
Simulation examples

2

Chapter overview

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

This chapter includes the following sections:

- [Example circuit creation](#) on page 62
- [Performing a bias point analysis](#) on page 69
- [DC sweep analysis](#) on page 72
- [Transient analysis](#) on page 79
- [AC sweep analysis](#) on page 84
- [Parametric analysis](#) on page 89
- [Performance analysis](#) on page 97

Example circuit creation

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

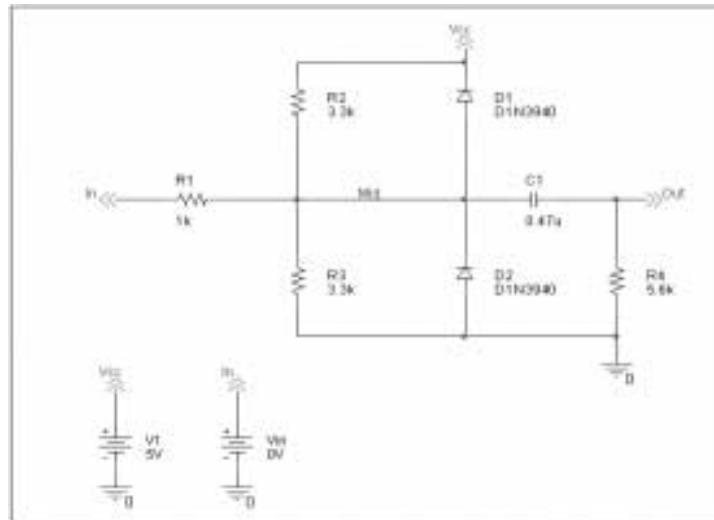


Figure 2-1 Diode clipper circuit.

To create a new PSpice project

- 1 From the Windows Start menu, choose Release OrCAD 10.0 from the Programs folder and then the Capture shortcut to start Capture.
- 2 In the Project Manager, from the File menu, point to New and choose Project.
- 3 Select Analog or Mixed-Signal Circuit Wizard.
- 4 In the Name text box, enter the name of the project (CLIPPER).
- 5 Use the Browse button to select the location for the project files, then click OK.
- 6 In the Create PSpice Project dialog box, select Create a blank project.
- 7 Click OK.

No special libraries need to be configured at this time. A new page will be displayed in Capture and the new project will be configured in the Project Manager.

To place the voltage sources

- 1 In Capture, switch to the schematic page editor.
- 2 From the Place menu, choose Part to display the Place Part dialog box.
- 3 Add the library for the parts you need to place:
 - a. Click the Add Library button.
 - b. Select `SOURCE.OLB` (from the PSpice library) and click Open.

Note: There are two sets of library files supplied with Capture and PSpice. The standard schematic part libraries are found in the directory `\TOOLS\CAPTURE\LIBRARY`. The part libraries that are designed for simulation with PSpice are found in the sub-directory `\TOOLS\CAPTURE\LIBRARY\PSPICE`. In order to have access to specific parts, you must first configure the library in Capture using the Add Library function.

- 4 In the Part text box, type `VDC`.
- 5 Click OK.
- 6 Move the pointer to the correct position on the schematic page (see Figure 2-1) and click to place the first part.
- 7 Move the cursor and click again to place the second part.
- 8 Right-click and choose End Mode to stop placing parts.

To place the diodes

- 1 From the Place menu, choose Part to display the Place Part dialog box.
- 2 Add the library for the parts you need to place:


- a. Click the Add Library button.
 - b. Select `DIODE.OLB` (from the PSpice library) and click Open.
- 3 In the Part text box, type `D1N39` to display a list of diodes.
When placing parts:
 - Leave space to connect the parts with wires.
 - You will change part names and values that do not match those shown in [Figure 2-1](#) later in this section.
- 4 Select `D1N3940` from the Part List and click OK.
- 5 Press R to rotate the diode to the correct orientation.
- 6 Click to place the first diode (D1), then click to place the second diode (D2).
- 7 Right-click and choose End Mode to stop placing parts.

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


- 1 Click the text to select it, then drag the text to a new location.

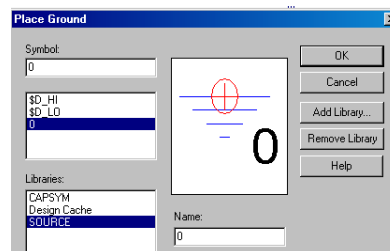
To place the other parts

- 1 From the Place menu, choose Part to display the Place Part dialog box.
- 2 Add the library for the parts you need to place:
 - a. Click the Add Library button.
 - b. Select `ANALOG.OLB` (from the PSpice library) and click Open.
- 3 Follow similar steps as described for the diodes to place the parts listed below, according to [Figure 2-1](#) on page 62. The part names you need to type in the Part name text box of the Place Part dialog box are shown in parentheses:
 - resistors (R)

- capacitor (C)
- 4 To place the off-page connector parts (OFFPAGELEFT-R), click the Place Off-Page Connector button  on the tool palette.
 - 5 Add the library for the parts you need to place:
 - a. Click the Add Library button.
 - b. Select CAPSYM.OLB (from the Capture library) and click Open.
 - 6 Place the off-page connector parts according to [Figure 2-1](#) on page 62.
- Note:** To rotate the part so the arrows are pointing in the correct direction, place the part, select it, then press *R* one or more times to rotate the part to the desired orientation.

To place the zero ground part

- 1 To place the ground parts (0), click the GND button  on the tool palette.



- 2 Add the library for the parts you need to place:
 - a. Click the Add Library button.
 - b. Select SOURCE.OLB (from the PSpice library) and click Open.
- 3 Place the 0 ground part from SOURCE.OLB as shown in [Figure 2-1](#) on page 62.

Important

You must use the 0 (zero) ground part from the SOURCE.OLB part library. You can use any other ground part only if you change its name to 0 (zero).

