Since V_0 and α are both constants, the linear relationship between the applied stress P and the output voltage V_{out} makes the piezoresistor a natural sensor for detecting or measuring mechanical stress. However, we should examine the sensitivity of such a sensor. As a reference, a finger can apply about 50 N of force across an area of 1 cm^2 (10^{-4} m^2), which is equivalent to a pressure $P = 5 \times 10^5$ N/m^2. If the piezoresistor is made of silicon with $\alpha = -1 \times 10^{-9}$ m^2/N and if the dc source in the Wheatstone bridge is $V_0 = 1$ V, Eq. (2.51) yields the result that $V_{out} = -125$ μV, which is not impossible to measure but quite small nevertheless. How then are such pressure sensors used?

The answer is simple: We need a mechanism to amplify the signal. We can do so electronically by feeding V_{out} into a high-gain amplifier, or we can amplify the mechanical pressure itself before applying it to the piezoresistor. The latter approach can be realized by constructing the piezoresistor into a cantilever structure, as shown in Fig. 2-38 (a cantilever is a fancy name for a "diving board" with one end fixed and the other free). Deflection of the cantilever tip induces stress at the base of the cantilever near the attachment point. If properly designed, the cantilever—which usually is made of silicon or metal—can amplify the applied stress by several orders of magnitude (see Example 2-16).

Example 2-16: A Realistic Piezoresistor Sensor

When a force F is applied on the tip of a cantilever of width W, thickness H, and length L (as shown in Fig. 2-38) the corresponding stress exerted on the piezoresistor attached to the cantilever base is given by

$$P = \frac{FL}{WH^2}. \tag{2.52}$$

Determine the output voltage of a Wheatstone-bridge circuit if $F = 50$ N, $V_0 = 1$ V, the piezoresistor is made of silicon, and the cantilever dimensions are $W = 0.5$ cm, $H = 0.5$ mm, and $L = 1$ cm.

Solution: Combining Eqs. (2.51) and (2.52) gives

$$V_{out} = \frac{V_0}{4} \alpha \cdot \frac{FL}{WH^2}$$
$$= \frac{1}{4} \times (-1 \times 10^{-9}) \times \frac{50 \times 10^{-2}}{(5 \times 10^{-3}) \times (5 \times 10^{-4})^2}$$
$$= -0.1 \text{ V}.$$

The integrated piezoresistor–cantilever arrangement generates an output voltage whose magnitude is on the order of 800 times greater than that generated by pressing on the resistor directly!

Concept Question 2-15: Does compression along the current direction increase or decrease the resistance? Why?

Concept Question 2-16: Why are piezoresistors placed at the base of cantilevers and other deflecting structures?

Exercise 2-12: What would the output voltage associated with the circuit of Example 2-16 change to, if the cantilever thickness is reduced by a factor of 2?

Answer: $V_{out} = -0.4$ V. (See ✍)

2-7 Introducing Multisim

Multisim 11 is the latest edition of National Instrument's *SPICE* simulator software. SPICE, originally short for Simulation Program with Integrated Circuit Emphasis, was developed by Larry Nagel at the University of California, Berkeley, in the early 1970s. It since has inspired and been used in many academic and commercial software packages to simulate analog, digital, and mixed-signal circuits. Modern SPICE simulators like Multisim are indispensable in integrated circuit design; ICs are so complex that they cannot be built and tested on a breadboard ahead of production (see Technology Brief 5 on page 120). With SPICE, you can draw a circuit from a library of components, specify how the components are connected, and ask the program to solve for all voltages and currents at any point in time. Modern SPICE packages like Multisim include very intuitive graphic user interface (GUI) tools that make both circuit design and analysis very easy. Multisim allows the user to simulate a laboratory experience on his/her computer ahead of actually working with real components.

In this section, you will learn how to:

• Set up and analyze a simple dc circuit in Multisim.

• Use the Measurement Probe tool to quickly solve for voltages and currents.

• Use the Analysis tools for more comprehensive solutions.

