Chapter 1

Getting Started with LTspice

1.1 Introduction

SPICE is a computer program used for numerical analysis of electric circuits. SPICE is an acronym for Simulation Program with Integrated Circuit Emphasis. Developed in the early 1970s at the University of California at Berkley, SPICE is generally regarded to be the most widely used circuit simulation program [1]. LTspice is a version of SPICE for personal computers produced by Analog Devices (originally by Linear Technology) [2].

Figure 1.1 shows the opening screen of LTspice. Figure 1.1 identifies three ways of interacting with LTspice. We issue commands to LTspice using the Window Tabs and Toolbar icons. The Status bar displays messages from LTspice. (The LTspice toolbar can be docked, as shown in Figure 1.1 or undocked as shown in Figure 1.2. Double-click on the toolbar to undock a docked toolbar or to dock an undocked toolbar.)

Figure 1.2 identifies the commands associated with some of the icons on the toolbar. Many of those same commands are available from the Window Tabs. Figure 1.3 shows the commands that are available from the "Edit" Window tab.

A circuit diagram is called a schematic in LTspice. Consider representing the circuit shown in Figure 1.4 as a schematic in LTspice. Begin by clicking on the New Schematic icon, , on the toolbar. The LTspice screen will change in a couple of ways (compare Figures 1.1 and 1.5). In particular, the LTspice Workspace will now appear white because it contains a new, blank schematic. Also, additional tabs will be available in the LTspice Window Tabs.

LTspice® for Linear Circuits, First Edition. James A. Svoboda.

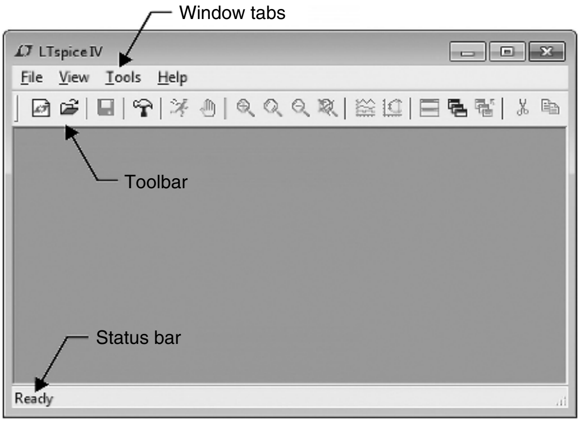


Figure 1.1 The opening screen of LTspice.

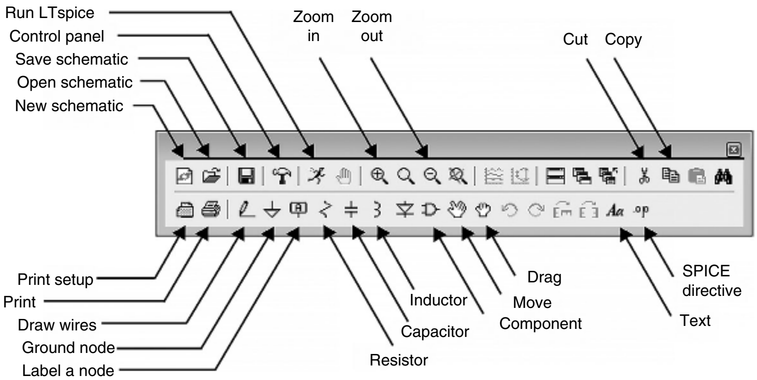


Figure 1.2 Identifying icons on the LTspice toolbar.

Drafting a schematic in LTspice requires four activities:

1) Place symbols representing the circuit elements on the schematic.
2) Adjust the values of the circuit element parameters, e.g. the resistances of the resistors, the voltage of the voltage source, and the current of the current source.
3) Place a ground symbol to identify the bottom node of the schematic as the ground node. (Notice that the bottom node of the circuit in Figure 1.4 has been identified as the ground node.)
4) Draw the wires that connect the circuit elements.

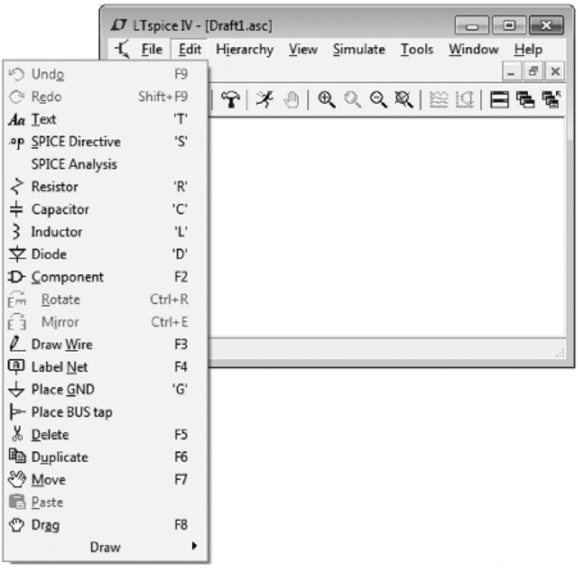


Figure 1.3 Icons on the "Edit" Window Tab.

Figure 1.4 The circuit for the first example.

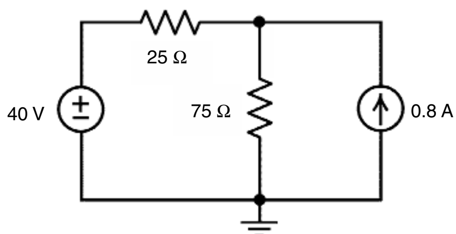


Figure 1.5 A new, blank schematic in LTspice.

