Appendix C: MATLAB's Simulink Tutorial

◖ C.1 Introduction

Readers who are studying MATLAB may want to explore the functionality and convenience of MATLAB's Simulink. Before proceeding, the reader should have studied Appendix B, the MATLAB Tutorial, including Section B.1, which is applicable to this appendix.

MATLAB's Simulink Version 7.4 and MATLAB Version 7.9(R2009b) are required in order to use Simulink.

The models described in this appendix, which are available at www.wiley.com/college/nise, were developed on a PC using MATLAB Version 7.9 and Simulink Version 7.4. The code will also run on workstations that support MATLAB. Consult the MATLAB Installation Guide for your platform for minimum system hardware requirements.

Simulink is used to simulate systems. It uses a graphical user interface (GUI) for you to interact with blocks that represent subsystems. You can position the blocks, resize the blocks, label the blocks, specify block parameters, and interconnect blocks to form complete systems from which simulations can be run.

Simulink has block libraries from which subsystems, sources (that is, function generators), and sinks (that is, scopes) can be copied. Subsystem blocks are available for representing linear, nonlinear, and discrete systems. LTI objects can be generated if the Control System Toolbox is installed.

Help is available on the menu bar of the **MATLAB** Window. Under **Help** select **Product Help.** When the help screen is available, choose **Simulink** under the **Contents** tab. Help is also available for each block in the block library and is accessed either by right-clicking a block's icon in the **Simulink Library Browser** and selecting **Help for** . . . or by double-clicking the block's icon and then clicking the **Help** button. Finally, screen tips are available for some toolbar buttons. Let your mouse's pointer rest on the button for a few seconds to see the explanation.

◖ C.2 Using Simulink

The following summarize the steps to take to use Simulink. Section C.3 will present four examples that demonstrate and clarify these steps.
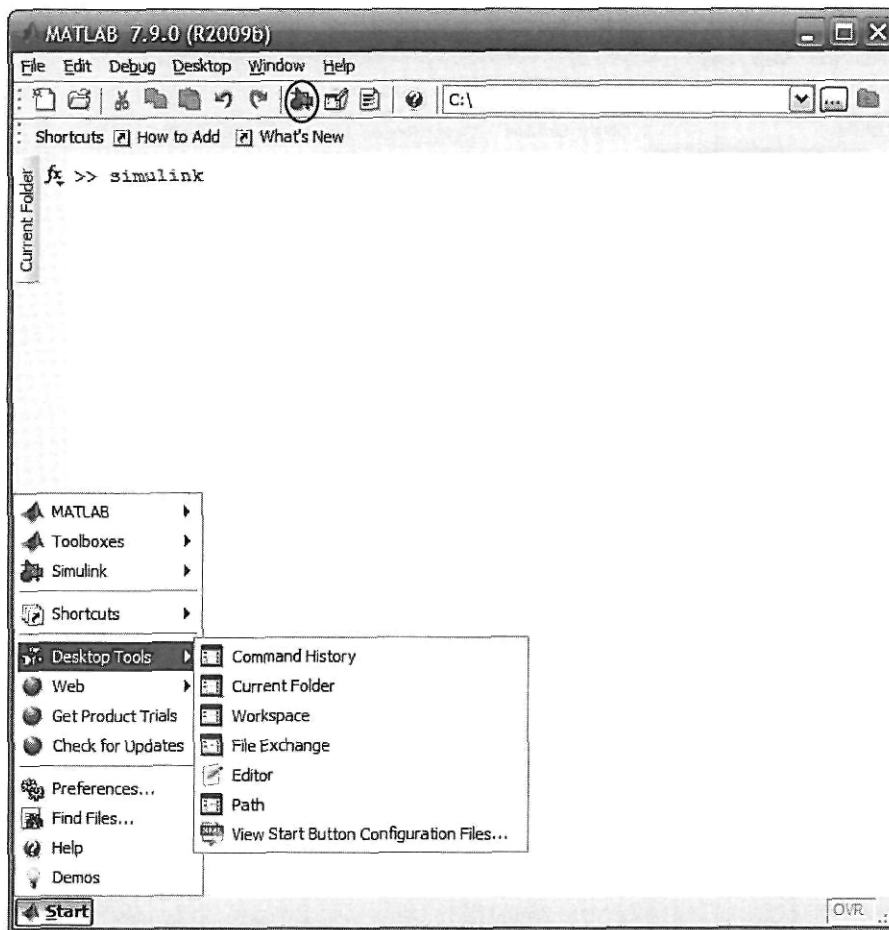
836

FIGURE C.1 MATLAB Window showing how to access Simulink. The **Simulink Library Browser** button is shown circled.
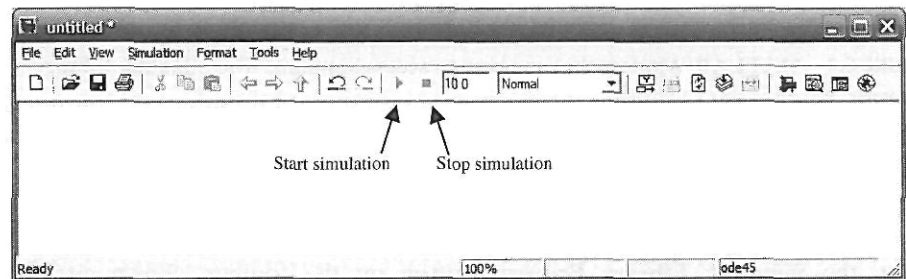
1. **Access Simulink** The **Simulink Library Browser**, from where we begin Simulink, is accessed by typing *simulink* in the **MATLAB Command Window** or by clicking on the **Simulink Library Browser** button on the toolbar, shown circled in Figure C.1.

   In response, MATLAB displays the **Simulink Library Browser** shown in Figure C.2(*a*). We now create an **untitled** window, Figure C.2(*b*), by clicking on the **Create a new model** button (shown circled in Figure C.2(*a*)) on the toolbar of the **Simulink Library Browser**. You will build your system in this window. Existing models may be opened by clicking on the **Open a model** button on the **Simulink Library Browser** toolbar. This button is immediately to the right of the **Create a new model** button. Existing models may also be opened by selecting the **Current Folder** from the **Command Window Start** menu or the tab on the left side of the **Command Window as** shown in Figure C.1, selecting your file names, and then dragging them to the **MATLAB Command Window**.

2. **Select blocks** Figure C.2(*a*) shows the **Simulink Library Browser** from which all blocks can be accessed. The left-hand side of the browser shows major libraries, such as **Simulink**, as well as underlying block libraries, such as **Continuous**. The
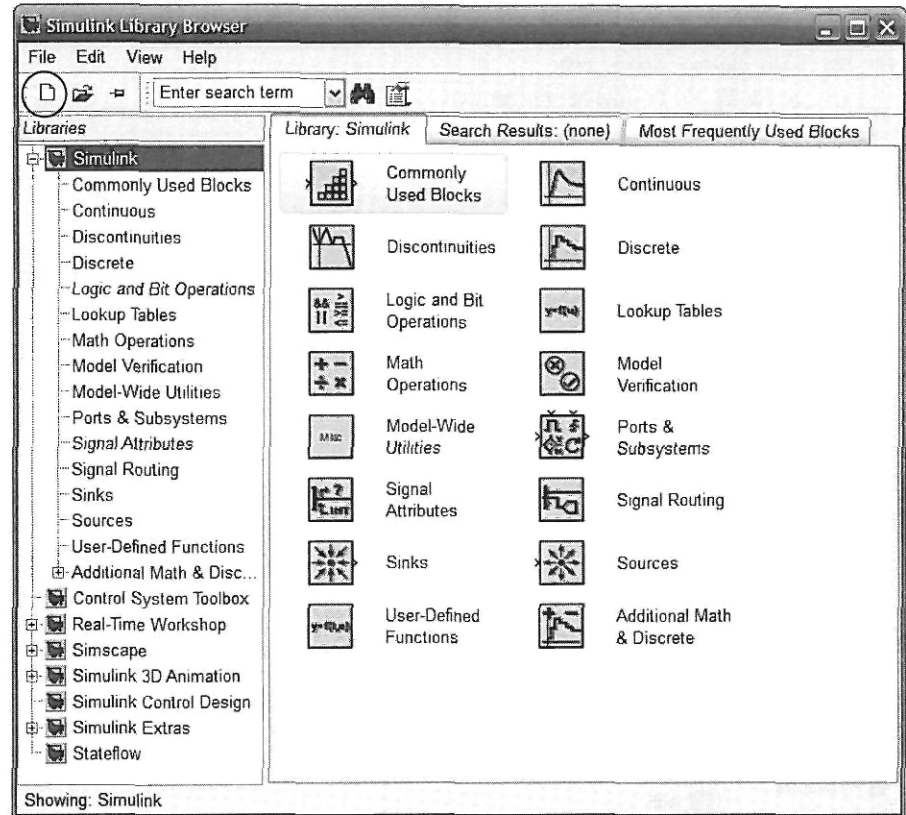
(a)

(b)

FIGURE C.2   a. Simulink Library Browser window showing the **Create a new model** button encircled **b.** resulting **untitled** model window

right-hand side of Figure C.2(a) also shows the underlying block libraries. To reveal a block library's underlying blocks, select the block library on the left-hand side or double-click the block library on the right-hand side. As an example, the **Continuous** library blocks under the **Simulink** major library are shown exposed in Figure C.3(a). Figures C.3(b) and C.3(c) show some of the **Sources** and **Sinks** library blocks, respectively.

