

OrCAD 10.5 Tutorial

One of the earliest circuit-analysis programs, known as SPICE (an acronym for *Simulation Program with Integrated Circuit Emphasis*), was developed by the Electronics Research Laboratory at the University of California in the early 1970s. Several commercial versions of SPICE have since been produced that include a wide variety of useful extensions to the original program.

In this appendix, we show how to use OrCAD 10.5, a suite of programs by Cadence Design Systems, to analyze most of the types of circuits discussed in this book. This suite of programs is a very sophisticated and powerful tool for electronic-circuit designers.

Using OrCAD 10.5, you can check your answers to many of the problems in this book.

G.1 ANALYSIS OF DC CIRCUITS

First, you should use the OrCAD 10.5 Demo disk included with this book to install the software on your computer.

Then, use the start menu on your computer to start **Capture CIS Demo**, which is located in the **OrCAD 10.5 Demo** program group. (If you wish to learn more about the software after finishing this appendix, you can use the **Help/Learning Capture CIS** command to bring up the Capture tutorial. Then, you can work your way through the lessons to become familiar with additional features of Capture.)

Using Capture/PSpice to Solve DC Circuits

Next, we illustrate how to solve dc circuits using Capture and PSpice. As a first example, we will solve the circuit shown in Figure 2.7(a) on page 55. You will learn best if you follow along on your computer.

First, create your own project folder named **Student OrCAD Projects** in a convenient location on your hard drive. Then, return to the Capture window and use the **File/New/Project** command to bring up the window shown in Figure G.1(a). Type in the name of the project: **Figure 2_7a**. Then, check the **Analog or Mixed A/D** option, click on **Browse . . .**, navigate to your project folder as shown in Figure G.1(a), and left-click on **OK**. This brings up the window shown in Figure G.1(b). Left-click on **Create a blank project**, then on **OK**. This produces the screen shown in Figure G.2, which contains several windows. We will draw the circuit in the window titled **SCHEMATIC1: PAGE 1**.

Menu selections are printed in bold with slashes separating successive selections. Thus, **Help/Learning Capture CIS** indicates that we should place the cursor on **Help** (at the top of the **Capture** window), click the left mouse button to pull down a menu, move the cursor to **Learning Capture CIS**, and click the left mouse button.

This font (Courier) is used to indicate material that you should type in from your keyboard.

Placing Parts

Next, use the **Place/Part** command to bring up the window shown in Figure G.3. The first time you use this program, you will need to add libraries from which parts are selected. To accomplish this, click on **Add Library. . .** as indicated in Figure G.3. Then, in the window that opens, select all of the libraries shown, and click on **Open**. Type the letter **R** in the **Part** window and left-click on **OK**. At this point, you can move the cursor around on the **SCHEMATIC** window and place a resistor each time you click the left mouse button. The orientation of the resistor can be rotated by pressing the “control” and “r” keys simultaneously. Place four resistors as shown in Figure G.4.



Figure G.1 We use these windows to create a new project.

Then, click the right mouse button and select **End Mode** from the pop-up menu. Next, place the voltage source (its name is VDC) by again using the **Place/Part** command.

All of this might take a few tries. The steps (and some helpful tips) are as follows:

1. Use the **Place/Part** command to bring up the window from which parts can be selected. For now, we only need R for resistors and VDC for dc voltage sources.
2. After selecting the desired part, click on **OK**. If necessary, depress the “control” and “r” keys simultaneously to rotate the part to the desired orientation. Then, move the cursor to the desired location and left-click to place a part. After all the parts of a given type are placed, click the right mouse button and select **End Mode** on the pop-up menu.
3. If you accidentally get an unneeded part, you can eliminate it after you leave the place-part mode. Select the unwanted part by placing the cursor on it and clicking the left mouse button. Then, press the delete key.
4. If a part has the wrong name (such as R7 instead of R4), change it after you leave the place-part mode. Double left-click on the name and type in the correct name.

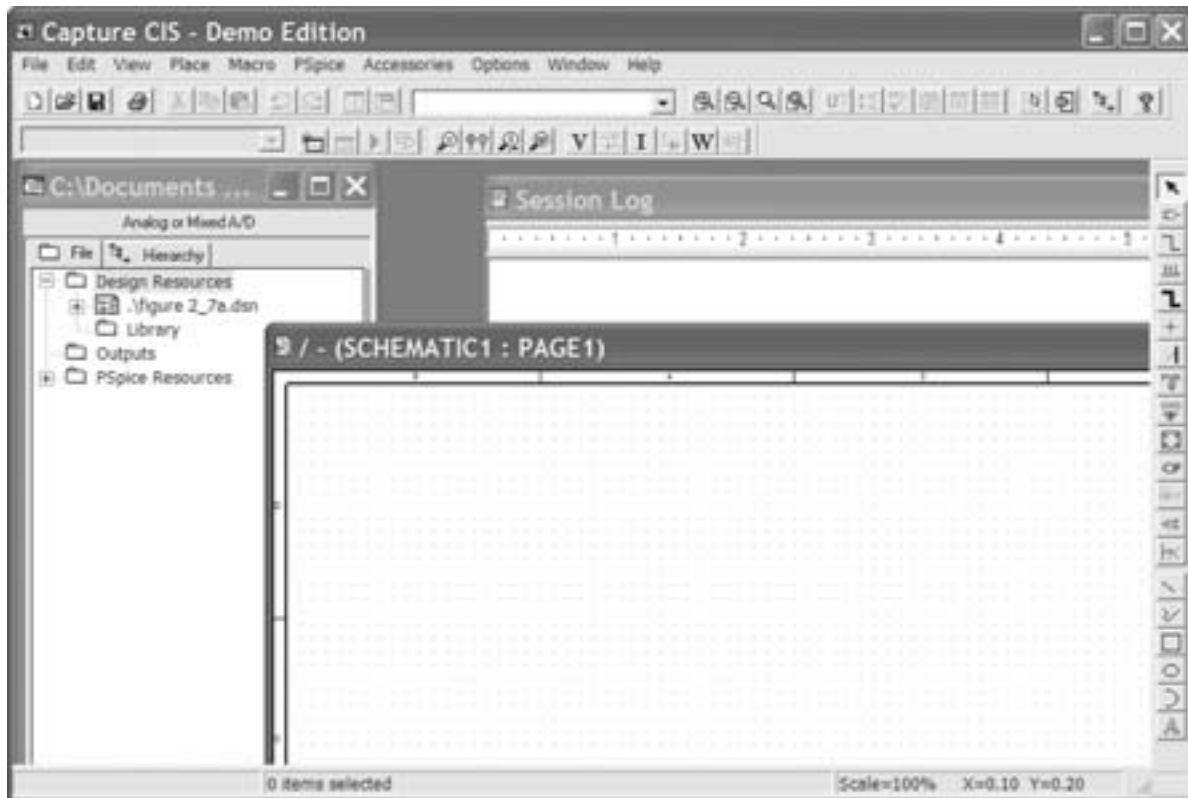


Figure G.2 We use these windows to draw the circuit.



Figure G.3 We use this window to select parts to be placed in the circuit diagram.