The circuit shown in Figure 1.4 consists of two resistors: a voltage source and a current source. Let us begin by placing a symbol representing the voltage source in the LTspice schematic. Click on the Component icon, , on the toolbar to pop up the Select Component Symbol dialog box shown in Figure 1.6. Start typing "voltage source" in the search box. LTspice will find the symbol for a voltage source as soon as "vo" has been typed. Click the "OK" button on the Select Component Symbol dialog box. A voltage source symbol will appear on the schematic as shown in Figure 1.7. Position the voltage source symbol as desired by dragging the mouse, and then left-click to place the voltage source symbol on the schematic. (To move a symbol that was placed previously, click first on the Move toolbar icon, ,

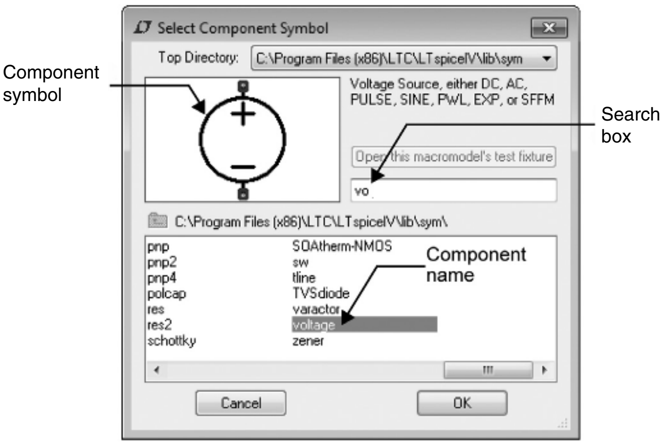


Figure 1.6 The "Select Component Symbol" dialog box.

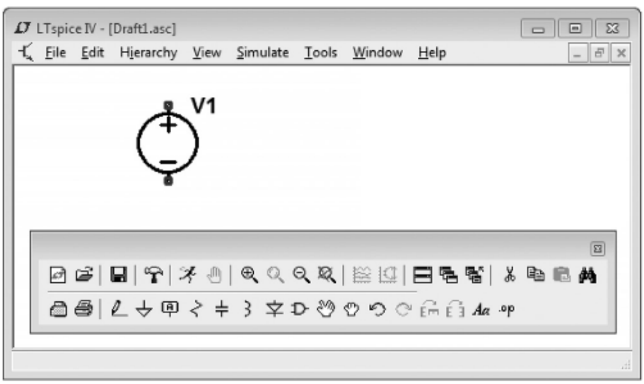


Figure 1.7 A voltage source symbol placed on the schematic.

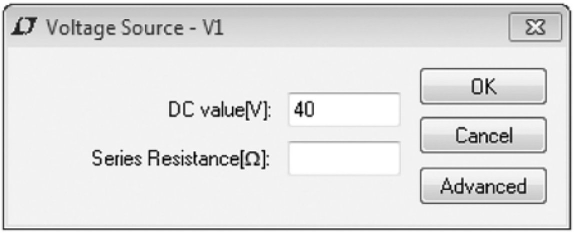


Figure 1.8 Specify the voltage of voltage source V1.

and then click the component symbol to be moved. Position the component symbol as desired by dragging the mouse, and then left-click again to place the voltage source symbol on the schematic.)

Notice that this voltage source has been labeled as element V1 in Figure 1.7. We want the voltage of V1 to be 40 V – the voltage of the voltage source in Figure 1.4. Right-click on the voltage source symbol on the schematic to pop up the dialog box shown in Figure 1.8 and set the voltage of V1 to 40 V.

Click on the resistor icon, , twice to place symbols for resistors R1 and R2 on the schematic. (The resistor icon on the toolbar provides a shortcut to clicking on the component icon, , and then typing "resistor" in the search box in the Select Component Symbol dialog box.) Right-click on R1 and use the resulting dialog box to set the resistance of R1 to 25 Ω. Right-click on R2 and set the resistance of R2 to 75 Ω.

Click on the Component icon, , on the toolbar to pop up the Select Component Symbol dialog box. Start typing "current source" in the search box. LTspice will find the symbol for a current source as soon as "cu" has been typed. Click the "OK" button on the Select Component Symbol dialog box. A current source symbol will appear on the schematic as shown in Figure 1.9. Position the current source symbol as desired. Right-click on the current source and set the current to 0.8 A.

Click on the Ground icon, , to place a ground symbol on the schematic.

Figure 1.9 shows the schematic after placing symbols representing the circuit elements that comprise the circuit in Figure 1.4 and adjusting the values associated with those symbols. We need to rotate the symbol of the 25 Ω resistor by 90° and to rotate the symbol of the current source by 180° to make the schematic correspond to the circuit diagram in Figure 1.4. Click on the move icon, , and then click the symbol of the 25 Ω resistor and type <Ctrl>R to rotate the symbol of the 25 Ω resistor by 90°. Position the symbol of the 25 Ω resistor as desired and left-click to finish. Similarly, Click on the move icon, , and then click the current source symbol and type <Ctrl>R twice to rotate the symbol of the current source by 180°.

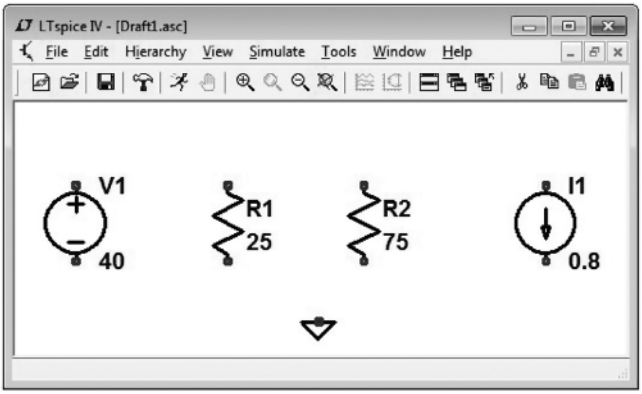


Figure 1.9 The schematic after placing symbols.

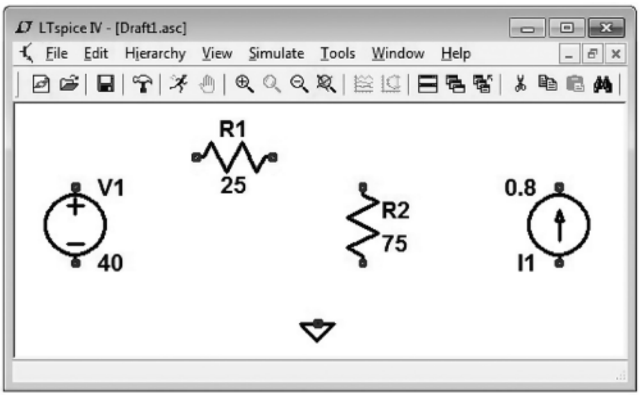


Figure 1.10 The schematic after rotating symbols.

