Example The input to the circuit shown in Figure 1 is the voltage source voltage $v_s(t)$. The response is the voltage, $v_o(t)$, across the 20 kΩ resistor. The network function of this circuit is

$$\mathbf{H}(\omega) = \frac{\mathbf{V}_o(\omega)}{\mathbf{V}_s(\omega)} = -\frac{R_2}{R_1 + j\omega C R_1 R_2} = \frac{-4}{1 + j0.008\,\omega}$$

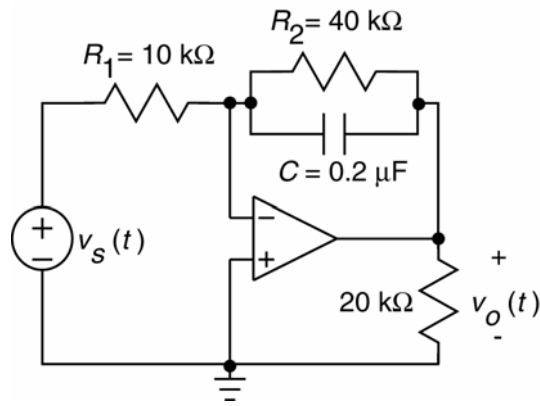Graph the frequency response of this circuit.

Figure 1 A circuit.

Step 1 Formulate a circuit analysis problem.

Obtain the gain and phase Bode plots of the circuit shown in Figure 1.

Step 2 Describe the circuit using Orcad Capture.

Figure 2 shows the circuit as described in Capture.

Two nodes of this circuit have been named using the using a PSpice part called an "off-page connector". The particular off-page connector used in Figure 2 is called a "OFFPAGELEFT-R" part and is found in the part library named CAPSYM. To label a node, select **Place / Off-Page Connector…** from the Orcad capture menus to pop up the "Place Off-Page Connector" dialog box as shown in Figure 3. Select the library CAPSYM from the list of libraries. (If CAPSYM does not appear in the list of libraries, left-click the "Add Libraries" button to pop up a "Browse File" dialog box. Select capsym.olb from the list of libraries in the "Libraries" folder, then left-click the "Open"

button to return to "Place Off-Page Connector" dialog box.) Left-click the "OK" button to place the part in the Orcad workspace.

The new connector will be labeled as "OFFPAGELEFT-R". Use the property editor to change this name to something descriptive such as "Vo". Wire the connector to the appropriate node of the circuit to name that node "Vo".
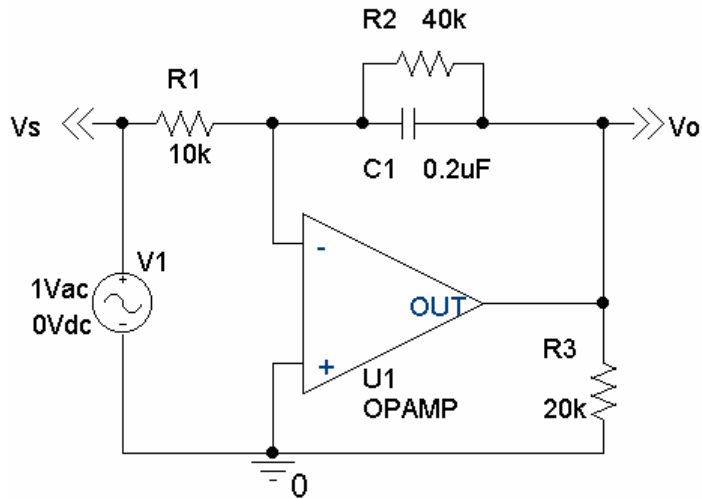