We 1
other ar
an intr
that ac
if you l
selectic
Sub-M

2-7.1

After i
with th
circuit
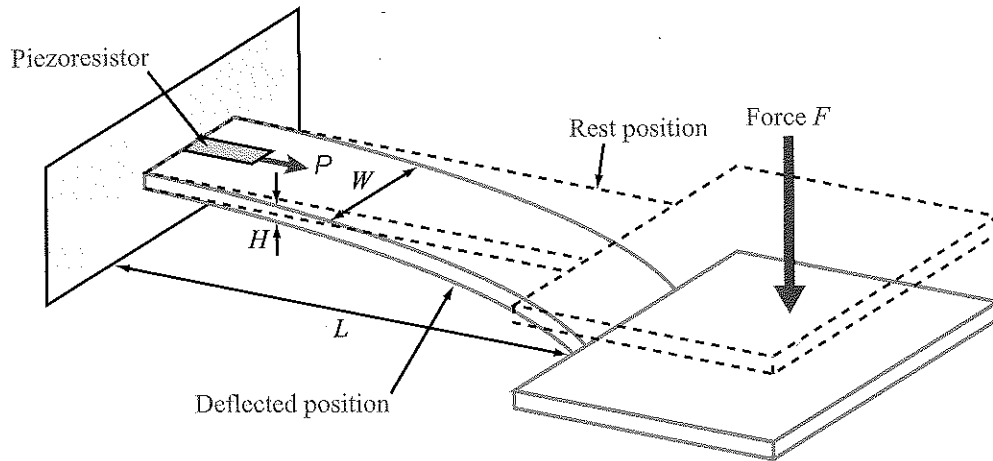Tutoria
much a

Placin

C
g
G
y
P
w
c
a
o
c

Figure 2-38: A cantilever structure with integrated piezoresistor at the base.

We will return to these concepts and learn to apply many other analysis tools throughout the book. Appendix C provides an introduction to the Multisim Tutorial available on the CD that accompanies the book. The Tutorial is a useful reference if you have never used Multisim before. When defining menu selections starting from the main window, the format Menu → Sub-Menu1 → Sub-Menu2 will be used.

2-7.1 Drawing the Circuit

After installing and running Multisim, you will be presented with the *basic user interface* window, also referred to as the *circuit window* or the *schematic capture window* (see Multisim Tutorial on accompanying CD). Here, we will draw our circuits much as if we were drawing them on paper.

Placing Resistors in the Circuit

Components in Multisim are organized into a hierarchy going in a descending general order from Database → Group → Family → Component. Every component that you use in Multisim will fit into this hierarchy somewhere.

Place → Component opens the Select a Component window. (Ctrl-W is the shortcut key for the place-component command. Multisim has many shortcut keys, and it will be worthwhile for you to learn some of the basic ones to improve your efficiency in creating and testing circuits.)

Choose Database: Master Database and Group: Basic in the pulldown menus.

Now select Family: RESISTOR.

You should see a long list of resistor values under Component and the schematic symbol for a resistor (Fig. 2-39). Note that the Family menu contains other components like inductors, capacitors, potentiometers, and many more. We will use these in later chapters.

Scroll down and select a 1k value (the units are in ohms) and then click OK. You should see a resistor in the capture window. Before clicking in the window, Ctrl-R allows you to rotate the resistor in the window. Rotate the resistor such that it is vertical and then click anywhere on the window to place it. Repeat this operation; this time place a vertical 100-ohm resistor directly below the first one (as in Fig. 2-40). How to connect them together will be described shortly. Once you are finished placing components, click Close to return to the schematic capture window.

Note that the components have symbolic names (R1 and R2) and values displayed next to them (1k and 100). Also, by double-clicking on a specific component, you can access many details of the component model and its values. For now, it is sufficient to know that the Resistance value can be altered at any time through the Value menu.

