression.

Note: For the current design example, work in the Simulation Results view in PSpice.

1 In the side toolbar, click on  .

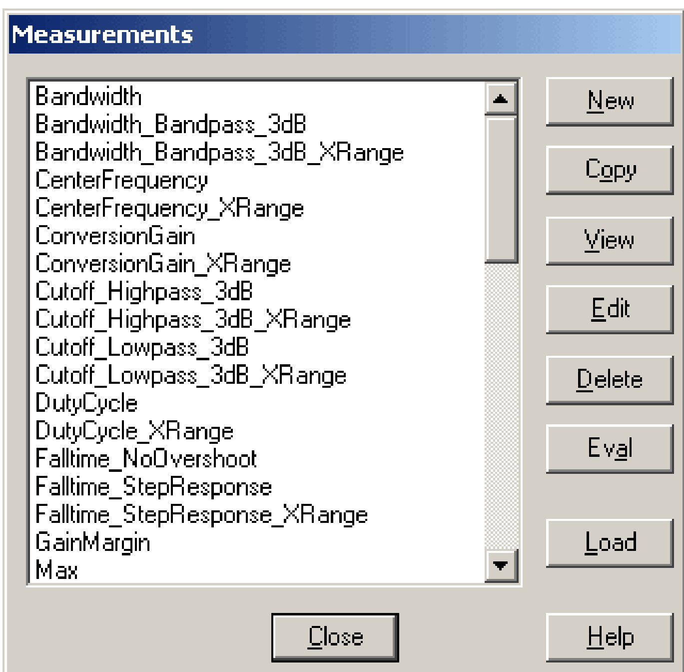
2 From the **Trace** menu in PSpice, select **Measurements**.

The **Measurements** dialog box appears.



3 Select the measurement definition you want to evaluate.

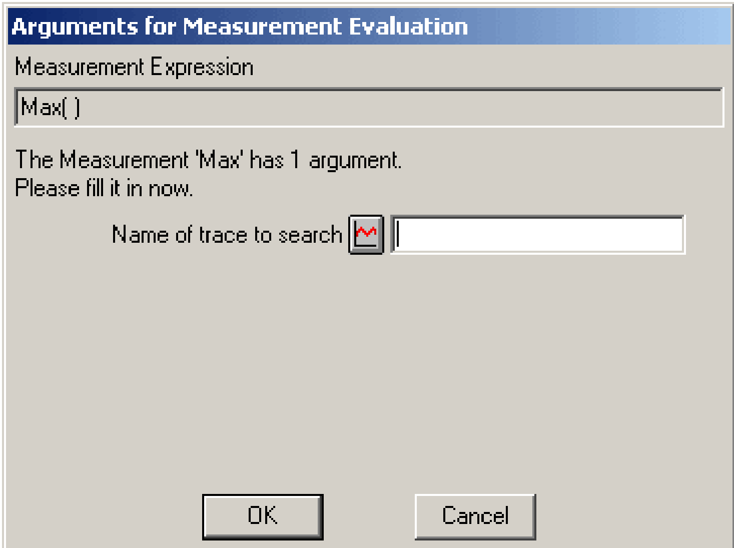
4 Click **Eval** (evaluate).



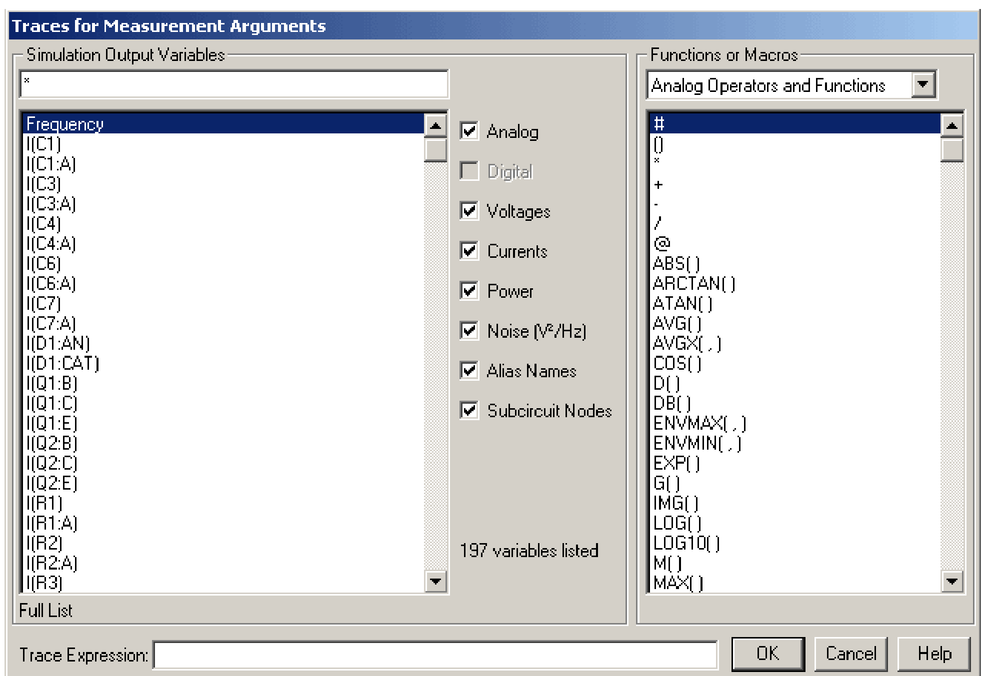
The **Arguments for Measurement Evaluation** dialog box appears.

5 Click the **Name of trace to search** button.

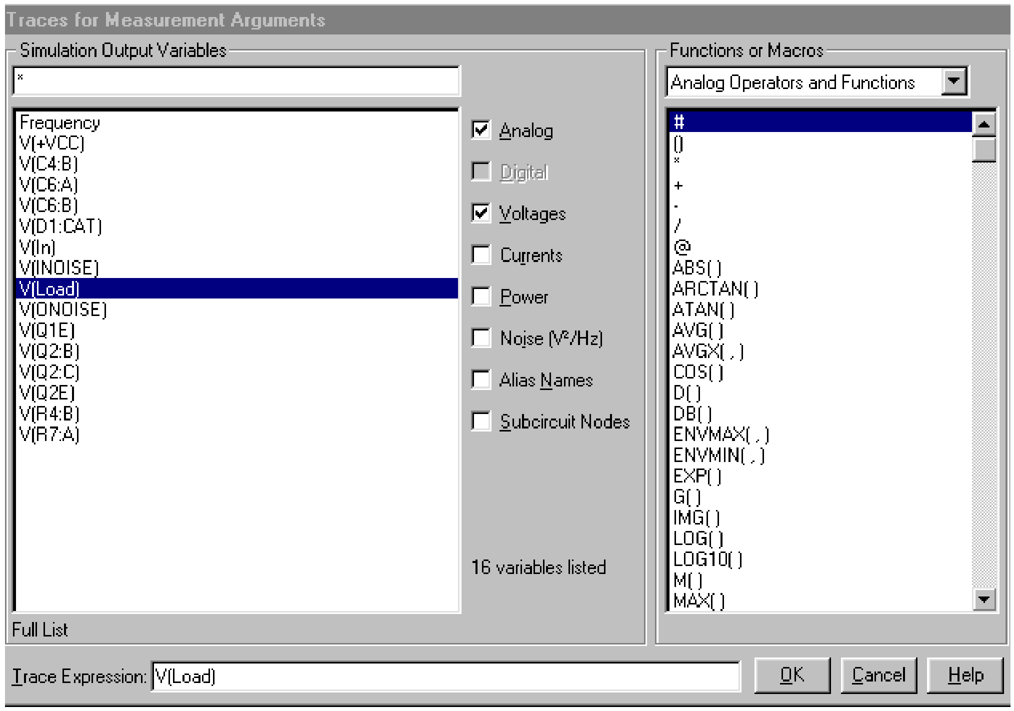
The **Traces for Measurement Arguments** dialog box appears.



Note: You will only be using the Simulation Output Variables list on the left side. Ignore the Functions or Macros list.



6 Uncheck the output types you don't need (if you want to simplify the list).

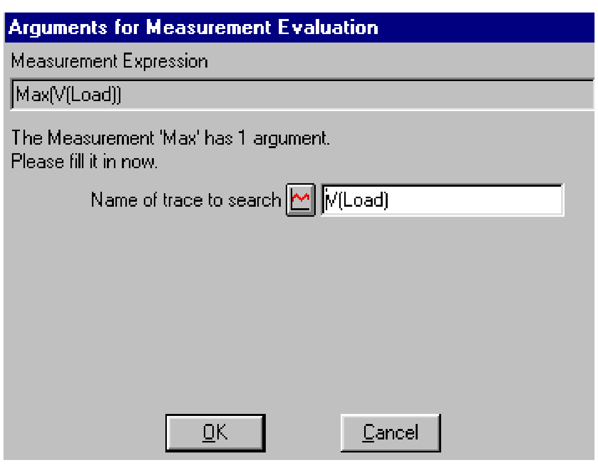


7 Click on the output variable you want to evaluate.

The output variable appears in the **Trace Expression** field.

8 Click **OK**.

The **Arguments for Measurement Evaluation** dialog box reappears with the output variable you chose in the **Name of trace to search** field.

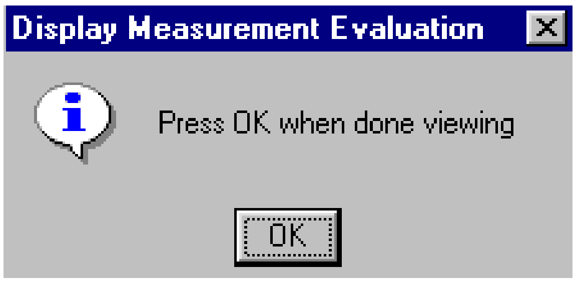


9 Click **OK**.

Your new measurement expression is evaluated and displayed 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 does not display in the window. The only way to get another graphical display is to redo these steps. You can see a numerical evaluation by following the next steps.



11 Click **Close**.

Viewing the results of measurement evaluations

1 From the **View** menu in PSpice, select **Measurement Results**.

The **Measurement Results** table displays below the plot window.

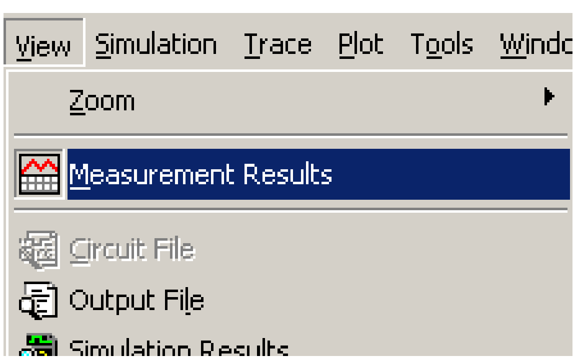
2 Click the box in the **Evaluate** column.

The PSpice calculation for your measurement expression appears in the **Value** column.