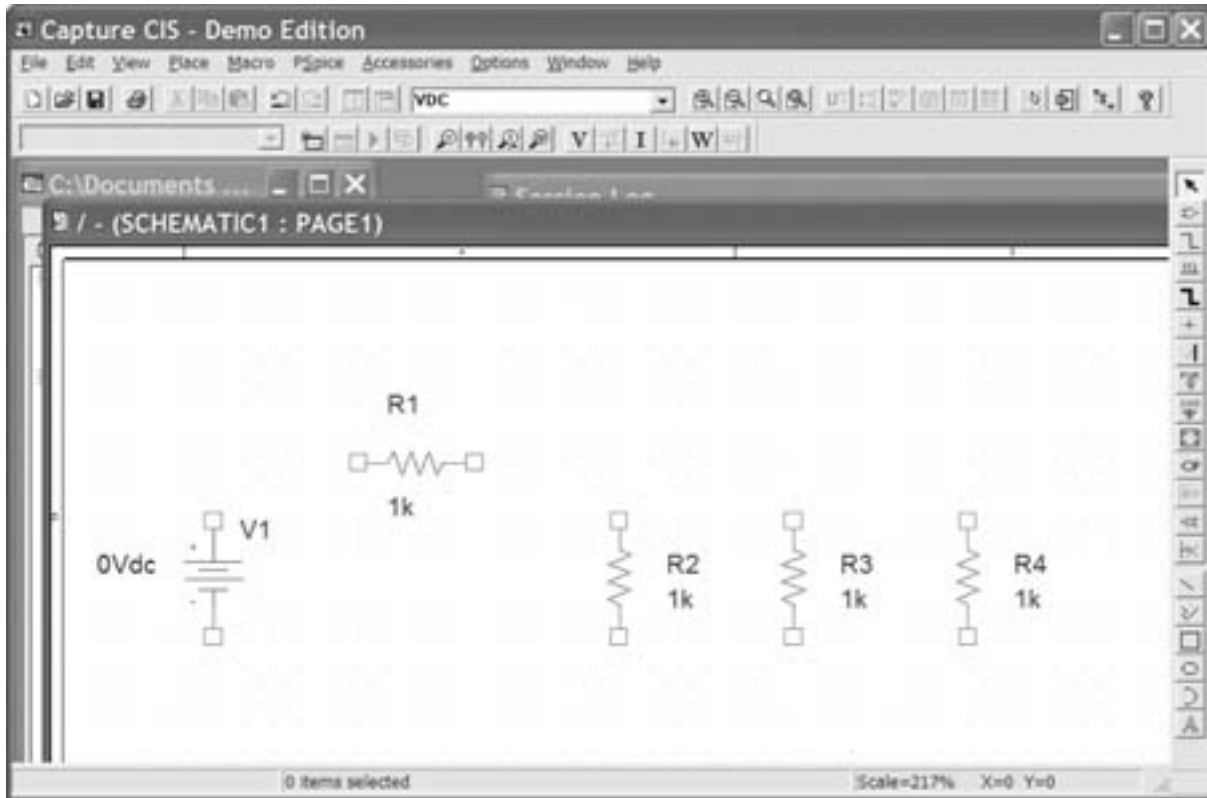


Figure G.4 Parts have been placed.

(However, two or more parts must not have the same name when we do the PSpice analysis.)

5. Similarly, we can rotate a part that has been placed with a wrong orientation by selecting it, right-clicking, and selecting **Rotate** on the pop-up menu.

Wiring the Circuit

Next, we use the **Place/Wire** command to change the cursor into a wiring tool. Move the cursor to the top end of the voltage source, click the left mouse button, move the cursor to the left end of R_1 , and click to wire the voltage source to R_1 . Continue wiring components until the circuit appears as shown in Figure G.5. Then, click the right mouse button and select **End Wire** to quit the wiring mode. If you end up with unneeded wire segments, you can eliminate them after you quit the wiring mode. Click on the unwanted wire segment and press the delete key. Also, the **Edit/Undo Place** command is useful for correcting mistakes.

Unless we specify the right mouse button, click means to click the left mouse button.

Adding the Ground Node

All circuits analyzed by PSpice must have a ground or reference node, such as we used in node-voltage analysis in Section 2.4. After the analysis is completed, PSpice can report the voltage at each of the other nodes with respect to the reference node. These are the steps in adding the ground node:

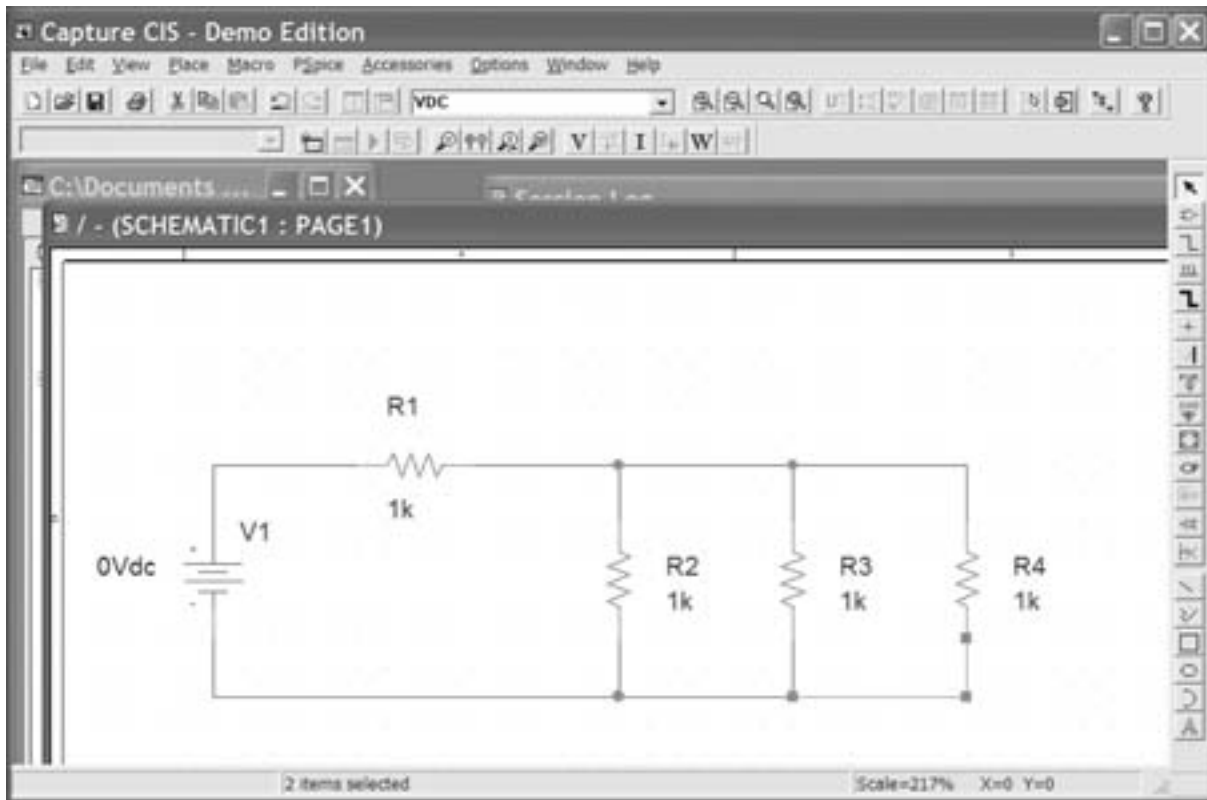


Figure G.5 Use the **Place/Wire** command to bring up the wiring tool and wire the circuit.

1. Left-click on the ground tool on the right-hand side of the screen. (See Figure G.6.) This brings up the *Place Ground* window.
2. Select the ground named **0/Source**, click on **OK**, and place the ground at the bottom of the circuit as shown in Figure G.6. In case the selection 0 or 0/Source does not appear on the menu, click **Add Library ...**, select the file `source.olb` from the PSpice folder, and click on **Open**. The complete path to this file is

```
c:\OrCAD\OrCAD_10.5_Demo\tools\capture\library\pspice\
source
```

After placing the ground symbol, right-click and select **End Mode**.

3. Use the **Place/Wire** command to change the cursor to a wiring tool and wire the ground to the circuit as shown in Figure G.6.

Changing Component Values

Next, we change the default component values to those shown in Figure 2.7(a) on page 55. Double left-click on 0Vdc next to the voltage source to bring up the window shown in Figure G.7. Enter 20V as the new value, and then click on **OK** to close the window. In a similar fashion, change the values of the resistances so that R_1 is 10, R_2 is 20, R_3 is 30, and R_4 is 40. (If you want to, you can change the names of the elements as well. For example, we could change V1 to Vs to better match the element names shown in Figure 2.7.) Now, the circuit should appear as shown in Figure G.8.