Placing an Independent Voltage Source

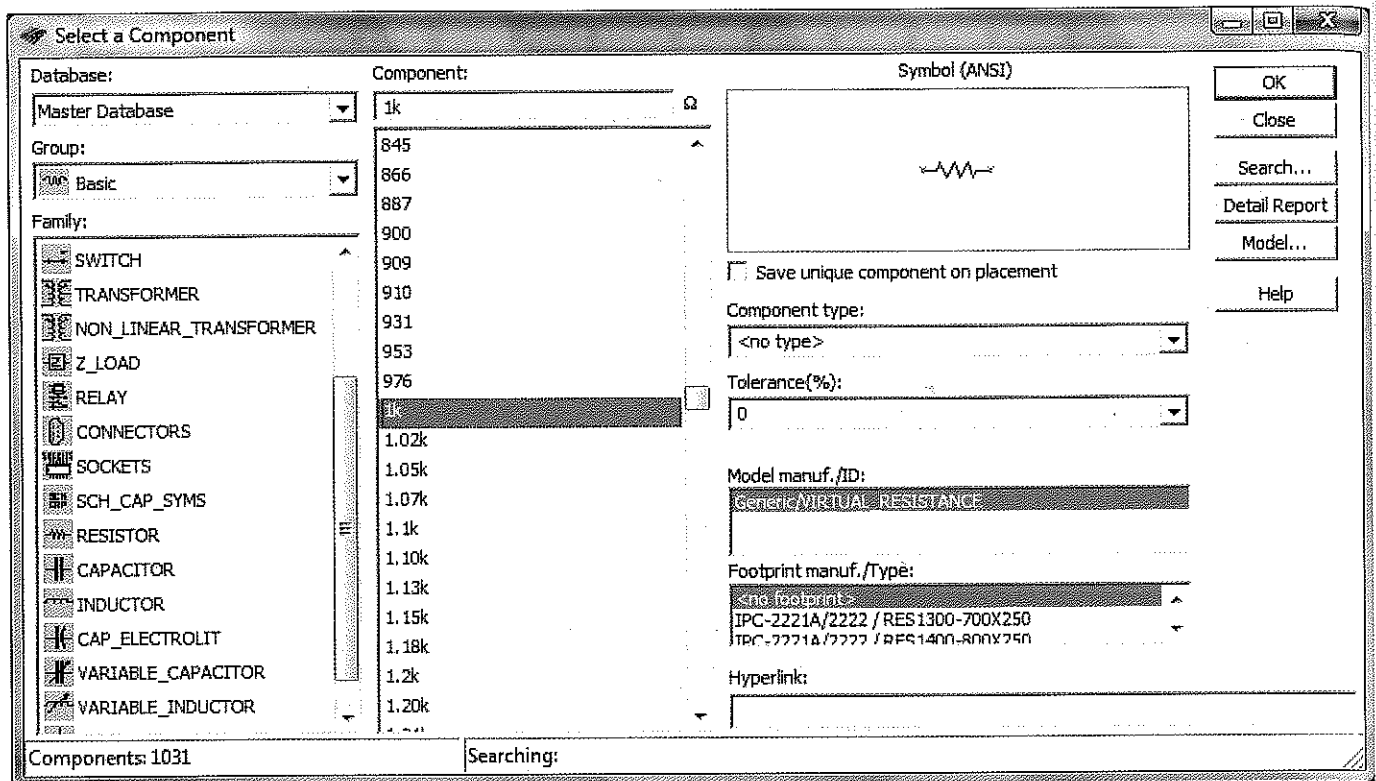Just as you did with the resistors, open up the Select a Component window.

Figure 2-39: Multisim screen for selecting and placing a resistor.

Choose Database: Master Database and Group: Sources in the pulldown menus.

Select Family: POWER_SOURCES.

Under Component select DC_POWER and click OK.

Place the part somewhere to the left of the two resistors (Fig. 2-40).

Once placed, close the component window, then double-click on component V1. Under the Value tab, change the Voltage to 10 V. Click OK.

Wiring Components Together

Place → Wire allows you to use your mouse to wire components together with click-and-drag motions (Ctrl-Q is the shortcut key for the wire command). You can also enable the wire tool automatically by moving the cursor very close to a component node; you should see the mouse pointer change into a black circle with a cross-hair.