Example; Viewing the results of measurement evaluations

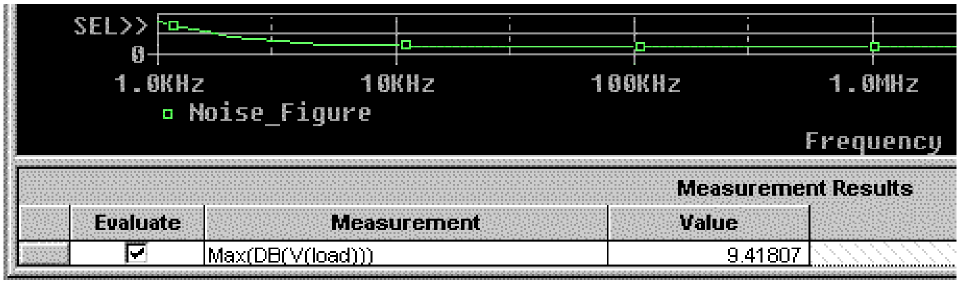
1 From the **View** menu, select **Measurement Results**.

The **Measurement Results** table displays below the plot window.



2 Click the box in the **Evaluate** column.

A checkmark appears in the **Evaluate** column checkbox and the PSpice calculation for your measurement expression appears in the **Value** column.



Measurement definitions included in PSpice

Definition	Finds the. . .
Bandwidth	Bandwidth of a waveform (you choose dB level)
Bandwidth_Bandpass_3dB	Bandwidth (3dB level) of a waveform
Bandwidth_Bandpass_3dB_XRange	Bandwidth (3dB level) of a waveform over a specified X-range
CenterFrequency	Center frequency (dB level) of a waveform
CenterFrequency_XRange	Center frequency (dB level) of a waveform over a specified X-range
ConversionGain	Ratio of the maximum value of the first waveform to the maximum value of the second waveform
ConversionGain_XRange	Ratio of the maximum value of the first waveform to the maximum value of the second waveform over a specified X-range
Cutoff_Highpass_3dB	High pass bandwidth (for the given dB level)
Cutoff_Highpass_3dB_XRange	High pass bandwidth (for the given dB level)
Cutoff_Lowpass_3dB	Low pass bandwidth (for the given dB level)

Definition	Finds the. . .
Cutoff_Lowpass_3dB_XRange	Low pass bandwidth (for the given dB level) over a specified range
DutyCycle	Duty cycle of the first pulse/period
DutyCycle_XRange	Duty cycle of the first pulse/period over a range
Falltime_NoOvershoot	Falltime with no overshoot.
Falltime_StepResponse	Falltime of a negative-going step response curve
Falltime_StepResponse_XRange	Falltime of a negative-going step response curve over a specified range
GainMargin	Gain (dB level) at the first 180-degree out-of-phase mark
Max	Maximum value of the waveform
Max_XRange	Maximum value of the waveform within the specified range of X
Min	Minimum value of the waveform
Min_XRange	Minimum value of the waveform within the specified range of X
NthPeak	Value of a waveform at its nth peak
Overshoot	Overshoot of a step response curve
Overshoot_XRange	Overshoot of a step response curve over a specified range
Peak	Value of a waveform at its nth peak
Period	Period of a time domain signal
Period_XRange	Period of a time domain signal over a specified range
PhaseMargin	Phase margin
PowerDissipation_mW	Total power dissipation in milli-watts during the final period of time (can be used to calculate total power dissipation, if the first waveform is the integral of V(load)

Definition	Finds the. . .
Pulsewidth	Width of the first pulse
Pulsewidth_XRange	Width of the first pulse at a specified range
Q_Bandpass	Calculates Q (center frequency / bandwidth) of a bandpass response at the specified dB point
Q_Bandpass_XRange	Calculates Q (center frequency / bandwidth) of a bandpass response at the specified dB point and the specified range
Risetime_NoOvershoot	Risetime of a step response curve with no overshoot
Risetime_StepResponse	Risetime of a step response curve
Risetime_StepResponse_XRange	Risetime of a step response curve at a specified range
SettlingTime	Time from <begin_x> to the time it takes a step response to settle within a specified band
SettlingTime_XRange	Time from <begin_x> to the time it takes a step response to settle within a specified band and within a specified range
SlewRate_Fall	Slew rate of a negative-going step response curve
SlewRate_Fall_XRange	Slew rate of a negative-going step response curve over an X-range
SlewRate_Rise	Slew rate of a positive-going step response curve
SlewRate_Rise_XRange	Slew rate of a positive-going step response curve over an X-range
Swing_XRange	Difference between the maximum and minimum values of the waveform within the specified range
XatNthY	Value of X corresponding to the nth occurrence of the given Y_value, for the specified waveform

Definition	Finds the. . .
XatNthY_NegativeSlope	Value of X corresponding to the nth negative slope crossing of the given Y_value, for the specified waveform
XatNthY_PercentYRange	Value of X corresponding to the nth occurrence of the waveform crossing the given percentage of its full Y-axis range; specifically, nth occurrence of Y=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

  So it will come when it's called.

- A marked point expression

These are the calculations that compute 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 waveform.
5. Test the search commands and measurements.

Note: An easy way to create a new definition:

From the PSpice **Trace** menu, select **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.

Writing a new measurement definition

To write a new measurement definition:

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.

   Make sure **local file** is selected.

   This stores the new measurement in a .prb file local to the design.

4. Click **OK**.

   The **Edit New Measurement** dialog box appears.

5 Type in the marked expression.

6 Type in any comments you want.

7 Type in the search function.

Your new measurement definition is now listed in the **Measurements** dialog box.

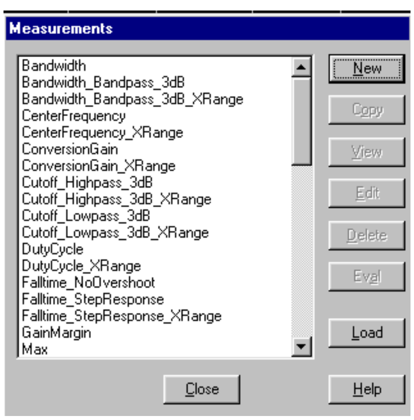
You can see the following for more information:

How to create custom measurement expressions	Composing custom measurement expressions
Syntax	Measurement definition syntax

Example: Writing a new measurement definition

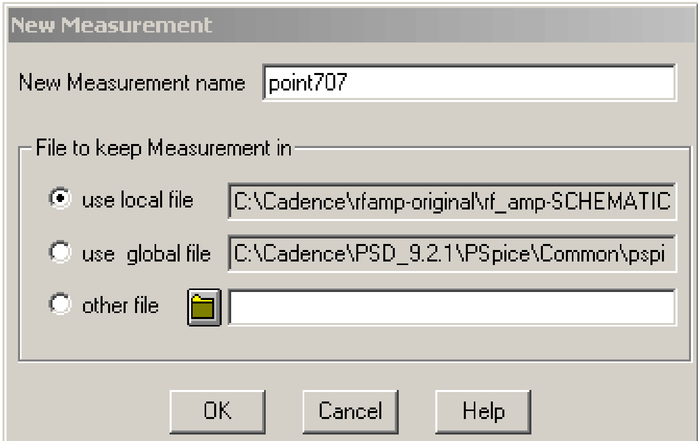
1 From the PSpice or PSpice **Trace** menu, choose **Measurements**.

The **Measurements** dialog box appears.



2 Click **New**.

The **New Measurement** dialog box appears.



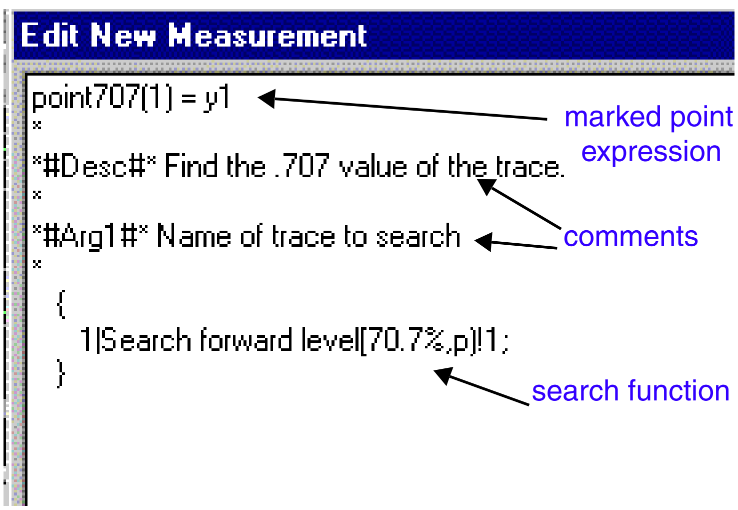
3 Type in a name in the **New Measurement name** field.

4 Make sure **use 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 expression:

point707(1) = y1

7 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.

8 Type in any explanatory comments you want:

```
*
*#Desc#* Find the .707 value of the trace.
*
*#Arg1#* Name of trace to search
*
```

Your new measurement definition is now listed in the Measurements dialog box.

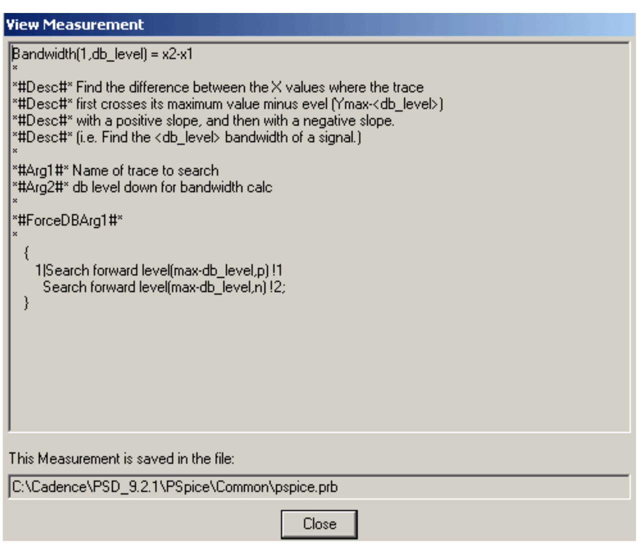
Using measurement definition syntax

Check out the existing measurement definitions in PSpice for syntax examples.

1 From the **Trace** menu, choose **Measurements**.

The **Measurement** dialog box appears.

2 Highlight any example, and select **View** to examine the syntax.



You can see the following for more information:

Syntax rules for measurement name syntax	Measurement name syntax
Syntax rules for comments	Comments syntax
Syntax rules for marked point expressions	Marked point expression syntax
Syntax rules for search commands	Search command syntax

Measurement definition: fill in the place holders

measurement_name (1, [2, …, n][, subarg1, subarg2, …, subargm]) =
marked_point_expression

{

1| 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;

}

Measurement name syntax

Can contain any alphanumeric character (A-Z, 0-9) or underscore _ , up to 50 characters in length. The first character should be an upper or lower case letter.

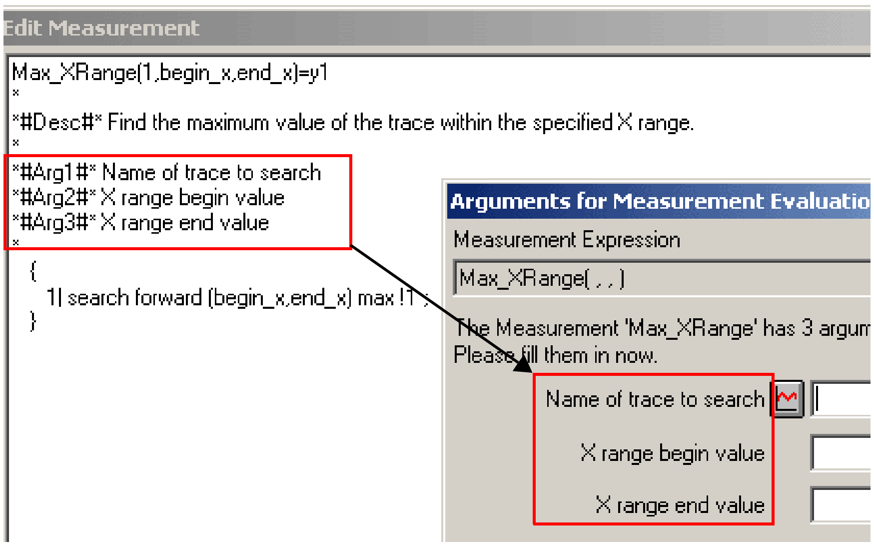
Examples of valid function names: Bandwidth, CenterFreq, delay_time, DBlevel1.