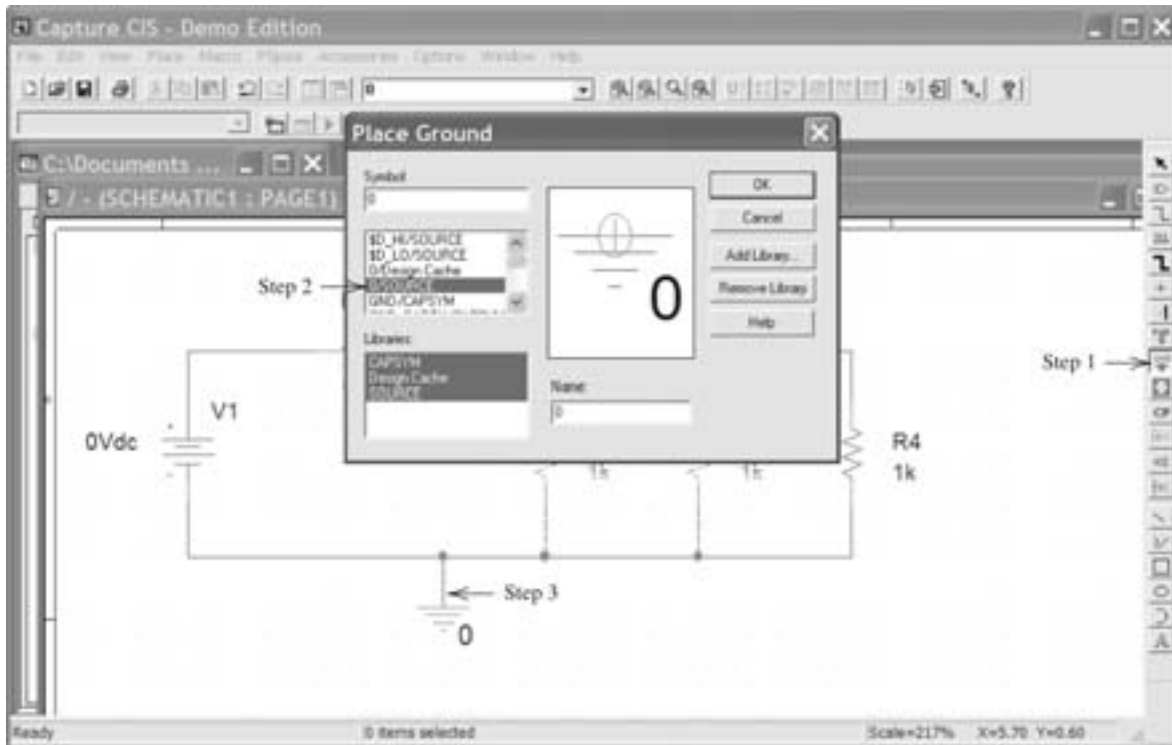


Figure G.6 Select, place, and wire the ground.



Figure G.7 Change the value of the source to 20 V.

Setting Up a New Simulation Profile

Simulation profiles tell PSpice what type of analysis is desired. Next, use the **PSpice/New Simulation Profile** command to bring up the window shown in Figure G.9(a). Type in **DC Solution** as the name of the simulation profile, make sure that the *Inherit From* window contains the word *none*, and left-click on **Create**. This brings up the window shown in Figure G.9(b). Use the pull-down menu and select **Bias Point** as the analysis type. Then, click on **OK**. The simulation profile we have created instructs PSpice to compute the voltages at the circuit nodes, the current through each element, and the power dissipated in each element.

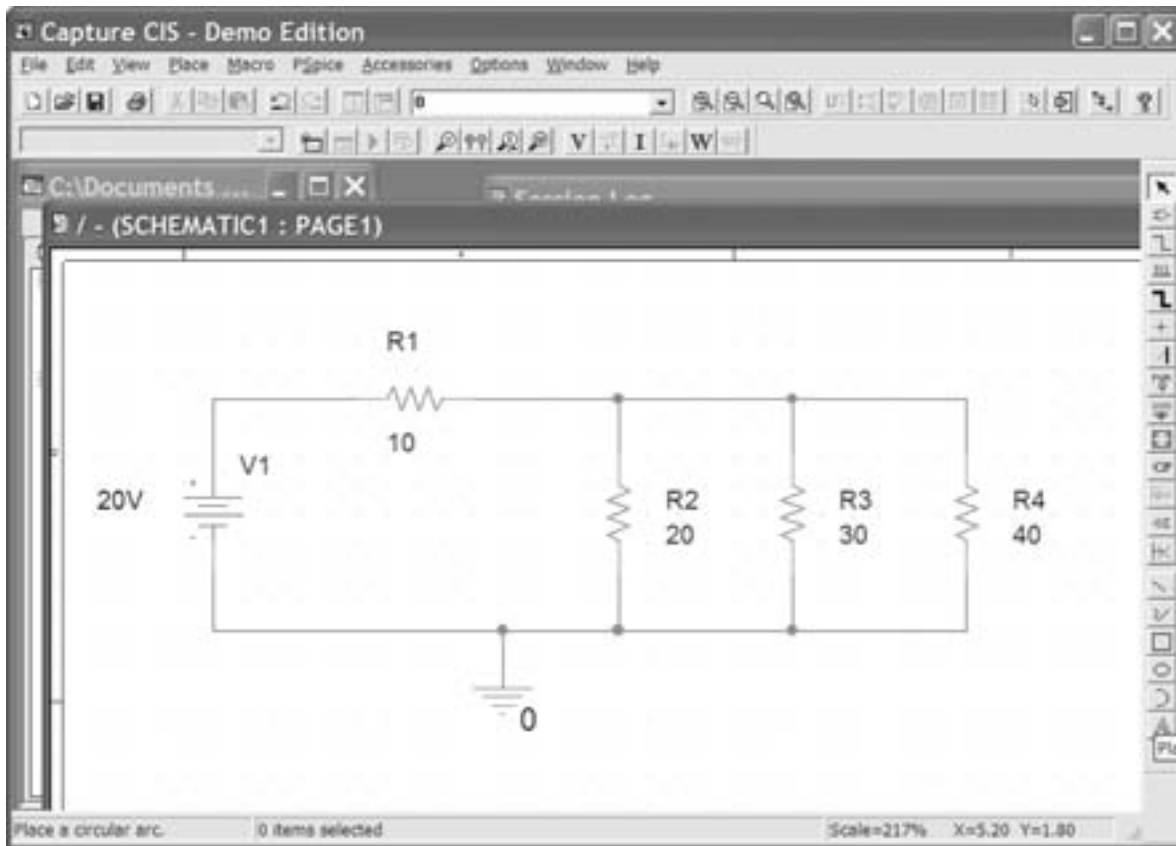


Figure G.8 The completed circuit.

Running the PSpice Simulation

Next, use the **PSpice/Run** command to run the simulation. Close the *SCHEMATIC1-DC Solution* window that appears. Next, click on the V, I, and W icons (shown in Figure G.10) to toggle the display of currents, voltages, and powers on and off. The current, voltage, and power for each element are shown in Figure G.10. For example, the current entering the left end of R_1 is 1.040 A, the voltage at the top end of R_4 is 9.6 V with respect to ground, and the power dissipated in R_4 is 2.304 W. The currents agree with the values found manually in Exercise 2.2.

Exercise G.1 Solve the circuits shown in Figure 2.7(b) and 2.7(c) on page 55 using Capture/PSpice. (*Hint:* The part name for a dc current source is IDC.)

Answer The project files are named Figure 2_7b and Figure 2_7c, which can be found on the OrCAD CD in the folder named **Hambley OrCAD Projects**. □

Exercise G.2 Determine the equivalent resistance seen looking into the terminals of the circuit shown in Figure 2.4(a) on page 51. (*Hint:* Attach a 1-A source to the input terminals and solve the circuit. The voltage across the current source equals the equivalent resistance.)

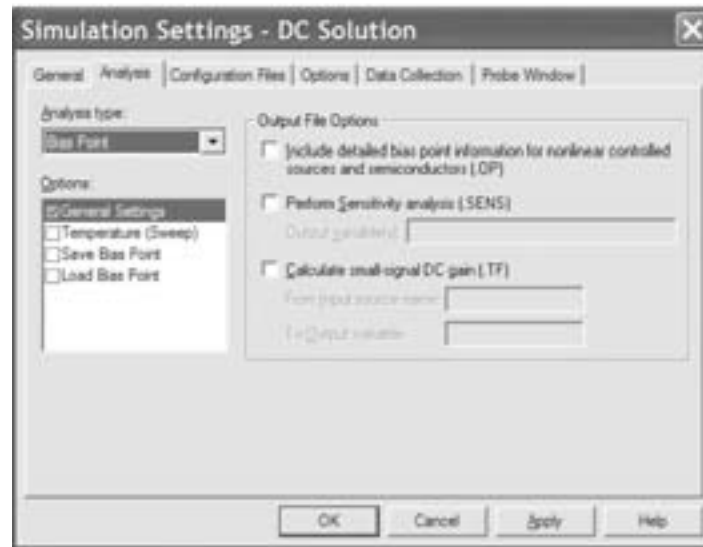
Answer The project file is named Figure 2_4a and is in the folder named **Hambley OrCAD Projects** on the OrCAD CD. The voltage across the current source is 3 V, so the equivalent resistance is $3\ \Omega$, which is the value found manually in Exercise 2.1. □

If you place the cursor on the indicator for a current, hold the left button down, and move the mouse, the current indicator can be moved. The dotted line points to one terminal of an element. By PSpice convention, the current is referenced positive entering that terminal.