Figure 1.10 shows the schematic after component symbols have been placed and rotated and the values of component parameters have been specified. The wires connecting these component symbols will be represented by horizontal and vertical straight lines. Notice that each component symbol identifies the locations of the terminals of the component by small squares. Consider, for example, drawing the wire connecting the top node of the voltage source symbol to the left node of the symbol of the 25 Ω resistor. This wire will be represented by two connected line segments: one vertical and one horizontal. Click on the "Draft wires" icon, , on the toolbar, and then click on the top terminal of the voltage source symbol to begin drawing

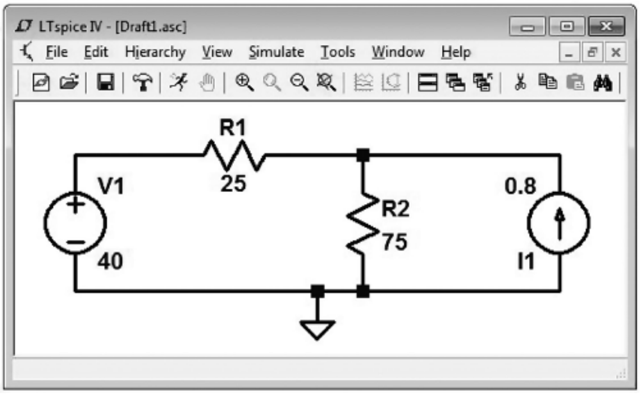


Figure 1.11 The schematic after wiring symbols.

the vertical line segment. Drag the mouse upward to draw the vertical line segment. Click to change directions, and then drag the mouse to the left terminal of the symbol of the 25 Ω resistor.

Figure 1.11 shows the circuit after adding wires to connect the circuit elements.

1.2 Six Steps

We will use a six-step procedure to organize circuit analysis using LTspice. This procedure is illustrated in Figure 1.12 and is stated as follows.

Step 1. **Formulate** a circuit analysis problem.
Step 2. **Describe** the circuit using an LTspice schematic.
Step 3. **Simulate** the circuit using LTspice.
Step 4. **Display** the results of the simulation.
Step 5. **Verify** that the simulation results are correct.
Step 6. **Report** the answer to the circuit analysis problem.

The second, third, and fourth steps of this procedure use LTspice. In the first and fifth steps, the user identifies the problem that is to be solved and verifies that it has indeed been solved. It would be hard to overemphasize the importance of steps 1 and 5.

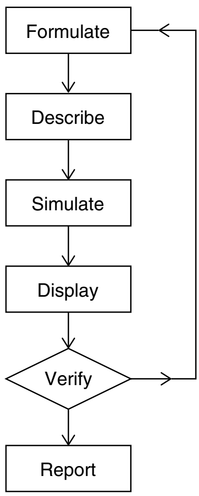


Figure 1.12 A six-step procedure for circuit analysis using LTspice.

If the simulation is not correct, the error must be identified and eliminated so that a correct simulation can be performed. Only after verifying that the simulation is correct can the answer to the circuit analysis problem be reported.

Example 1.1

This example illustrates the six-step procedure for circuit analysis from Figure 1.12 by using it to analyze the circuit shown in Figure 1.13.

Step 1. Formulate a circuit analysis problem.

Determine the value of v_2, the voltage across the 75 Ω resistor of the circuit shown in Figure 1.13. Notice that the bottom node of this circuit is identified as the reference node. Consequently, v_2 is a node voltage of the circuit.

Step 2. Describe the circuit using an LTspice schematic.

In general, the goal of circuit analysis is to determine the values of the voltages across, and currents in, the devices of the circuit. In contrast, analysis of a circuit using LTspice provides the values of the node voltages and the device currents. We can easily determine the values of the device voltages from the values of the node voltages.

Click on the New Schematic toolbar icon, , to start a new schematic. Place the component symbols, adjust the values of the circuit parameters, place a ground symbol, and wire the circuit as described on pages 1–7 to obtain the schematic shown in Figure 1.14.

Step 3. Simulate the circuit using LTspice.

LTspice is capable of performing several types of simulation. To specify the desired type of simulation, click on the Edit Tab and then select "Spice Analysis" (see Figure 1.3) to pop up the Edit Simulation Command dialog box as shown in Figure 1.15.

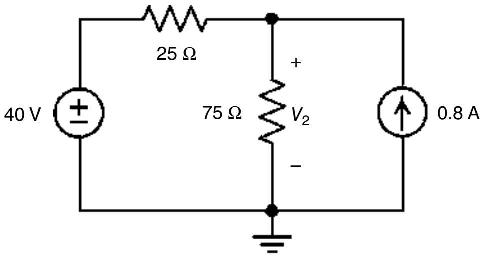


Figure 1.13 The circuit considered Example 1.1.

Example 1.1 (Continued)

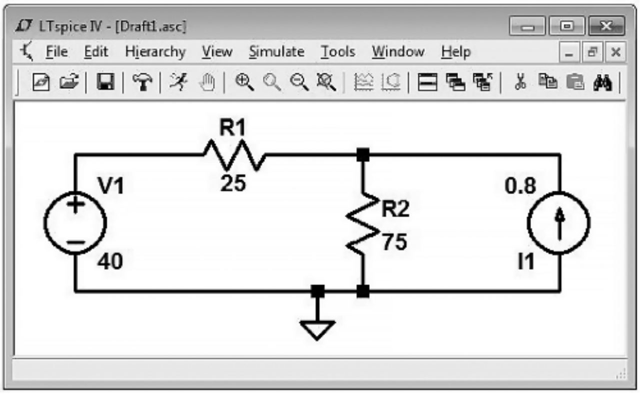


Figure 1.14 The schematic corresponding to the circuit in Figure 1.13.

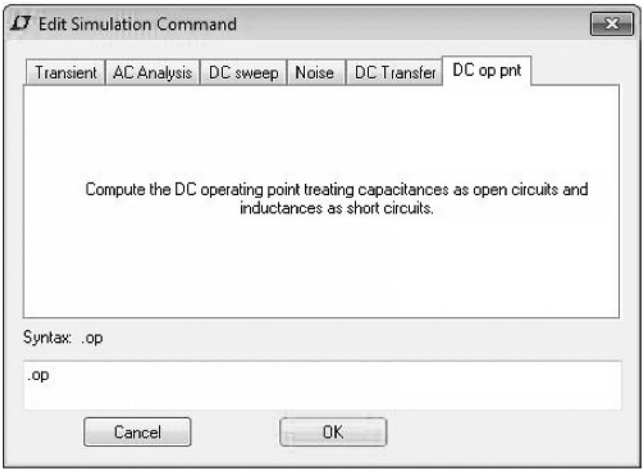


Figure 1.15 Edit simulation command dialog box.