Another approach to revealing the Simulink block library is to type *open_ system ('simulink.mdl')* in the **MATLAB Command Window**. The window shown in Figure C.4 is the result. Double-clicking any of the libraries in Figure C.4
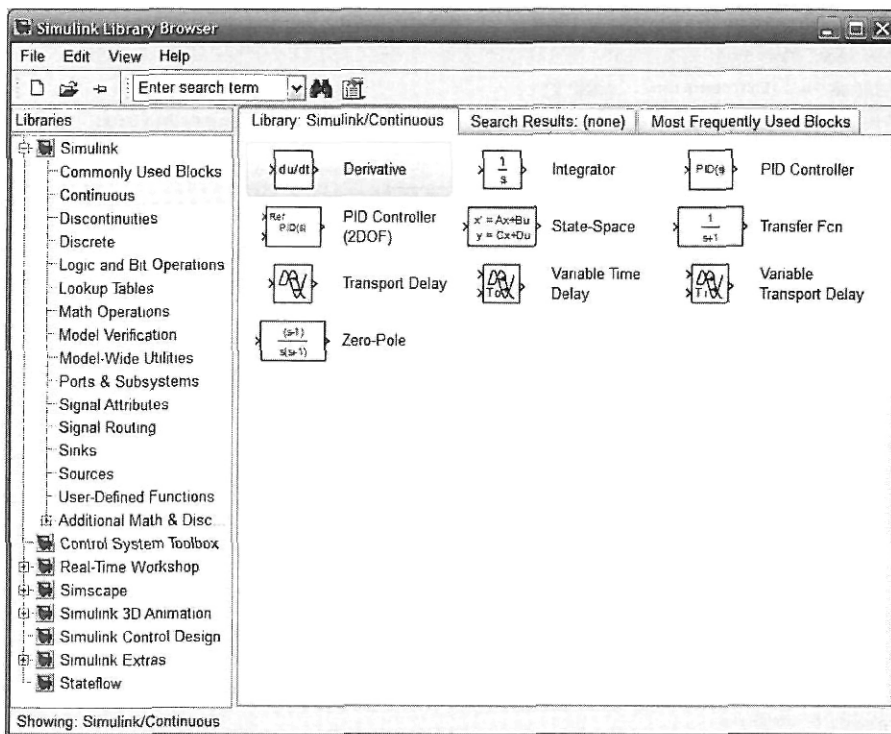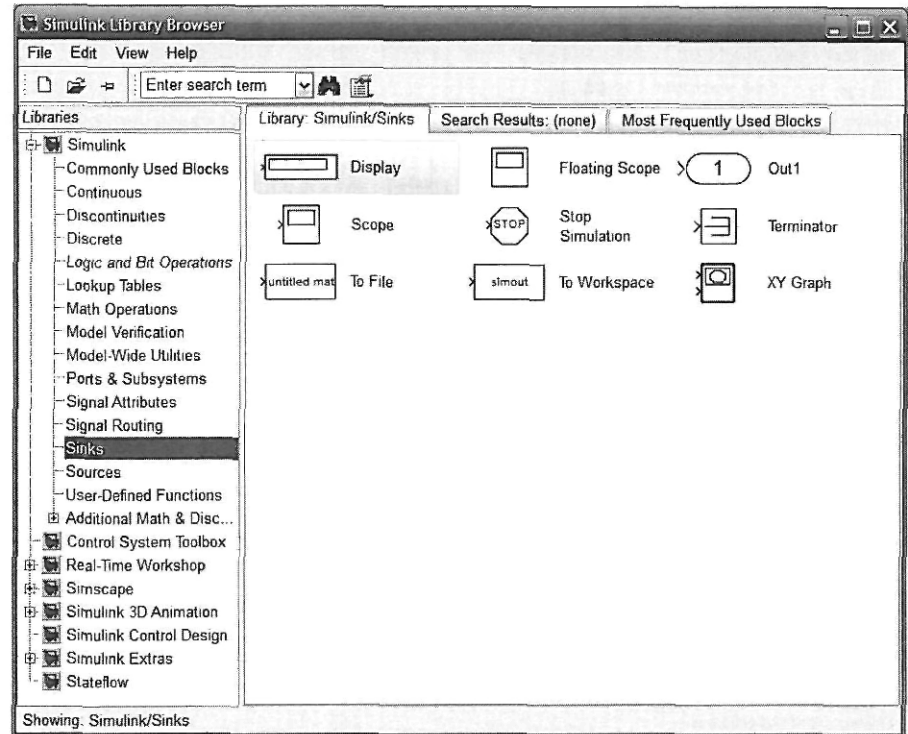
(a)
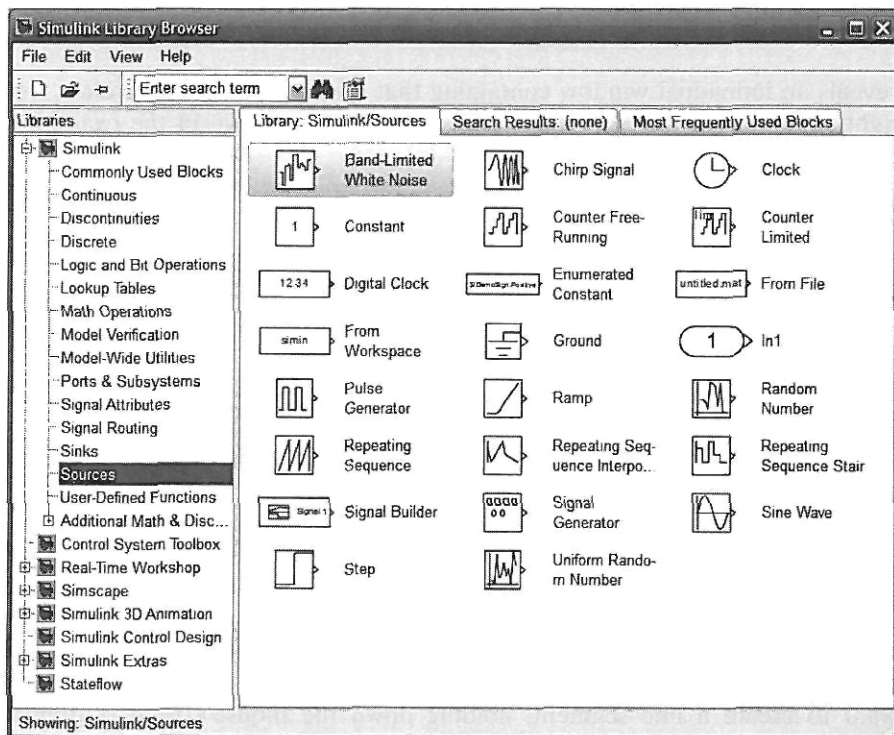
(b)

FIGURE C.3 Simulink block libraries: **a.** Continuous systems **b.** Sources (*figure continues*)

(c)

FIGURE C.3   (*Continued*)   c. Sinks

reveals an individual window containing that library's blocks, equivalent to the right-hand side of the **Simulink Library Browser** as shown in the examples of Figure C.3.

3. **Assemble and label subsystems** Drag required subsystems (blocks) to your model window from the browser, such as those shown in Figure C.3. Also, you may access the blocks by double-clicking the libraries shown in Figure C.4. You can position, resize, and rename the blocks. To position, drag with the mouse; to resize, click on the subsystem and drag the handles; to rename, click on the existing name, select the existing text, and type the new name. The text can also be repositioned to the top of the block by holding the mouse down and dragging the text.

