
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 2-56](#)
- [Performing a bias point analysis on page 2-62](#)
- [DC sweep analysis on page 2-66](#)
- [Transient analysis on page 2-72](#)
- [AC sweep analysis on page 2-77](#)
- [Parametric analysis on page 2-82](#)
- [Performance analysis on page 2-89](#)

Example circuit creation

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

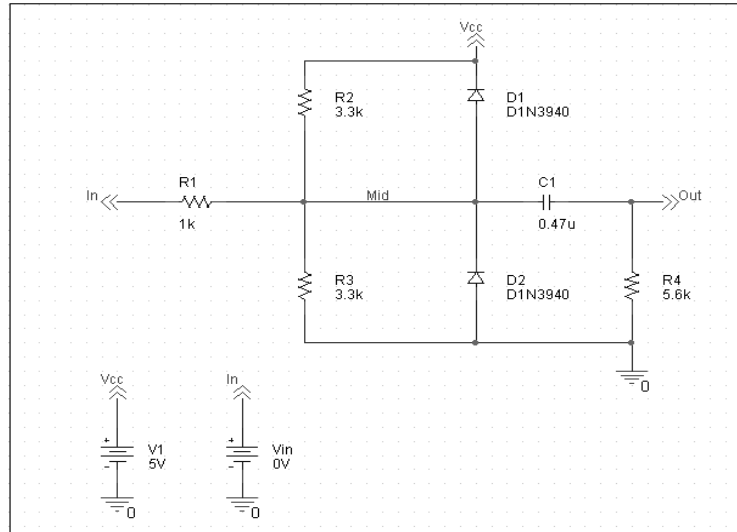


Figure 2 *Diode clipper circuit.*

To create a new PSpice project

- 1 From the Windows Start menu, choose the OrCAD Release 9 program 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 Click OK, then click Finish.

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.
- 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) 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.



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

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.
- 4 Select D1N3940 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.



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 later in this section.

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. 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.



- 7 To place the ground parts (0), click the GND button on the tool palette.
- 8 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.
- 9 Place the ground parts according to Figure 2.

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.
- 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 on page 2-56.

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 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 3).

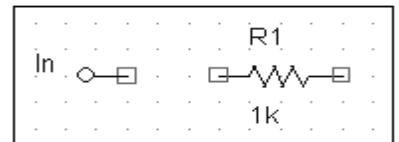


Figure 3 Connection points.

If you make a mistake when placing or connecting components:

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

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

Label the off-page connectors as shown in Figure 2 on page 2-56.

- 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 on page 2-56.

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 on page 2-56.

Your schematic page should now have the same parts, wiring, labels, and properties as Figure 2 on page 2-56.

To save your design

- 1 From the File menu, choose Save.



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:

- 1 Select all of the parts to be modified by pressing **Ctrl** and clicking each part.
- 2 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.

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 A/D

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

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](#).

While PSpice A/D is running, the progress of the simulation appears and is updated in the PSpice A/D simulation output window (see Figure 4).

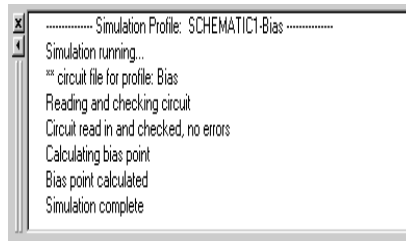


Figure 4 PSpice A/D simulation output window.

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.

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

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

To simulate the circuit from within Capture

- 1 From the PSpice menu, choose Run.



PSpice A/D 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.*

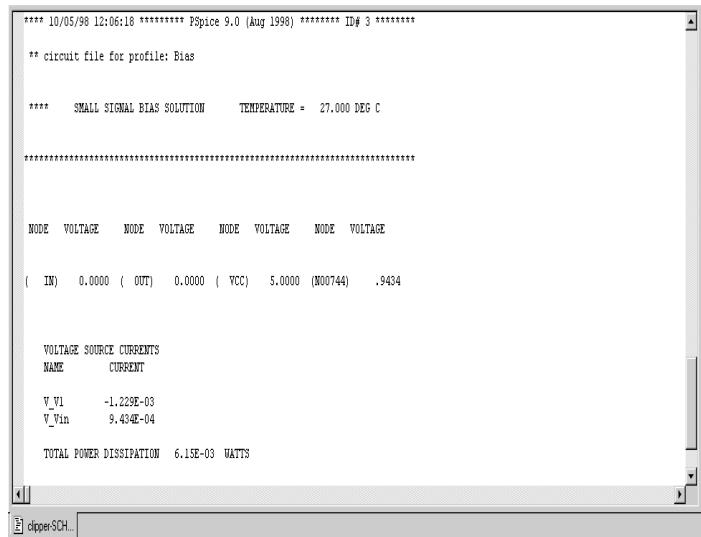
Using the simulation output file

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 A/D writes error or warning messages to the output file.