To connect the parts

- 1 From the Place menu, choose Wire to begin wiring parts.

The pointer changes to a crosshair.

- 2 Click the connection point (the very end) of the pin on the off-page connector at the input of the circuit.
- 3 Click the nearest connection point of the input resistor R1.

To stop wiring, right-click and choose End Wire. The pointer changes to the default arrow.

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

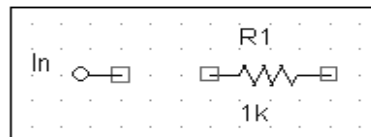



Figure 2-2 Connection points.

If you make a mistake when placing or connecting components:

From the Edit menu, choose Undo, or click .

- 4 Connect the other end of R1 to the output capacitor.
- 5 Connect the diodes to each other and to the wire between them:
 - a. Click the connection point of the cathode for the lower diode.
 - b. Move the cursor straight up and click the wire between the diodes. The wire ends, and the junction of the wire segments becomes visible.

- c. Click again on the junction to continue wiring.
 - d. Click the end of the upper diode's anode pin.
- 6 Continue connecting parts until the circuit is wired as shown in [Figure 2-1](#) on page 62.

To assign names (labels) to the nets

- 1 From the Place menu, choose Net Alias to display the Place Net Alias dialog box.
- 2 In the Name text box, type Mid.
- 3 Click OK.
- 4 Place the net alias on any segment of the wire that connects R1, R2, R3, the diodes, and the capacitor. The lower left corner of the net alias must touch the wire.
- 5 Right-click and choose End Mode to quit the Net Alias function.

To assign names (labels) to the off-page connectors

Label the off-page connectors as shown in [Figure 2-1](#) on page 62.

- 1 Double-click the name of an off-page connector to display the Display Properties dialog box.
- 2 In the Name text box, type the new name.
- 3 Click OK.
- 4 Select and relocate the new name as desired.

To assign names to the parts

- 1 Double-click the second VDC part to display the Parts spreadsheet.
- 2 Click in the first cell under the Reference column.
- 3 Type in the new name `Vin`.

- 4 Click Apply to update the changes to the part, then close the spreadsheet.
- 5 Continue naming the remaining parts until your schematic looks like [Figure 2-1](#) on page 62.



A more efficient way to change the names, values and other properties of several parts in your design is to use the Property Editor, as follows:

- a. Select all of the parts to be modified by pressing *Ctrl* and clicking each part.
- b. From the Edit menu, choose Properties.

The Parts Spreadsheet appears.

Change the entries in as many of the cells as needed, and then click Apply to update all of the changes at once.

To change the values of the parts

- 1 Double-click the voltage label (0V) on V1 to display the Display Properties dialog box.
- 2 In the Value text box, type 5V.
- 3 Click OK.
- 4 Continue changing the Part Value properties of the parts until all the parts are defined as in [Figure 2-1](#) on page 62.

Your schematic page should now have the same parts, wiring, labels, and properties as [Figure 2-1](#) on page 62.

To save your design

- 1 From the File menu, choose Save.

Finding out more about setting up your design

About setting up a design for simulation

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

Running PSpice

When you perform a simulation, PSpice generates an output file (*.OUT).

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

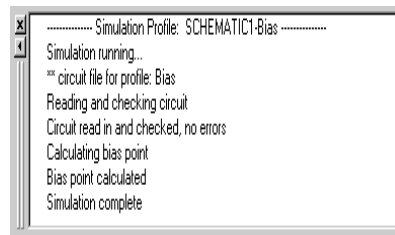


Figure 2-3 PSpice simulation output window.

You can set up a simulation profile to run one analysis at a time. To run multiple analyses (for example, both DC sweep and transient analyses), set up a batch simulation. For more information, see [Chapter 8, “Setting up analyses and starting simulation.”](#)

Performing a bias point analysis

To set up a bias point analysis in Capture

- 1 In Capture, switch to `CLIPPER.OPJ` in the schematic page editor.

- 2 From the PSpice menu, choose New Simulation Profile to display the New Simulation dialog box.
- 3 In the Name text box, type `Bias`.
- 4 From the Inherit From list, select None, then click Create.

The Simulation Settings dialog box appears.

The root schematic listed is the schematic page associated with the simulation profile you are creating.

- 5 From the Analysis type list, select Bias Point.
- 6 Click OK to close the Simulation Settings dialog box.

To simulate the circuit from within Capture

- 1 From the PSpice menu, choose Run.

PSpice simulates the circuit and calculates the bias point information.

Note: Because waveform data is not calculated during a bias point analysis, you will not see any plots displayed in the Probe window for this simulation. To find out how to view the results of this simulation, see Using the simulation output file below.

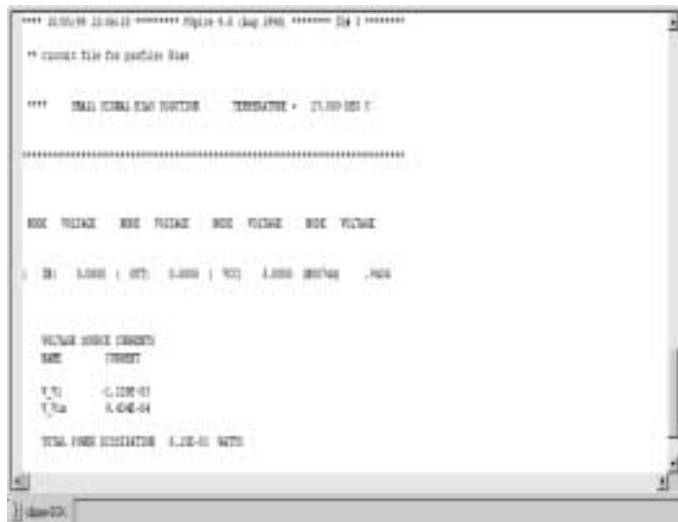
Using the simulation output file

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

To view the simulation output file

- 1 In PSpice, from the View menu, choose Output File.