4. **Interconnect subsystems and label signals** Position the pointer on the small arrow on the side of a subsystem, press the mouse button, and drag the resulting cross-hair pointer to the small arrow of the next subsystem. A line will be drawn between the two subsystems. Blocks may also be interconnected by single-clicking the first block followed by single-clicking the second block while holding down the control key. You can move line segments by positioning the pointer on the line, pressing the mouse button, and dragging the resulting four-arrow pointer. Branches to line segments can be drawn by positioning the pointer where you want to create a line segment, holding down the mouse's right button, and dragging the resulting cross hairs. A new line segment will form. Signals can be labeled by double-clicking the line and typing into the resulting box. Finally, labels can be placed anywhere by double-clicking and typing into the resulting box.
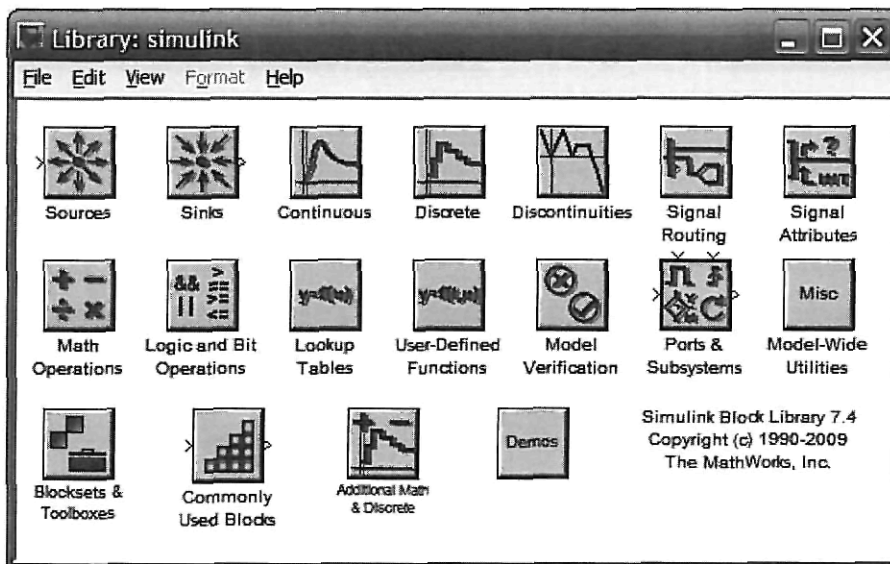
FIGURE C.4 Simulink Block Library window

5. **Choose parameters for the subsystems** Double-click a subsystem in your model window and type in the desired parameters. Some explanations are provided in the **Block Parameters** window. Press the Help button in the **Block Parameters** window for more details. The parameters can be read later without opening the block. Let your mouse's pointer rest on the block for a few seconds, and a screen tip will appear, identifying the block and listing its parameters. The information displayed in the screen tip first must be selected in the **Block Data Tips Options** in the model window's **View** menu. Explore other options by right-clicking on a block.

6. **Choose parameters for the simulation** Select **Configuration parameters** . . . under the **Simulation** menu in your model window to set additional parameters, such as simulation time. Press the **Help** button in the **Configuration parameters** window for more details.

7. **Start the simulation** Make your model window the active window. Double-click the **Scope** block (typically, the scope is used to view the simulation results) to display the **Scope** window. Select **Start** under the **Simulation** menu in your model window or click on the **Start simulation** icon on the toolbar of your model window as shown in Figure C.2(*b*). Clicking the **Stop simulation** icon will stop the simulation before completion.

8. **Interact with the plot** In the **Scope** window, using the toolbar buttons, you can zoom in and out, change axes ranges, save axis settings, and print the plot. Right-clicking on the **Scope** window brings up other choices.

9. **Save your model** Saving your model, by choosing **Save** under the **File** menu, creates a file with an .mdl extension, which is required.

(C.3 Examples

This section will present four examples of the use of Simulink to simulate linear, nonlinear, and digital systems. Examples will show the Simulink block diagrams as well as explain the settings of parameters for the blocks. Finally, the results of the simulations will be shown.

Example C.1

Simulation of Linear Systems

Our first example develops a simulation of three linear systems to compare their step responses. In particular, we solve Example 4.8 and reproduce the responses shown in Figure 4.24. Figure C.5 shows a Simulink block diagram formed by following Steps 1 through 5 in Section C.2 as follows:
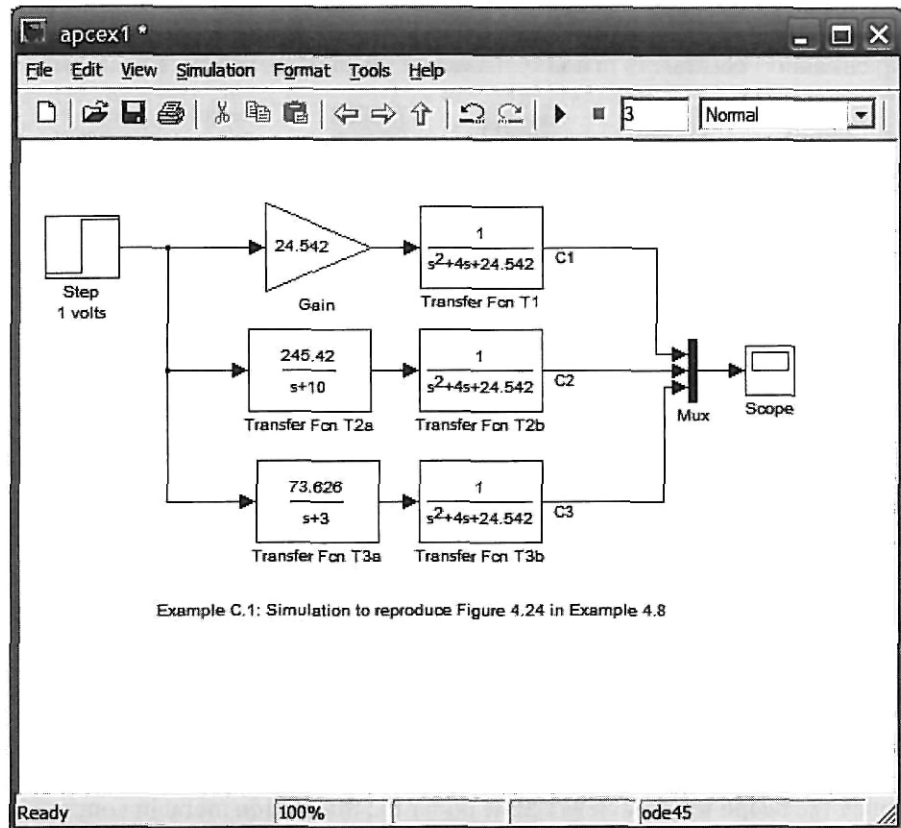

FIGURE C.5   Simulink block diagram for Example C.1

Access Simulink; select, assemble, and label subsystems   The source is a 1-volt step input, obtained by dragging the **Step** block from the **Simulink Library Browser** under **Sources** to your model window.

The first system, T1, consists of two blocks, **Gain** and **Transfer Fcn**. Gain is obtained by dragging the **Gain** block from the **Simulink Library Browser** under **Math Operations** to your model window. Transfer function, T1, is obtained by dragging the **Transfer Fcn** block from the **Simulink Library Browser** under **Continuous** to your model window. Systems T2 and T3 are created similarly.

The three output signals, C1, C2, and C3, are multiplexed for display into the single input of a scope. The Mux (multiplexer) is obtained by dragging the **Mux** block from the **Simulink Library Browser** under **Signal Routing** to your model window.

The sink is a scope, obtained by dragging the **Scope** block from the **Simulink Library Browser** under **Sinks** to your model window.

Alternatively, all blocks can be dragged from the **Library: simulink** window shown in Figure C.4. The **Mux** can be found under **Signal Routing** in the **Library: simulink** window.

The labels for the blocks can be changed to those shown in Figure C.5 by following Step 3 in Section C.2.

Interconnect subsystems and label signals Follow Step 4 to interconnect the subsystems and label the signals. You must set the mux's parameters before the wiring can be completed. See the next paragraph.

Choose parameters for the subsystems Let us now set the parameters of each block using Step 5. The **Block Parameters** window for each block is accessed by double-clicking the block on your model window. Figure C.6 shows the **Block Parameters** windows for the 1 volt step input, gain, transfer function 1, and mux. Set the parameters to the required values as shown.
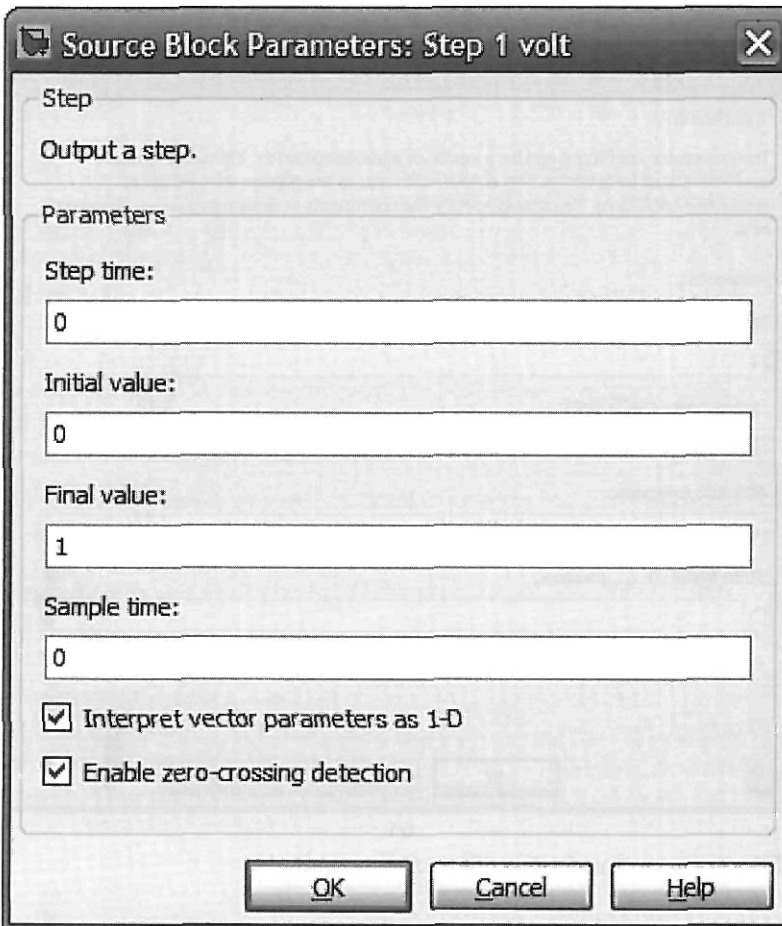
(a)