Comments syntax

A comment line always starts with an asterisk. Special comment lines include the following examples:

#Desc#	The measurement description
#Arg1#	Description of an argument used in the measurement definition.

These comment lines will be used in dialog boxes, such as the **Arguments for Measurement Evaluation** box.



Marked point expressions 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 expressions 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.

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.

Search command syntax

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]

forward or backward

The direction of the search. Search commands can specify either a forward or reverse direction. The search begins at the origin of the curve.

[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

Use this…	To start the search at this…
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_points#]` 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.

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

This range…	Means this…
(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:]

Specifies which occurrence of *<condition>* to find.

If repeat is greater than the number of found instances of *<condition>*, then the last *<condition>* found is used.

Example

The argument `2:LEvel` would find the second level crossing.

<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 expression is not evaluated.

LEvel(vahlue[,posneg])

[,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.

(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 below startvalue .+3db ⇒ 3db above startvalue
a value relative to max or min	max-3 ⇒ maxrng -3 min+3 ⇒ minrng +3
a db value relative to max or min	max-3db ⇒ 3db below maxrng min+3db ⇒ 3db above minrng

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 expression. If this is the first search command, then it represents the Y value of the startpoint of the search.

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	x1 y1
or an expression of marked points	(x1+x2)/2
a value relative to startvalue	.-3 ⇒ startvalue -3 .+3 ⇒ startvalue +3
a db value relative to startvalue	.-3db ⇒ 3db below startvalue .+3db ⇒ 3db above startvalue
a value relative to max or min	max-3 ⇒ maxrng -3 min+3 ⇒ minrng +3

Example: Using measurement definition syntax

The measurement definition is made up of:

- A measurement name
- A marked point expression
- One or more search commands enclosed within curly braces

This example also includes comments about:

- The measurement definition
- What arguments it expects when used
- A sample command line for its usage

Any line beginning with an asterisk is considered a comment line.

Risetime definition

```
Risetime(1) = x2-x1
    *
    *#Desc#* Find the difference between the X values
    *#Desc#* where the trace first crosses 10% and then
    *#Desc#* 90% of its maximum value with a positive
    *#Desc#* slope.
    *#Desc#* (i.e. Find the risetime of a step response
    *#Desc#* curve with no overshoot. If the signal has
    *#Desc#* overshoot, use GenRise().)
    *
    *#Arg1#* Name of trace to search
    *
    * Usage:
    *Risetime(<trace name>)
    *
       {
          1|Search forward level(10%, p) !1
            Search forward level(90%, p) !2;
       }
```

The name of the measurement is Risetime. Risetime will take 1 argument, a trace name (as seen from the comments).

The first search function searches forward (positive x direction) for 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 x2-x1. This means the measurement calculates the X value of point 2 minus the X value of point 1 and returns that number.

Advanced Analysis Engines

You can see the following for more information:

How to select an engine	Selecting an engine
Engine overview	Optimizer Engine Overview

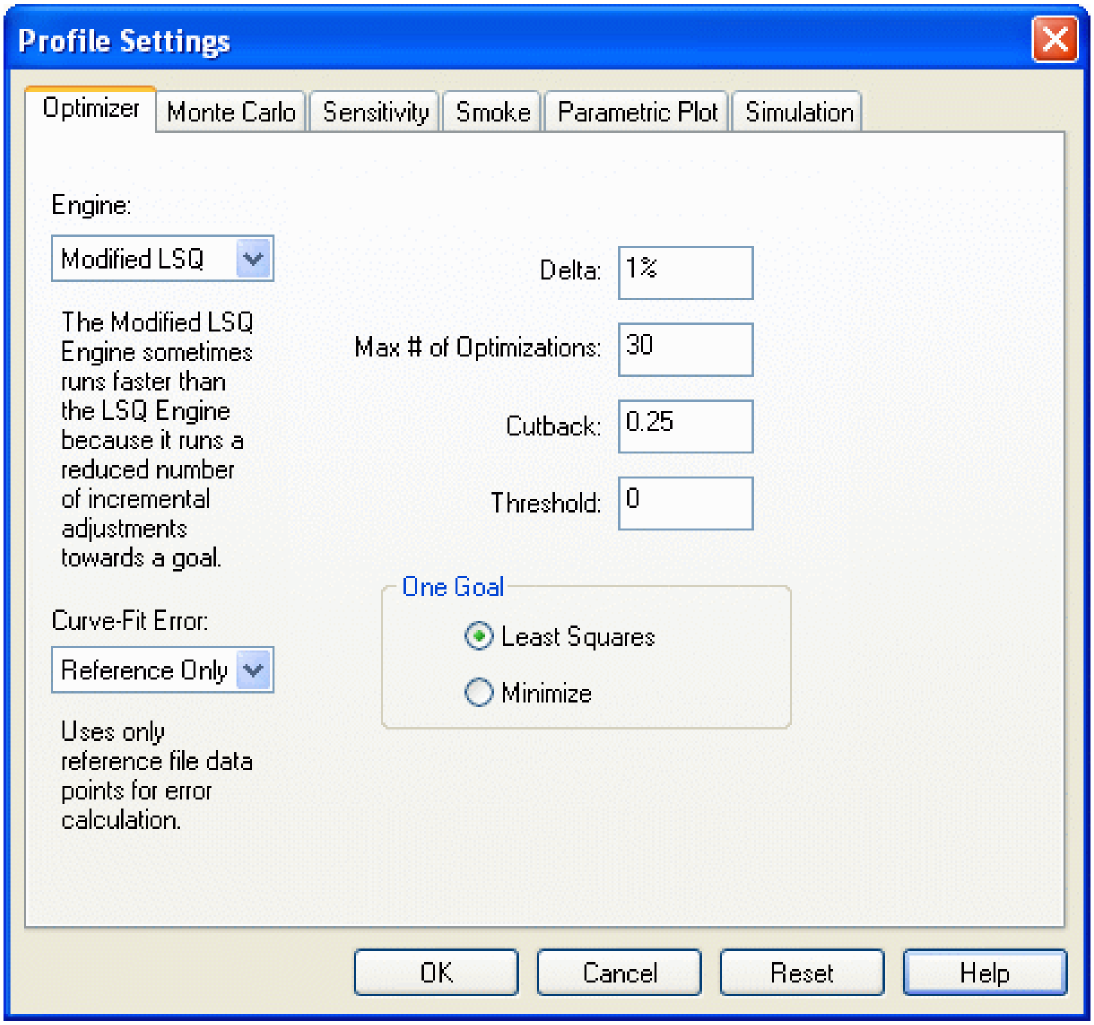
The Modified LSQ engine

The Modified LSQ engine uses both constrained and unconstrained minimization algorithms, which allow it to optimize goals subject to nonlinear constraints. The Modified LSQ engine runs faster than the LSQ engine because it runs a reduced number of incremental adjustments toward the goal.

Configuring the Modified LSQ engine

1. From the Advanced Analysis **Edit** menu, select **Profile Settings**.
2. Click the **Optimizer** tab.

3 From the **Engine** drop-down list, select **Modified LSQ**.



4 Edit default values in the text boxes.

See detailed explanations provided on the next few pages.

5 Select the **One Goal** option that you prefer: **Least Squares** or **Minimize**.

See Single goal optimization settings for details.

6 Click **OK**.

The Modified LSQ engine options

Modified LSQ Engine Options	Function	Default Value
Delta	The relative amount (as a percentage of current parameter value) the engine moves each parameter from the proceeding value when calculating the derivatives.	1%
Max # of Optimizations	The most attempts the Modified LSQ Engine should make before *giving up* on the solution (even if making progress).	20
Cutback	The minimum fraction by which an internal step is reduced while the Modified LSQ Engine searches for a reduction in the goal's target value. If the data is noisy, consider increasing the Cutback value from its default of 0.25.	0.25
Threshold	The minimum step size the Modified LSQ engine uses to adjust the optimization parameters.	0

Delta calculations

The optimizer uses gradient-based optimization algorithms that use a finite difference method to approximate the gradients (gradients are not known analytically). To implement finite differencing, the Modified LSQ engine:

1. Moves each parameter from its current value by an amount Delta.
2. Evaluates the function at the new value.
3. Subtracts the old function value from the new.
4. Divides the result by Delta.

Note: There is a trade-off. If Delta is too small, the difference in function values is unreliable due to numerical inaccuracies. However if Delta is too large, the result is a poor approximation to the true gradient.

Editing Delta

Enter a value in the Delta text box that defines a fraction of the parameter's total range.

> Example: If a parameter has a current value of 10^{-8}, and Delta is set to 1% (the default), then the Modified LSQ Engine moves the parameter by 10^{-10}.

The 1% default accuracy works well in most simulations.

If the accuracy of your simulation is very different from typical (perhaps because of the use of a non-default value for either RELTOL or the time step ceiling for a Transient analysis), then change the value of Delta as follows:

- If simulation accuracy is better, smaller adjustments are needed; decrease Delta by an appropriate amount.
- If simulation accuracy is worse, larger adjustments are needed; increase Delta by an appropriate amount.

Note: The optimum value of Delta varies as the square root of the relative accuracy of the simulation. For example, if your simulation is 100 times more accurate than typical, you should reduce Delta by a factor of 10.

Threshold calculations

The Threshold option defines the minimum step size the Modified LSQ Engine uses to adjust the optimization parameters.

The optimizer assumes that the values measured for the specifications change continuously as the parameters are varied. In practice, this assumption is not justified. For some analyses, especially transient analyses, the measurement expression values show discontinuous behavior for small parameter changes. This can be caused by accumulation of errors in iterative simulation algorithms.

The hypothetical data glitch figure demonstrates a typical case. The effect of the glitch is serious—the optimizer can get stuck in the spurious local minimum represented by the glitch. The optimizer’s threshold mechanism limits the effect of unreliable data.

Between iterations

Enter a value that defines a fraction of the current parameter value.

Example: A Threshold value of 0.01 means that the Modified LSQ Engine will change a parameter value by 1% of its current value when the engine makes a change.

By default, Threshold is set to 0 so that small changes in parameter values are not arbitrarily rejected. To obtain good results, however, you may need to adjust the Threshold value. When making adjustments, consider the following:

- If data quality is good, and Threshold is greater than zero, reduce the Threshold value to find more accurate parameter values.
- If data quality is suspect (has potential for spurious peaks or glitches), increase the Threshold value to ensure that the optimizer will not get stuck during the run.

Least squares / minimization

The Modified LSQ Engine implements two general classes of algorithm to measure design performance: least squares and minimization. These algorithms are applicable to both unconstrained and constrained problems.