To view the simulation output file

- 1 From PSpice's View menu, choose Output File.

Figure 5 shows the results of the bias point calculation as written in the simulation output file.



```

**** 10/05/98 12:06:18 ***** PSpice 9.0 (Aug 1998) ***** ID# 3 *****

** circuit file for profile: Bias

**** SMALL SIGNAL BIAS SOLUTION      TEMPERATURE = 27.000 DEG C

*****

NODE  VOLTAGE  NODE  VOLTAGE  NODE  VOLTAGE  NODE  VOLTAGE
( IN)  0.0000  ( OUT) 0.0000  ( VCC) 5.0000  (M00744) .9434

VOLTAGE SOURCE CURRENTS
NAME      CURRENT
V_V1      -1.229E-03
V_V1in    9.434E-04

TOTAL POWER DISSIPATION 6.115E-03 WATTS

```

Figure 5 *Simulation output file.*

- 2 When finished, close the window.

PSpice A/D 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

Table 2-1

To find out more about this...	See this...
bias point calculations	Bias point on page 9-315

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 6.

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.

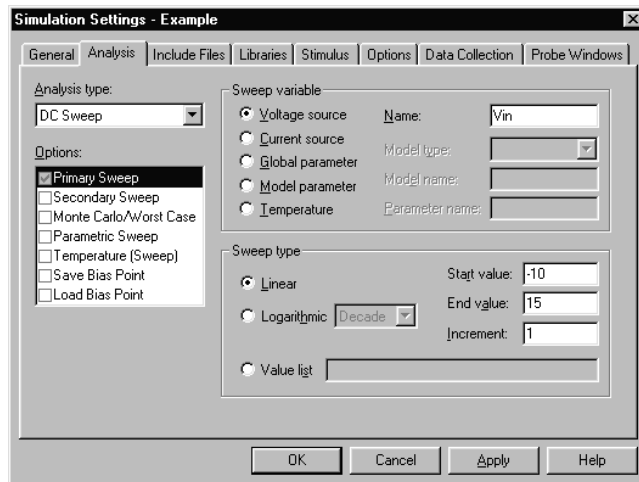


Figure 6 *DC sweep analysis settings.*

- 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.

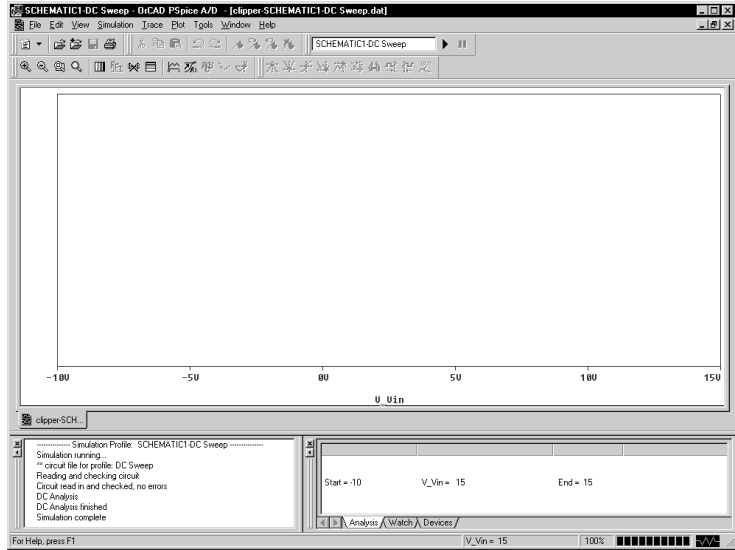


Figure 7 *Probe window.*

To plot voltages at nets In and Mid

press **Insert**



- 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

press **Ctrl + M**

- 1 From Capture's PSpice menu, point to Markers and choose Voltage Level.
- 2 Click to place a marker on net Out, as shown in Figure 8.