Click on one of the nodes of the dc source with the wire tool activated (you should see the mouse pointer change from a black cross to a black circle with a cross hair when you hover it over a node). Additional clicks anywhere in the schematic window will make corners in the wire. Double-clicking will terminate the wire. Additionally, when not already dragging a wire, double-clicking on any blank spot of the schematic will generate a wire based at the origin of clicking.

Wire the components as shown in Fig. 2-40. Add a GROUND reference point as shown in Fig. 2-41. The Ground can be found in the Component list of POWER_SOURCES. We now have a resistive divider.

2-7.2   Solving the Circuit

In Multisim, there are two broad ways in which to solve a circuit. The first, called *Interactive Simulation*, allows you to utilize virtual instruments (such as ohmmeters, oscilloscopes,
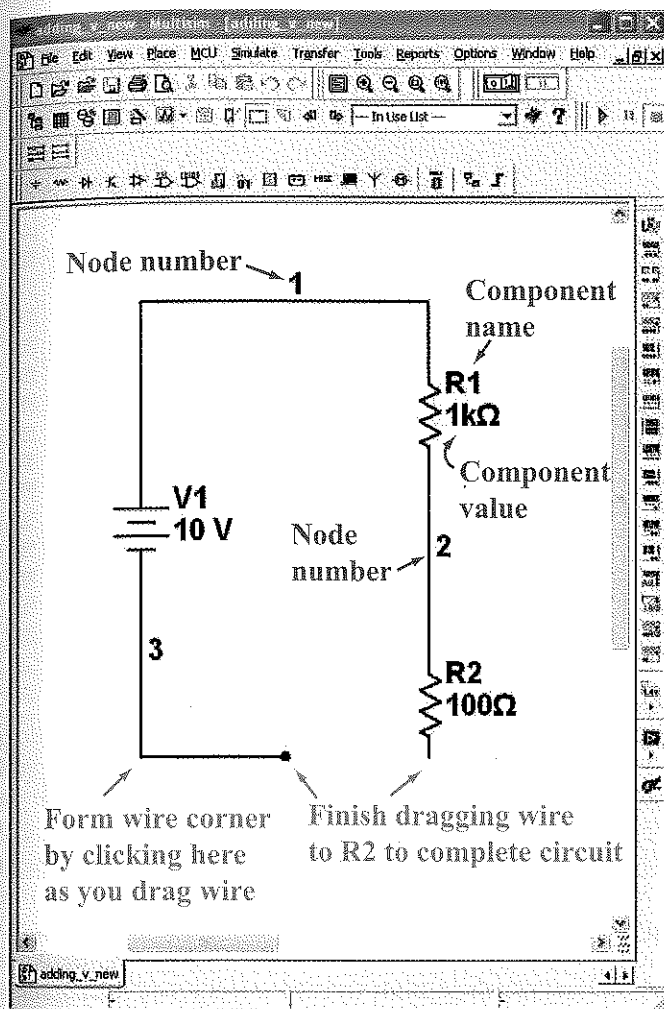
Figure 2-40: Adding a voltage source and completing the circuit.

Figure 2-41: Executing a simulation.