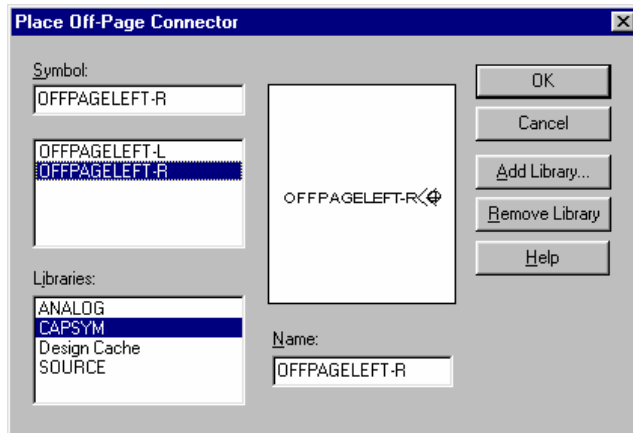**Figure 2** The circuit as described in Orcad Capture.

Figure 3 The Place Off-Page Connector dialog box.

Step 3 Simulate the circuit using PSpice.

Select **PSpice / New Simulation Profile** from the Orcad Capture menus to pop up the "New Simulation" dialog box. Specify a simulation name, then select "Create" to close the "New Simulation" dialog box and pop up the "Simulation Settings" dialog box as shown in Figure 4. In the "Simulation Settings" dialog box, select "AC Sweep/Noise" as the analysis type.

AC Sweep analysis allows us to simulate a circuit over a range of frequencies. Some trial and error may be required to determine an appropriate range of frequencies. In this example, the network function has only one corner frequency, a pole at $\frac{1}{0.008} = 125$ rad/s $= 19.89$ Hertz. It's reasonable to use a "Start Frequency" of 1 Hertz and an "End Frequency" of 1000 Hertz. Select a logarithmic frequency axis to graph the Bode plots. Finally, select "OK" to close the "Simulation Settings" dialog box and return to the Capture screen.

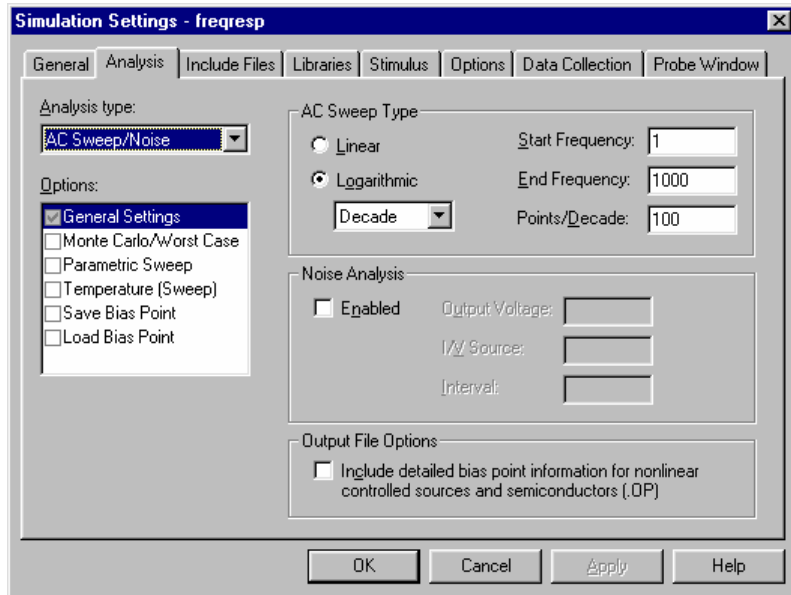Select **PSpice / Run** from the Orcad Capture menus to run the simulation.

Figure 4 The Simulation Settings dialog box.

Step 4 Display the results of the simulation, for example, using Probe.

After a successful AC Sweep simulation, Probe, the graphical post-processor, will open automatically in a Schematics window. Select **Plot / Add Plot to Window** from the Schematics menus to add a second plot. Two empty plots will appear, one above the other. Select the top plot by left-clicking the mouse on the top plot.