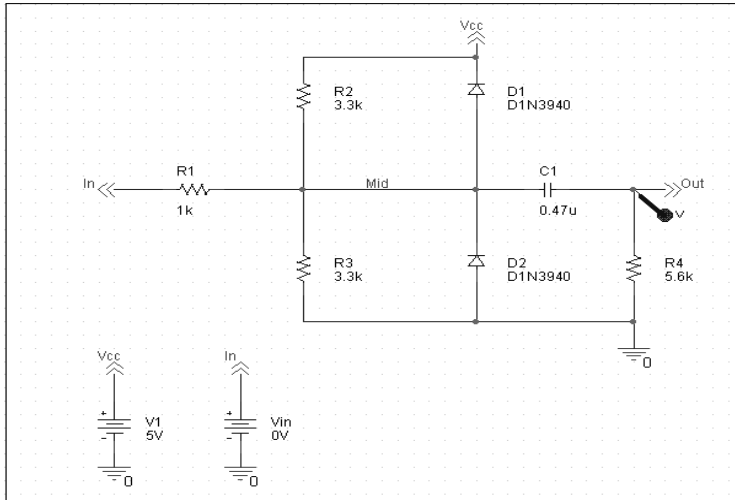


Figure 8 *Clipper circuit with voltage marker on net Out.*

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

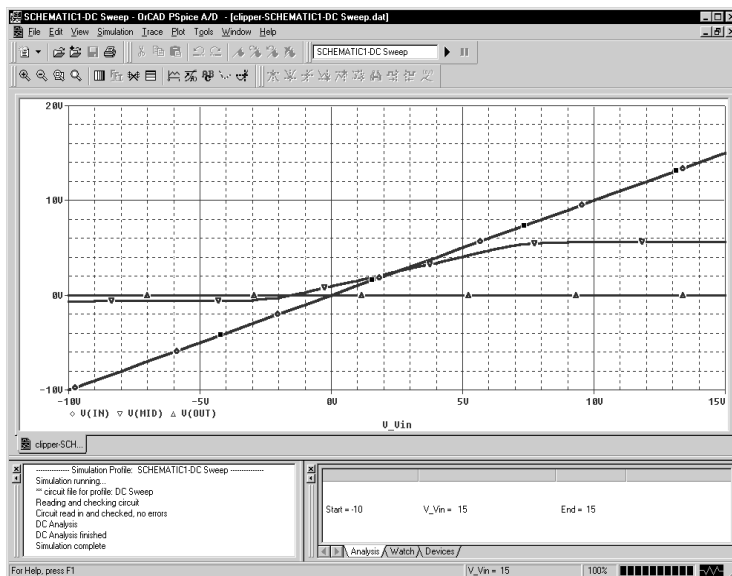


Figure 9 *Voltage at In, Mid, and Out.*

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

Table 10 Association of cursors with mouse buttons.

cursor 1	left mouse button
cursor 2	right mouse button



\square V(In) \diamond V(Mid) ∇ V(Out)

Figure 11 Trace legend with cursors activated.

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



\square V(In) \diamond V(Mid) ∇ V(Out)

Figure 12 Trace legend with V(Mid) symbol outlined.

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

- 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.

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 11.

- 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 \rightarrow and \leftarrow for tighter control.
- 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 12.
 - 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**+**→** and **Shift**+**←** for tighter control.

Figure 13 shows the Probe window with both cursors placed.

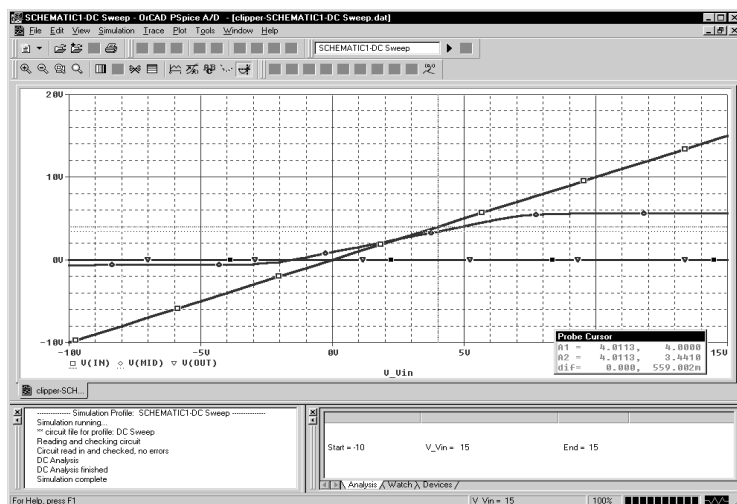


Figure 13 Voltage difference at $V(In) = 4$ volts.