and function generators) to measure aspects of a circuit in a time-based environment. It is best to think of the Interactive Simulation as a simulated "in-lab" experience. Just as in real life, time proceeds in the Interactive Simulation as you analyze the circuit (although the rate at which time proceeds is heavily dependent on your computer's processor speed and the resolution of the simulation). The Interactive Simulation is started using the F5 key, the ▷ button, or the ▣ toggle switch. The simulation is paused using the F6 key, the ▮▮ button, or the ▮▮ button. The simulation is terminated using either the ▮ button or the ▣ toggle switch.
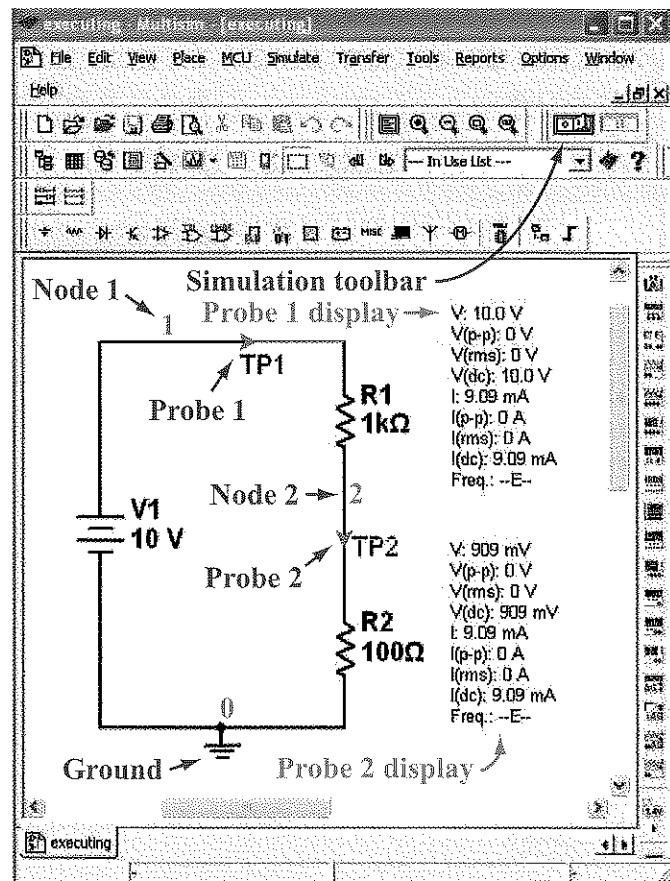
The other main way in which to solve a circuit in Multisim is through *Analyses*. These simulations display their outputs not in instruments, but rather in the *Grapher* window (which may produce tables in some instances). These simulations are run for controlled amounts of time or over controlled sweeps of specific variables or other aspects of the circuit. For example, a dc sweep simulates the values of a specified voltage or current in the circuit over a defined range of dc input values.

Each of the methods described has its own advantages and disadvantages, and in fact, both varieties can perform many of the same simulations, albeit with different advantages. The choice of method to be used for a given circuit really comes down to your preferences, which will be formed as you gain more experience with Multisim.