Least squares

When optimizing for more than one goal, the Modified LSQ Engine always uses the least-squares algorithm. A reliable measure of performance for a design with multiple targets is to take the deviation of each output from its target, square all deviations (so each term is positive) and sum all of the squares. The Modified LSQ Engine then tries to reduce this sum to zero.

This technique is known as least squares. Note that the sum of the squares of the deviations becomes zero only if all of the goals are met.

Minimization

Another measure of design performance considers a single output and reduces it to the smallest value possible.

Example: Power or propagation delay, each of which is a positive number with ideal performance corresponding to zero.

Single goal optimization settings

When optimizing for more than one goal, the Modified LSQ Engine always uses the least-squares algorithm. For a single goal, however, you must specify the algorithm for the optimizer.

1 Do one of the following:

- ❑ Select the **Least Squares** option button to minimize the square of the deviation between the measured and target value.

  Or:

- ❑ Select the **Minimize** option button to reduce a value to the smallest possible value.

If your optimization problem is to maximize a single goal, then set up the specification to minimize the negative of the value.

For example: To maximize gain, set up the problem to minimize –*gain*.

You can see the following for more information:

How to select an engine	Selecting an engine
Engine overview	Optimizer Engine Overview

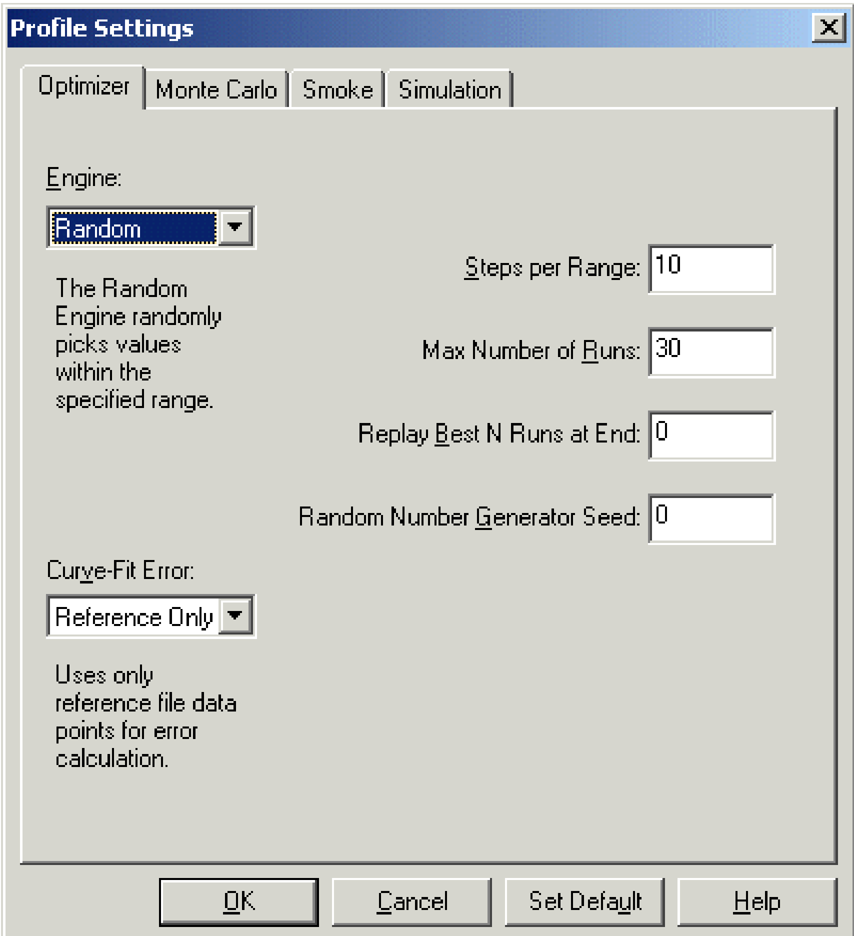
The Random engine

When you use the LSQ or Modified LSQ engines, it is sometimes difficult to determine where your starting points for optimization should be. The Random engine provides a good way to find these points.

The Random engine applies a grid to the design space and randomly runs analysis at the grid points. It keeps track of the grid points already run so that it never runs a duplicate set of parameter values. Once it finishes its initial analysis, it reruns the best points so you can easily use them for LSQ or Modified LSQ.

Configuring the Random engine

The Random Engine defaults are listed in a dialog box available from the **Optimizer** tab's, **Engine**, **Random** options.



To view and change the default options:

1 From the Advanced Analysis **Edit** menu, select **Profile Settings**

2 Click the **Optimizer** tab and select **Random** from the **Engine** drop-down list.

3 Edit the default value in the text box.

4 Click **OK**.

Configuring the Random engine options

Random Engine Options	Default Value
Steps per Range	10
Max Number of Runs	10
Replay Best N Runs at End	0
Random Number Generator Seed	0

Steps per Range

Specifies the number of steps into which each parameter's range of values should be divided.

For example, if this option is set to 7 and you have the following parameters

Parameter	Min	Max
A	1	4
B	10	16

The possible parameter values would be

Parameter A = 1, 1.5, 2, 2.5, 3, 3.5, 4

Parameter B = 10, 11, 12, 13, 14, 15, 16

Max Number of Runs

Specifies the maximum number of random trial runs that the engine will run. The engine will run either the total number of all grid points or the number specified in this option, whichever is less.

Note: With 10 parameters, the number of grid points in the design exploration ($NumSteps^{\#params}$) would be 8^{10} = 1,073,741,824.

For example, if Max Number of Runs is 100, Steps per Range is 8, and you have one parameter being optimized, there will be 8 trial runs. However, if you have 10 parameters being optimized, then there will be 100 runs.

Replay Best N Runs at End

Specifies the number of “best” runs the engine should rerun and display at the end of the analysis.

Note: The Replay runs are done after the trial runs. If Max Number of Runs is 100 and Replay is 10, there may be up to 110 runs total.

Random Number Generator Seed

Specifies the seed for the random number generator. Unlike the Monte Carlo tool, the seed in this engine does not automatically change between runs. Therefore, if you rerun the Random engine without changing any values, you will get the same results.

You can see the following for more information:

How to select an engine	Selecting an engine
Engine overview	Optimizer Engine Overview

The Discrete engine

The Discrete engine finds the nearest commercially available value for a component. The other engines calculate component values, but those values might not be commercially available.

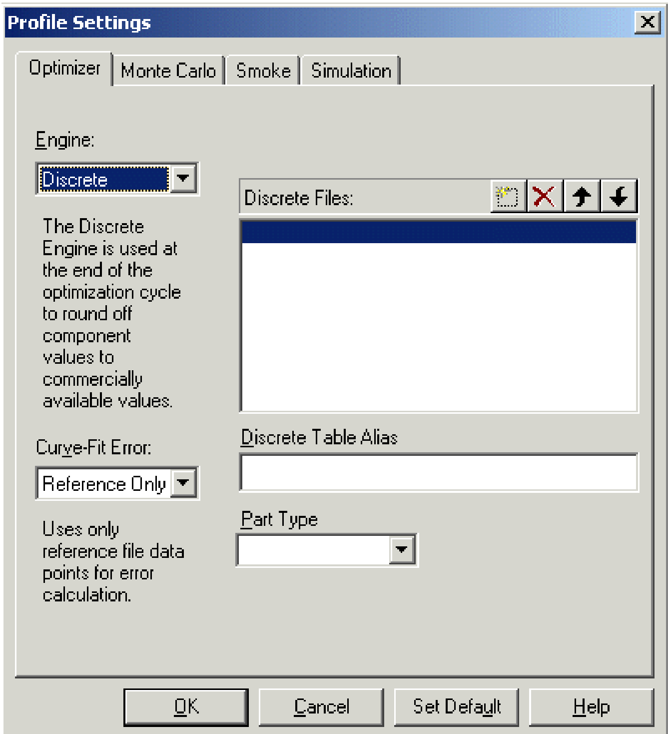
The discrete engine is a conceptual engine, rather than a true engine in that it does not actually perform an optimization, it finds available values from lists.

An example is a resistor that is assigned an optimal value of 1.37654328K ohms, which is not a standard value. Depending on the parameter tolerance and the manufacturer’s part number, the only values available might be 1.2K and 1.5K ohms. The Discrete engine selects parameter values based on discrete value tables for these parameters.

Once a value is selected, the engine makes a final run that lets you review the results in both the Optimizer and the output tools. If the results of the discrete analysis are not acceptable, the design can be optimized again to find another global minimum that might be less sensitive.

Configuring the Discrete engine

Advanced Analysis includes discrete tables of commercially available values for resistors, capacitors, and inductors. These tables are text files with a .table file extension.
In addition, you can add your own discrete values tables to an Advanced Analysis project using the dialog box shown below. To know more about the adding user-defined discrete value tables, see



After you have found commercial values for your design, you should run Monte Carlo and Sensitivity to ensure that the design is producible. Occasionally, the optimization process can find extremely good results, but it can be sensitive to even minor changes in parameter values.

You can see the following for more information:

How to select an engine	Selecting an engine
Engine overview	Optimizer Engine Overview

Troubleshooting

The Advanced Analysis troubleshooting feature returns you to PSpice[1] to analyze any measurement specification that is causing a problem during optimization.

When an Optimizer analysis fails, the error message displayed in the output window or a yellow or red flag in the Specifications table shows you which measurement and simulation profile is associated with the failure.

If the failure is a simulation failure (convergence error) or a measurement evaluation error, the troubleshooting feature can help track down the problem.

From the Optimizer tool in Advanced Analysis, you can right click on a measurement specification and select **Troubleshoot in PSpice**. PSpice will display two curves, one with the data from the original schematic values and one with the data of the last analysis run.

Using the troubleshooting feature

When an optimization analysis fails, you can use the troubleshooting feature to troubleshoot a problem specification.

Read the error message in the output window to locate the specification to troubleshoot, or look for a yellow or red flag in the first cell of a specification row.

1. Right click anywhere in the specification row you want to troubleshoot.

   A pop-up menu appears.

2. Select **Troubleshoot in PSpice**.

   PSpice opens and the measurement specification data is displayed in the window.

   The first trace shows the data from the run with the original schematic values.

   The second trace shows the data from the last run.

3. Right click on a trace, and from the pop-up menu select **Information**.

   A message appears about the trace data.

4. Make any needed edits:

---

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

- ❑ In the PSpice window, check the measurement plot or click on  to view the simulation output file.
- ❑ In the PSpice Measurements Results table, check the measurement syntax and the variables used.
- ❑ In PSpice, click  to edit the simulation profile.
- ❑ In the schematic editor, make changes to parameter values.

5 Rerun the simulation in the schematic editor.

6 Return to Advanced Analysis.

7 If you made changes:

- ❑ To a measurement in PSpice, copy the edited measurement from PSpice to the Advanced Analysis Specifications table (Use Windows copy and paste)
- ❑ To parameter values in your schematic editor, import the new parameter data by clicking on the Optimizer Parameters table row titled “Click here to import a parameter...”

8 Right click in the Error Graph and from the pop-up menu select **Clear History**.

9 Rerun Optimizer.

Example: Using the troubleshooting feature

To show how to use the troubleshooting feature, we need an optimization project that fails to find a solution. We’ll use the example in the Troubleshoot folder from the Tutorial directory. This example results in an unresolved optimization.