To delete all of the traces

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

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.

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

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

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

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

Finding out more about DC sweep analysis

Table 2-1

To find out more about this...	See this...
DC sweep analysis	DC Sweep on page 9-306

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 14.

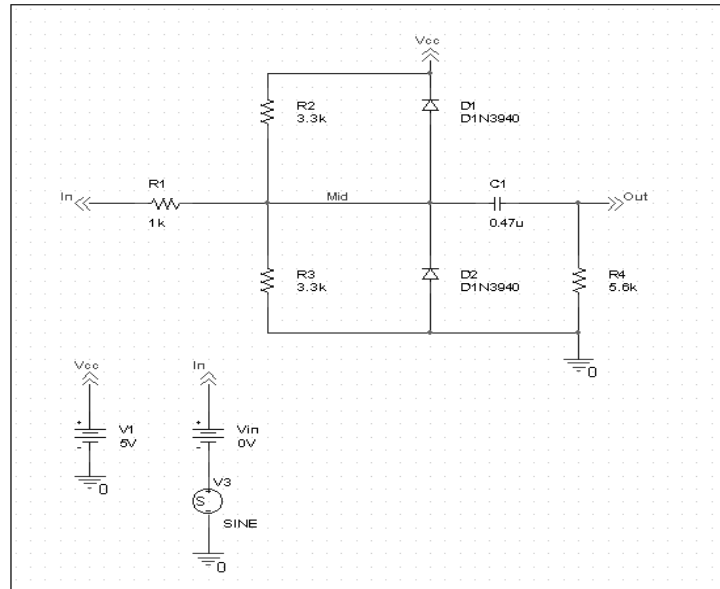


Figure 14 *Diode clipper circuit with a voltage stimulus.*

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 SOURCESTM.OLB) as shown in Figure 14.
- 6 From the Edit menu, choose Paste.
- 7 Place the ground part under the VSTIM part as shown in Figure 14.
- 8 From the View menu, point to Zoom, then choose All.
- 9 From the File menu, choose Save to save the design.



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

To set up the stimulus

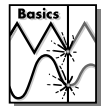
- 1 Select the VSTIM part (V3).
- 2 From the Edit menu, choose PSpice Stimulus.
The New Stimulus dialog box appears.
- 3 In the New Stimulus dialog box, type `SINE`.
- 4 Click SIN (sinusoidal), then click OK.
- 5 In the SIN Attributes dialog box, set the first three properties as follows:

Offset Voltage = 0
Amplitude = 10
Frequency = 1kHz

- 6 Click Apply to view the waveform.

The Stimulus Editor window should look like Figure 15.

Note The Stimulus Editor is not included in PSpice A/D Basics.



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.

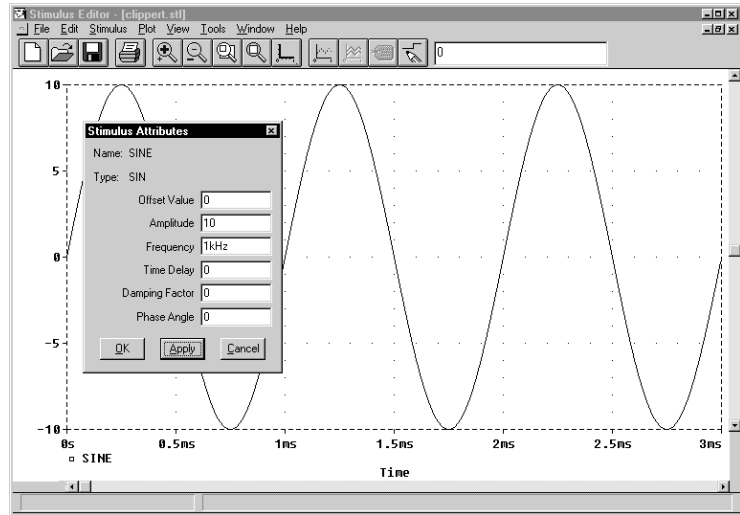


Figure 15 *Stimulus Editor window.*

7 Click OK.

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



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.