The circuit in Figure 1.13 is a DC circuit because the values of both the voltage-source voltage and the current-source current are constants. For historical reasons SPICE refers to analyzing a DC circuit as "finding the DC operating point" of the circuit.

Select the "DC op pnt" tab as shown in Figure 1.15, and then click the "OK" button. LTspice will generate a "spice directive" consisting of the command ".op." Position the spice directive on the schematic as desired.

(*Continued*)

Example 1.1 (Continued)

Click on the Run LTspice toolbar icon, , to perform a DC analysis of the circuit.

Step 4. Display the results of the simulation.

The results of a DC analysis display automatically as shown in Figure 1.16. The results of the DC analysis are labeled as the “Operating Point” in Figure 1.16. The operating point consists of node voltages in Volts and the circuit element currents in Amps.

Node voltages at two nodes, labeled nodes n001 and n002, are given in Figure 1.16. We expect the node voltage at the top node of the voltage source to be 40 V. Apparently node n001 is the top node of the voltage source and node n002 is the top node of the 75 Ω resistor.

Matching the node numbers generated by LTspice to circuit nodes becomes more tedious when LTspice is used to analyze larger circuits. Instead, let us choose convenient node numbers and label the nodes of the schematic using those convenient node numbers. For example, let us label the node at the top of the voltage source to be node “a” and the node at the top of the 75 Ω resistor to be node “b.” Click on the “Label a Node” toolbar icon, , (see Figure 1.2) to pop up the dialog box shown in Figure 1.17. Enter “a” in the text box and select “Output” from the drop-down menu as shown in Figure 1.17. Click the OK button to add a node label to the schematic. Position this symbol as desired and wire it to the top node of the voltage source as shown in Figure 1.18. Next, label the top node of the 75 Ω resistor to be node b as shown in Figure 1.18.

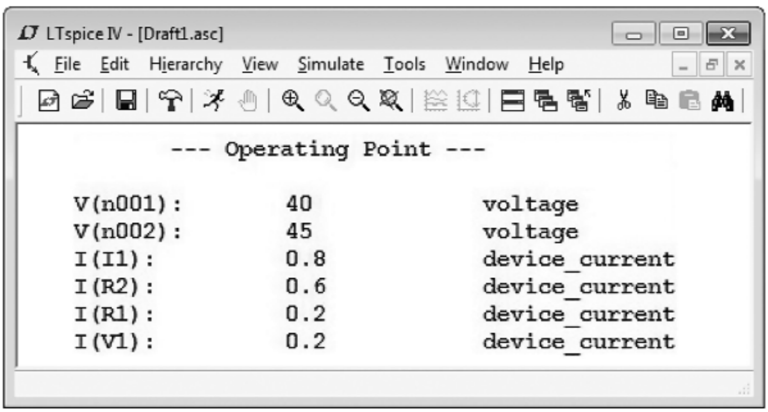
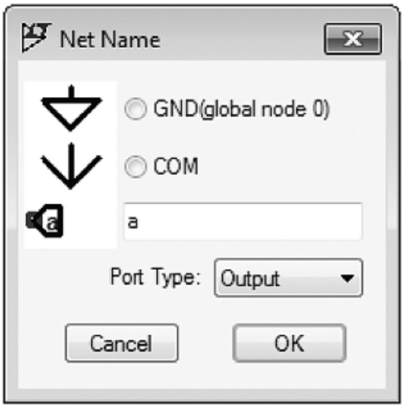


Figure 1.16 Simulation results.

Example 1.1 (Continued)

Figure 1.17 Labeling node "a."



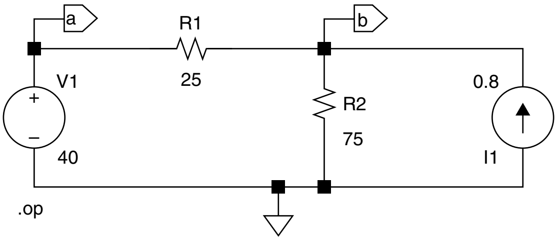


Figure 1.18 The Example 1.1 schematic after labeling nodes "a" and "b."

```
          - - - Operating Point - - -

v(a) :                    40                  voltage
v(b) :                    45                  Voltage
I(I1) :                   0.8                 device_current
I(R2) :                   0.6                 device_current
I(R1) :                   0.2                 device_current
I(V1) :                   0.2                 device_current
```

Figure 1.19 Revised simulation results.

Run the simulation again to obtain the revised simulation results shown in Figure 1.19. Notice that the names of the nodes are now "a" and "b."

(Continued)

Example 1.1 (Continued)

Step 5. Verify that the simulation results are correct.

The node voltages and device currents shown in Figure 1.19 are correct if, and only if, they satisfy both Ohm's and Kirchhoff's laws. We will label the circuit diagram from Figure 1.13 using the simulation results and then check if those currents and voltages satisfy Ohm's and Kirchhoff's laws.

We see from Figure 1.19 that I(R1) = 6 A. Is this the value of the current directed from right to left in the 25 Ω resistor or the current directed from left to right?

Consider the resistor shown in Figure 1.20. This resistor is connected to node a on the left and to node b on the right. The corresponding node voltages are V(a) and V(b). The currents i_1, directed from node a toward node b, and i_2, directed from node b toward node a, are given by

$$i_1 = \frac{V(\mathrm{a}) - V(\mathrm{b})}{R} \quad \text{and} \quad i_2 = \frac{V(\mathrm{b}) - V(\mathrm{a})}{R}$$

When $R > 0$ and $V(\mathrm{a}) > V(\mathrm{b})$, then $i_1 > 0$ and $i_2 < 0$. The rule for assigning a direction to I(R1) is simple: when R1 > 0 and $V(\mathrm{a}) > V(\mathrm{b})$ then I(R1) is the current directed from node a toward node b.

A different rule is used to assign a direction to a voltage-source current such as I(V1) in Figure 1.19. Consider the voltage source shown in Figure 1.21. The current I(V1) is directed from the node near the plus sign of the voltage polarity toward the node near the minus sign.

We can label the node voltages and device currents given in the simulation results (Figure 1.16) on the circuit diagram (Figure 1.13) as shown in Figure 1.22. We readily verify that the node voltages and device currents satisfy Ohm's and Kirchhoff's laws. (See the appendix to this chapter for an introduction to Ohm's and Kirchhoff's laws.)