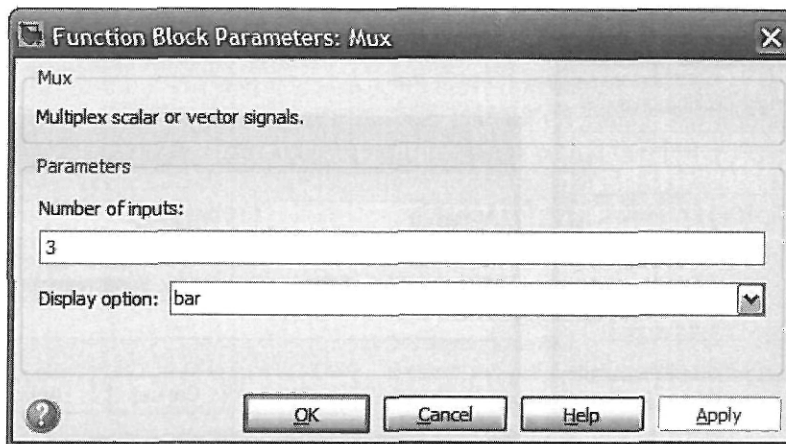FIGURE C.6 **Block parameters** windows for **a.** 1 volt step source; (*figure continues*)
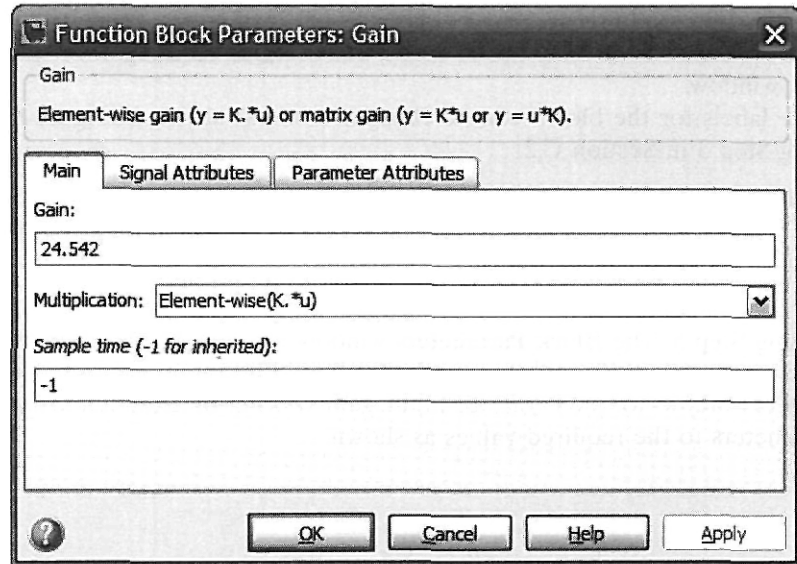
(b)
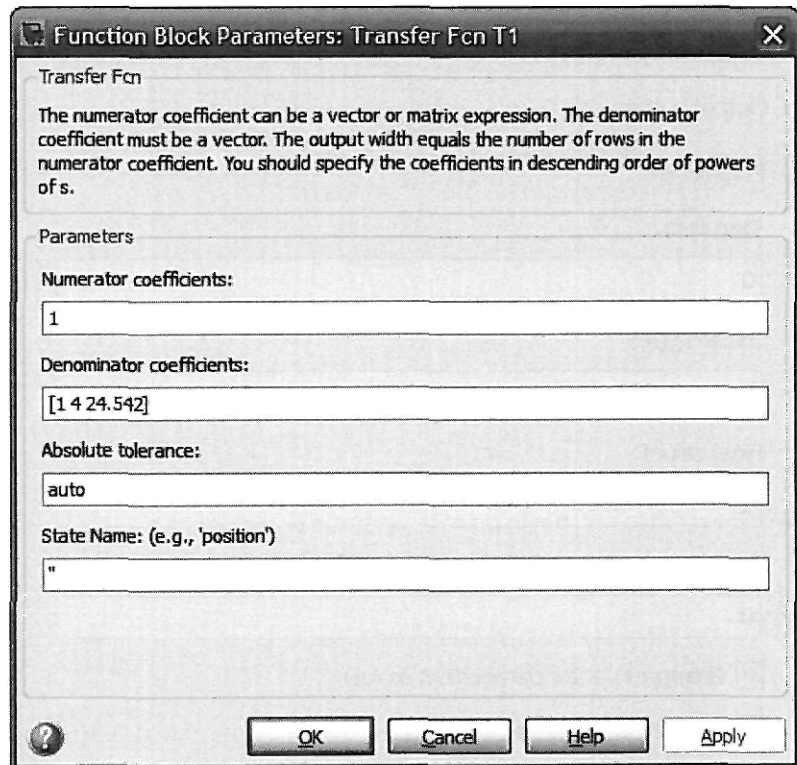
(c)

FIGURE C.6   **b**. gain; **c**. transfer function 1; (*figure continues*)

(d)

FIGURE C.6 (*Continued*) **d**. mux

The scope requires further explanation. Double-clicking the **Scope** block in your model window accesses the scope's display, Figure C.7(*a*).

Clicking the **Parameters** icon on the **Scope** window toolbar, shown in Figure C.7(*a*), accesses the **'Scope' parameters** window as shown in Figure C.7(*b*). The **'Scope' parameters** window contains two tabs, **General** and **Data history**, as shown in Figure C.7(*b*) and (*c*), respectively.
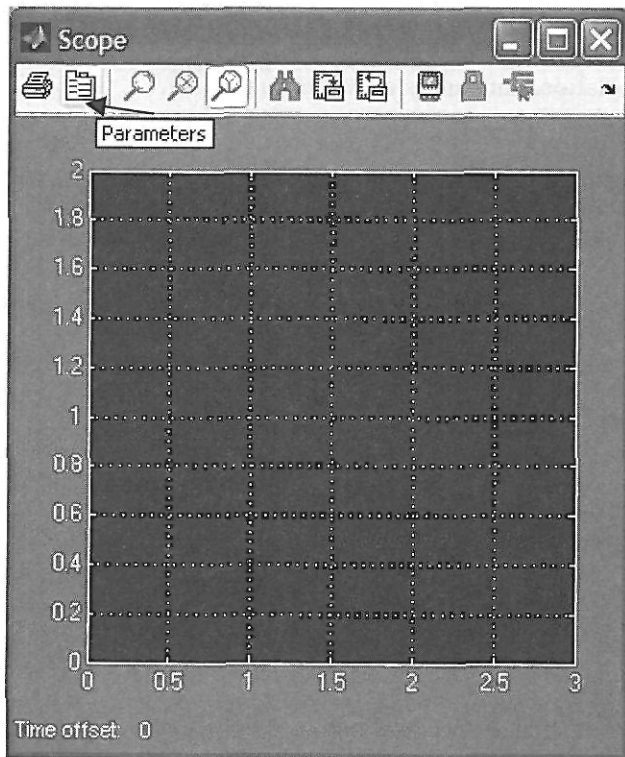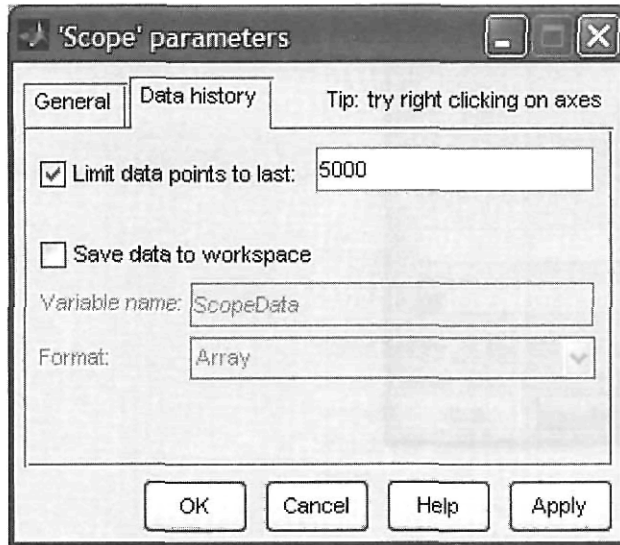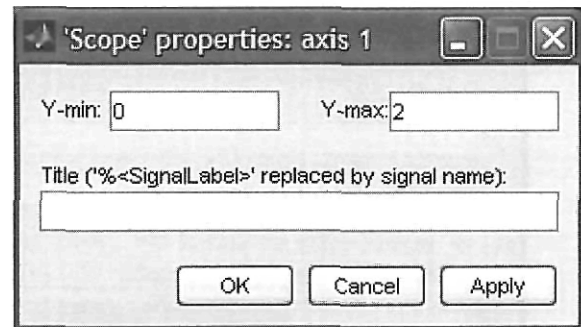