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 16.

TSTOP = 2ms

Start saving data after = 20ns

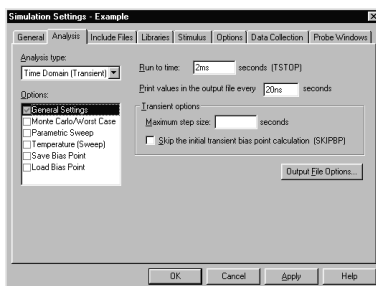


Figure 16 *Transient analysis simulation settings.*

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

PSpice A/D 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 A/D saves data to the waveform data file for each internal time step.



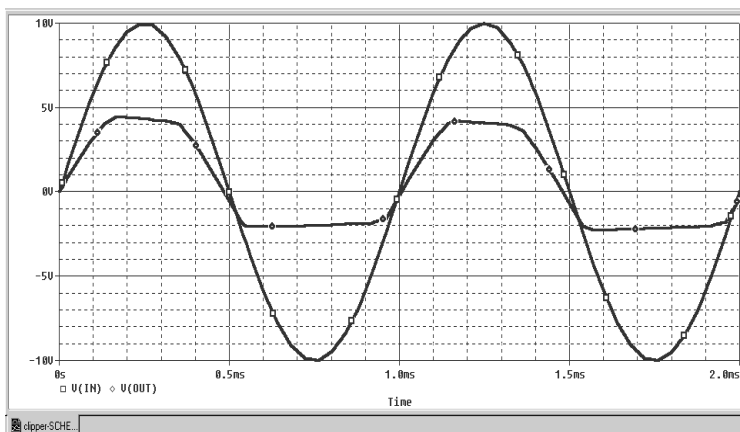
Note The internal time step is different from the Print Step value. Print Step controls how often optional text format data is written to the simulation output file (*.OUT).

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.



or press **Insert**



These waveforms illustrate the clipping of the input signal.

Figure 17 Sinusoidal input and clipped output waveforms.

Finding out more about transient analysis

Table 2-1

To find out more about this...	See this...
transient analysis for analog and mixed-signal designs*	Chapter 11, Transient analysis
transient analysis for digital designs*	Chapter 14, Digital simulation

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

AC sweep analysis

The AC sweep analysis in PSpice A/D 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 18) and by setting up AC sweep parameters.

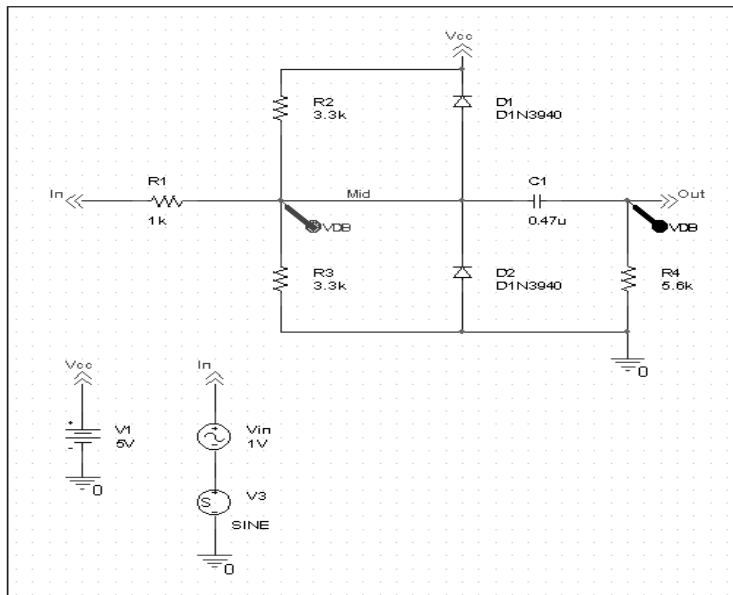


Figure 18 *Clipper circuit with AC stimulus.*

To change Vin to include the AC stimulus signal

- 1 In Capture, open CLIPPER.OPJ.
- 2 Select the DC voltage source, Vin, and press 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 17.
- 6 Double-click the VAC part (0V) to display the Parts spreadsheet.
- 7 Change the Reference cell to `Vin` and change the ACMAG cell to `1V`.
- 8 Click Apply to update the changes and then close the spreadsheet.