Step 6. Report the result.

The value of v_2, the voltage across the 75 Ω resistor in Figure 1.13, is v_2 = 45 V.

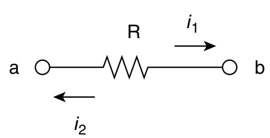


Figure 1.20 Node voltages and resistor currents.

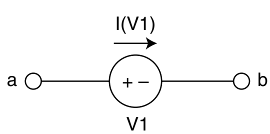


Figure 1.21 Node voltages and voltage-source currents.

Example 1.1 (Continued)

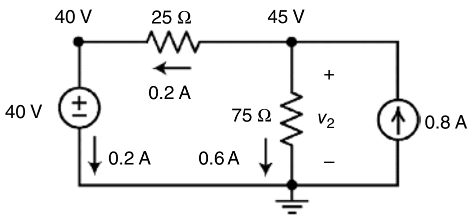


Figure 1.22 Node voltages and device currents.

1.2.1 LTspice Notation

Figure 1.23 shows an LTspice component, in this case, a resistor. The component is labeled twice by a “Instance Name” and also by a “Value.” The instance name identifies a particular component and distinguishes it from all other components in the same schematic. The instance name is a sequence of characters (letters and numbers) beginning with a letter. The initial letter corresponds to the component type as shown in Table 1.1.

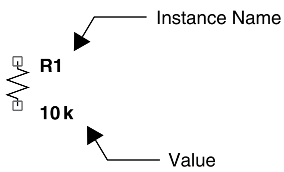


Figure 1.23 An LTspice component labeled by its “Instance Name” and its “Value.”

The “Value” is a real number or an expression. In Figure 1.23 the value is “10k” where “k” is a multiplying factor abbreviation indicating 1000. That is, the value of 10k is 10 * 1000 = 10 000. Table 1.2 lists the available multiplying factors. The “Value” of a resistor represents the resistance of the

Table 1.1 LTspice components and their labels.

Component	Instance name	Value
Current source	I. . .	Source current, A
Voltage source	V. . .	Source voltage, V
Capacitor	C. . .	Capacitance, F
Inductor	L. . .	Inductance, H
Resistor	R. . .	Resistance, Ω
VCVS	E. . .	Gain, V/V
CCCS	F. . .	Gain, A/A
VCCS	G. . .	Gain, A/V
CCVS	H. . .	Gain, V/A

Table 1.2 LTspice scale factor abbreviations.

Letter suffix	Multiplying factor	Name of suffix
T	1E12	tera
G	1E9	giga
MEG	1E6	mega
K	1E3	kilo
M	1E-3	milli
MIL	25.4E-6	mil
U	1E-6	micro
N	1E-9	nano
P	1E-12	pico
F	1E-15	femto

resistor in Ω. The third column of Table 1.1 gives interpretation of the Value for several component types.

Both the Instance Name and the Value are properties of LTspice components and are specified and changed using the Component Attribute Editor. For example, to change the resistance from "10k" to "25k," <CNTL>right-click on resistor R1 to pop up the Component Attribute Editor as shown in Figure 1.24. (The Xs on the right-hand column indicate

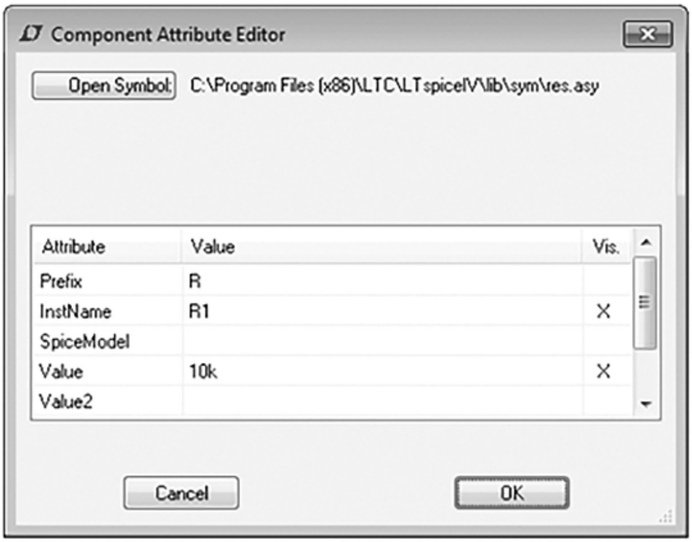


Figure 1.24 The component attribute editor.

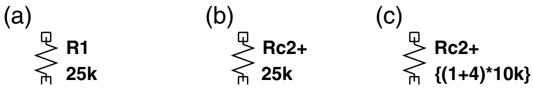


Figure 1.25 An LTspice component labeled by its "Instance Name" and its "Value."

that both the Instance Name and the Value of Component R1 will be visible on the schematic.) Edit the Value to change "10k" to "25k" and click the "OK" button. Component R1 will now appear as shown in Figure 1.25a.

<CNTL>Right-click on Component R1 to pop up the Component Attribute Editor again. Edit the Instance Name to change "R1" to "Rc2+" and click the "OK" button. Component R1 will now appear as shown in Figure 1.25b.

<CNTL>right-click on Component Rc2+ to pop up the Component Attribute Editor again. Edit the Value to change "25k" to "{(1 + 4) * 10k}" and click the "OK" button. Component Rc2+ will now appear as shown in Figure 1.25c. Braces indicate that the Value is an expression rather than a number. The resistance of component Rc2+ is $(1+4) * 10\text{k} = 50\,\text{k}\Omega$.

1.A Appendix: Verifying LTspice Simulation Results

1.A.1 Node Voltages and Device Currents

A circuit is a collection of interconnected devices. (Devices are sometimes called "circuit elements" or just "elements.") These devices are connected to each other using "leads" that have been attached to the devices for that purpose. Figure 1.A.1a shows a device that has two leads that can be connected to other devices. Figure 1.A.1b shows three devices connected to each other by their leads. The place where these leads are connected together is called a "node," as shown in Figure 1.A.1b.