Figure 2-4 shows the results of the bias point calculation as written in the simulation output file.



The screenshot shows a window titled 'Output File' displaying the following text:

```
**** SIMULATION REPORT **** PSpice 10.0 (Aug 1999) ****  
**** CONDUCT FILE FOR BIAS POINT CALC ****  
**** SMALL SIGNAL BIAS POINT CALC **** TEMPERATURE = 27.000000 C  
*****  
BIAS POINT RESULTS  
-----  
NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE  
1 20 0.000000 470 0.000000 700 0.000000 8000 0.000000  
  
VOLTAGE SOURCE CURRENTS  
NAME CURRENT  
V12 -6.128E-03  
V12a 6.62E-04  
  
TOTAL POWER DISSIPATION 6.12E-02 WATT
```

Figure 2-4 Simulation output file.

- 2 When finished, close the window.

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

Finding out more about bias point calculations

To find out more about this...	See this...
Bias point calculations	Bias point on page 409

DC sweep analysis

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

Setting up and running a DC sweep analysis

To set up and run a DC sweep analysis

- 1 In Capture, from the PSpice menu, choose New Simulation Profile.
The New Simulation dialog box appears.
- 2 In the Name text box, type DC Sweep.
- 3 From the Inherit From list, select Schematic1-Bias, then click Create.

The Simulation Settings dialog box appears.

- 4 Click the Analysis tab.
- 5 From the Analysis type list, select DC Sweep and enter the values shown in Figure 2-5.

Note: The default settings for DC Sweep simulation are Voltage source as the swept variable type and Linear as the sweep type. To use a different swept variable type or sweep type, choose different options under Sweep

variable and Sweep type.



Figure 2-5 DC sweep analysis settings.

- 6 Click OK to close the Simulation Settings dialog box.
- 7 From the File menu, choose Save.
- 8 From the PSpice menu, choose Run to run the analysis.

Displaying DC analysis results

Probe windows can appear during or after the simulation is finished.

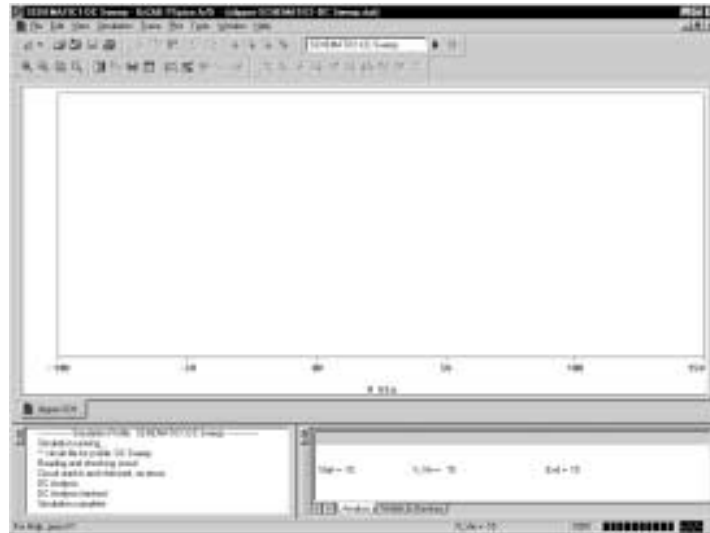


Figure 2-7 Probe window.

To plot voltages at nets In and Mid

- 1 From PSpice's Trace menu, choose Add Trace.
- 2 In the Add Traces dialog box, select V(In) and V(Mid).
- 3 Click OK.

To display a trace using a marker

- 1 From Capture's PSpice menu, point to Markers and choose Voltage Level.

- Click to place a marker on net Out, as shown in Figure 2-8.

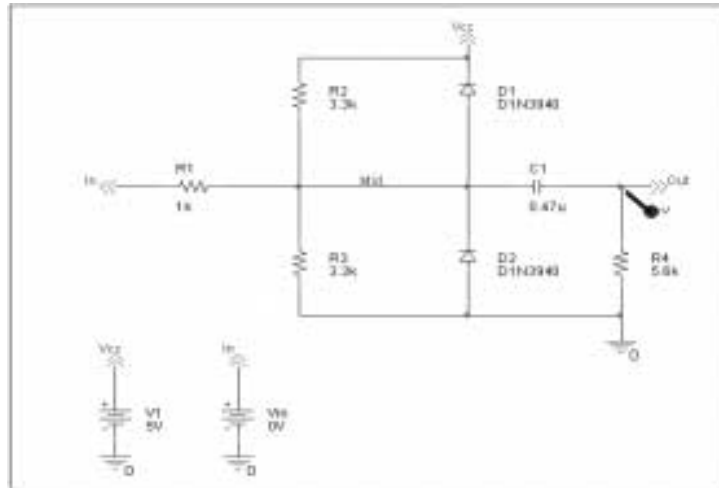


Figure 2-8 Clipper circuit with voltage marker on **Out**.

- Right-click and choose End Mode to stop placing markers.
- From the File menu, choose Save.
- Switch to PSpice. The V(Out) waveform trace appears, as shown in Figure 2-9.

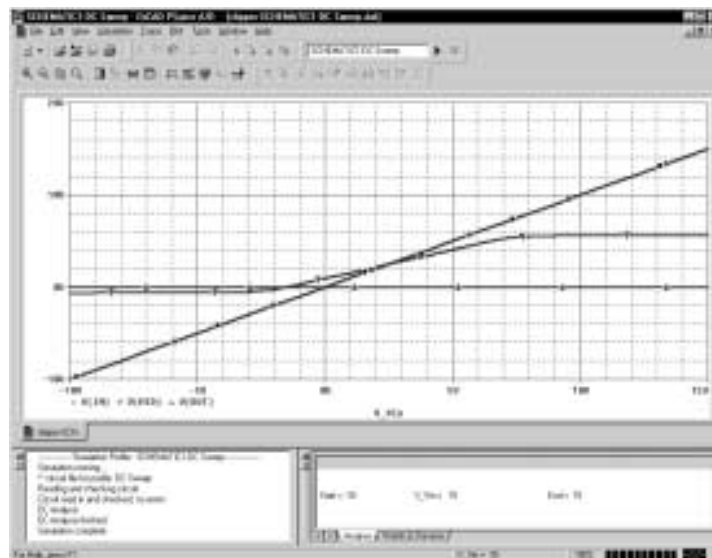


Figure 2-9 Voltage at In, Mid, and Out.