To set up and run the AC sweep simulation

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.

- 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 19.

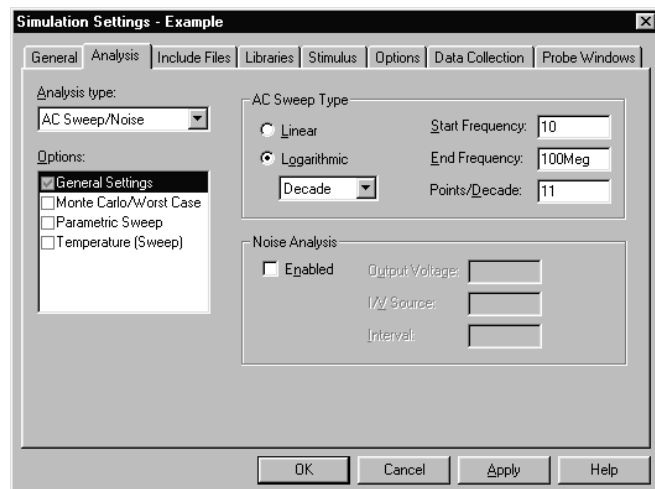


Figure 19 AC sweep and noise analysis simulation settings.

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



PSpice A/D 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.
- 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.

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

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 20 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.

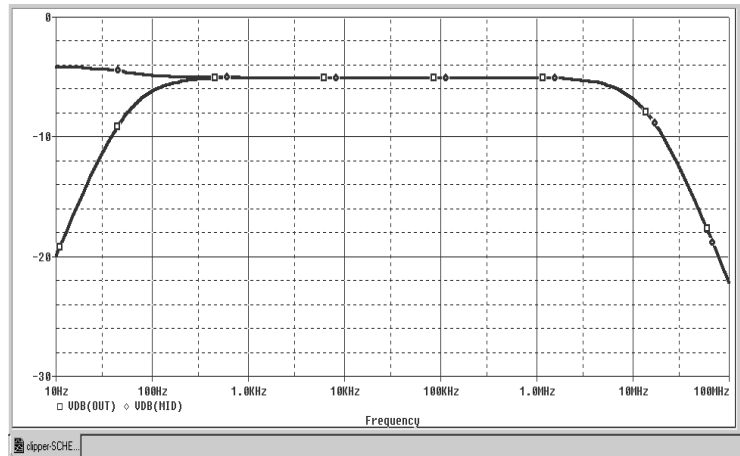


Figure 20 dB magnitude curves for “gain” at Mid and Out.

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

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

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



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



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.
- 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.

- 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 21.

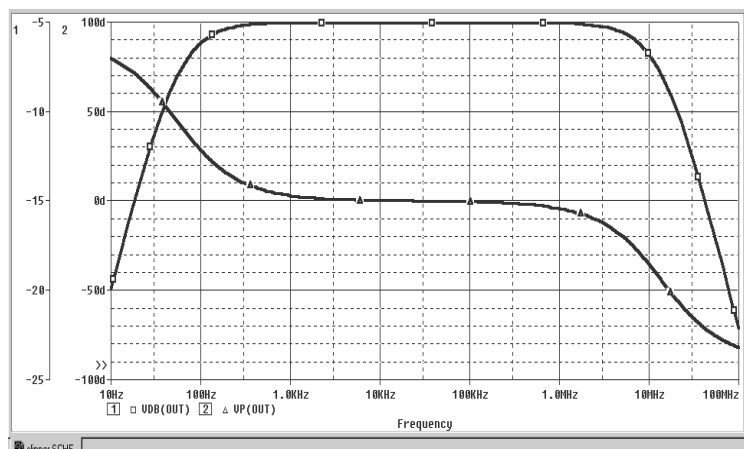
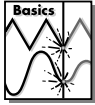


Figure 21 *Bode plot of clipper's frequency response.*

Finding out more about AC sweep and noise analysis

Table 2-2

To find out more about this...	See this...
AC sweep analysis	AC sweep analysis on page 10-324
noise analysis based on an AC sweep analysis	Noise analysis on page 10-333



Note Parametric Analysis is not supported in PSpice A/D Basics.