If you copy the folder named **Hambley OrCAD Projects** to your C drive, you can use Capture to open and run simulations for the projects contained in the folder.



(a)



(b)

Figure G.9 Windows used to set up the simulation profile.

Opening Existing Projects

To open the projects provided as solutions for the Exercises, first copy the folder named **Hambley OrCAD Projects** from the OrCAD 10.5 Demo CD to your hard drive. Then, to open the solution for Exercise G.2, start Capture CIS, use the **File/Open/Project . . .** command, navigate to the desired project, which in this case is named Figure_2_4a, and double-click on it. Then, navigate to Page 1 of Schematic 1 as shown in Figure G.11 and double-click on it to bring up the window containing the circuit diagram.

Controlled Sources

Capture and PSpice can also simulate circuits that contain controlled sources. Except for setting the gain parameters of the controlled sources, the steps are very similar to those we used previously. The part names of the controlled sources are as follows:

- E voltage-controlled voltage source
- G voltage-controlled current source
- H current-controlled voltage source
- F current-controlled current source

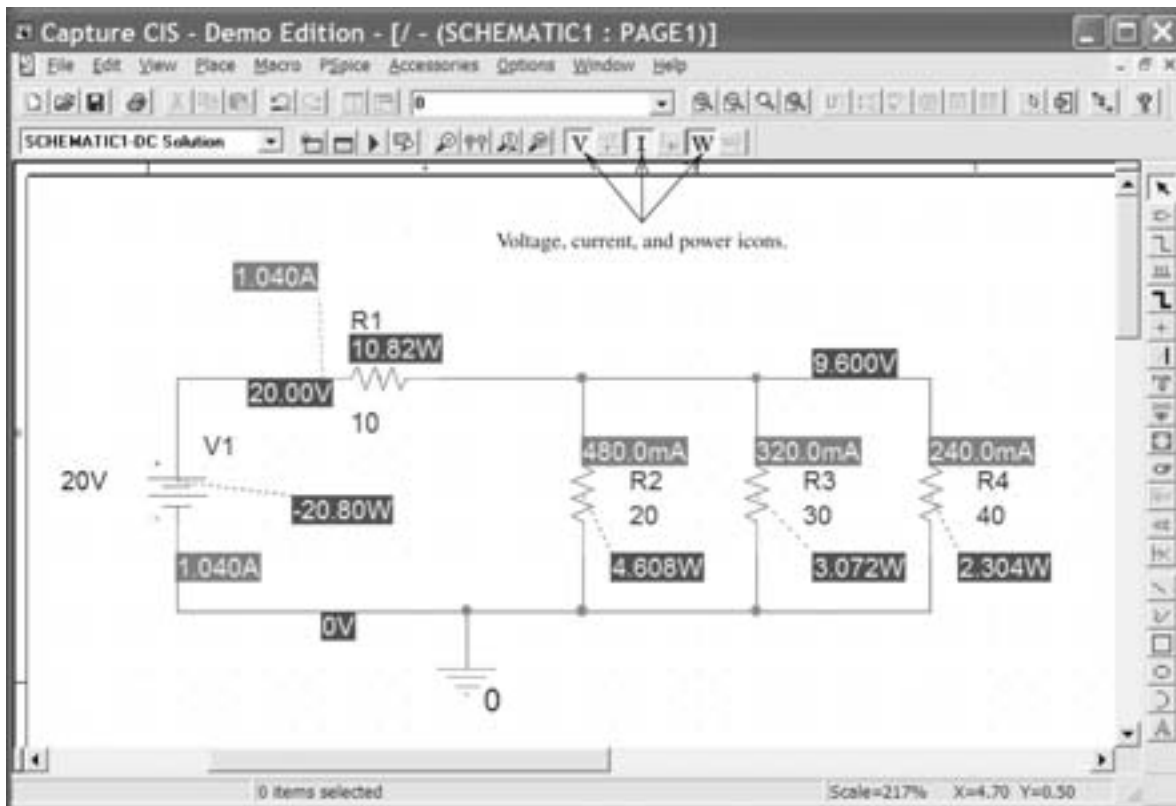


Figure G.10 After running PSpice, the voltages, currents, and powers can be displayed on the schematic.

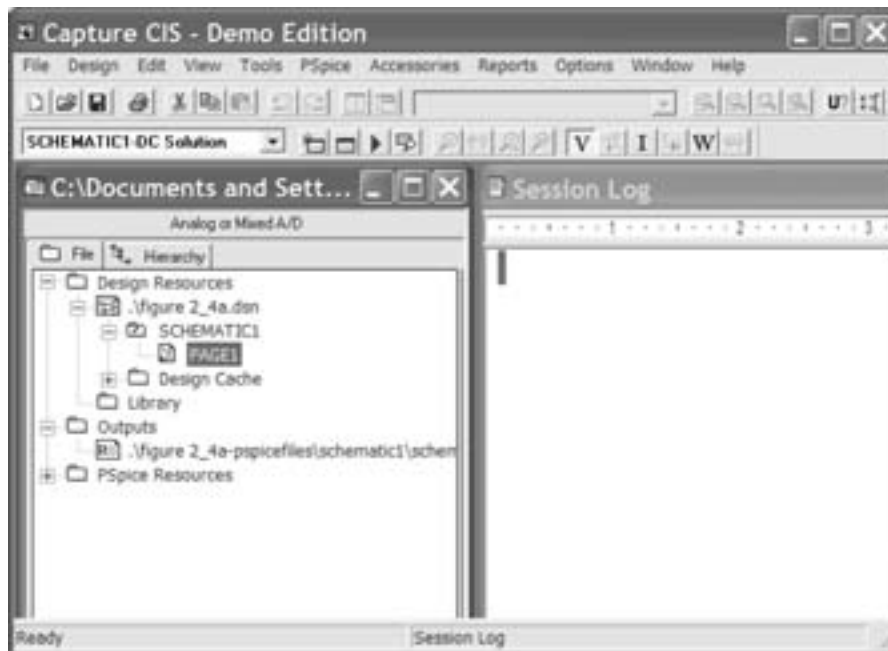


Figure G.11 Double-click on PAGE1 to open the schematic (if it is not open already).

Figure G.12 shows the appearance of the various controlled sources in Capture. As the symbols are oriented in Figure G.12, the controlling variable should be wired to the terminals on the left-hand side, and the source terminals are on the right-hand side. (Of course, if we flip or rotate the symbol, either set of terminals may be at the top, bottom, left, or right.)

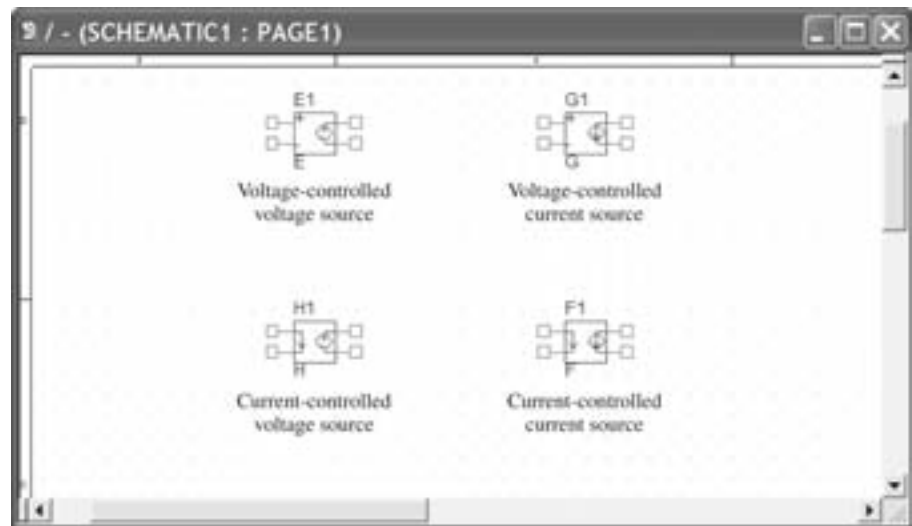


Figure G.12 Controlled sources.