Strategy

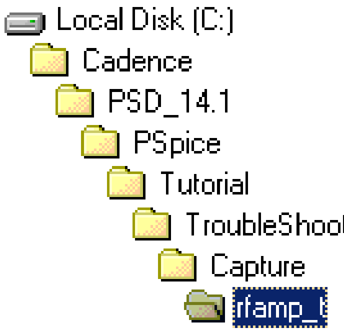
In this example we’ll:

- Open the RF amp circuit in the Troubleshooting directory
- Run the AC simulation and open Optimizer
- Use the troubleshoot function to view waveforms of the problem measurement

Setting up the example

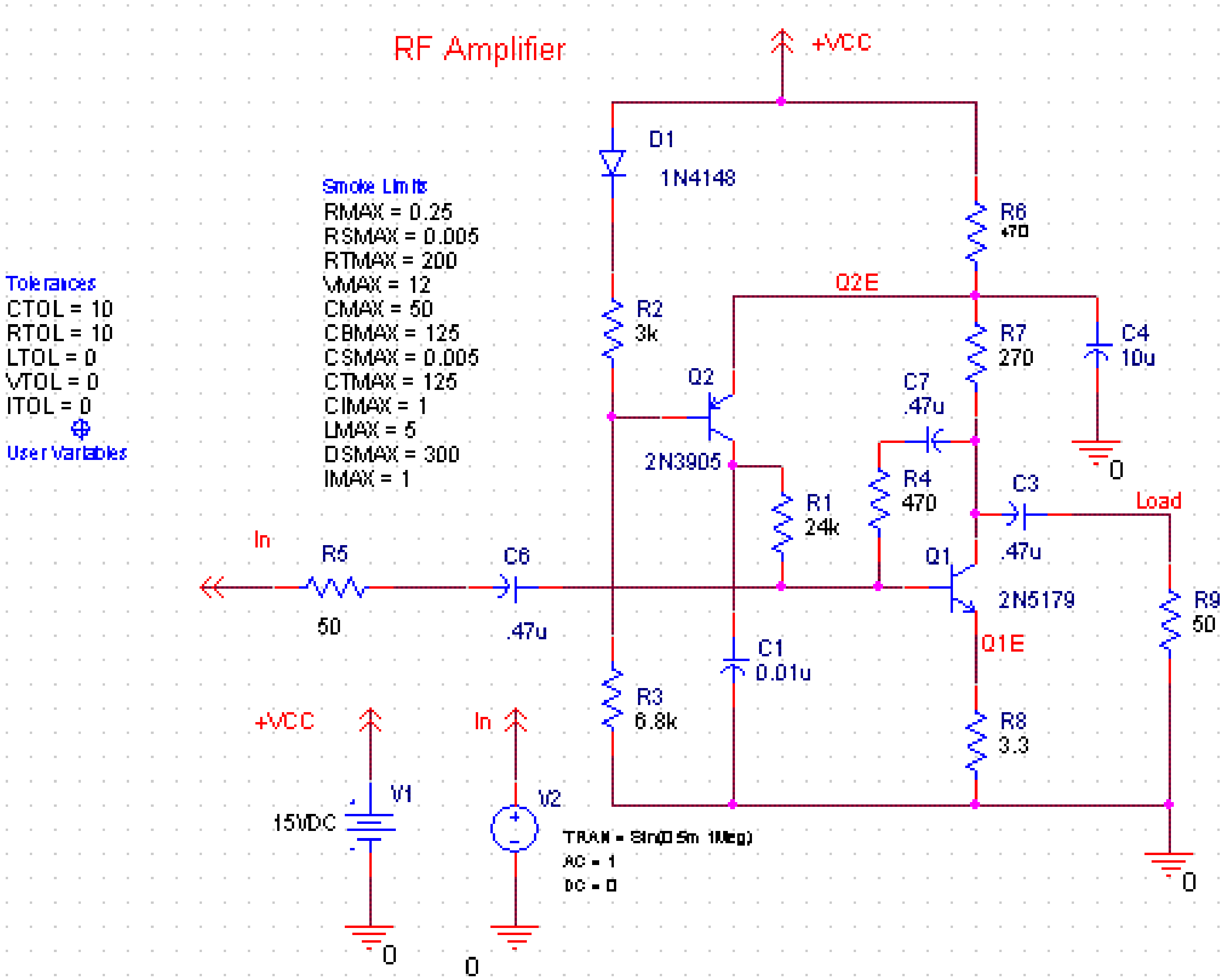
1 In your schematic editor, browse to the TroubleShoot directory:

<target directory> \ PSpice \ Tutorial \



2 From your schematic editor, open the rfampt project from the rfampt folder.

3 Open the schematic page.



4 With the SCHEMATIC1-AC simulation profile selected, click to run the simulation.

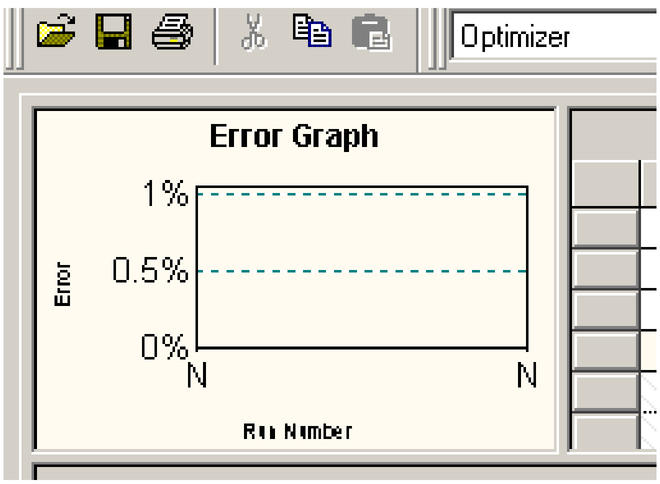
5 From **PSpice** menu in the schematic editor, select **Advanced Analysis** / **Optimizer**.

Advanced Analysis opens to the Optimizer view.

There are four measurement goals included in this example.

			Specifications [Next Run]					
	+	On/Off	Profile	Measurement	Min	Max	Type	Weight
		✓	rf_amp-schematic1...	Max(DB(V(Load)))	5	5.5000	Constraint	20
▶		✓	rf_amp-schematic1...	Bandwidth(V(Load),3)	200meg		Goal	1
		✓	rf_amp-schematic1...	Min(10*Log10(V(inoise...		5	Constraint	1
		✓	rf_amp-schematic1...	Max(V(onoise))		3n	Constraint	20

6 If there is any history in the Error Graph, right click in the error graph window and select **Clear History** from the pop-up menu.

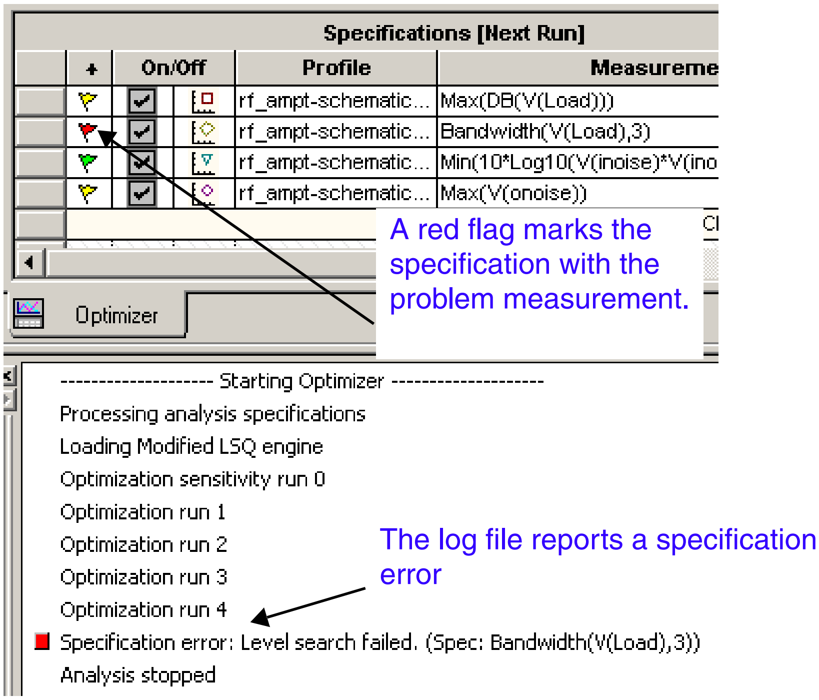


Error Graph with history cleared

7 Make sure the **Modified LSQ** engine is selected and click on the top toolbar to start the optimization.

Click to start optimization

The optimization starts and makes four run attempts.

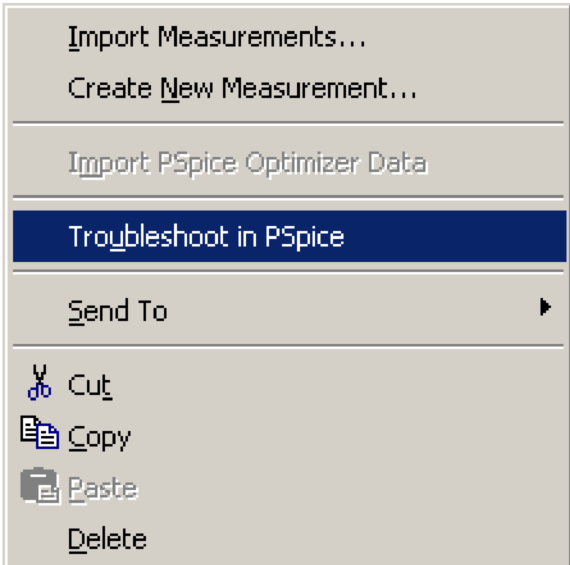


The Optimizer failed to find a solution. Let's troubleshoot the problem measurement in PSpice.