Figure 1.A.2a shows a generic circuit element connected between two nodes labeled as node "a" and node "b." There are three nodes in Figure 1.A.2b: nodes a and b of the generic circuit element and a third node

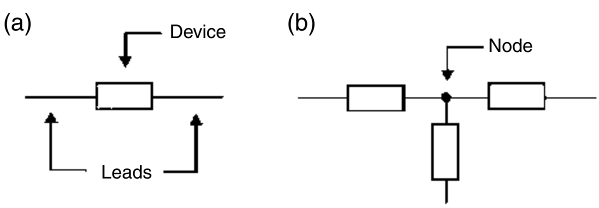


Figure 1.A.1 Leads and nodes.

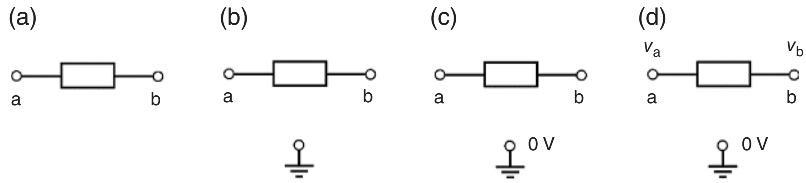


Figure 1.A.2 Nodes and node voltages.

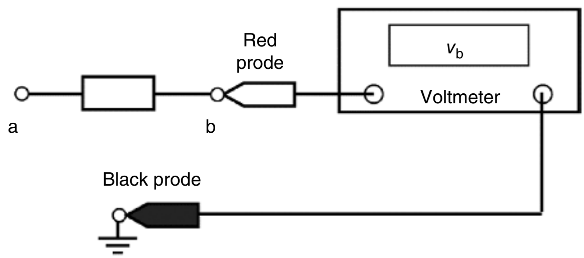


Figure 1.A.3 Measuring a node voltage in the laboratory using a voltmeter.

that has been designated as "the ground node" by attaching the ground symbol shown in Figure 1.A.2b to that node. By convention the voltage at the ground node is 0 V as shown in Figure 1.A.2c. Having identified the ground node, we can associate a "node voltage" with each node of the circuit as shown in Figure 1.A.2d.

Figure 1.A.3 shows how to measure a **node voltage** using an instrument called a voltmeter. The voltmeter has two color-coded probes: a black probe and a red probe. (In a black and white drawing such as Figure 1.A.3, the red probe will be represented by a white probe.) When the black probe is connected to the ground node and the red probe is connected to node b as shown in Figure 1.A.3, the voltmeter displays the value, v_b, of the node voltage at node b. The value of v_a, the node voltage at node a, is obtained by moving the red probe from node b to node a. Often we label the nodes of a circuit with the corresponding node voltages as shown in Figure 1.A.2d.

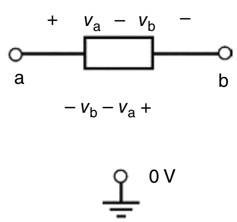


Figure 1.A.4 Labeling element voltages and their polarities.

The voltage across a circuit element can be labeled in either of the two ways as shown in Figure 1.A.4. The + and − signs in these labels indicate the polarity of the element voltage. Figure 1.A.5 shows how to measure a voltage across a circuit element using a voltmeter. Notice

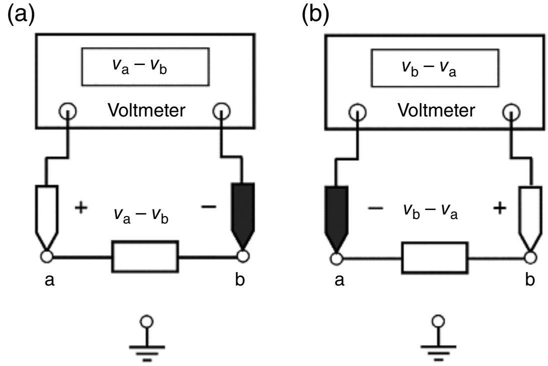


Figure 1.A.5 Measuring element voltages in the laboratory using a voltmeter.

that the locations of the color-coded probes of the voltmeter correspond to the polarity of the element voltage. Also, since

$$v_a - v_b = -(v_b - v_a) \tag{1.A.1}$$

reversing the polarity of an element voltage requires multiplying the value of that element voltage by −1.

The current in a circuit element can be labeled in either of the two ways as shown in Figure 1.A.6. Figure 1.A.7 shows how to measure a current in a circuit element using an ammeter. Reversing the direction of an element current requires multiplying that current by −1. Consequently, $i_{ab} = -i_{ba}$ in Figure 1.A.7.

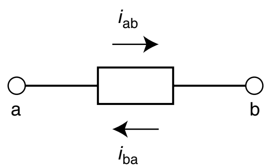


Figure 1.A.6 Labeling element currents and their directions.

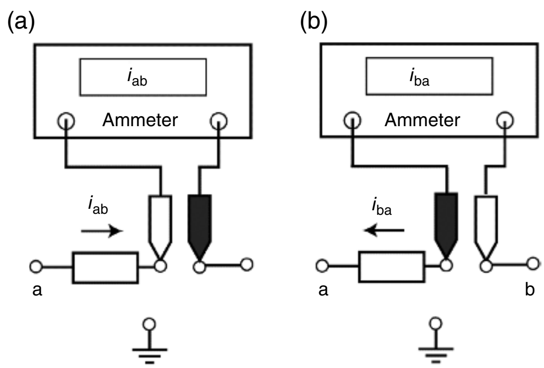


Figure 1.A.7 Measuring element currents in the laboratory using an ammeter.

The terms "passive convention" and "active convention" describe the relationship between the polarity of the element voltage of a circuit element and the direction of the element current of that circuit element. The element current and voltage are said to adhere to the **passive convention** when the element current is directed from the node near the plus sign of the element voltage polarity toward the node near the minus sign of the element voltage polarity. Conversely, the element current and element voltage are said to adhere to the **active convention** when the element current is directed from the node near the minus sign of the element voltage polarity toward the node near the plus sign of the element voltage polarity.