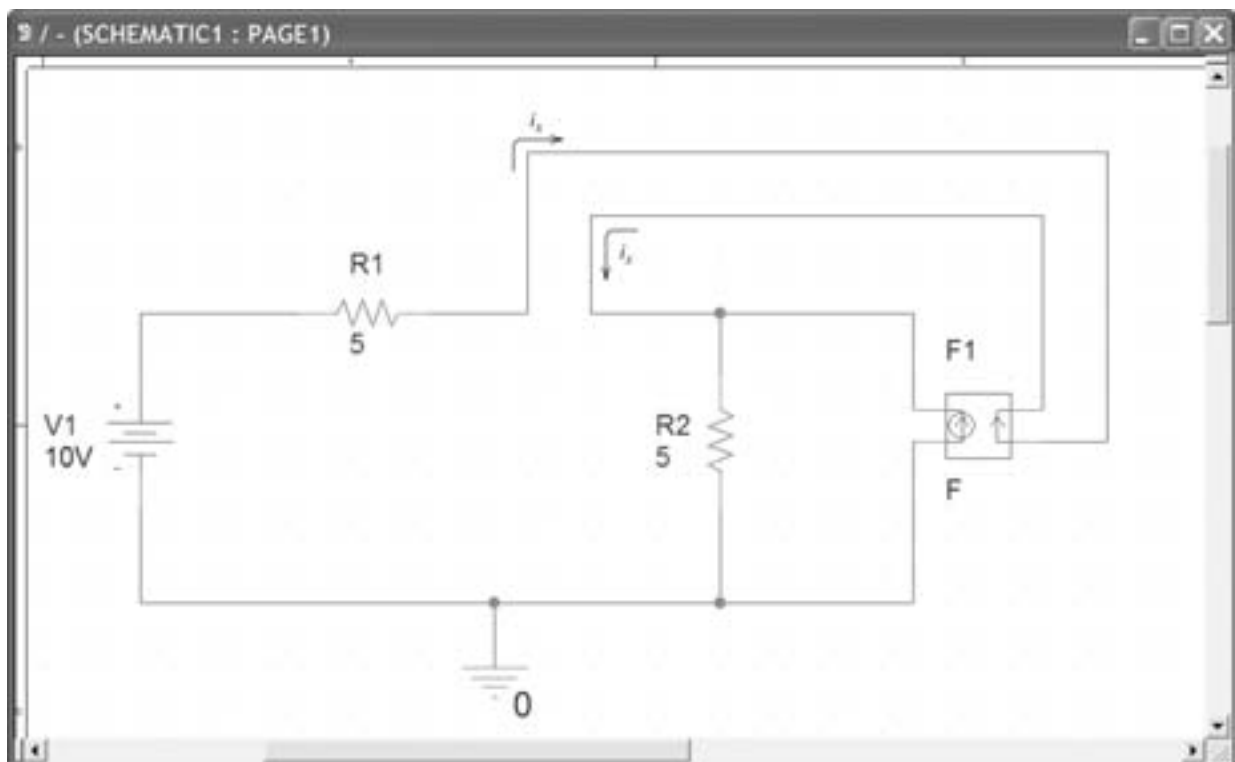


Figure G.13 The circuit of Figure 2.31(a) on page 77 drawn with Capture.

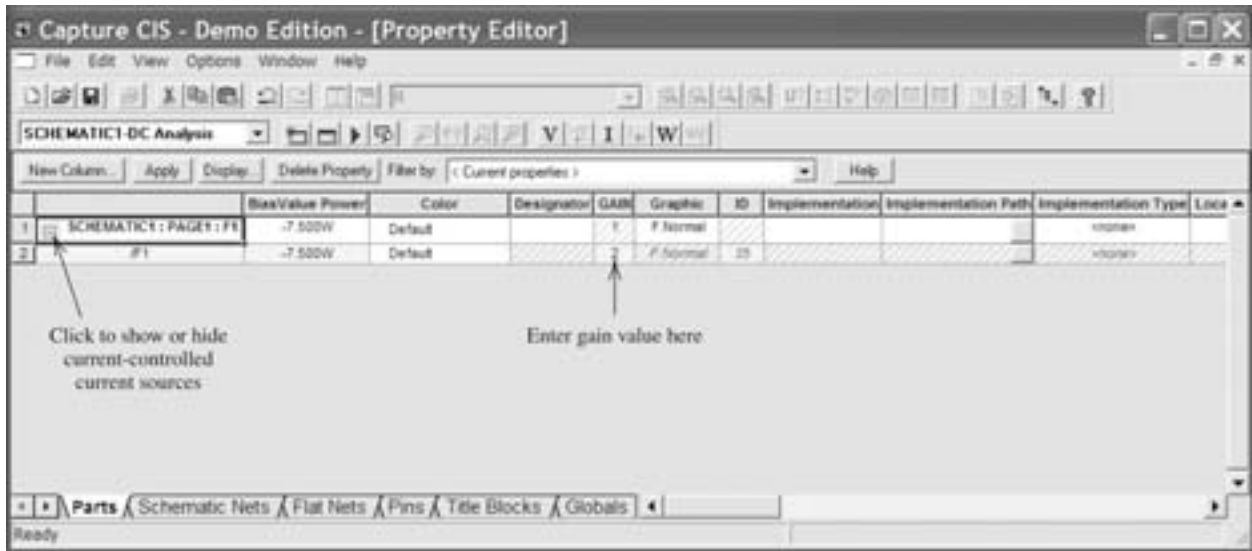


Figure G.14 This window is used to set the gain of the controlled source.

As an example, we solve for i_x in the circuit shown in Figure 2.31(a) on page 77 which contains a current-controlled current source. First, we place the elements (the name of the controlled source is F), place the ground, and wire the circuit, as we did earlier in this appendix. The Capture screen is shown in Figure G.13. Notice that we have flipped and rotated the controlled source, so that the source terminals are on the left-hand side. Then, the controlling variable i_x is wired to the input terminals. Compare the screen display with the original circuit shown in Figure 2.31(a).

To set the gain parameter of the controlled source, we double left-click on the controlled source, bringing up the window shown in Figure G.14. Here, we enter the gain as 2, click on **Apply**, and close the window. Then, we set up the simulation profile to perform a **Bias Point** analysis, run PSpice, and observe the results on the Capture screen. As in the manual analysis that we did for Exercise 2.13, we obtain $i_x = 0.5$ A. This project is stored in the file named Figure_2_31a.

Exercise G.3 Solve the circuit shown in Figure 2.31(b) on page 77 using Capture/PSpice. (*Hint:* The part name for a current-controlled voltage source is H, and the name for a constant current source is IDC.)

Answer The project file is Figure_2_31b, which can be found on the OrCAD CD in the folder named **Hambley OrCAD Projects**. As in the manual analysis, we find $i_y = 2.31$ A. □

G.2 TRANSIENT ANALYSIS

Next, we show how to perform transient analysis of RLC circuits, as we did by traditional analysis in Chapter 4. In transient analysis, we often have switches that open or close at $t = 0$. We draw the circuit with the switches in their final positions and specify the initial voltages for capacitors and initial currents for inductors. Then, PSpice analyzes the circuit for $t > 0$.

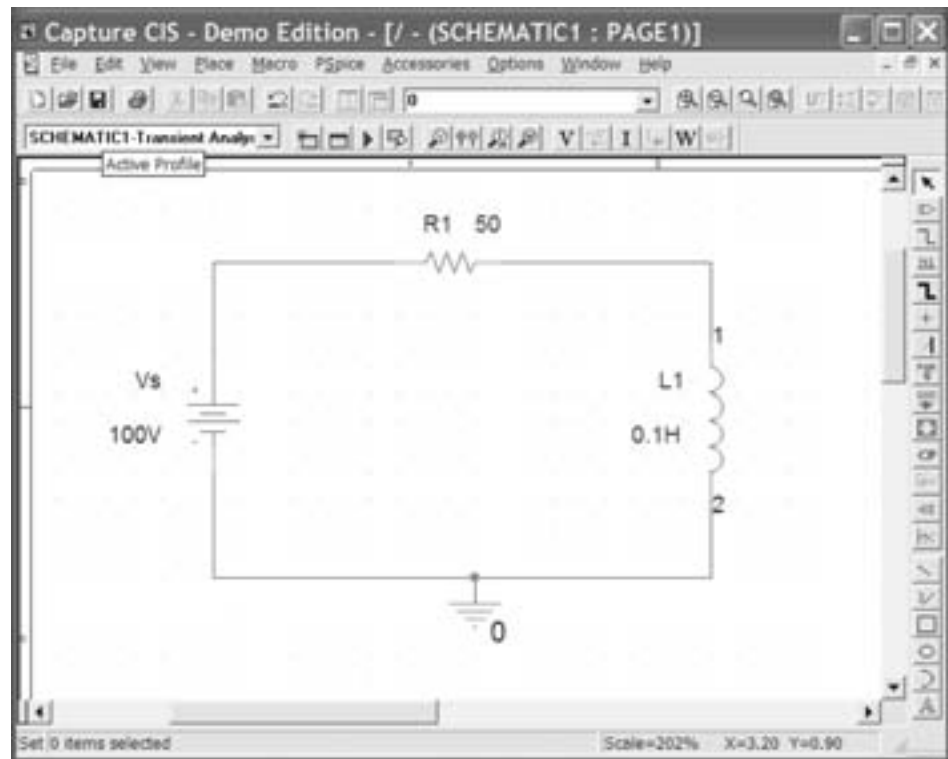


Figure G.15 The circuit of Figure 4.7 on page 173 drawn with Capture.