Using the troubleshooting function

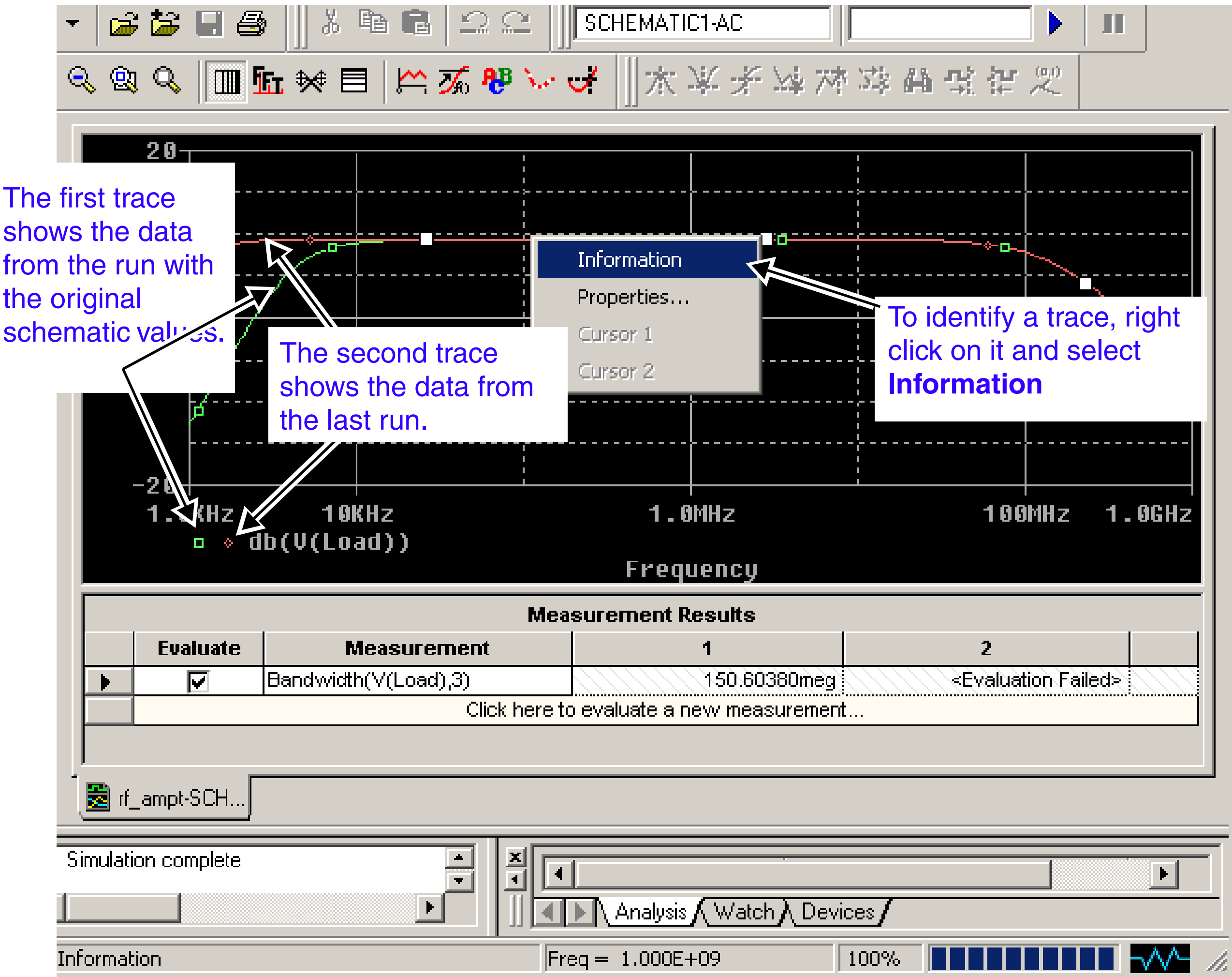
1 Right click in the specification row marked by the red flag (second row, Bandwidth(V(Load),3)).

A pop-up menu appears.

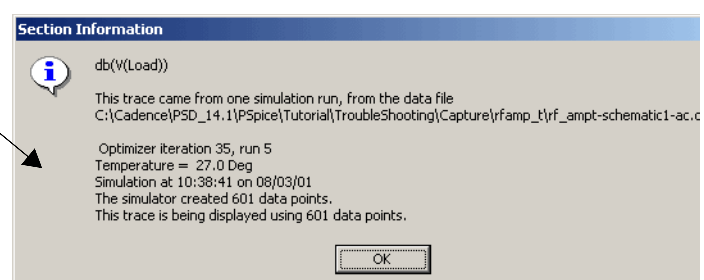


2 From the pop-up menu, select **Troubleshoot in PSpice**.

PSpice opens and the measurement specification data displays in the window.



A message appears about the trace data

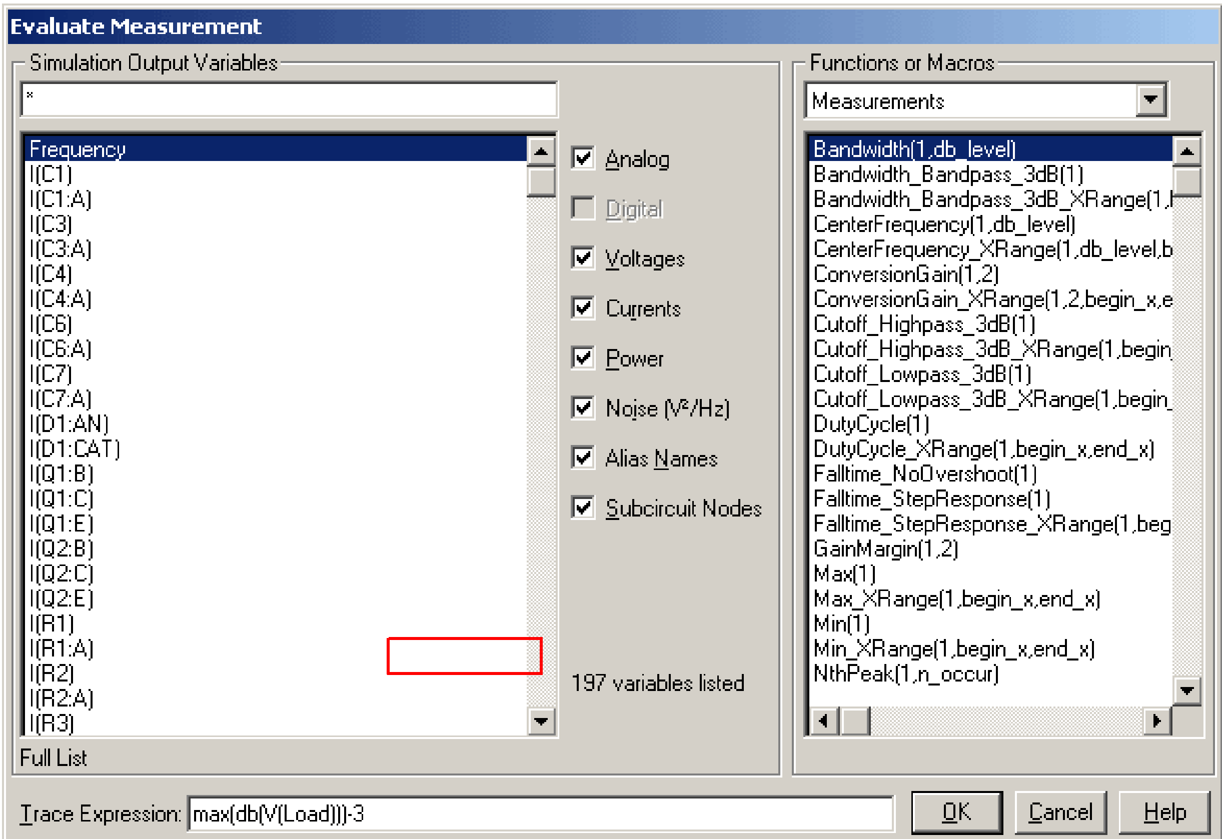


Analyzing the trace data

We know the bandwidth constraint failed. We'll add a measurement in PSpice to find the -3dB point of the trace.

1 Click at the bottom of the Measurements Results table.

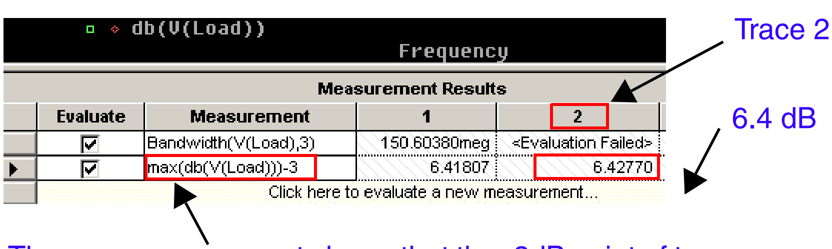
The Evaluate Measurement dialog box appears.



2 In the Trace Expression field at the bottom, type in:

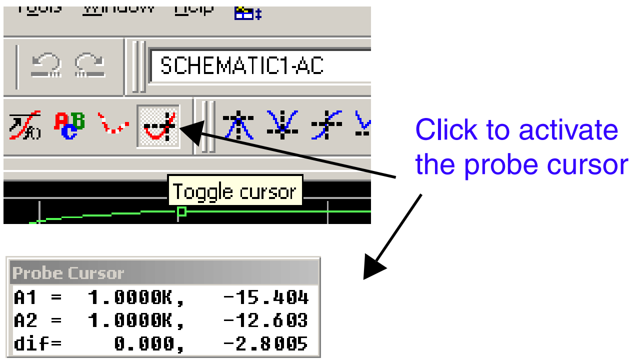
max(db(v(load)))-3

A measurement that calculates the -3dB point appears in the Measurement Results table.

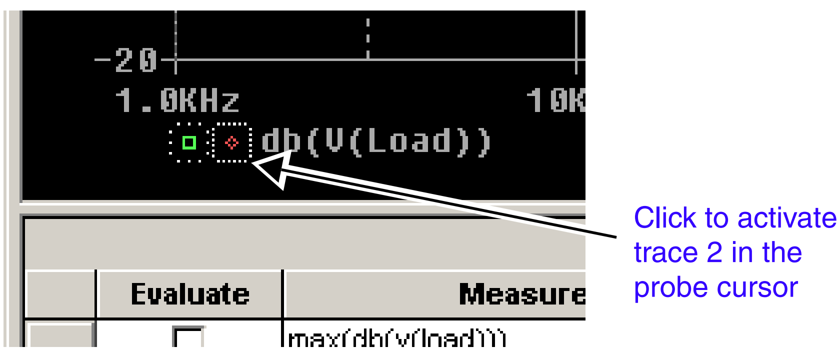


The new measurement shows that the -3dB point of trace 2 is at 6.4 dB

3 Click  to enable the Probe cursor.



4 Activate trace 2 in the probe cursor.



5 Click at the left end of trace 2.

The probe cursor shows that trace 2's -3dB point (6.4dB) occurs before 1kHz.

The Optimizer is increasing the bandwidth as we asked it to in the measurement specification, but not exactly in the way we wanted.

While this results show a slightly higher bandwidth, we are more interested in increasing the cut-off frequency.

Resolving the optimization

One solution may be to introduce a specification that keeps the low frequency cutoff above 1kHz, but this would complicate the optimization and take longer to complete.

Another solution may be to simplify things. It could be that we have given the optimizer too many degrees of freedom (parameters), some of which may not be necessary for meeting our goals.

Let's check out the bandwidth measurement in Sensitivity to see which components are the most sensitive.

Sensitivity check

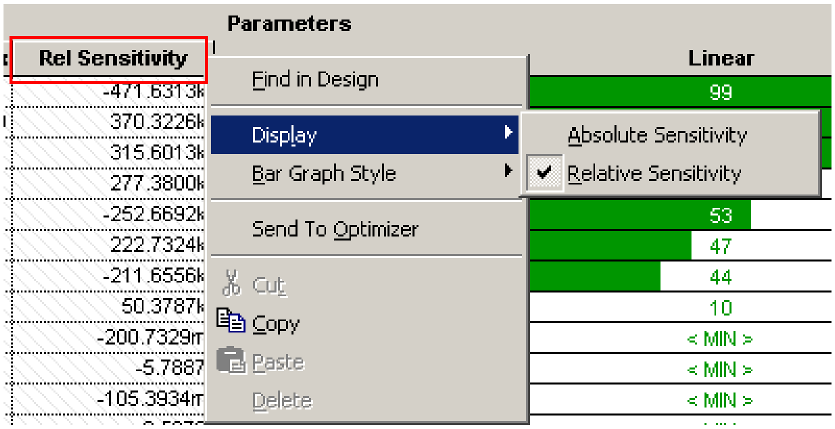
1 Return to Advanced Analysis and from the View menu, select **Sensitivity**.

The Sensitivity tool opens.

2 Make sure **Rel Sensitivity** is displayed in the Parameters table.

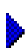
If you need to change the display from absolute to relative sensitivity:

❑ Right click and from the pop-up menu choose Display / Relative Sensitivity.