(a)
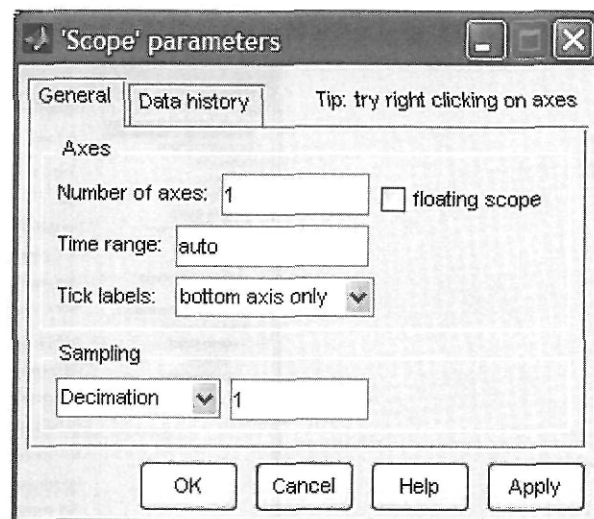
(b)

FIGURE C.7 Windows for the scope: **a. Scope; b. 'Scope' parameters, General** tab; (*figure continues*)

(c)

(d)

FIGURE C.7 (*Continued*) **c. 'Scope' parameters, Data history** tab; **d. 'Scope' properties: axis 1**

Finally, right-clicking in the plotting area in the Scope window and selecting **Axis properties** . . . reveals the **'Scope' properties: axis 1** window, Figure C.7(*d*). We now can set the display parameters, such as amplitude range.

Choose parameters for the simulation   Follow Step 6 to set simulation parameters. Figure C.8 shows the resulting **Configuration Parameters** window. Among other parameters, the simulation start and stop times can be set.
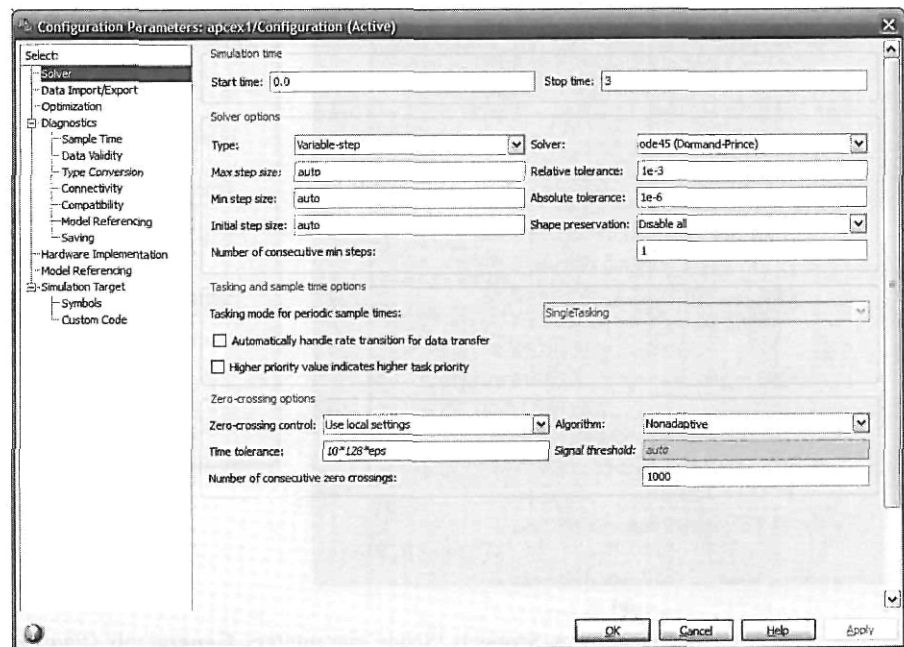

FIGURE C.8   Configuration Parameters window for **Solver tab**

Start the simulation   Now run the simulation by following Step 7. Figure C.9 shows the result in the **Scope** window. Plots are color coded in the order in which they appear at the mux input as follows: yellow, magenta, cyan, red, green, and dark blue. If the mux has more inputs, the colors recycle.

Interact with the plot   The toolbar of the **Scope** window shown in Figure C.9 has several buttons that can be used to interact with the plot. Let us summarize the function and operation of each, starting with the left-most button:

Button 1 executes a plot print.

Button 2 has already been explained and is used to set scope parameters.

Button 3 permits zooming into the plot in both the x and y directions. Press the button and drag a rectangle over the portion of the curve you want to expand.

Button 4 allows zooming in the x direction only. Drag a horizontal line over the plot covering the extent of x you want to expand.

Button 5 allows zooming in the y direction only. Drag a vertical line over the plot covering the range of y you want to expand.

Button 6 autoscales axis for use after zooming.

Button 7 saves current axis settings.

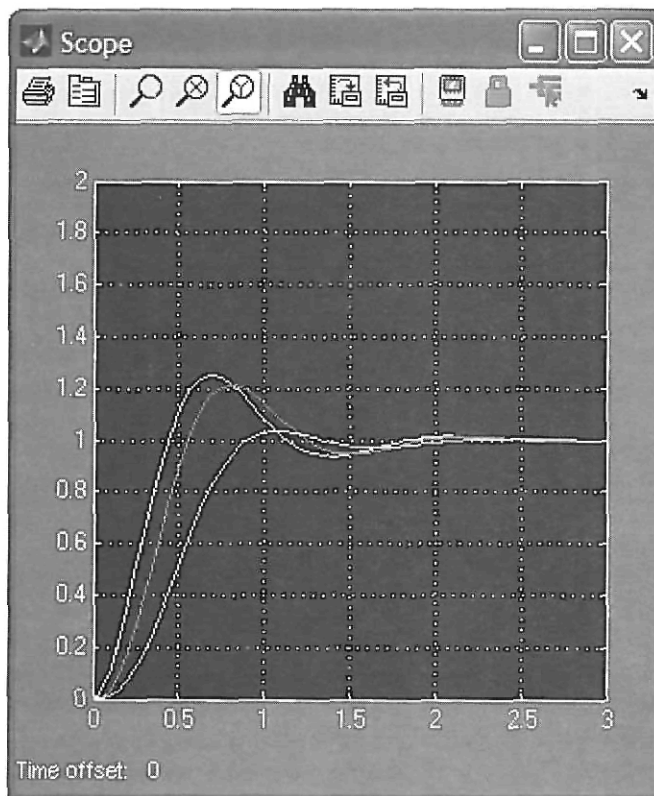**Button 8** restores saved axis settings.

FIGURE C.9   Scope window after Example C.1 simulation stops

Button 9 toggles floating scope. It must be turned off to use zooming. See documentation for use of floating scopes.

Button 10 *toggles lock for current axis selection.*

Button 11 allows selection of signals to view when using floating scope.

Example C.2

Effect of Amplifier Saturation on Motor's Load Angular Velocity

This example, which generated Figure 4.29 in the text, shows the use of Simulink to simulate the effect of saturation nonlinearity on an open-loop system. Figure C.10 shows a Simulink block diagram formed by following Steps 1 through 5 in Section C.2 above.

Saturation nonlinearity is an additional block that we have not used before. Saturation is obtained by dragging to your model window the **Saturation** block in the **Simulink Library Browser** window under **Discontinuities** as shown in Figure C.11(*a*) and setting its parameters to those shown in Figure C.11(*b*).