1.A.2 Ohm's and Kirchhoff's Laws

In 1827, Georg Simon Ohm determined that the current in a conducting wire of uniform cross section could be expressed as $i = \dfrac{Av}{\rho L}$ where A is the cross-sectional area, ρ is the resistivity, and L is the length of the wire. Also, the direction of the current, i, in the wire is required to adhere to the passive convention with the voltage, v, across the resistor. Ohm defined the resistance, R, of the wire to be $R = \dfrac{A}{\rho L}$, after which Ohm's law [3] is

$$v = Ri \tag{1.A.2}$$

Suppose that the circuit element in Figure 1.A.8 is a resistor, as shown in Figure 1.A.9. According to Ohm's law, the voltage across a resistor is proportional to the current in the resistor. (Notice that the resistor

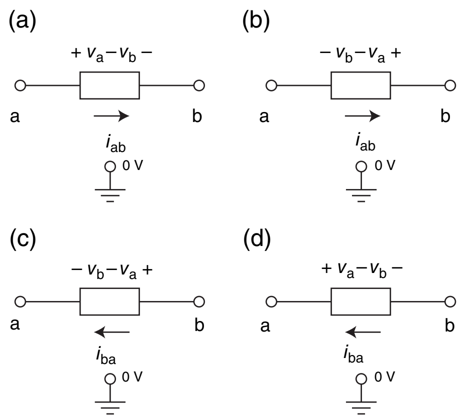


Figure 1.A.8 The element current and element voltage adhere to the passive convention in (a) and (c) and to the active convention in (b) and (d).

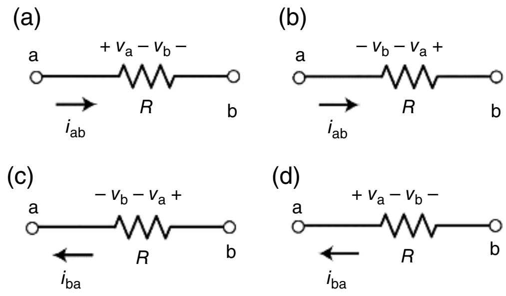


Figure 1.A.9 The resistor current and resistor voltage adhere to the passive convention in (a) and (c) and to the active convention in (b) and (d).

voltage and current labeled in Figure 1.A.8a and c adhere to the **passive convention**.) In this case

$$v_a - v_b = Ri_{ab} \tag{1.A.3}$$

where

$$R = \frac{v_a - v_b}{i_{ab}} \tag{1.A.4}$$

is called the resistance of the resistor. The unit of resistance is Ohms = Volt/Amp.

Using Eqs. (1.A.1) and (1.A.2) with Eq. (1.A.4), we obtain

$$R = \frac{v_a - v_b}{i_{ab}} = -\frac{v_b - v_a}{i_{ab}} = +\frac{v_b - v_a}{i_{ba}} = -\frac{v_a - v_b}{i_{ba}} \tag{1.A.5}$$

In 1847, Gustav Robert Kirchhoff, a professor at the University of Berlin, formulated two important laws that provide the foundation for analysis of electric circuits. These laws are referred to as *Kirchhoff's current law* (KCL) and *Kirchhoff's voltage law* (KVL) in his honor.

Kirchhoff's current law (KCL): The algebraic sum of the currents directed into a node at any instant is zero.

Kirchhoff's voltage law (KVL): The algebraic sum of the voltages around any loop in a circuit is identically zero for all time.

Kirchhoff's laws are a consequence of conservation of charge and conservation of energy [4].

The phrase *algebraic sum* in Kirchhoff's current law indicates that we must take current directions into account as we add up the currents of the elements connected to a particular node. One way to take current directions into account is to use a plus sign when the current is directed toward the

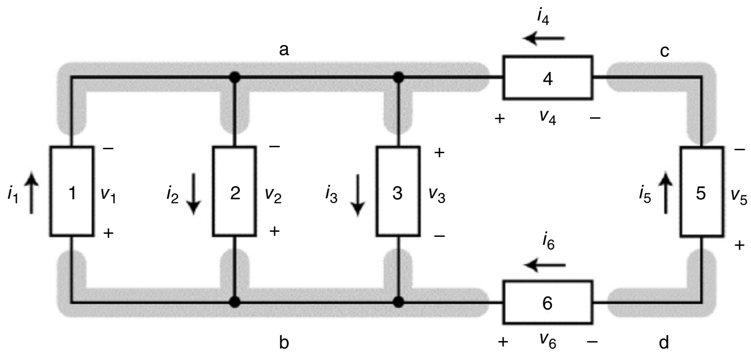


Figure 1.A.10 A circuit consisting of six elements connected together at four nodes.

node and a minus sign when the current is directed away from the node. For example, consider the circuit shown in Figure 1.A.10.

Four elements of this circuit – elements 1, 2, 3, and 4 – are connected to node a. By KCL, the algebraic sum of the element currents i_1, i_2, i_3, and i_4 must be zero. Currents i_2 and i_3 are directed away from node a, so we will use a plus sign for i_2 and i_3. In contrast, currents i_1 and i_4 are directed toward node a, so we will use a minus sign for i_1 and i_4. The KCL equation for node a of Figure 1.A.10 is

$$-i_1 + i_2 + i_3 - i_4 = 0 \tag{1.A.6}$$

An alternate way of obtaining the algebraic sum of the currents into a node is to set the sum of all the currents directed away from the node equal to the sum of all the currents directed toward that node. Using this technique, we find that the KCL equation for node a of Figure 1.A.10 is

$$i_2 + i_3 = i_1 + i_4 \tag{1.A.7}$$

Similarly, the KCL equation for node b of Figure 1.A.10 is

$$i_1 = i_2 + i_3 + i_6$$

KVL refers to "a *loop* in a circuit." A *loop* is a closed path through a circuit that does not encounter any intermediate node more than once. For example, starting at node a in Figure 1.A.10, we can move through element 4 to node c, then through element 5 to node d, through element 6 to node b, and finally through element 3 back to node a. We have a closed path, and we did not encounter any of the intermediate nodes – b, c, or d – more than once. Consequently, elements 3, 4, 5, and 6 comprise a loop.

Similarly, elements 1, 4, 5, and 6 comprise a loop of the circuit shown in Figure 1.A.10. Elements 1 and 3 comprise yet another loop of this circuit. The circuit has three other loops: elements 1 and 2, elements 2 and 3, and elements 2, 4, 5, and 6.