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

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

- 1 From PSpice's Trace menu, point to Cursor and choose Display.

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

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

Table 2-1 Association of cursors with mouse buttons.

cursor 1	left mouse button
cursor 2	right mouse button

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


 V(In) ♦ V(Mid) ▽ V(Out)

Figure 2-10 Trace legend with cursors activated.

- 3 Place the first cursor on the V(In) waveform:
 - a. Click the portion of the V(In) trace in the proximity of 4 volts on the x-axis. The cursor crosshair appears,

and the current X and Y values for the first cursor appear in the cursor window.

- b. To fine-tune the cursor location to 4 volts on the x-axis, drag the crosshairs until the x-axis value of the A1 cursor in the cursor window is approximately 4.0. You can also press Right arrow key and Left arrow key for tighter control.

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

- 4 Place the second cursor on the V(Mid) waveform:
 - a. Right-click the trace legend part (diamond) for V(Mid) to associate the second cursor with the Mid waveform. The crosshair pattern for the second cursor outlines the V(Mid) trace part as shown in Figure 2-11.



V(In) V(Mid) V(Out)

Figure 2-11 Trace legend with V(Mid) symbol outlined.

- b. Right-click the portion on the V(Mid) trace that is in the proximity of 4 volts on the x-axis. The X and Y values for the second cursor appear in the cursor window along with the difference (dif) between the two cursors' X and Y values.
- c. To fine-tune the location of the second cursor to 4 volts on the x-axis, drag the crosshairs until the x-axis value of the A2 cursor in the cursor window is approximately 4.0. You can also press *Shift*+Right arrow key and *Shift*+Left arrow key for tighter control.

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

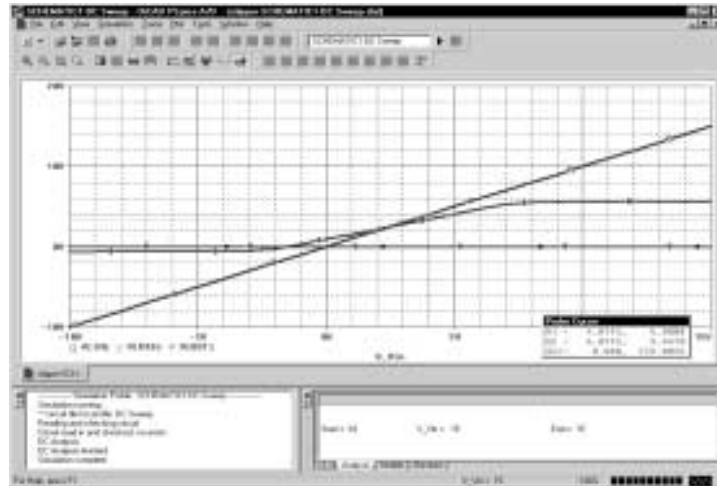


Figure 2-12 Voltage difference at $V(\text{In}) = 4$ volts.

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

- In PSpice, add the trace expression $V(\text{In})-V(\text{Mid})$.
- In Capture, from the PSpice menu, point to Markers and choose Voltage Differential. Place the two markers on different pins or wires.

To delete all of the traces

- 1 From the Trace menu, choose Delete All Traces.

Note: You can also delete an individual trace by selecting its name in the trace legend and then pressing Delete. Example: To delete the $V(\text{In})$ trace, click the text, $V(\text{In})$, located under the plot's x-axis, and then press *Delete*.

At this point, the design has been saved. If needed, you can quit Capture and PSpice and complete the remaining analysis exercises later using the saved design.

Finding out more about DC sweep analysis

To find out more about this...

See this...

DC sweep analysis

[DC Sweep](#) on page 400

Transient analysis

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

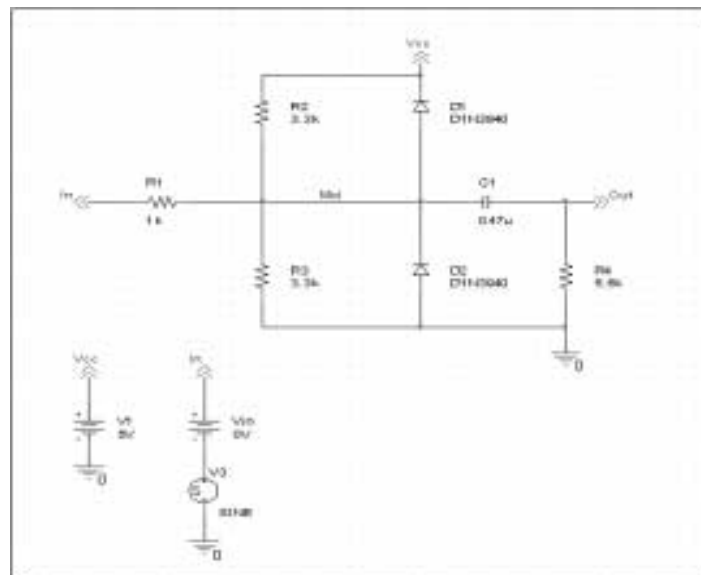


Figure 2-13 Diode clipper circuit with a voltage stimulus.

To add a time-domain voltage stimulus

- 1 From Capture's PSpice menu, point to Markers and choose Delete All.
- 2 Select the ground part beneath the VIN source.
- 3 From the Edit menu, choose Cut.