For the circuit in Fig. 2-41, we wish to solve for the voltages at every node and the currents running through every branch. As you will often see in Multisim, the solution can be obtained
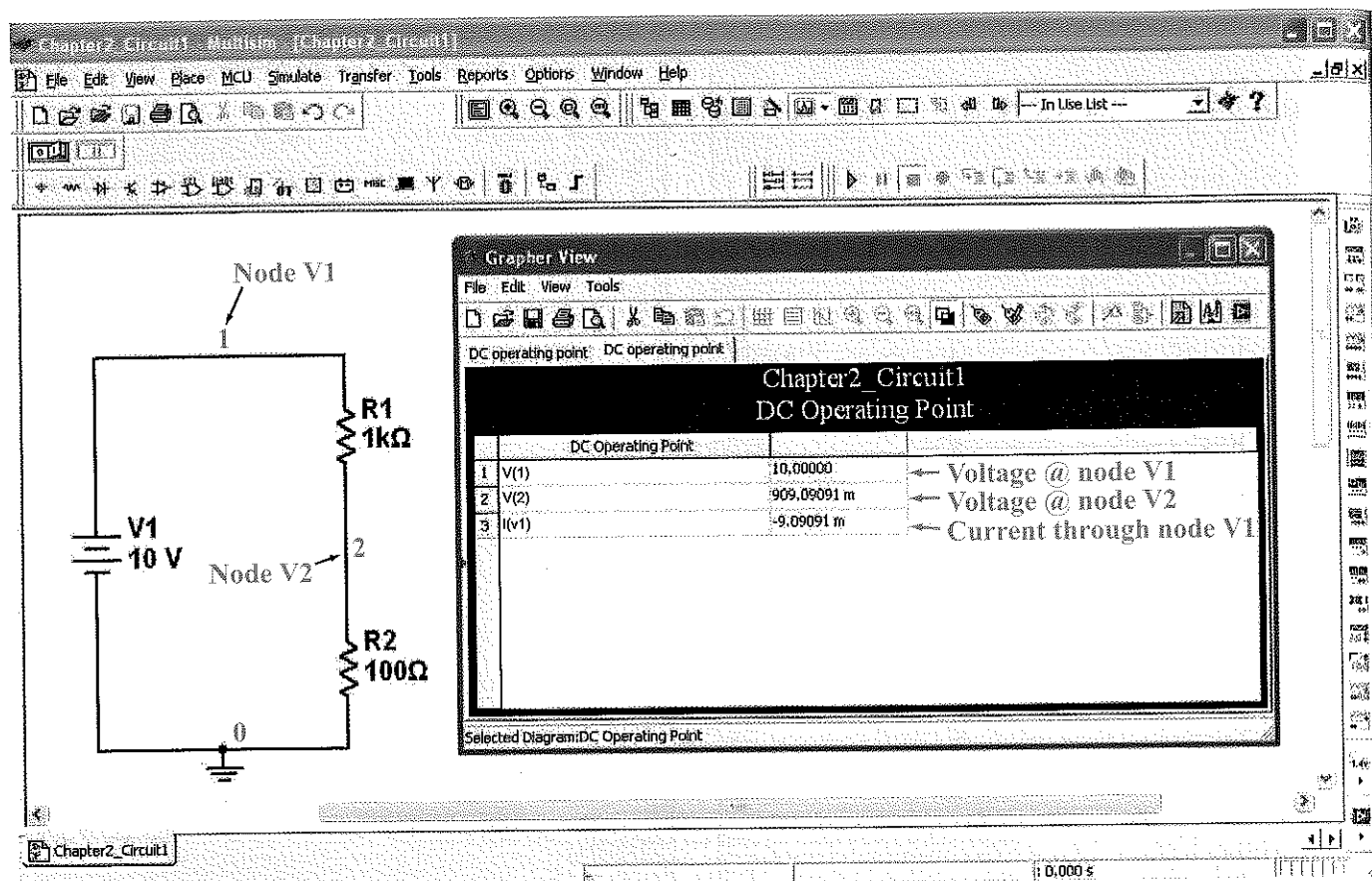
Figure 2-42: Solution window.

using either the Interactive Simulation or through one of the Analyses. We will demonstrate both approaches.

Interactive Simulation

Selecting Simulate → Instruments → Measurement Probe allows you to drag and place a measurement probe onto any node in the circuit. (Note that the Instruments menu contains many common types of equipment used in an electronics laboratory.) The Measurement Probe constantly reports both the current running through the branch to which it is assigned and the voltage at that node. Place two probes into the circuit as shown in Fig. 2-41. When placed, by default, the probes should be pointing in the direction shown in Fig. 2-41. If they are not, you can reverse a probe's direction by right-clicking on it and pressing Reverse Probe Direction. Once the probes are in place, you must run the simulation using the commands for Interactive Simulations.

As expected, the current running through both wires is the same since the circuit has only one loop.

$$I = \frac{V_1}{R_1 + R_2} = \frac{10}{1000 + 100} = 9.09 \text{ mA}.$$

The voltage at node 1 is 10 V, as defined by the source. Application of voltage division (Fig. 2-16) gives

$$V_2 = \left(\frac{R_2}{R_1 + R_2}\right) V_1 = \left(\frac{100}{1100}\right) 10 = 0.909 \text{ V}.$$