3 In the Specifications table, select the bandwidth measurement (second row).

Specifications			
+	On/Off	Profile	Measurement
	☑	rf_ampt-schematic1-ac.sim	max(db(v(load)))
	☑	rf_ampt-schematic1-ac.sim	bandwidth(v(load),3)
	☑	rf_ampt-schematic1-ac.sim	min(10*log10(v(inoise)*v(inoise)/8.28e-19))
	☑	rf_ampt-schematic1-ac.sim	max(v(onoise))
			Click here to import a measurement created within PS

4 Click  on the top toolbar to start the sensitivity analysis.

Sensitivity runs.

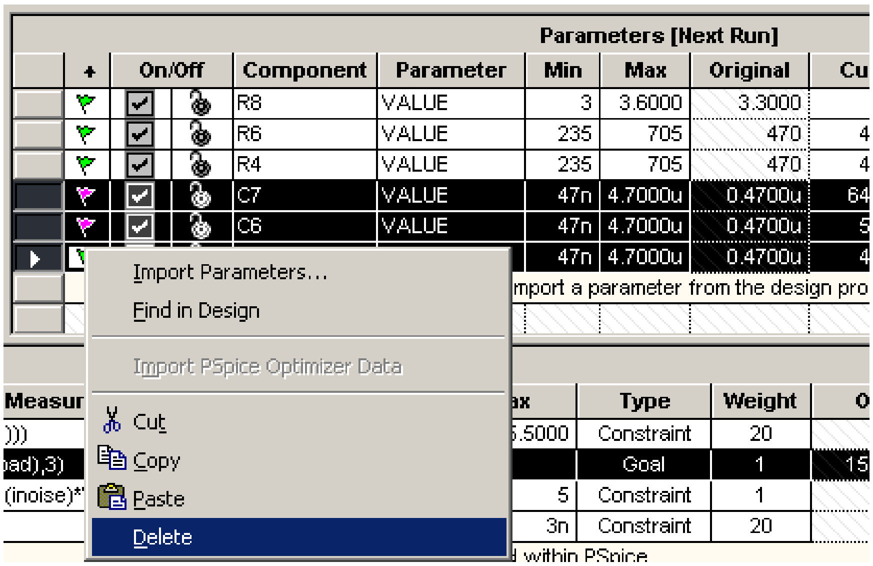
We can see that in the relative sensitivity analysis, Capacitors 3, 6, and 7 are not critical to the bandwidth response.

We'll return to Optimizer and remove the capacitors from the optimization analysis. Reducing variables may help Optimizer reach a solution.

Optimizer rerun

1 Return to the Optimizer tool and in the Parameters table, hold down your shift key and select the capacitor rows.

2 Right click and select **Delete** from the pop-up menu.



3 If there is any history in the Error Graph, right click in the Error Graph window and select **Clear History** from the pop-up menu.

4 Select the Modified LSQ engine and click on the top toolbar to start the optimization.

The optimization starts and finds a solution.

This section suggests solutions to problems you may encounter in any of the Advanced Analysis tools.

Common problems and solutions.o.htm

Check the following tables for answers these problems:

- Analysis fails
- Results are not what you expected
- Can’t make user interface do what you want
- Not enough disk space or memory

Analysis fails

Problem: Analysis fails	Possible cause	Solution
Smoke analysis won't run.	May not have a transient profile in the design. If a transient profile is included in the design, Smoke automatically picks the first transient profile for the analysis.	Smoke analysis only works if you have one or more transient profiles. Smoke does not work on AC or DC sweeps.
Smoke analysis won't run: message says "cannot find .dat file."	Transient analysis simulation may not be done.	Simulate the transient analysis in PSpice, review the waveform and measurement results, then run Smoke.
Smoke analysis fails: Output window displays the following error for smoke parameters: "Data not found for Smoke test. Please verify Save Data and Data Collection options in the simulation profile"	Data save start time is not zero or data collection options for voltages, currents and power is not set to **All**.	From the **Simulation** menu in PSpice, choose **Edit Profile** to open the **Simulation Settings** dialog box. Ensure that the data save start time in the **Analysis** tab is 0. Smoke analysis works only if data save start time is zero seconds. Or From the **Simulation** menu in PSpice, choose **Edit Profile** to open the **Simulation Settings** dialog box. Ensure that the data collection options in the **Data Collection** tab is set to **All** for voltages, currents and power.

Problem: Analysis fails	Possible cause	Solution
Monte Carlo analysis takes too long.	The number of runs may be too large.	Decrease the number of runs in the Monte Carlo settings tab (from the **Edit** menu, select **Profile Settings** and click the **Monte Carlo** tab).
I get an evaluation error message.	You might be using the wrong profile for the type of measurement you're evaluating.	Check the selected profile and change it to the profile that applies to your measurement. For example, change to an AC profile to evaluate bandwidth.
Optimization didn't converge.	The engine may have found a local minimum, which may not be the best solution.	Use the Random engine to search for alternate starting points. Go to the Error Graph history and copy the best Random engine result to the Nth run (the end). Then switch to the Modified LSQ engine to pinpoint the final answer.

Problem: Analysis fails	Possible cause	Solution
Optimization didn't converge after running through several iterations.	The parameters have changed the circuit's behavior, so the simulation results may not provide the information needed to meet the measurement goal.	Use the **Troubleshoot in PSpice** feature to check the shapes of the traces and make sure they are appropriate for the desired measurement (right click on a measurement row and select the Troubleshoot command from the pop-up menu). For example, do the traces show that the filter still looks like a bandpass? Try changing the simulation settings to increase the range of frequencies. Or Restrict the parameter ranges in the Optimizer **Parameters** table to prevent the problem.
Optimization didn't converge, but it looked like it was improving.	Too few iterations.	Increase number of iterations in the Optimizer engine settings tab (from the **Edit** menu, select **Profile Settings** and click the **Optimizer** tab.)
Optimization didn't converge. Parameters didn't change much from their original values.	Selected parameters may not be sensitive to the chosen measurement.	Choose different parameters more sensitive to the chosen measurement.

Problem: Analysis fails	Possible cause	Solution
Optimization didn't converge. It was improving for a few iterations, then the Error Graph traces flattened out.	One or more parameters may have reached its limit.	If appropriate, change the range of any parameter that is near its limit, to allow the parameter to exceed the limit. If the limit cannot be changed, you may want to disable that parameter because it is not useful for optimization and will make the analysis take longer.

Results are not what you expected

Return to top of table.

Problem: Results are not what you expected	Possible cause	Solution
I set up my circuit and ran Smoke, but I'm not getting the results I expected.	Your components may not have smoke parameters specified.	Check the online *Advanced Analysis Library List* and *PSpice library list* for a complete list of components supplied with smoke parameters. Replace your existing components with those containing smoke parameters. or For R,L and C components, add the design variables table (default variables) to your schematic. This table contains default smoke parameters and values. See the Libraries chapter of this manual for instructions on how to add this table to your schematic. or Add smoke parameters to your component models using the instructions provided in our technical note, "Creating Models with Smoke Parameters," which is available on .

Problem: Results are not what you expected	Possible cause	Solution
Smoke analysis peak results don’t look right: measured values are too small.	Transient analysis may not be long enough to include the expected peaks or may not have sufficient resolution to detect sharp spikes.	Check the transient analysis results in PSpice. Make sure the analysis includes any expected peaks. If necessary, edit the simulation profile to change the length of the simulation or to take smaller steps for better resolution.
Smoke analysis average or RMS measured results are not what I expected.	Transient analysis may not be set up correctly.	Check the transient analysis results in PSpice. Make sure the average of voltages and currents over the entire range is the average value you’re looking for. If you want the measurement average to be based on steady-state operation, make sure the analysis runs long enough and that you only save data for the period over which you want to average.
I selected a custom derating or standard derating file in Smoke, but my %Derating and %Max values didn’t change.	Need to click the Run button to recalculate the Smoke results with the new derating factors.	In Smoke, click  on the top toolbar and wait for the new values to appear.
My Smoke result has a yellow flag and a cell is grey.	The limit (average, RMS, or peak) is not typically defined for this parameter. Grey results show the calculated simulation values; however, grey results also indicate that comparison with the limit may not be valid.	The information is provided this way for user convenience, to show all calculated simulation values (average, RMS, and peak), but comparison to limits requires user interpretation. The color coding is intended to help.

Problem: Results are not what you expected	Possible cause	Solution
The derating factor for the PDM smoke parameter isn't 100% even though I'm using No Derating.	This is OK. Smoke applies a thermal correction to the calculation.	None needed. This is normal behavior.
My Optimizer results don't look right. The current results are missing.	Your cursor might be set on a prior run in the Error Graph. The results you see are history.	In the Error Graph, click on the Nth (end) run's vertical line. Current results will appear in the **Parameters** table.
In Optimizer, I finally get a good parameter value, but as I continue optimizing other things, the good parameter value keeps changing.	The good parameter value needs to be locked in so it won't change for the next runs.	In the Optimizer **Parameters** table, click the icon for the applicable parameter. This will close the lock and the parameter value will not change for subsequent runs.
In Optimizer, there aren't any discrete values listed for my component.	Discrete values tables are provided for RLCs. If your component is not an RLC, you'll have to create a discrete values table.	Create a discrete values table for your non-RLC component using instructions provided in the *Adding User-Defined Discrete Table* section of the *Optimizer* chapter in *PSpice Advanced Analysis User Guide*.
Can't see the Optimizer discrete tables column.	Optimizer engine is not set to **Discrete**.	Change the Optimizer engine to **Discrete** in the drop-down list.
I can't find my individual Monte Carlo run results.	Raw measurement tab is not selected.	Click on the tab labeled **Raw Meas** to bring individual run results to the foreground on your screen.
I want more detail on my Monte Carlo graph.	Bin size is too small for desired detail.	Increase bin size in the Monte Carlo setting tab (from the **Edit** menu, select **Profile Settings** and click the **Monte Carlo** tab).

Problem: Results are not what you expected	Possible cause	Solution
The Monte Carlo PDF / CDF graph doesn't look right for my measurement.	The applicable measurement row may not be highlighted.	Click on the measurement row. The resulting graph corresponds to that measurement.
I can't see the CDF graph.	Graph defaults to PDF view.	Right click the graph and select **CDF graph** from the pop-up menu.
I can't find the parameter values for my Monte Carlo runs.	Monte Carlo parameter values are only available in the log file.	From the **View** menu, select **Log File** / **Monte Carlo** and scroll through the file to the applicable run.

Can't make user interface do what you want

Return to top of table.

Problem: Can't make user interface do what you want	Possible cause	Solution
I can't get all my red bar graphs to appear at the top of my Smoke or Sensitivity tables.	Data isn't sorted.	Click twice on the bar graph column header. The first click puts all the red bars at the bottom. The second click puts them at the top.
I don't want to see the grey bars in Smoke.	Average, RMS, or peak limits that don't apply to your parameter may be selected for viewing.	Double click the message flag column header. This will sort the grey bars so they appear at the bottom of the data display. or Right click and uncheck the average, RMS, or peak values on the right click pop-up menu.

Problem: Can’t make user interface do what you want	Possible cause	Solution
Why can’t I use my Monte Carlo settings and results from PSpice?	The programs are separate and use different input.	Advanced Analysis Monte Carlo provides more information and can be run on more than one specification simultaneously. This is the trade-off.
Monte Carlo cursor won’t drag to a new location.	The cursor can be moved, but it doesn’t use the drag and drop method.	Click once on the cursor. Click in your desired location. The cursor moves to the location of the second click.