- 4 Scroll down (or from the View menu, point to Zoom, then choose Out).
- 5 Place a VSTIM part (from the PSpice library SOURCSTM.OLB) as shown in Figure 2-13.
- 6 From the Edit menu, choose Paste.
- 7 Place the ground part under the VSTIM part as shown in Figure 2-13.
- 8 From the View menu, point to Zoom, then choose All.
- 9 From the File menu, choose Save to save the design.

To set up the stimulus

Note: The Stimulus Editor is not available in PSpice A/D Basics. If you are using PSpice A/D Basics, see [If you do not have the Stimulus Editor](#) on page 81.

- 1 Select the VSTIM part (V3).
- 2 From the Edit menu, choose PSpice Stimulus.
The New Stimulus dialog box appears.
- 3 In the New Stimulus dialog box, type `SINE`.
- 4 Click SIN (sinusoidal), then click OK.
- 5 In the SIN Attributes dialog box, set the first three properties as follows:
Offset Voltage = 0
Amplitude = 10
Frequency = 1kHz
- 6 Click Apply to view the waveform.

The Stimulus Editor window should look like Figure 2-14.

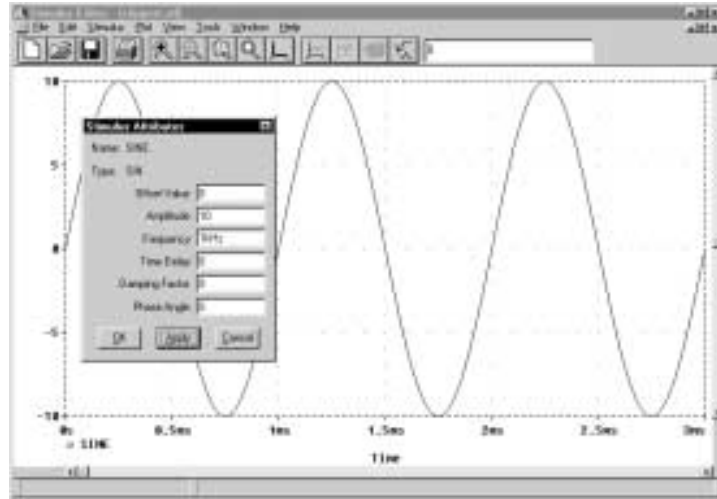


Figure 2-14 Stimulus Editor window.

- 7 Click OK.
- 8 From the File menu, choose Save to save the stimulus information. Click Yes to update the schematic.
- 9 From the File menu, choose Exit to exit the Stimulus Editor.

If you do not have the Stimulus Editor

- 1 Place a VSIN part instead of VSTIM and double-click it.
- 2 In the Edit Part dialog box, click User Properties.
- 3 Set values for the VOFF, VAMPL, and FREQ properties as defined in step 5. When finished, click OK.

To set up and run the transient analysis

- 1 From Capture's PSpice menu, choose New Simulation Profile.
The New Simulation dialog box appears.
- 2 In the Name text box, type Transient.

- 3 From the Inherit From list, select Schematic1-DC Sweep, then click Create.

The Simulation Settings dialog box appears.

- 4 Click the Analysis tab.
- 5 From the Analysis list, select Time Domain (Transient) and enter the settings shown in Figure 2-15.



Figure 2-15 Transient analysis simulation settings.

TSTOP = 2ms

Start saving data after = 20ns

- 6 Click OK to close the Simulation Settings dialog box.
- 7 From the PSpice menu, choose Run to perform the analysis.

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

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

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

- 1 From PSpice's Trace menu, choose Add Trace.
- 2 In the trace list, select V(In) and V(Out) by clicking them.

- 3 Click OK to display the traces.
- 4 From the Tools menu, choose Options to display the Probe Options dialog box.
- 5 In the Use Symbols frame, click Always if it is not already enabled.
- 6 Click OK.

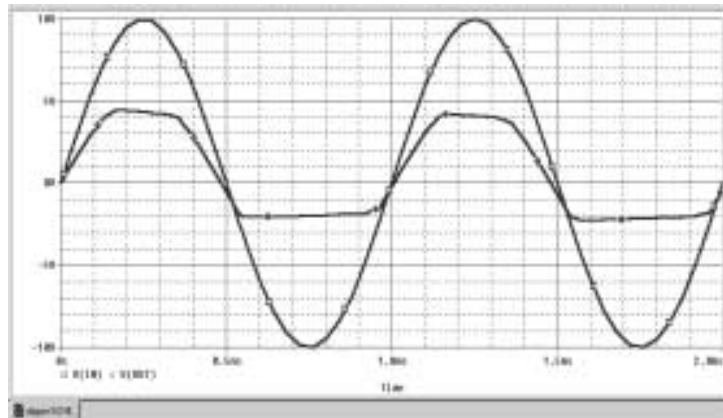


Figure 2-16 Sinusoidal input and clipped output waveforms.

The waveforms illustrate the clipping of the input signal.

Finding out more about transient analysis

To find out more about this...	See this...
transient analysis for analog and mixed-signal designs ¹	Chapter 12, “Transient analysis”
transient analysis for digital designs ¹	Chapter 14, “Digital simulation”

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

AC sweep analysis

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

Setting up and running an AC sweep analysis

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

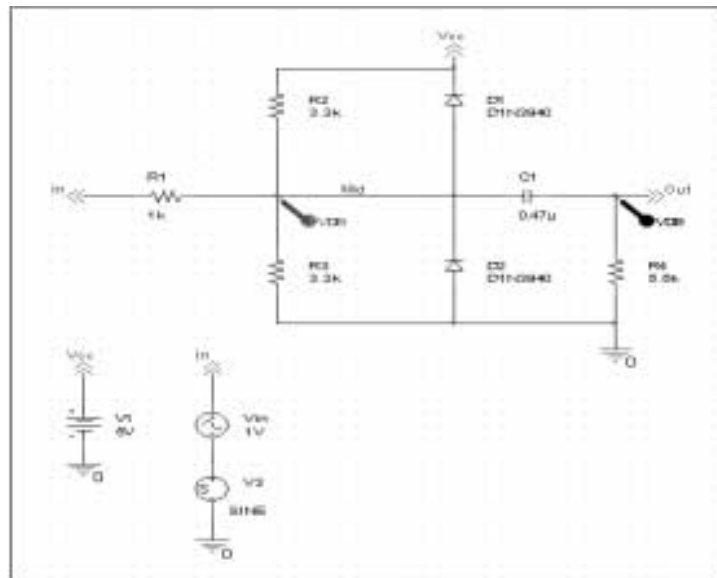


Figure 2-17 Clipper circuit with AC stimulus.