Parametric analysis

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

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

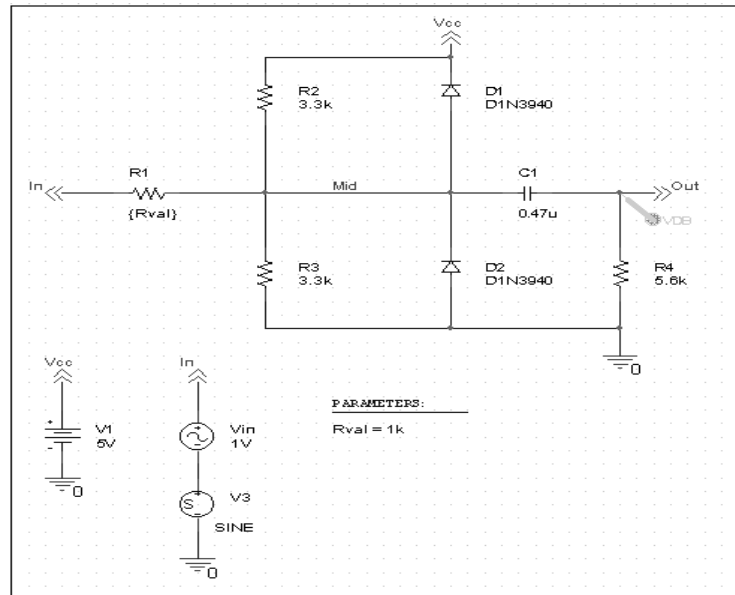


Figure 22 Clipper circuit with global parameter Rval.

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}.
- 4 Click OK.

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

To add a PARAM 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.
- 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 to remove the marker from the schematic page.
- 12 From the File menu, choose Save to save the design.

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

This example is only interested in the magnitude of the response.

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.

- 4 Click the Analysis tab.

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

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

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.

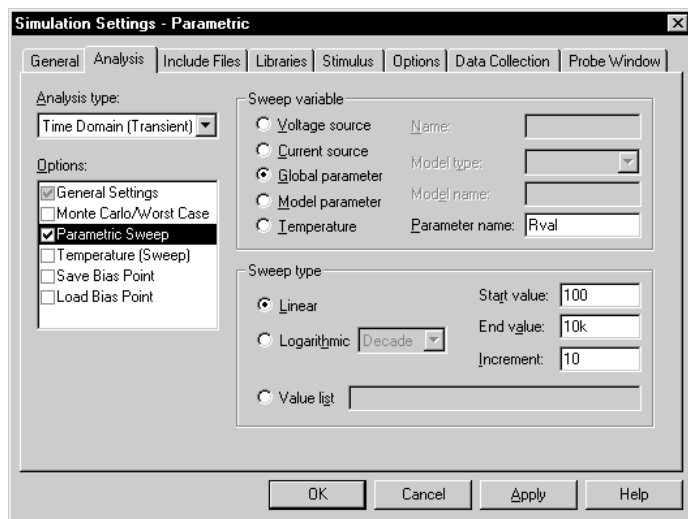


Figure 23 Parametric simulation settings.

- 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 A/D 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 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 24.

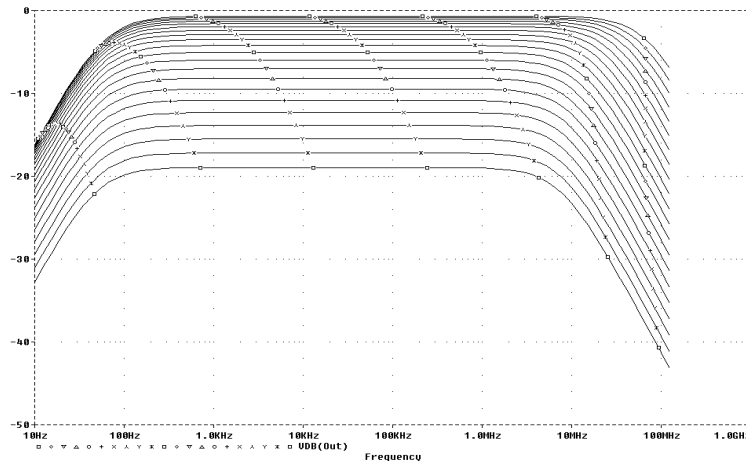


Figure 24 *Small signal response as R1 is varied from 100Ω to 10 kΩ*

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

To select individual runs, click each one separately.