The element names for inductors and capacitors are L and C, respectively.

As an example, we use PSpice to analyze the circuit of Figure 4.7 on page 173. First, we set up a new project named Figure 4_7, place the elements, and wire the circuit. (Don't forget to include a ground.) Because the switch is closing, it is a short circuit for $t > 0$ and appears as a wire in Capture. The resulting Capture window is shown in Figure G.15.

Next, we specify the initial (i.e., immediately after $t = 0$) current flowing in the inductor. In this circuit, the initial current is zero. Double left-click on the inductor to bring up the window shown in Figure G.16. Enter 0 for the initial condition (IC) as shown. Then, click on **Apply** and close the window.

Setting Up a Simulation Profile for Transient Analysis

Next, use the **PSpice/New Simulation Profile** command to bring up the window shown in Figure G.17(a). Type in **Transient Analysis** as the name of the simulation profile, make sure that the *Inherit From* window contains the word *none*, and left-click on **Create**. This brings up the window shown in Figure G.17(b). Use the pull-down menu, and select **Time Domain (Transient)** as the analysis type.

PSpice performs transient analysis by using the initial values of the voltages and currents to compute the circuit responses a short time later. Then, these values are used to compute the responses at the next time point. PSpice adjusts the time step between computed values as the solution proceeds. Higher accuracy is provided by a small step size, but it results in long execution times. Therefore, PSpice uses a small time step if the response is changing rapidly, and a larger step if the response is changing more slowly. Also, when we plot the results, a small time step results in a

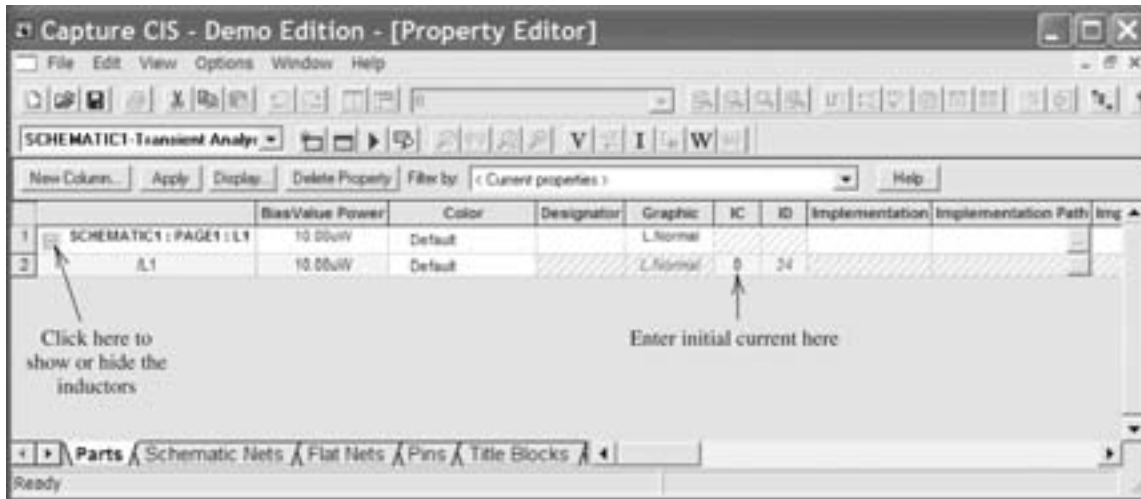


Figure G.16 We use this window to specify the initial current in the inductor.

smooth curve. In the *Simulation Settings* window, we can specify the maximum time step to be used as well as how long we want the simulation to proceed.

We must select appropriate parameters for the time-domain analysis for the circuit at hand. In this circuit, we know that the current approaches its final value with a time constant of $L/R = 2$ ms. A smooth plot can be obtained by selecting the maximum step to be about 1 percent of the time constant. Thus, we select the *Maximum step size* to be $20 \mu\text{s}$. (In Capture, the letter u represents the Greek letter μ in numerical values.) Furthermore, we know that the response approaches its final value within a negligible error in about five time constants. Therefore, we select the *Run to time* value as 10 ms. (This instructs PSpice to stop the analysis when t reaches 10 ms.)

In more complex circuits for which the results are not known in advance, we may need to play around with these parameters (*Run to time* and *Maximum step size*). Usually, engineers are designing circuits for which they know the approximate response in advance, and selecting the analysis parameters is not a major problem.

Running the Simulation and Viewing the Results

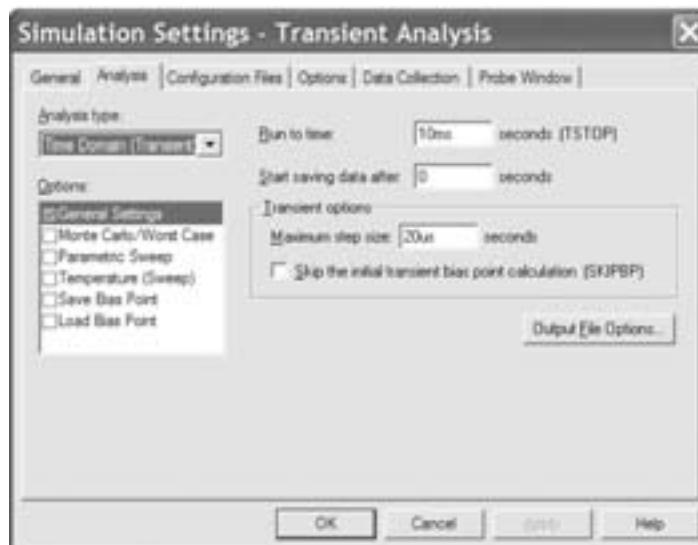
Next, we attach markers to the circuit to specify which currents and voltages we wish to observe. Markers are available to observe the voltage at any node (with respect to ground), the voltage between any two nodes, the power dissipated in any element, and the current entering an element terminal. The icons needed to bring up these markers are shown in Figure G.18. We attach a marker to observe the voltage across the resistor and a marker to observe the voltage at the top node of the inductor, as shown in the figure.

Next, use the **PSpice/Run** command to run the simulation. This brings up a window that displays the voltages. The voltage plots are shown in Figure G.19. Notice that the plot for the voltage across the inductor agrees with the plot shown in Figure 4.8(b) on page 174.

Exercise G.4 Use PSpice to observe the voltage across the capacitor in the circuit shown in Figure 4.21 on page 187 for $R = 100 \Omega$. Repeat for $R = 200$ and 300Ω .



(a)



(b)

Figure G.17 We use these windows to set up the simulation profile.

Answer The project is named Figure 4_21 and is stored on the OrCAD CD. The plots are similar to those shown in Figure 4.26 on page 191. □

G.3 FREQUENCY RESPONSE

Another type of analysis that PSpice can perform is to determine and plot the frequency response of a circuit as we did by traditional analysis in Chapter 6. As an example, we analyze the circuit shown in Figure 6.17 on page 307.

First, we set up a new project named Figure 6_17, place the elements (the name of the ac voltage source is VAC), and wire the circuit. We included a ground at the bottom of the circuit. Next, we name the input and output nodes of the circuit. To do this, we use the **Place/Net Alias** command to bring up the window shown in Figure G.20(a). Next, we type **in** as the name of the input node, click on **OK**, place the name field on the wire connected to the top of the source, and left-click. Then, we right-click and select **End Mode**. Similarly, we name the output node **out**. The resulting Capture window is shown in Figure G.20(b).