Select **Trace / Add Trace** from the Schematics menus to pop up the "Add Traces" dialog box shown in Figure 5. Select first V(VO) and then V(VS) from the list of "Simulation Output Variables". The "Trace Expression", near the bottom of the dialog box, will be "V(VO)V(VS)". Edit the trace expression to be "Vdb(VO)-Vdb(VS)". Because

$$20 \log_{10} \frac{\mathbf{V}_o(\omega)}{\mathbf{V}_s(\omega)} = 20 \log_{10} \mathbf{V}_o(\omega) - 20 \log_{10} \mathbf{V}_s(\omega)$$

Vdb(VO)-Vdb(VS) is the gain in decibles. Left-click "OK" to close the "Add Traces" dialog box.

Select the bottom plot by left-clicking the mouse on the bottom plot. Select **Trace / Add Trace** to pop up the "Add Traces" dialog box. Select first V(VO) and then V(VS) from the list of "Simulation Output Variables". The "Trace Expression", near the bottom of the dialog box, will be "V(VO)V(VS)". Edit the trace expression to be "Vp(VO)-Vp(VS)". Vp(VO)-Vp(VS) is the phase shift in degrees. Left-click "OK" to close the "Add Traces" dialog box. The Bode plots are shown in Figure 6.

Probe provides tools to label some points on the Bode plots. Select the top plot, the gain Bode plot, by left-clicking on it. Select **Trace / Cursor / Display** from the Schematic menus to activate the cursors. (This is a toggle, so selecting **Trace / Cursor / Display** will also deactivate an active cursor.) The cursor is positioned using the arrow keys or by left-clicking on a trace using the mouse. Move the cursor to a low frequency (for example, 1.1415 Hertz is used in Figure 5) to measure the low frequency gain of the circuit. When the cursor is positioned as desired, select **Plot / Label / Mark** from the Schematics menus to mark the point. Deactivate the cursor by selecting **Trace / Cursor / Display** from the Schematics menus. Then position the label as desired using the mouse. The point on the gain Bode plot is labeled as

(1.1415, 12.027)

This label indicates that at a frequency of 1.1415 Hertz the gain is 12.027 dB. The gain will be 3 dB lower at the corner frequency. Activate the cursor and use it to locate and label the point on the gain Bode plot where the gain is 9.027 dB. (The gain changes from one value to another in small increments as the cursor moves along the trace. Figure 6 shows that the gain is 9.0296 dB, the available value closest to 9.027 dB, at a frequency of 19.900 Hertz.)