To change Vin to include the AC stimulus signal

- 1 In Capture, open CLIPPER.OPJ.
- 2 Select the DC voltage source, Vin, and press D to remove the part from the schematic page.
- 3 From the Place menu, choose Part.
- 4 In the Part text box, type VAC (from the PSpice library SOURCE.OLB) and click OK.

- 5 Place the AC voltage source on the schematic page, as shown in Figure 2-17.
- 6 Double-click the VAC part (0V) to display the Parts spreadsheet.
- 7 Change the Reference cell to V_{in} and change the ACMAG cell to 1V.

Note: PSpice simulation is not case-sensitive, so both M and m can be used as “milli,” and MEG, Meg, and meg can all be used for “mega.” However, waveform analysis treats M and m as mega and milli, respectively.

- 8 Click Apply to update the changes and then close the spreadsheet.

To set up and run the AC sweep simulation

- 1 From Capture's PSpice menu, choose New Simulation Profile.
- 2 In the Name text box, enter AC Sweep, then click create. The Simulation Settings dialog box appears.
- 3 Click the Analysis tab.
- 4 From the Analysis type list, select AC Sweep/Noise and enter the settings shown in Figure 2-18.

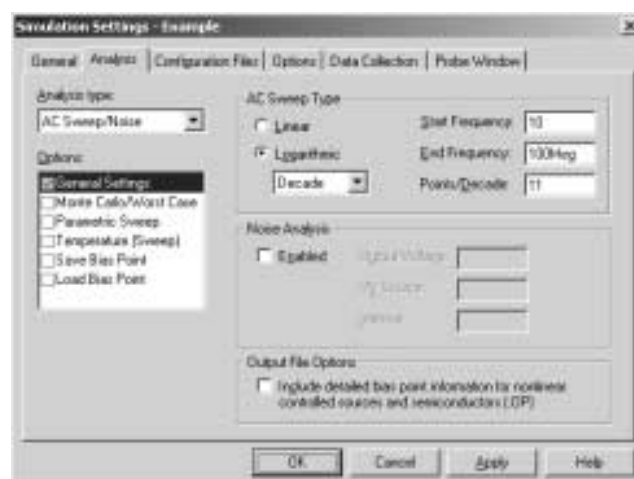


Figure 2-18 AC sweep and noise analysis simulation settings.

- 5 Click OK to close the Simulation Settings dialog box.
- 6 From the PSpice menu, choose Run to start the simulation.

PSpice performs the AC analysis.

To add markers for waveform analysis

- 1 From Capture's PSpice menu, point to Markers, point to Advanced, then choose db Magnitude of Voltage.

Note: You must first define a simulation profile for the AC Sweep/Noise analysis in order to use advanced markers.

- 2 Place one Vdb marker on the Out net, then place another on the Mid net.
- 3 From the File menu, choose Save to save the design.

AC sweep analysis results

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

to 1V, the output voltage is the same as the gain (or attenuation) of the circuit.

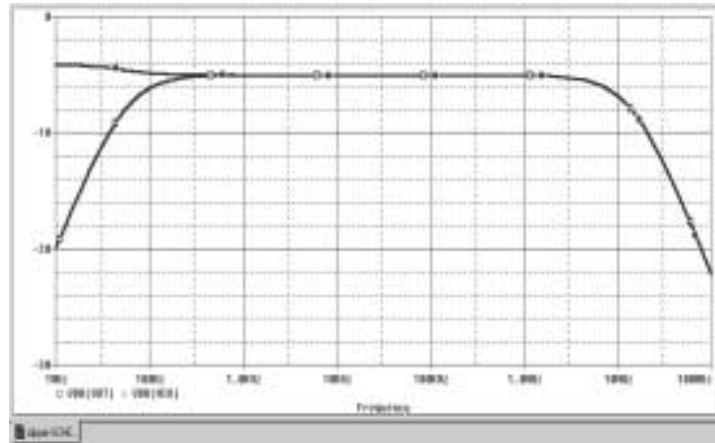


Figure 2-19 dB magnitude curves for “gain” at Mid and Out.

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

- 1 From Capture’s PSpice menu, point to Markers, point to Advanced and choose Phase of Voltage.
- 2 Place a Vphase marker on the output next to the Vdb marker.

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

- 3 Delete the Vdb marker on Mid.
- 4 Switch to PSpice.

In the Probe window, the gain and phase plots both appear on the same graph with the same scale.

For more information on Probe windows and trace expressions, see [Chapter 17, “Analyzing waveforms.”](#)

- 5 Click the trace name VP(Out) to select the trace.
- 6 From the Edit menu, choose Cut.

- 7 From the Plot menu, choose Add Y Axis.
- 8 From the Edit menu, choose Paste.

The Bode plot appears, as shown in Figure 2-20.