Not enough disk space or memory

Return to top of table.

Problem: Not enough disk space or memory	Possible cause	Solution
I get a disk space error message or an out of memory message and I’m running a Monte Carlo analysis.	Too much data is being saved for the Monte Carlo runs. For example, in a 10,000-run Monte Carlo analysis where all data is collected and saved, the data file and memory usage may become very large.	Turn off the option to save all simulation waveform data in Advanced Analysis. By doing this, saved data will be limited to just the current run. However, at this setting, the simulation will run slower. To turn off the data storage: **1.** From the Advance Analysis menu select: Edit / Profile Settings/ Simulation tab **2.** From the Monte Carlo field, select **Save None** from the drop-down list Advanced Analysis will overwrite the data file for each run.

Problem: Not enough disk space or memory	Possible cause	Solution
I get a disk space error message or an out of memory message and I'm running a Monte Carlo analysis (continued).	Too much data is being collected for each simulation run. For instance, collecting voltages, currents, power, digital data, noise data, and all of these for internal subcircuit components results in a large data file and large memory use. Note: You can also place markers on nets, pins, and devices on the schematic and collect data at these marker locations. In PSpice, set the data collection option to **At Markers Only** for all the data types you want. See the schematic editor help for more information on how to use markers on the schematic.	Limit data collection to only the information that is needed to perform Advanced Analysis. You can do this in conjunction with the data file solution mentioned on the previous page or do just this and save data for all Monte Carlo runs. To change data collection options for each simulation, do the following for each simulation profile used in Advanced Analysis: **1.** From the PSpice Simulation menu, select Edit Profile. **2.** In the Simulation Settings dialog box, select the Data Collection tab. **3.** Set the data collection option to **None** for all the data types that are not required. Use the drop-down list to select the option. **4.** Set the data collection option to **All but Internal Subcircuits** for data required for Advanced Analysis. Use the drop-down list to select the option.

Printing Results from Advanced Analysis

To print results from Advanced Analysis:

– Click .

Or:

From the File menu, select **Print**.

Customizing Toolbars

Use the Customize dialog box to customize the look and feel of the Advanced Analysis toolbars. Use the Toolbars tab to customize toolbars and use the Commands tab to add button to the toolbar.

The Toolbars tab of the Customize dialog box has the following elements:

`Toolbars`	Lists available toolbars. To display a new toolbar, select the checkbox to the left of the toolbar. Clearing the checkbox hides the toolbar.
`Show Tooltips`	Select to display tooltips for buttons.
`Cool Look`	Select to change the look of the toolbar.
`Large Buttons`	Select to increase the size of the buttons in the toolbars.
`New`	Click to define a new toolbar.
`Reset`	Click to restore default settings.

The New Toolbar dialog box that appears on clicking the New button in the Toolbars tab allows you to specify a name for the toolbar.

The Commands tab of the Customize dialog box has the following elements:

Categories	Lists available categories.
`Buttons`	Displays the buttons in each category.

Saving results from Advanced Analysis

To save results from Advanced Analysis:

1 Click .

Or:

Choose *File - Save*.

The final results will be saved in the Advanced Analysis profile (`.aap`).

Simulation Tab

The Simulation tab of the Profile Settings dialog box allows you to control the amount of data stored in the simulation results data file. You can specify the number of runs for which data is to be stored for Sensitivity, Optimizer, and Monte Carlo. You can specify the following two options:

- Save All Runs: Data for all runs is stored in the data file.
- Save None: Data is saved only for the last run.

Glossary

A|B|C|D|E|F|G|H|I|J|K|L|M|N|O|P|Q|R|S|T|U|V|W|X|Y|Z

A

absolute sensitivity

The change in a measurement caused by a unit change in parameter value (for example, 0.1V: 1Ohm).

The formula for absolute sensitivity is:

[(Ms - Mn) / (Pn * 0.4 * Tol)]

Where:

Mn = the measurement from the nominal run

Ms = the measurement from the sensitivity run for that parameter

Tol = relative tolerance of the parameter

B

bimodal distribution function

Related to Monte Carlo. This is a type of distribution function that favors the extreme ends of the values range. With this distribution function, there is a higher probability that Monte Carlo will choose values from the far ends of the tolerance range when picking parameter values for analysis.

C

component

A circuit device, also referred to as a part.

component parameter

A physical characteristic of a component. For example, a breakdown temperature is a parameter for a resistor. A parameter value can be a number or a named value, like a programming variable that represents a numeric value. When the parameter value is a name, its numerical solution can be varied within a mathematical expression and used in optimization.

constraint

Related to Modified LSQ optimization engine. An achievable numerical value in circuit optimization. A constraint is specified by the user according to the user's design specifications. The Modified LSQ engine works to meet the goals, subject to the specified constraints.

cumulative distribution function (CDF)

A way of displaying Monte Carlo results that shows the cumulative probability that a measurement will fall within a specified range of values. The CDF graph is a stair-step chart that displays the full range of calculated measurement values on the x-axis. The y-axis displays the cumulative number of runs that were below those values.

D

derating factor

A safety factor that you can add to a manufacturer's maximum operating condition (MOC). It is usually a percentage of the manufacturer's MOC for a specific component. "No derating" is a case where the derating factor is 100 percent. "Standard derating" is a case where derating factors of various percents are applied to different components in the circuit.

device

See component

distribution function

Related to Monte Carlo. When Monte Carlo randomly varies parameter values within tolerance, it uses that parameter's distribution function to make a decision about which value to select. See also: Flat (Uniform), Gaussian (Normal), Bimodal, and Skewed distribution functions. See also cumulative distribution function.

Discrete engine

Related to the Optimizer. The Discrete engine is a calculation method that selects commercially available values for components and uses these values in a final optimization run. The engine uses default tables of information provided with Advanced Analysis or tables of values specified by the user.

discrete values table

For a single component (such as a resistor), a discrete values table is a list of commercially available numerical values for that component. Discrete values tables are available from manufacturers, and several tables are provided with Advanced Analysis.

E

error graph

A graph of the error between a measurement's goal or constraint and the calculated value for the measurement. Sometimes expressed in percent.

Error =
(Calculated meas. value - Goal value) / Goal value

Error =
(Calculated meas. value - Constraint) / Constraint

F

flat distribution function

Also known as Uniform distribution function. Related to Monte Carlo. This is the default distribution function used by Advanced Analysis Monte Carlo. For a Flat (Uniform) distribution function, the program has an equal probability of picking any value within the allowed range of tolerance values.

G

Gaussian distribution function

Also known as Normal distribution function. Related to Monte Carlo. For a Gaussian (Normal) distribution function, the program has a higher probability of choosing from a narrower range within the allowed tolerance values near the mean.

global minimum

Related to the Optimizer. The global minimum is the optimum solution, which ideally has zero error. But factors such as cost and manufacturability might make the optimum solution another local minimum with an acceptable total error.

goal

A desirable numerical value in circuit optimization. A goal may not be physically achievable, but the optimization engine tries to find answers that are as close as possible to the goal. A goal is specified by the user according to the user's design specifications.

H

I

J

K

L

local minimum

Related to the Optimizer. Local minimum is the bottom of any valley in the error in the design space.

M

Maximum Operating Conditions (MOCs)

Maximum safe operating values for component parameters in a working circuit. MOCs are defined by the component manufacturer.

Modified Least Squares Quadratic (LSQ) engine

A circuit optimization engine that results in fewer runs to reach results, and allows goal- and constraint-based optimization.

measurement expression

An expression that evaluates a characteristic of one or more waveforms. A measurement expression contains a measurement definition and an output variable. For example, Max(DB(V(load))). Users can create their own measurement expressions.

model

A mathematical characterization that emulates the behavior of a component. A model may contain parameters so the component's behavior can be adjusted during optimization or other advanced analyses.

Monte Carlo analysis

Calculations that estimate statistical circuit behavior and yield. Uses parameter tolerance data. Also referred to as yield analysis.

N

nominal value

For a component parameter, the nominal value is the original numerical value entered on the schematic.

For a measurement, the nominal value is the value calculated using original component parameter values.

normal distribution function

See Gaussian distribution function

O

optimization

An iterative process used to get as close as possible to a desired goal.

original value

See nominal value

P

parameter

See component parameter

parameterized library

A library that contains components whose behaviors can be adjusted with parameters. The Advanced Analysis libraries include components with tolerance parameters, smoke parameters, and optimizable parameters in their models.

part

See component

probability distribution function (PDF) graph

A way of displaying Monte Carlo results that shows the probability that a measurement will fall within a specified range of values. The PDF graph is a bar chart that displays the full range of calculated measurement values on the x-axis. The y-axis displays the number of runs that met those values. For example, a tall bar (bin) on the graph indicates there is a higher probability that a circuit or component will meet the x-axis values (within the range of the bar) if the circuit or component is manufactured and tested.

Q

R

Random engine

Related to Optimizer. The Random engine uses a random number generator to try different parameter value combinations then chooses the best set of parameter values in a series of runs.

relative sensitivity

Relative sensitivity is the percent change in measurement value based on a one percent positive change in parameter value for the part.

The formula for relative sensitivity is:

[(Ms - Mn) / (0.4*Tol)]

Where:

Mn = the measurement from the nominal run

Ms = the measurement from the sensitivity run for that parameter

Tol = relative tolerance of the parameter

S

Safe Operating Limits (SOLs)

Maximum safe operating values for component parameters in a working circuit with safety factors (derating factors) applied. Safety factors can be less than or greater than 100 percent of the maximum operating condition depending on the component.

sensitivity

The change in a simulation measurement produced by a standardized change in a parameter value:

$$S(measurement) = \frac{\Delta_{measurement}}{\Delta_{parameter}}$$

See also relative and absolute sensitivity.

skewed distribution function

Related to Monte Carlo. This is a type of distribution function that favors one end of the values range. With this distribution function, there is a higher probability that Monte Carlo will choose values from the skewed end of the tolerance range when picking parameter values for analysis.

Smoke analysis

A set of safe operating limit calculations. Uses component parameter maximum operating conditions (MOCs) and safety factors (derating factors) to calculate if each component parameter is operating within safe operating limits. Also referred to as stress analysis.

specification

A goal for circuit design. In Advanced Analysis, a specification refers to a measurement expression and the numerical min or max value specified or calculated for that expression.

T

U

uniform distribution function

See flat distribution function

V

W

weight

Related to Optimizer. In Optimizer, we are trying to minimize the error between the calculated measurement value and our goal. If one of our goals is more important than another, we can emphasize that importance, by artificially making that goal's error more noticeable on our error plot. If the error is artificially large, we'll be focusing on reducing that error and therefore focusing on that goal. We make the error stand out by applying a weight to the important goal. The weight is a positive integer (say, 10) that is multiplied by the goal's error, which results in a "magnified" error plot for that goal.

worst-case maximum

Related to Sensitivity. This is a maximum calculated value for a measurement based on all parameters set to their tolerance limits in the direction that will increase the measurement value.

worst-case minimum

Related to Sensitivity. This is a minimum calculated value for a measurement based on all parameters set to their tolerance limits in the direction that will decrease the measurement value.

X

Y

yield

Related to Monte Carlo. Yield is used to estimate the number of usable components or circuits produced during mass manufacturing. Yield is a percent calculation based on the number of run results that meet design specifications versus the total number of runs. For example, a yield of 99 percent indicates that of all the Monte Carlo runs, 99 percent of the measurement results fell within design specifications.

Z

Y