Now run the simulation by making your model window active and selecting **Start** under the **Simulation** menu of your model window or clicking on the **Start simulation** button on your model window toolbar. Figure C.12 shows the result in the **Scope** window.
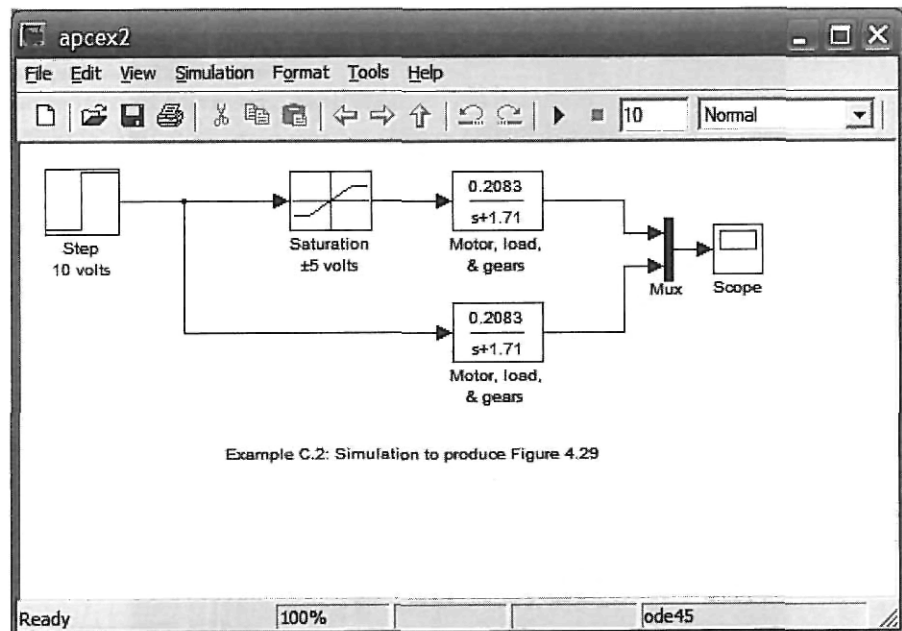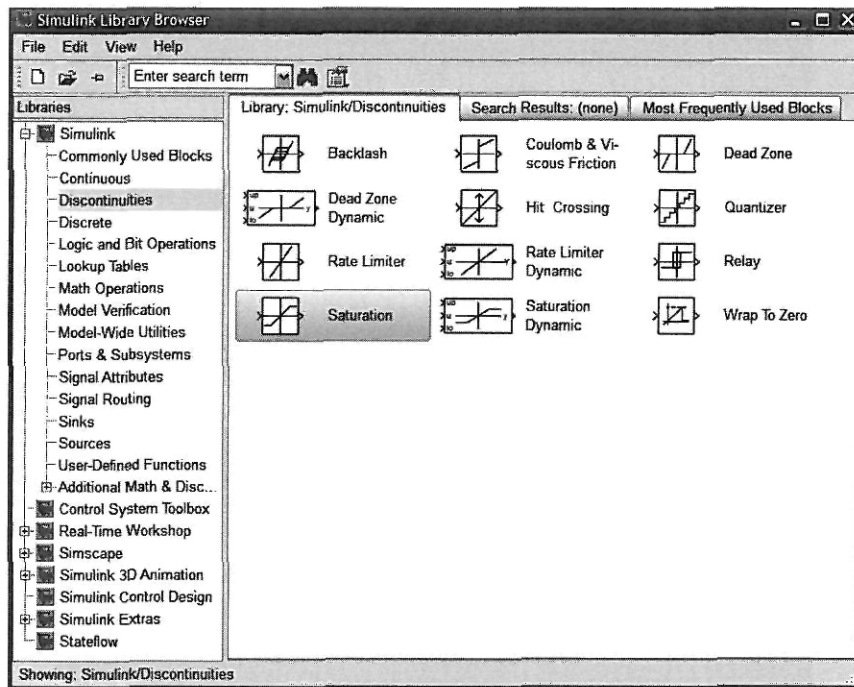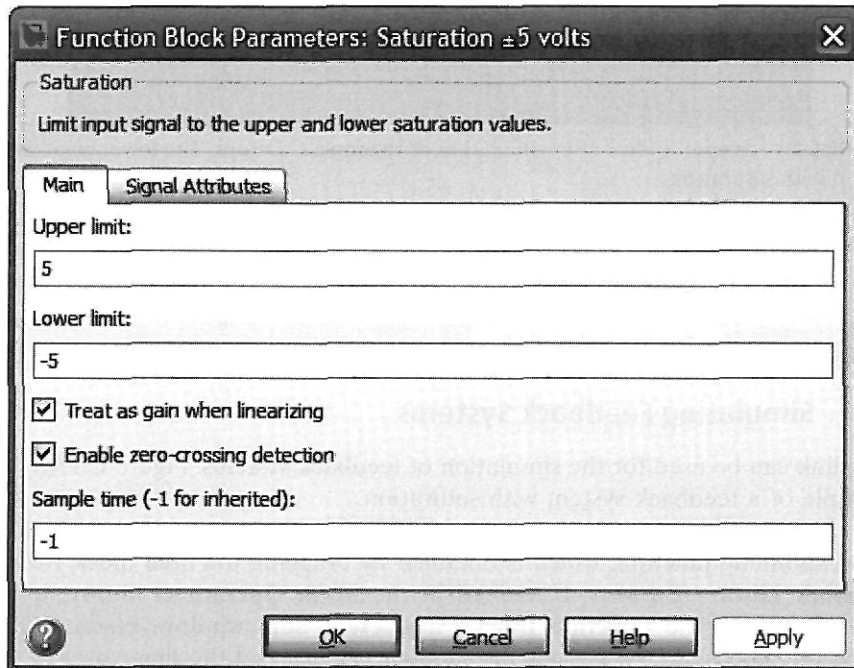

FIGURE C.10   Simulink block diagram for Example C.2

(a)

(b)

FIGURE C.11 **a.** Simulink library for nonlinearities; **b.** parameter settings for saturation
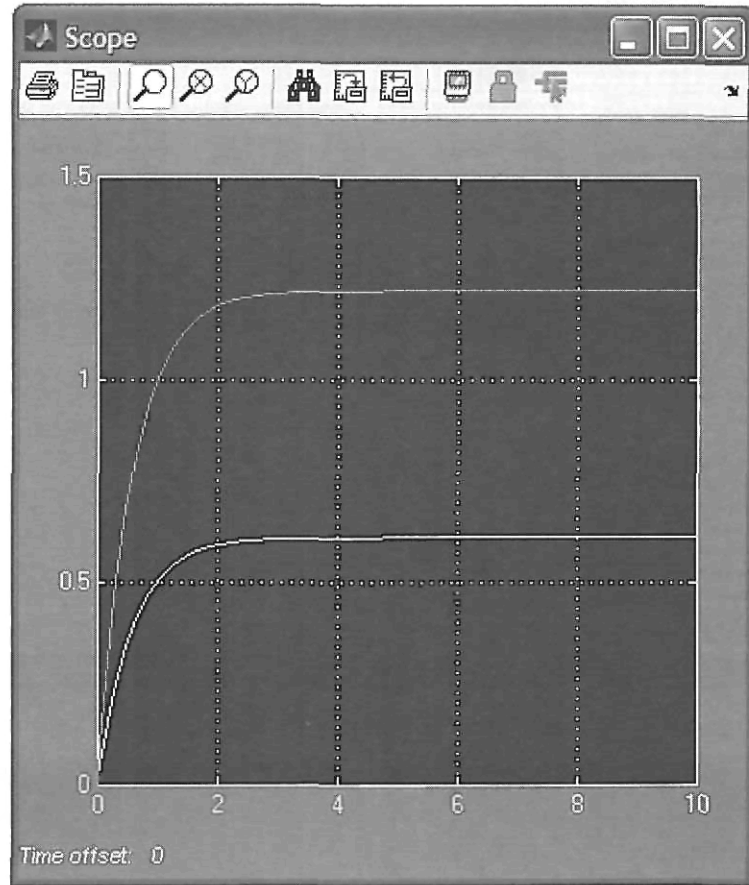
FIGURE C.12   Scope window after simulation of Example C.2 stops. The lower curve is the output with saturation
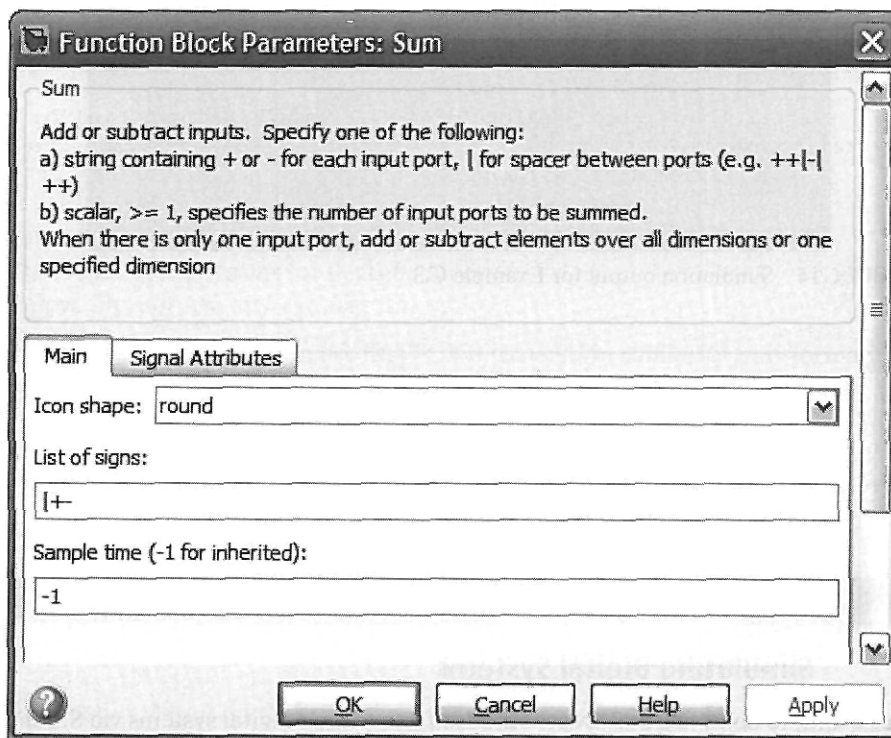
Example C.3

Simulating Feedback Systems

Simulink can be used for the simulation of feedback systems. Figure C.13(*a*) is an example of a feedback system with saturation.