Select the bottom plot, the phase Bode plot, by left-clicking on it. Activate the cursor and use it to locate and label the point on the phase Bode plot where the angle is 135°. The point on the phase Bode plot is labeled as (19.900, 134.992). This label indicates that at a frequency of 19.900 Hertz the phase shift is 134.992°. The labeled plot is shown in Figure 6.
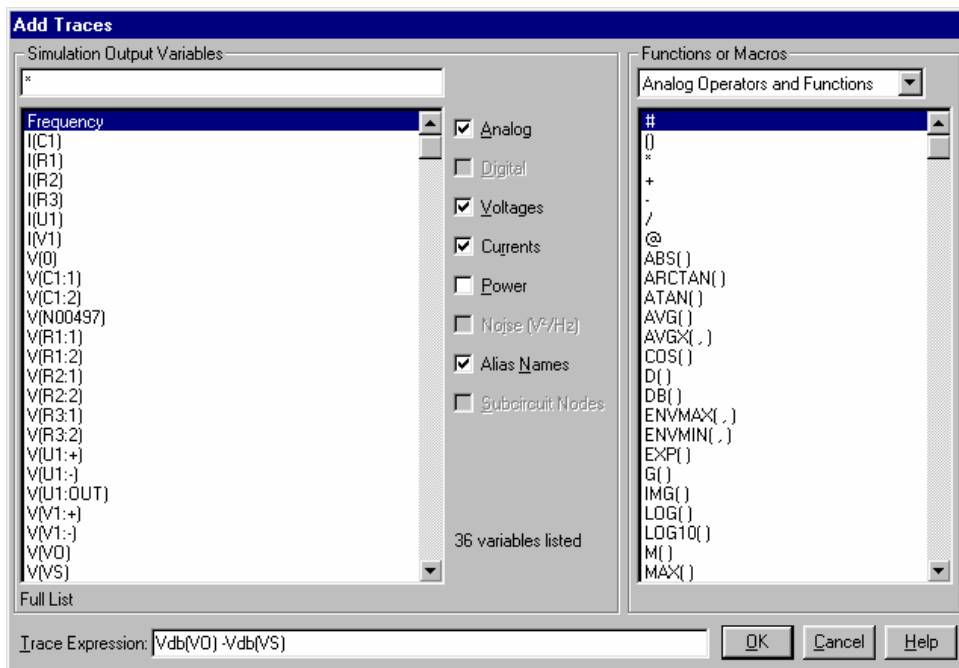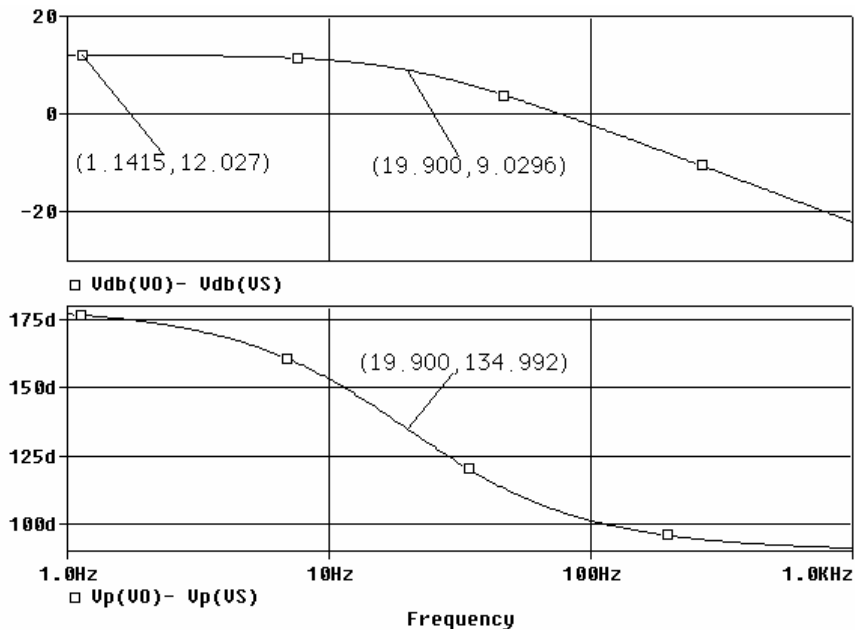

Figure 5 The Add Traces dialog box.

Figure 6 The Gain and Phase Bode plots.

Step 5   Verify that the simulation results are correct.

Let's check to see that the points labeled on the Bode plots agree with the network function of the circuit. The network function of the circuit is

$$\mathbf{H}(\omega) = \frac{\mathbf{V}_o(\omega)}{\mathbf{V}_s(\omega)} = -\frac{R_2}{R_1 + j\omega C R_1 R_2} = \frac{-4}{1 + j0.008\omega} = \frac{-4}{1 + j0.016\pi f}$$

When f = 1.1415 Hertz, $\mathbf{H}(\omega) = 3.993\angle176.7°$. Since 20 $\log_{10}$ 3.993 = 12.027 dB, the calculated gain agrees with the value labeled on the gain Bode plot.

Also, when f = 19.900 Hertz, $\mathbf{H}(\omega) = 2.828\angle134.992°$. The phase angle agrees with the value labeled on the phase Bode plot. Since 20 $\log_{10}$ 2.828 = 9.03 dB, the calculated gain agrees with the value labeled on the gain Bode plot.

The Bode plots agree with the network function so the simulation results are correct.