The phrase algebraic sum indicates that we must take polarity into account as we add up the voltages of elements that comprise a loop. One way to take polarity into account is to move around the loop in the clockwise direction while observing the polarities of the element voltages. We write the voltage with a plus sign when we encounter the + of the voltage polarity before the −. In contrast, we write the voltage with a minus sign when we encounter the − of the voltage polarity before the +. For example, consider the circuit shown in Figure 1.A.10. Elements 3, 4, 5, and 6 comprise a loop of the circuit. By KVL, the algebraic sum of the element voltages v_3, v_4, v_5, and v_6 must be zero. As we move around the loop in the clockwise direction, we encounter the + of v_4 before the −, the − of v_5 before the +, the − of v_6 before the +, and the − of v_3 before the +. Consequently, we use a minus sign for v_3, v_5, and v_6 and a plus sign for v_4. The KVL equation for this loop of Figure 1.A.10 is

$$v_4 - v_5 - v_6 - v_3 = 0$$

Similarly, the KVL equation for the loop consisting of elements 1, 4, 5, and 6 is

$$v_4 - v_5 - v_6 + v_1 = 0$$

The KVL equation for the loop consisting of elements 1 and 2 is

$$-v_2 + v_1 = 0$$

1.A.3 Verifying LTspice Simulation Results

Ohm's and Kirchhoff's laws provide a way to verify that we have simulated a circuit correctly. As an example, consider the circuit shown in Figure 1.A.11a, the corresponding LTspice schematic shown in Figure 1.A.11b, and simulation results shown Figure 1.A.11c. The first two lines of the simulation results indicate that the value of the node voltage at node a is 40 V and the value of the node voltage at node b is 45 V. These values are used in Figure 1.A.11d to label the nodes of the circuit with the values of their node voltages.

(Recall that Figure 1.A.3 shows how to measure the value of a node voltage using a voltmeter. The black probe of the voltmeter is connected to the ground node, and the red probe is connected to the node at which the node voltage is being measured. The voltmeter displays the value of the node voltage at the node to which the red probe is connected.)

In contrast, consider the currents in the 25 and 75 Ω resistors. The simulation results provided by LTspice specify the values of these currents but do not specify the directions of these currents. Instead, the directions of the resistor currents are determined from the simulation results using two consequences of Ohm's law:

1) The resistor current and voltage related by Ohm's law must adhere to the passive convention so the resistor current is directed from the node near the plus sign of the resistor voltage polarity toward node near the minus sign.
2) When the resistance is positive, and it almost always is, the resistor voltage and resistor current related by Ohm's law are either both positive or both negative.

Consider the 25 Ω resistor. The simulation results indicate that the resistor current is 0.2 A. This current is positive and the resistance, 25 Ω, is also positive so, using Ohm's law, the resistor voltage is also positive:

$$25(0.2) = 5 \text{ V}$$

The voltages at the nodes of the 25 Ω resistor are 40 and 45 V so the voltage across the 25 Ω resistor is $+45 - 40 = 5$ V rather than $+ \ 40 - 45 = -5$ V. The polarity of the voltage across the 25 Ω resistor will have the + near the node at which the node voltage is 45 V and the − near the node at which the node voltage is 40 V. The current in the 25 Ω resistor must be

$$\frac{5 \text{ V}}{2 \ \Omega} = 0.2 \text{ A}$$

directed from left-to-right to adhere to the passive convention with the resistor voltage.

Next consider the 75 Ω resistor. The simulation results indicate that the resistor current is 0.6 A. This current is positive and the resistance, 75 Ω, is also positive so, using Ohm's law, the resistor voltage is also positive:

$$75(0.6) = 45 \text{ V}$$

The voltages at the nodes of the 75 Ω resistor are 45 and 0 V so the voltage across the 75 Ω resistor is 45 V. The polarity of the voltage across the 75 Ω resistor will have the + near the top node and the – near the bottom node. The current in the 75 Ω resistor must be

$$\frac{45\ \text{V}}{75\ \Omega} = 0.6\ \text{A}$$

directed downward to adhere to the passive convention with the resistor voltage.

Having verified that the node voltages and device currents obtained by simulating the circuit shown in Figure 1.A.11 satisfy both Ohm's and Kirchhoff's laws, we are now confident that the simulation results are correct.

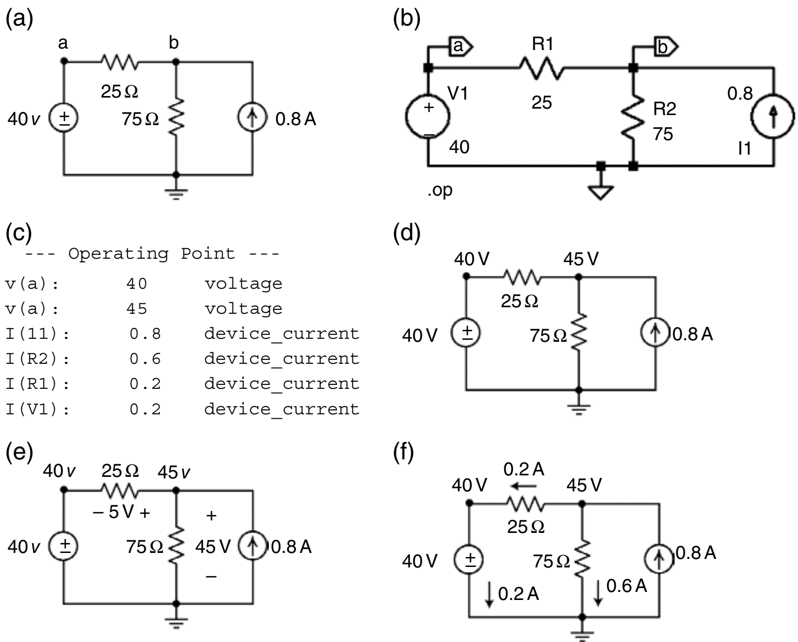


Figure 1.A.11 Interpreting LTspice simulation results. A DC circuit (a) is represented by an LTspice schematic (b). Simulating the schematic using LTspice produces node voltages and device currents tabulated in (c). Next, label the node voltages on the circuit diagram (d) and then determine the resistor voltages from the node voltages (e). The resistor currents, including the directions of the resistor currents, are determined using Ohm's law (f).

References

1. Perry, T.S. (1998). Donald Pederson: The Father of SPICE. *IEEE Spectrum* 35 (6): 22–27.
2. Mike Engelhardt (2022). Analog devices. Linear technology. LTspice, 17 August. https://en.wikipedia.org/wiki/LTspice (accessed 23 October 2022).
3. Svoboda, J.A. and Dorf, R.C. (2014). *Introduction to Electric Circuits*, 9e, 25. NJ: Wiley.
4. Svoboda, J.A. and Dorf, R.C. (2014). *Introduction to Electric Circuits*, 9e, 56. NJ: Wiley.