In this example, we have added a feedback path (see Step 4 in Section C.2) and a summing junction, which is obtained by dragging the **Sum** block from the **Simulink Library Browser**, contained in the **Math Operations** library, to your model window. The **Function Block Parameters: Sum** window, Figure C.13(*b*), shows the parameter settings for the summer. You can set the shape as well as set the plus and minus inputs. In the list of signs, the "|"symbol signifies a space. We place it at the beginning to start the signs at "nine o'clock," conforming to our standard symbol, rather than at "12 o'clock." The result of the simulation is shown in Figure C.14.
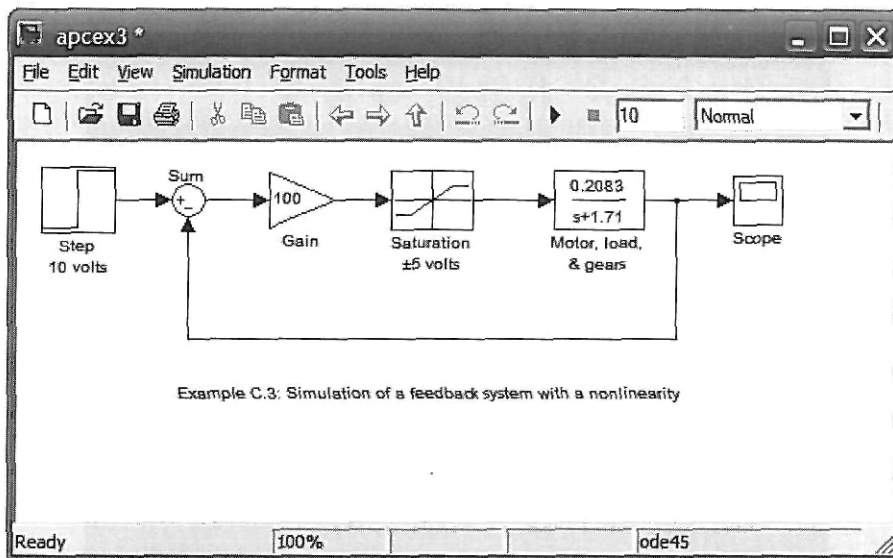
(a)

(b)

FIGURE C.13   a. Simulation block diagram for a feedback system with saturation; **b.** block parameter window for the summer
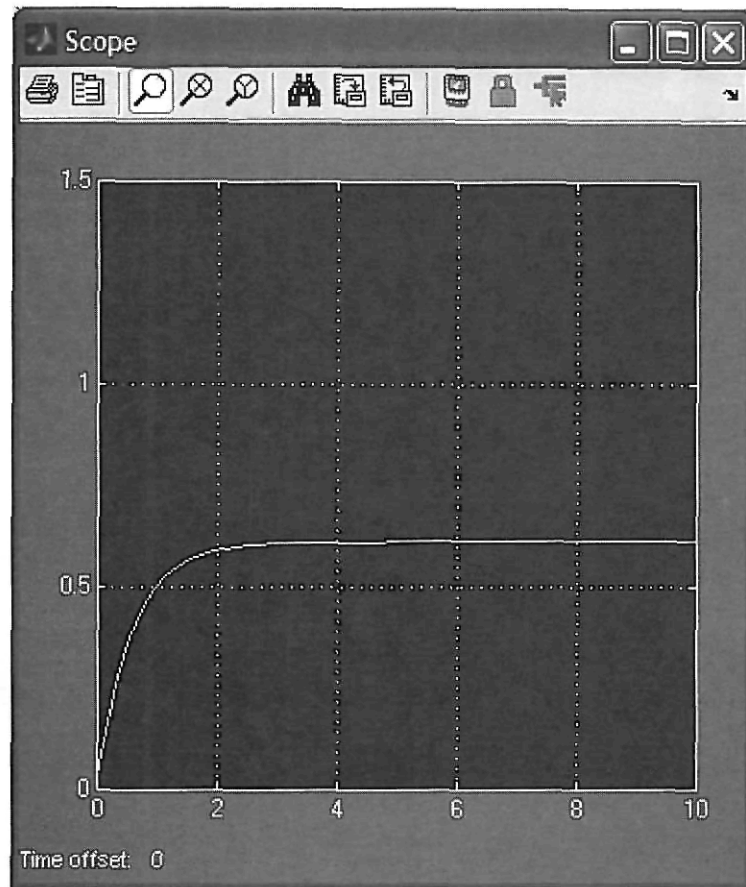
FIGURE C.14   Simulation output for Example C.3

Example C.4

Simulating Digital Systems

This example demonstrates two methods of generating digital systems via Simulink for the purpose of simulation, as shown in Figure C.15.

The first approach uses a linear transfer function cascaded with a **Zero-Order Hold** block obtained from the **Simulink Library Browser** under the **Discrete** block library, shown on the right-hand side of Figure C.16. The second method uses a discrete transfer function also obtained from the **Simulink Library Browser** under the **Discrete** block library. The remainder of the block diagram was obtained by methods previously described.
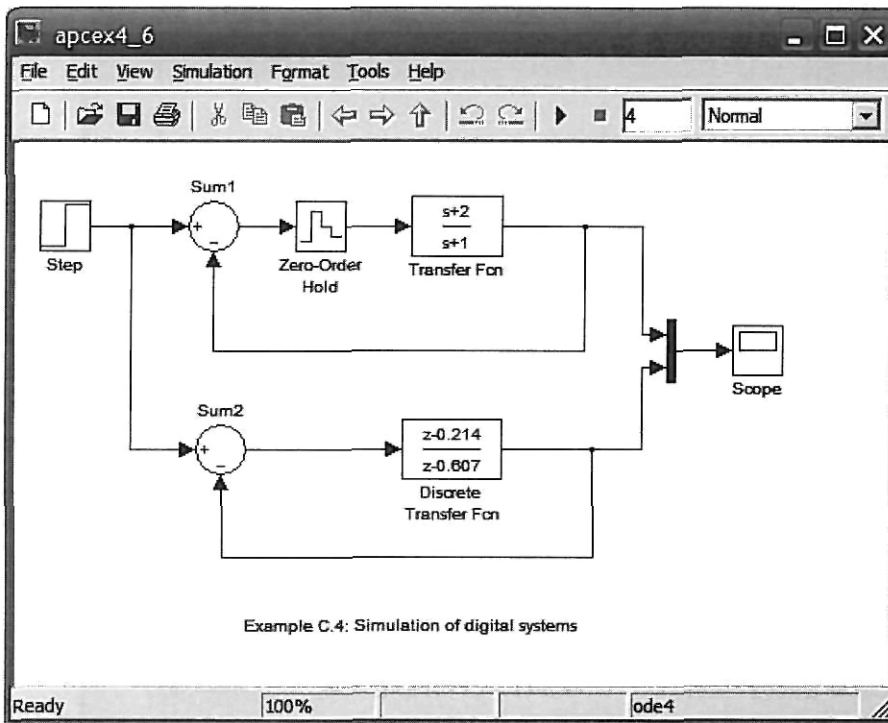
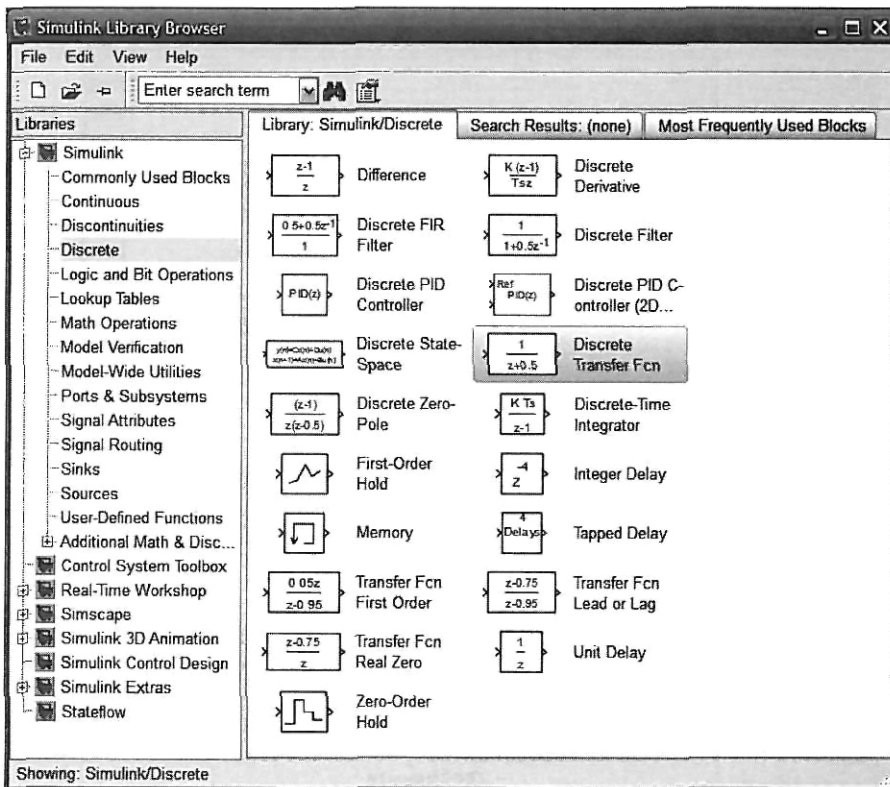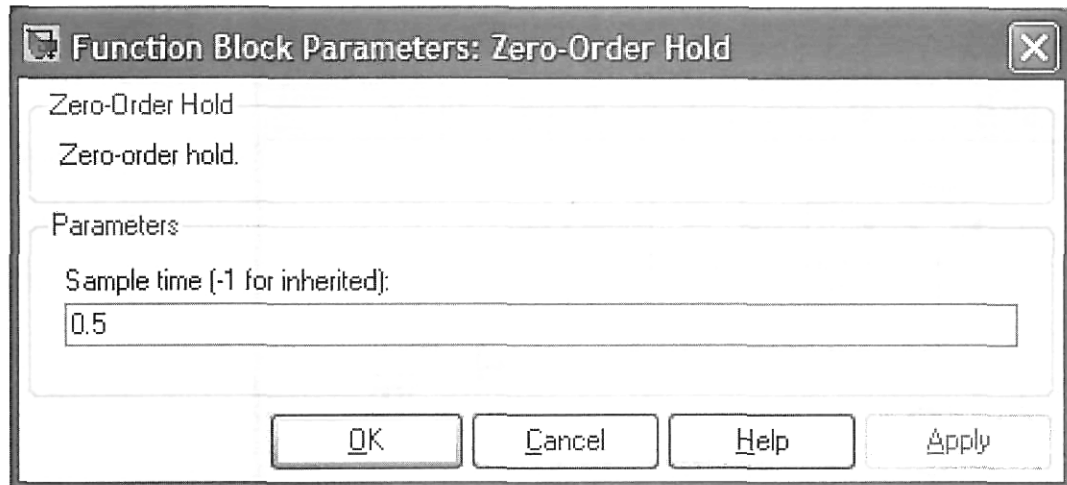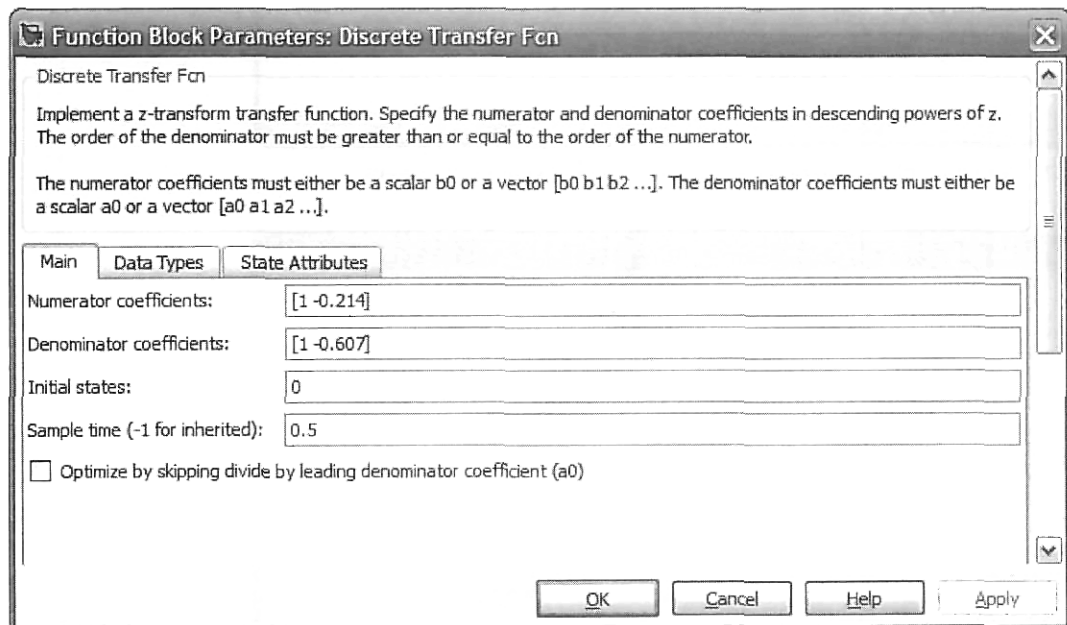**FIGURE C.15**   Simulink block diagram for simulating digital systems two ways