DC Operating Point Analysis

The circuit also can be solved using Simulate → Analyses → DC Operating Point. This method is more convenient than the Interactive Simulation when solving circuits with many nodes. After opening this window, you can specify which voltages and currents you want solved. (The Interactive Simulation mode
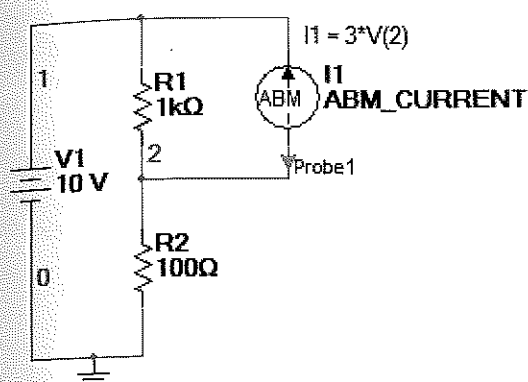
Figure 2-43: Creating a dependent source.

must be stopped, not just paused, in order for the DC Operating Point Analysis mode to work.) Under the Output tab, select the two node voltages and the branch current in the Variables in Circuit window. Make sure the Variables in Circuit pull-down menu is set to All Variables. Once selected, click Add and they will appear in the Selected variables for analysis window. Once you have selected all of the variables for which you want solutions, simply click Simulate. Multisim then solves the entire circuit and opens a window showing the values of the selected voltages and currents (Fig. 2-42).

2-7.3   Dependent Sources

Multisim provides both defined dependent sources (voltage-controlled current, current-controlled current, etc.) and a generic dependent source whose definition can be entered as a mathematical equation. We will use this second type in the following example.

Step 1:   The dependent sources are established as follows: Place → Component opens the Select a Component window.

Choose Database: Master Database and Group: Sources in the pulldown menus.

Select Family: CONTROLLED_VOLTAGE or CONTROLLED_CURRENT.

Under Component, select ABM_VOLTAGE or ABM_CURRENT and click OK.

The value of ABM sources (which stands for Analog Behavioral Modeling) can be set directly with mathematical expressions using any variables in the circuit. For information on the variable nomenclature, which may be somewhat confusing, see the Multisim Tutorial on the CD-ROM.

Step 2:   Using what you learned in Section 2-7.1, draw the circuit shown in Fig. 2-43 (including the probe at node 2).

Step 3:   Double-click the ABM_CURRENT source. Under the value tab, enter: 3*V(2). The expression V(2) refers to the voltage at node 2. This effectively defines this source as a voltage-controlled current source. Note that when making the circuit, if the node numbering in your circuit differs from that in the example (e.g., if nodes 1 and 2 are switched), then take care to keep track of the differences so that you will use the proper node voltage when writing the equation. To edit or change node labels, double-click any wire to open the Net Window. Under Net name enter the label you like for that node.

To write the expression for I1 next to the current source, go to Place → Text, and then type in the expression at a location near the current source. [Ctrl-T is the shortcut key for the place-text command.]

Referencing Currents in Arbitrary Branches

Now let us analyze the circuit using the DC Operating Point Analysis. Our goal is to solve for the voltages at every node and the current running through each branch. Remove the probe from the circuit (if you still have it in there) by clicking on it so it is highlighted, and press the Delete key.

To perform a DC operating point analysis, just as we did earlier in Section 2-7.2, go to Simulate → Analyses → DC Operating Point and transfer all available variables into the Selected variables for analysis window. You should notice that the only variables available are V(1), V(2), and I(v1); if Probe 1 is still connected to your circuit, you should also see I(Probe 1) and V(Probe 1). Where are the other currents, such as the current flowing through R1, the current through R2, or even the current coming out of the dependent source? In Multisim and most SPICE software in general, you can only measure/manipulate currents through a Voltage Source (there are some exceptions, but we will ignore them for now). This is why the current through V1, denoted I(v1), is available but

the currents through the other components are not. A simple trick, however, to obtain these currents is to add a 0 V dc source into the branches where you want to measure current. Do this to your circuit, so that it ends up looking like that shown in Fig. 2-44.

You will notice that there are new nodes in the circuit now, but since V2, V3, and V4 are 0 V sources, $V(3) = V(4) = V(1)$ and $V(5) = V(2)$.