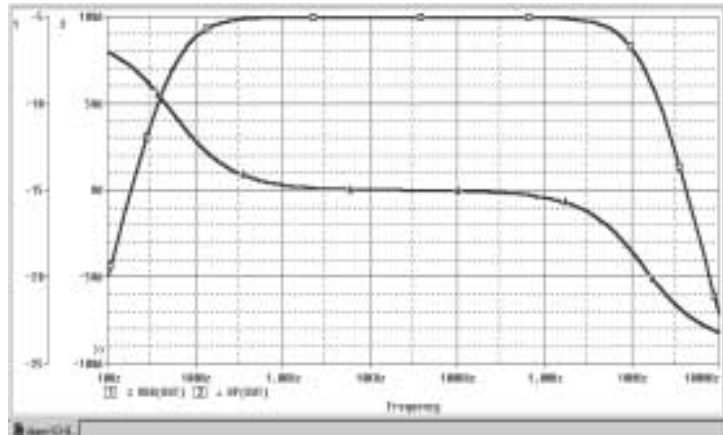


Figure 2-20 Bode plot of clipper's frequency response.

Finding out more about AC sweep and noise analysis

To find out more about this...	See this...
AC sweep analysis	AC sweep analysis on page 418
noise analysis based on an AC sweep analysis	Noise analysis on page 428

Parametric analysis

Note: Parametric analysis is not included in PSpice A/D Basics.

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

- Changing the value of R1 to the expression {Rval}.
- Placing a PARAM part to declare the parameter Rval.
- Setting up and running a parametric analysis to step the value of R1 using Rval.

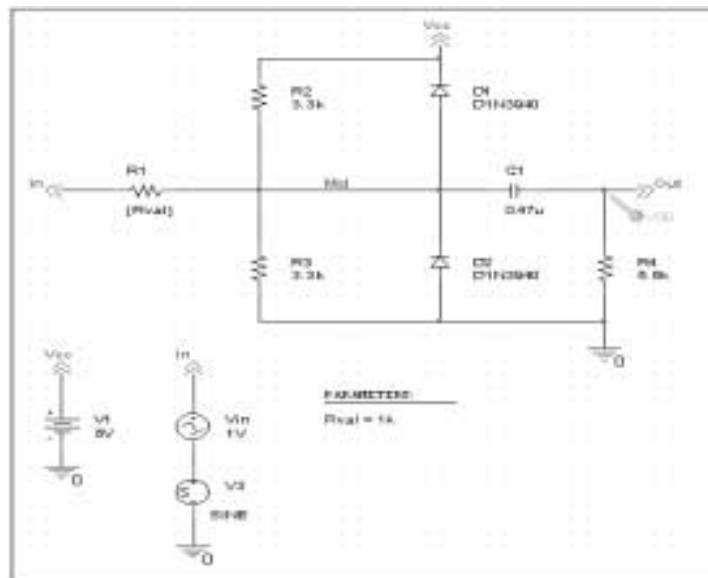


Figure 2-21 Clipper circuit with global parameter Rval.

This example produces multiple analysis runs, each with a different value of R1. After the analysis is complete, you can analyze curve families for the analysis runs using PSpice A/D.

Setting up and running the parametric analysis

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

- 1 In Capture, open `CLIPPER.OPJ`.
- 2 Double-click the value (1k) of part R1 to display the Display Properties dialog box.
- 3 In the Value text box, replace 1k with `{Rval}`.

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

- 4 Click OK.

To add a PARAM part to declare the parameter Rval

- 1 From Capture's Place menu, choose Part.
- 2 In the Part text box, type `PARAM` (from the PSpice library `SPECIAL.OLB`), then click OK.
- 3 Place one PARAM part in any open area on the schematic page.

- 4 Double-click the PARAM part to display the Parts spreadsheet, then click New.

For more information about using the Parts spreadsheet, see the *OrCAD Capture User's Guide*.

- 5 In the Property Name text box, enter `Rval` (no curly braces), then click OK.

This creates a new property for the PARAM part, as shown by the new column labeled Rval in the spreadsheet.

- 6 Click in the cell below the Rval column and enter 1k as the initial value of the parametric sweep.
- 7 While this cell is still selected, click Display.

- 8 In the Display Format frame, select Name and Value, then click OK.
- 9 Click Apply to update all the changes to the PARAM part.
- 10 Close the Parts spreadsheet.
- 11 Select the VP marker and press *Delete* to remove the marker from the schematic page.
Note: This example is only interested in the magnitude of the response.
- 12 From the File menu, choose Save to save the design.

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

- 1 From Capture's PSpice menu, choose New Simulation Profile.
The New Simulation dialog box appears.
- 2 In the Name text box, type *Parametric*.
- 3 From the Inherit From list, select AC Sweep, then click Create.
The Simulation Settings dialog box appears.
The root schematic listed is the schematic page associated with the simulation profile you are creating.
- 4 Click the Analysis tab.

- 5 Under Options, select Parametric Sweep and enter the settings as shown below.



Figure 2-22 Parametric simulation settings.

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

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

- 6 Click OK.
- 7 From the PSpice menu, choose Run to start the analysis.

Analyzing waveform families

Continuing from the example above, there are 21 analysis runs, each with a different value of R1. After PSpice completes the simulation, the Available Sections dialog box appears, listing all 21 runs and the Rval parameter value for each. You can select one or more runs to display. To select individual runs, click each one separately.

To display all 21 traces

- 1 In the Available Sections dialog box, click OK.

All 21 traces (the entire family of curves) for VDB(Out) appear in the Probe window as shown in Figure 2-23.

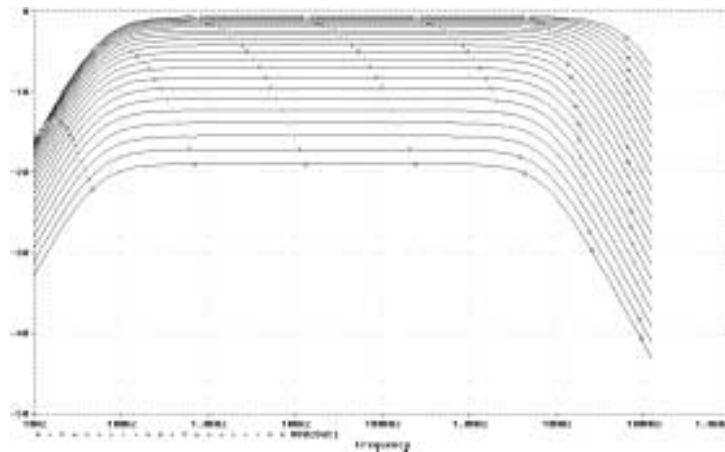


Figure 2-23 Small signal response as R1 is varied from 100Ω to 10 kΩ.