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

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

press **Insert**

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

press **Insert**

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

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
- 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 Tools 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)
 - f Select 2 as the Cursor to Move option.

g Click OK.

Figure 25 shows the Probe window with cursors placed.

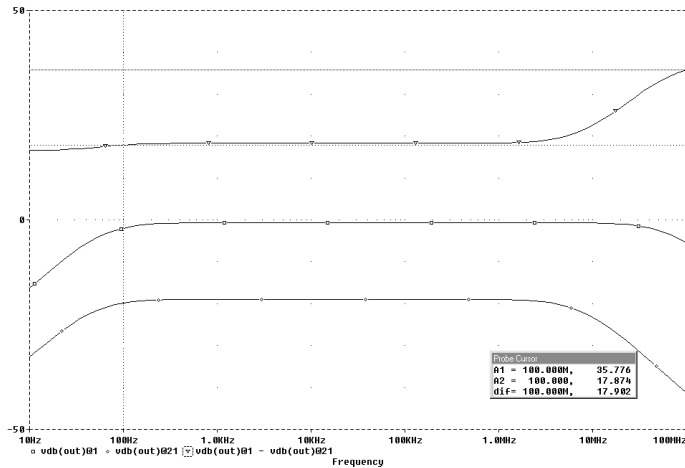


Figure 25 *Small signal frequency response at 100 and 10 k Ω input resistance.*

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

Table 2-3

To find out more about this...	See this...
parametric analysis	<u>Parametric analysis on page 12-364</u>
using global parameters	<u>Using global parameters and expressions for values on page 3-107</u>

Performance analysis

Performance analysis is an advanced feature in PSpice A/D 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.

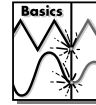
After you define these functions, you can apply them to a family of waveforms and produce traces that are a function of the variable that changed within the family.

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.
- 3 Click the Wizard button.
- 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.



Note *Performance Analysis is not supported in PSpice A/D Basics.*

At each step, the wizard provides information and guidelines.

Click , then double-click $V(\text{Out})$.

Double-click the x-axis.

- 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.



or press **Insert**

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

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 Goal Functions list, and then click the Max(1) goal function.
- 4 In the Simulation Output Variables list, click V(out).
- 5 In the Trace Expression text box, edit the text to be `Max(Vdb(out))`, then click OK.

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

Figure 26 shows the final performance analysis plot of 3dB bandwidth and gain in dB vs. the swept input resistance value.

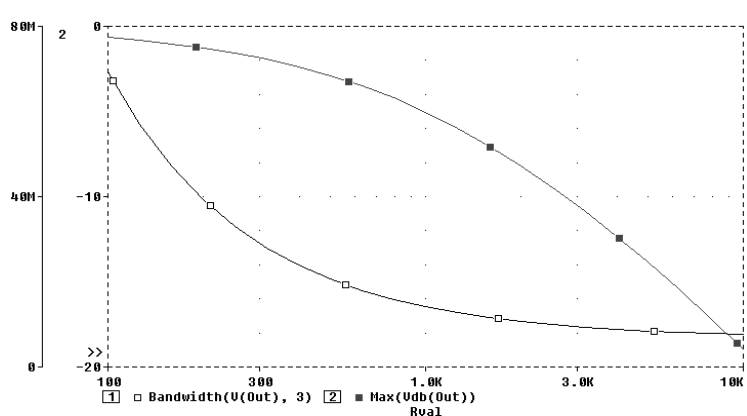


Figure 26 Performance analysis plots of bandwidth and gain vs. Rval.

Finding out more about performance analysis

Table 2-4

To find out more about this...	See this...
how to use performance analysis	RLC filter example on page 12-366 Example: Monte Carlo analysis of a pressure sensor on page 13-385
how to use search commands and create goal functions	PSpice A/D online Help