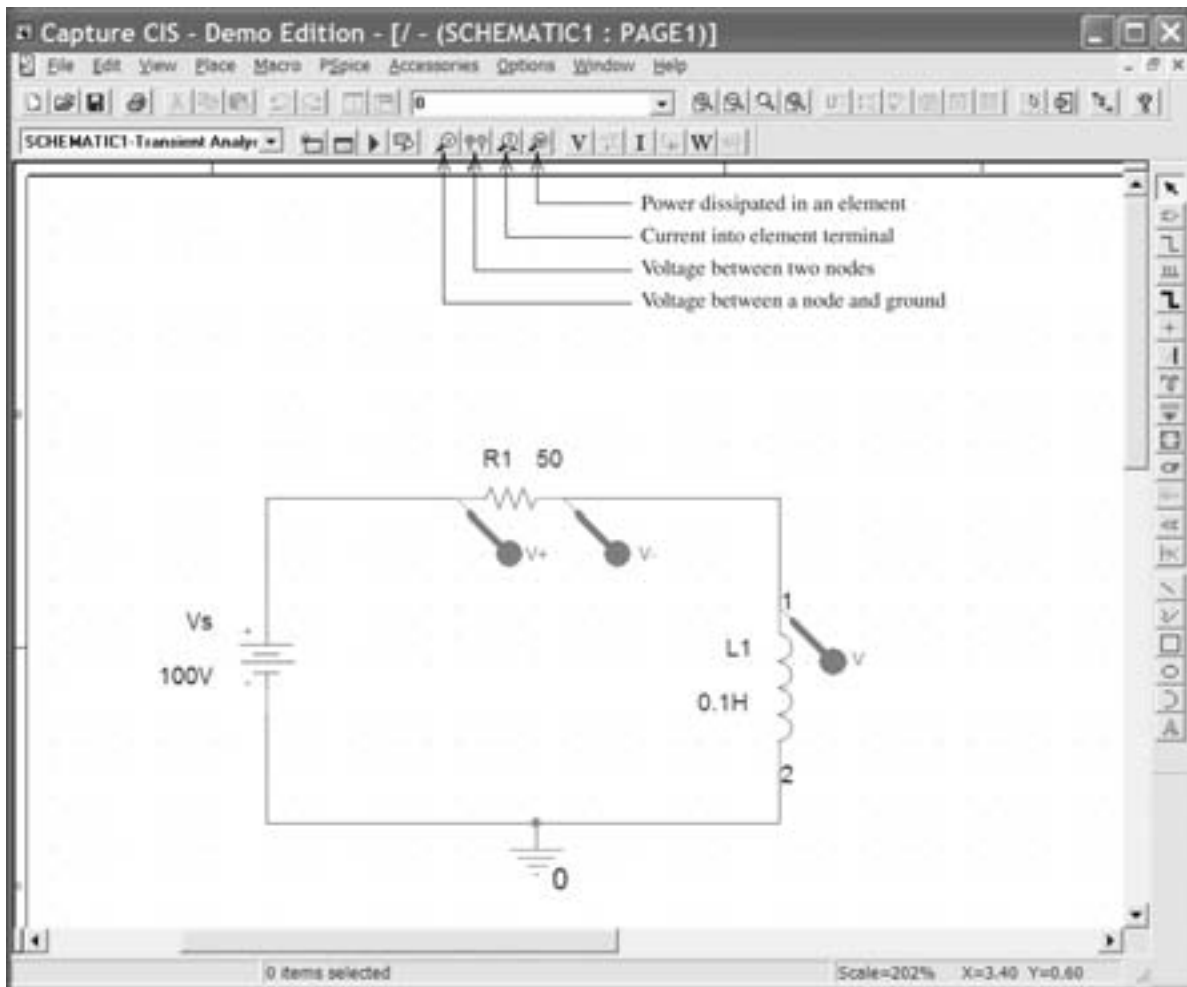


Figure G.18 If we attach markers to the circuit, PSpice automatically plots the corresponding currents, voltages, and powers.

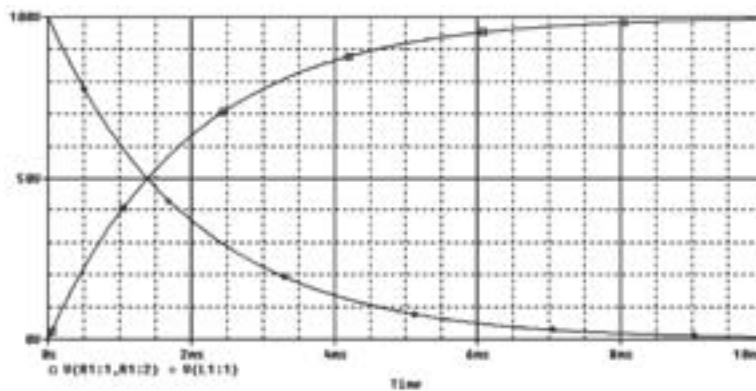
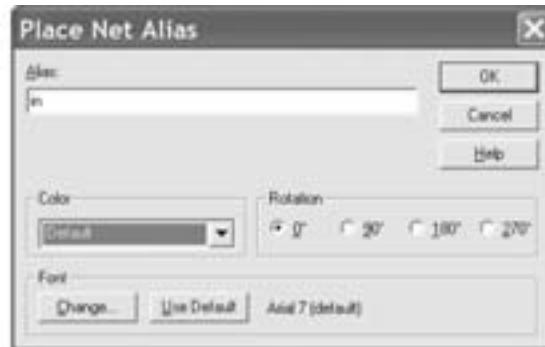
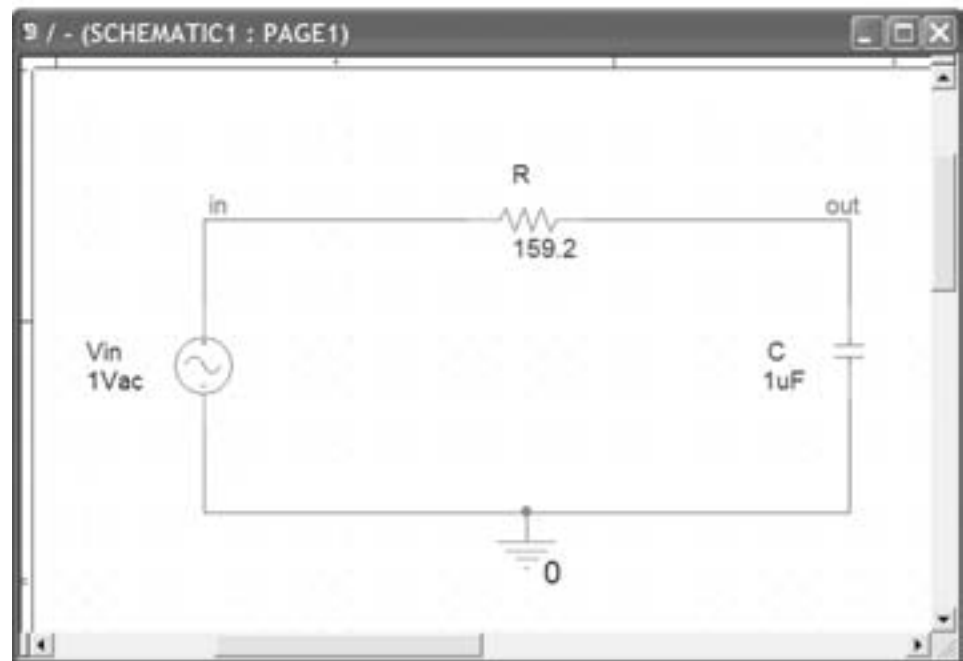


Figure G.19 Plots of the voltage across the resistor and across the inductor versus time.



(a)



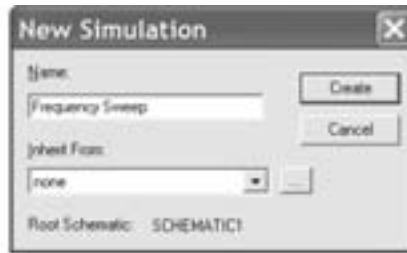
(b)

Figure G.20 We use the *Place Net Alias* window to name nodes in a circuit.