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

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

You can also remove the traces by removing the VDB marker from your schematic page in Capture.

To compare the last run to the first run

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

- 2 In the Trace Expression text box, type the following:

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



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

- 3 Click OK.

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

- 4 Plot the new trace by specifying a waveform expression:

- a. From the Trace menu, choose Add Trace.

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

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

- c. Click OK.

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

- a. From the Trace menu, point to Cursor and choose Display.

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

- c. From the Trace menu, point to Cursor and choose Max.

- d. From the Trace menu, point to Cursor and choose Search Commands.
- e. In the Search Command text box, type the following:

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

The search command tells PSpice to search for the point on the trace where the x-axis value is 100.
- f. Select 2 as the Cursor to Move option.
- g. Click OK.

Figure 2-24 shows the Probe window with cursors placed.

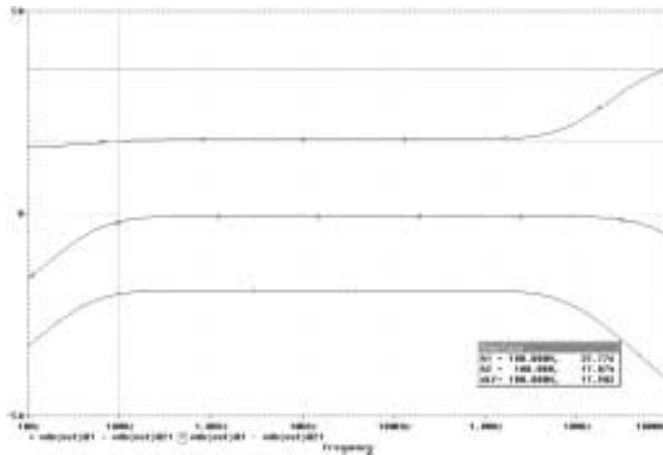


Figure 2-24 Small signal frequency response at 100 and 10 k Ω input resistance.

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

- 6 From the Trace menu, point to Cursor and choose Display to turn off the display of the cursors.
- 7 Delete the trace.

Finding out more about parametric analysis

To find out more about this...	See this...
parametric analysis	Parametric analysis on page 438
using global parameters	Using global parameters and expressions for values on page 117

Performance analysis

Note: Performance analysis is not available in PSpice A/D Basics.

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

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

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

To plot bandwidth vs. Rval using the performance analysis wizard

- 1 In Capture, open `CLIPPER.OPJ`.
- 2 From PSpice's Trace menu, choose Performance Analysis.

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

Note: The Performance Analysis menu item is only available if an analysis data file is available. In this case, the data from the parametric analysis of the previous example should still be open.

- 3 Click the Wizard button.

At each step, the wizard provides information and guidelines.

- 4 Click the Next> button.

- 5 In the Choose a Goal Function list, click Bandwidth, then click the Next> button.
- 6 Click in the Name of Trace to search text box and type $V(\text{Out})$.
- 7 Click in the db level down for bandwidth calc text box and type 3.
- 8 Click the Next> button.

The wizard displays the gain trace for the first run ($R=100$) and shows how the bandwidth is measured. This is done to test the goal function.

- 9 Click the Next> button or the Finish button.
A plot of the 3dB bandwidth vs. Rval appears.
- 10 Change the x-axis to log scale:
 - a. From the Plot menu, choose Axis Settings.
 - b. Click the X Axis tab.
 - c. Under Scale, choose Log.
 - d. Click OK.

To plot gain vs. Rval manually

- 1 From the Plot menu, choose Add Y Axis.
- 2 From the Trace menu, choose Add to display the Add Traces dialog box.
- 3 In the Functions or Macros frame, select the Measurements list, and then click the Max(1) goal function.

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

- 4 In the Simulation Output Variables list, click $V(\text{out})$.
- 5 In the Trace Expression text box, edit the text to be $\text{Max}(V_{\text{db}}(\text{out}))$, then click OK.

PSpice displays gain on the second y-axis vs. Rval.

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

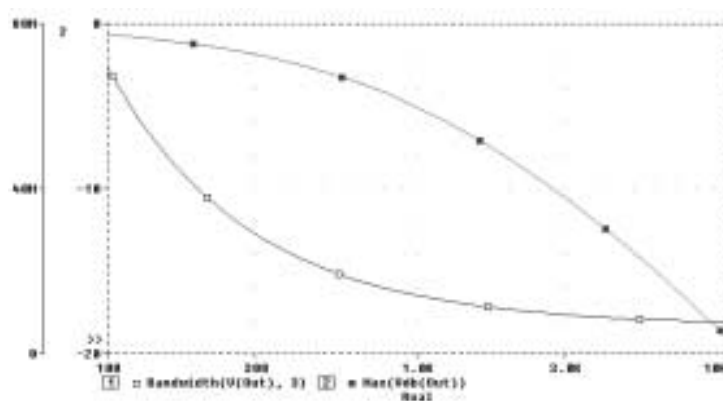


Figure 2-25 Performance analysis plots of bandwidth and gain vs. Rval.

Finding out more about performance analysis

To find out more about this...

how to use performance analysis

how to use search commands and create measurement expressions

See this...

[RLC filter example](#) on page 439

[Example: Monte Carlo analysis of a pressure sensor](#) on page 486

[Chapter 18, "Measurement expressions"](#)