Go back to the DC Operating Point Analysis window and under the Variables in Circuit window there should now be four currents [I(v1), I(v2), I(v3), and I(v4)] and the five voltages. Highlight all four currents as well as V(1) and V(2) and click Add and then click OK. This will bring up the Grapher window with the results of the analysis.

Note that when we analyze the currents through the branches, the current through a voltage source is defined as going *into* the positive terminal. For example, in source V1, this corresponds to the current flowing *from* Node 1 into V1 and then *out* of V1 to Node 0.
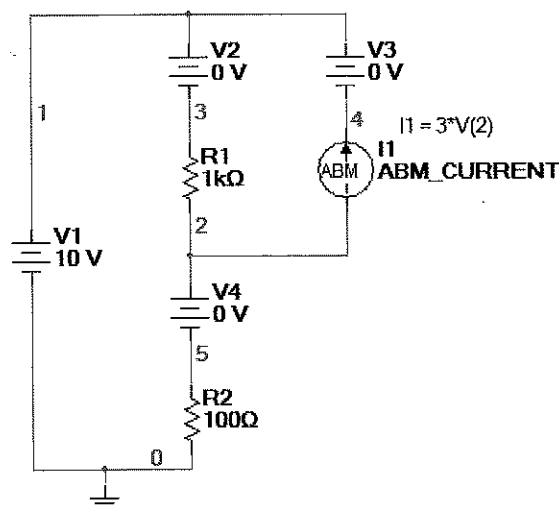

Figure 2-44: Circuit from Fig. 2-43 adapted to read out the currents through R1, R2, and the dependent source.

Concept Question 2-17: In Multisim, how are components placed and wired into circuits?

Concept Question 2-18: How do you obtain and visualize the circuit solution?

Exercise 2-13: The circuit in Fig. E2-13 is called a resistive bridge. How does $V_x = (V_3 - V_2)$ vary with the value of potentiometer R_1?
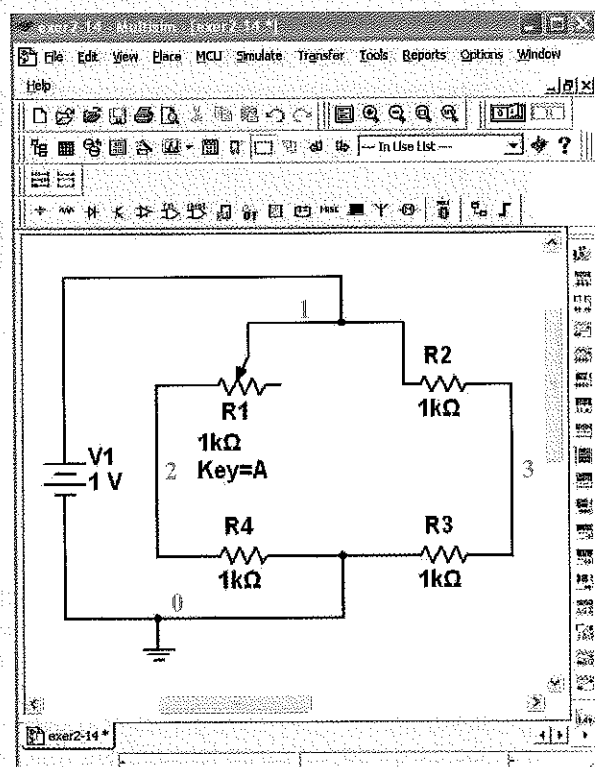

Figure E2-13

Answer: (See ⊘)

Exercise 2-14: Simulate the circuit shown in Fig. E2-14 and solve it for the voltage across R_3. The magnitude of the dependent current source is $V_1/100$.
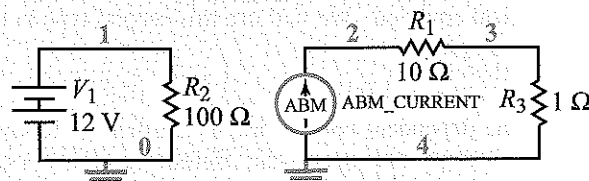

Figure E2-14

Answer: (See ⊘)