Setting Up a Simulation Profile for Frequency Response Analysis

Next, use the **PSpice/New Simulation Profile** command to bring up the window shown in Figure G.21(a). Type in **Frequency Sweep** as the name of the simulation profile, make sure that the *Inherit From* window contains the word *none*, and left-click on **Create**. This brings up the window shown in Figure G.21(b). Use the pull-down menu and select **AC Sweep/Noise** as the analysis type, select *Logarithmic*, select *Decade* in the pull-down menu, enter 10 for the *Start Frequency*, 1e5 for the *End Frequency*, and 100 for *Points/Decade* as shown in Figure G.21(b). Then click on **OK**. This simulation profile instructs PSpice to analyze the circuit for frequencies ranging logarithmically from 10 Hz to 10⁵ Hz with 100 steps per decade.

Then, use the **PSpice/Run** command to run the simulation. After the simulation is completed, the window used to plot results comes up. Then, we use the **Trace/Add**



(a)



(b)

Figure G.21 We use these windows to set up a simulation profile for frequency response analysis.

Trace command to bring up the window shown in Figure G.22. Here we type in $\text{DB}(V(\text{out})/V(\text{in}))$ and click on **OK**. This instructs the program to plot the magnitude of the output voltage divided by the input voltage in decibels versus frequency. In other words, we have asked for a magnitude Bode plot of the circuit transfer function. The resulting plot is shown in Figure G.23. This plot agrees well with the approximate plot we obtained manually in Figure 6.18(a) on page 307. Similarly, we could obtain a Bode phase plot by requesting a plot of $\text{P}(V(\text{out})/V(\text{in}))$ in which P instructs the program to plot phase.

Exercise G.5 Obtain a magnitude Bode plot for the filter designed in Exercise 6.20 and shown in Figure 6.39 on page 324. Allow frequency to range from 10 Hz to 100 kHz.

Answer The project is named Figure_6_39, and the plot is shown in Figure G.24. As intended by the design carried out for Exercise 6.20, this is a lowpass filter with a cutoff frequency of 5 kHz. □



Figure G.22 Type $DB(V(out)/V(in))$ in the *Trace Expression* window to obtain a magnitude Bode plot for the transfer function.

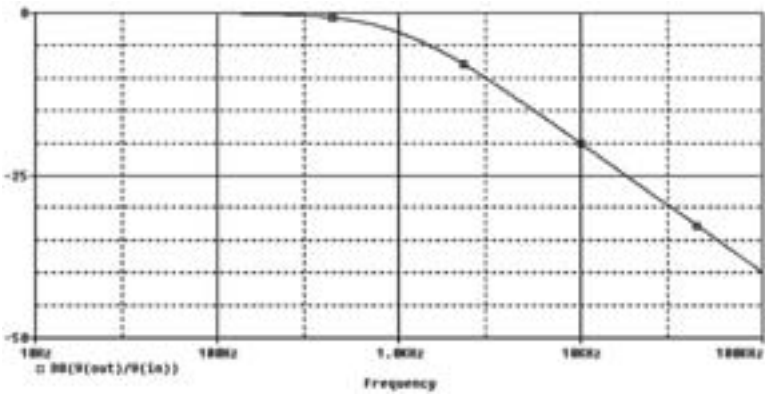


Figure G.23 Magnitude Bode plot produced by PSpice.

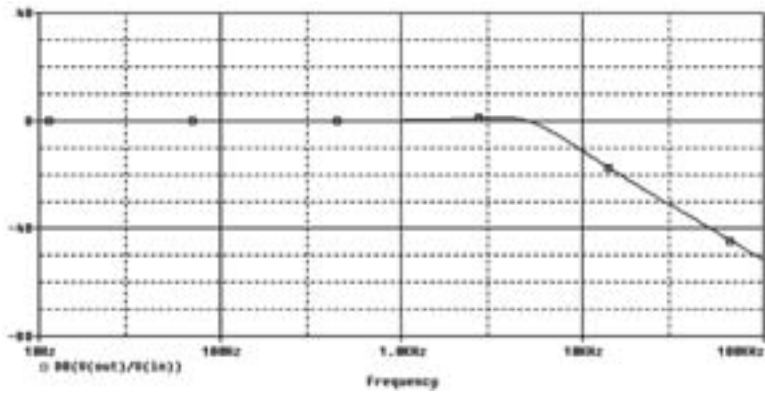


Figure G.24 See Exercise G.5.

G.4 OTHER EXAMPLES

Several other interesting examples showing analysis of circuits from this book are included in the folder named **Hambley OrCAD Projects** on the OrCAD CD. Each project file is named for a corresponding figure in the book.

Half-Wave Rectifier

The project named **Figure 10_24** is a simulation of a half-wave rectifier circuit. The model for a real diode (the 1N4002) is used, and the input is a 1-kHz 10-V-peak sinewave. The half-wave rectified signal is applied to a 100- Ω load.

Peak Rectifier

The project named **Figure 10_26** is a simulation of a peak rectifier. After the simulation runs, the diode and load current are plotted. Because the initial voltage on the capacitor is zero, the first current pulse through the diode is very large. After that, the waveforms are similar to those shown in Figure 10.26 on page 493.

Clipper Circuit

The project named **Figure 10_29** simulates the clipper circuit shown in Figure 10.29 on page 496 except that the voltage sources have been adjusted to account for the forward drop of a real diode. This project has two simulation profiles, one for displaying the waveforms as in Figure 10.29(b) and the other for displaying the transfer characteristic as in Figure 10.29(c). (You can select the active simulation profile by clicking on it in the project window shown in Figure G.25 and then by using the **PSpice/Make Active** command in the main window.)

MOSFET Curves

The project named **Figure 12_7** produces the device curves for the MOSFET of Exercise 12.2, which are shown in Figure 12.7 on page 581. If you open the project,

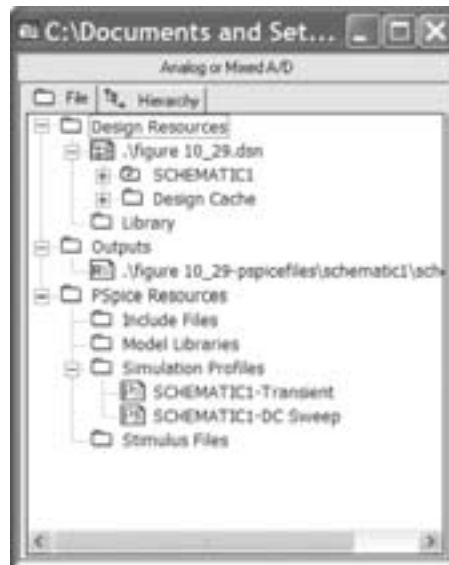


Figure G.25 Click on the desired simulation profile. Then use the **PSpice/Make Active** command in the main window.

you can view the device parameters (KP , V_{to} , L , and W) by first left-clicking on the MOSFET symbol and then right-clicking and selecting **Edit PSpice Model**. Also, look at the simulation profile, which calls for a primary sweep of V_{DS} and a secondary sweep for V_{GS} .

Simple MOSFET Amplifier

The project named Figure 12_10 simulates the amplifier shown in Figure 12.10 on page 583. You can modify the circuit to see the effects of changing the supply voltages and the input amplitude on the output waveform.

MOSFET Q-Point Determination

The project named Figure 12_15 solves for the dc bias point of the circuit shown in Figure 12.15 on page 587. We solved for the bias point by using traditional mathematical analysis in Example 12.2.

BJT Curves

The project named Figure 13_5 produces output characteristics for a BJT similar to Figure 13.5(b) on page 619.

Simple BJT Amplifier

The project named Figure 13_7 simulates the amplifier that we analyzed by using load-lines in Example 13.2 on page 623. You can modify the circuit to see the effects of changing the supply voltages and the input amplitude on the output waveform.

BJT Q-Point Determination

The project named Figure 13_18 solves for the dc bias point of the circuit shown in Figure 13.18 on page 632. We solved for the bias point using traditional mathematical analysis in Example 13.4.

Nonlinear Behavior of an Op Amp Circuit

The project named Figure 14_23 simulates the noninverting amplifier shown in Figure 14.23 on page 685. You can observe the output waveform for proper linear operation of the circuit. Then, by changing circuit parameters as given in the captions of Figures 14.25, 14.26, and 14.27, you can observe clipping because of the maximum voltage limit of the op amp, clipping because of the current limit of the op amp, and slew-rate limiting.