FIGURE C.16   Simulink library of discrete blocks

(a)

(b)

FIGURE C.17   Function Block parameter windows for: **a. Zero-Order Hold** block; **b. Discrete Transfer Fcn** block

The block parameters for the **Zero-Order Hold** and **Discrete Transfer Fcn** blocks are set as shown in Figures C.17(a) and (b), respectively.

Select **Configuration parameters** . . . under the **Simulation** menu in your model window and set the simulation stop time to 4 seconds, the type **to fixed-step**, and the solver to **ode4 (Runge-Kutta)**. The result of the simulation is shown in Figure C.18.
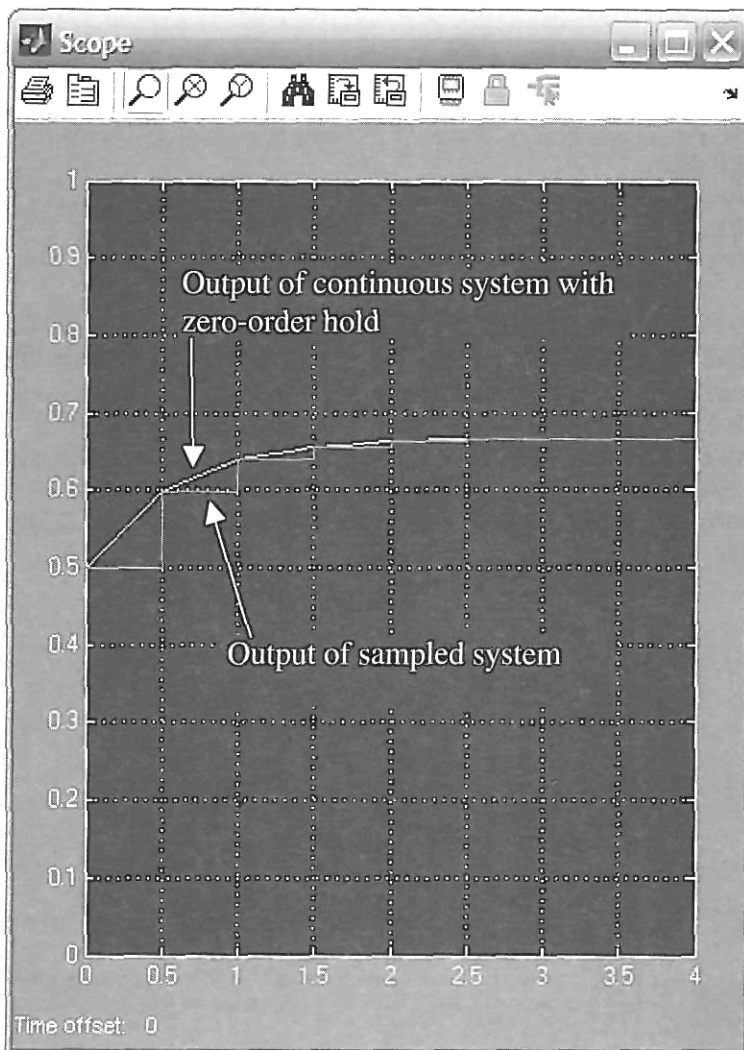
FIGURE C.18   Outputs of the digital systems

(Summary)

This appendix explained Simulink, its advantages, and how to use it. Examples were taken from Chapters 4, 5, and 13 and demonstrated the use of Simulink for simulating linear, nonlinear, and digital systems.

The objective of this appendix was to familiarize you with the subject and get you started using Simulink. There are many blocks, parameters, and preferences that could not be covered in this short appendix. You are encouraged to explore and expand your use of Simulinkby using the on-screen help that was explained earlier. The references in the Bibliography of this appendix also provide an opportunity to learn more about Simulink.

❰ Bibliography ❱

The MathWorks. *Control System Toolbox* ^{TM} *8 Getting Started Guide.* The MathWorks. Natick, MA. 2000–2009.

The MathWorks. *Control System Toolbox* ^{TM} *8 User's Guide.* The MathWorks. Natick, MA. 2001–2009.

The MathWorks. *MATLAB® 7 Getting Started Guide.* The MathWorks. Natick, MA. 1984–2009.

The MathWorks. *MATLAB® 7 Graphics.* The MathWorks. Natick, MA. 1984–2009.

The MathWorks. *MATLAB® 7 Mathematics.* The MathWorks. Natick, MA. 1984–2009.

The MathWorks. *MATLAB® 7 Programming Fundamentals.* The MathWorks. Natick, MA. 1984–2009.

The MathWorks. *Simulink® 7 Getting Started Guide.* The MathWorks. Natick, MA. 1990–2009.

The MathWorks. *Simulink® 7 User's Guide.* The MathWorks. Natick, MA. 1990–2009.