APPENDIX

B Simulink Basics

Simulink provides an efficient way for MATLAB users to model, simulate, and analyze feedback control systems [1–3]. Since Simulink is an interactive tool utilizing graphical interfaces effectively, the best way to learn about it is to jump right in and use it. In this appendix, you will walk through a sequence of steps to construct and simulate a simple system.

We attempt here to provide basic information about Simulink that is as loosely tied to specific releases of the software as possible. At the time of this printing, the latest version is Simulink 5.0. As different versions of Simulink are released, previous introductions to Simulink Basics will be posted on the *MCS* website—check there if you are having compatibility problems with the Simulink models in this book.

There are two ways to initiate a Simulink session:

- Click on the Simulink icon on the MATLAB toolbar.
- Enter the simulink command at the command prompt.

The two methods of starting Simulink are illustrated in Figure B.1.

When Simulink is activated, a window resembling Figure B.2 should appear on the desktop. Figure B.2a depicts the Library Browser, which shows a tree-structured view of the Simulink libraries installed on the computer. This is the screen that first

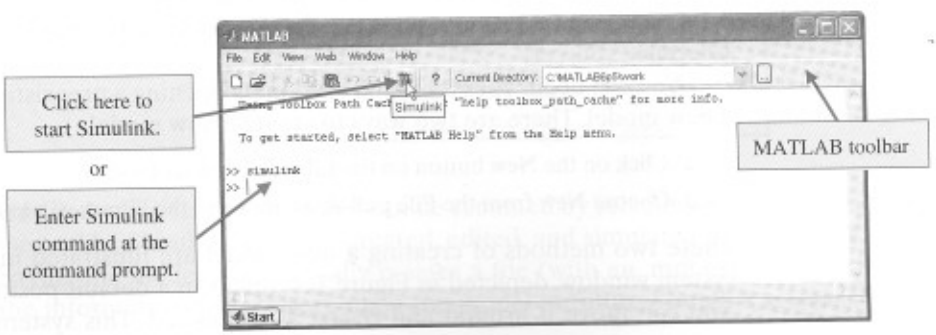

FIGURE B.1
Initiating a Simulink session.

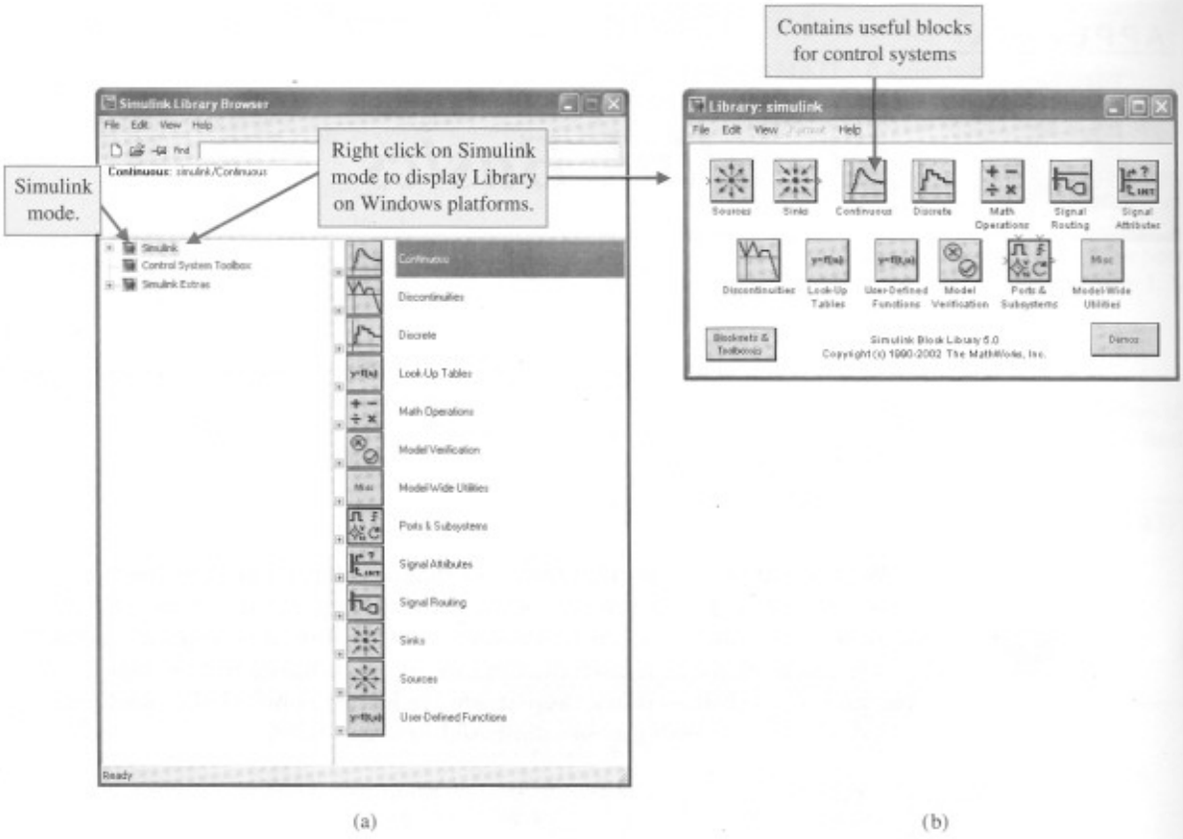

FIGURE B.2 First window presented by Simulink. (a) Library Browser. (b) Library window.

appears. The Simulink library window depicted in Figure B.2b can be displayed by right-clicking the Simulink node in the Library Browser window. In the case shown in Figure B.2a, the Simulink libraries associated with the *Control System Toolbox* appear since that toolbox is installed.

As an exercise to help learn about Simulink, you can follow the steps presented in the subsequent discussion to construct a simple system and to initiate a simulation. You should not rely solely on this material to get to know Simulink, as the printed literature and on-line help that comes with the software are also excellent reference sources.

You can begin your Simulink session by opening a preexisting model or creating a new model. There are two ways to create a new model:

- Click on the **New** button on the Library Browser toolbar.
- Choose **New** from the File pull-down menu in the library window and select **Model**.

These two methods of creating a new model are illustrated in Figure B.3. A new system window, depicted in Figure B.4, opens in a default position on the desktop. You can move it around and re-size it as desired. This system window is named

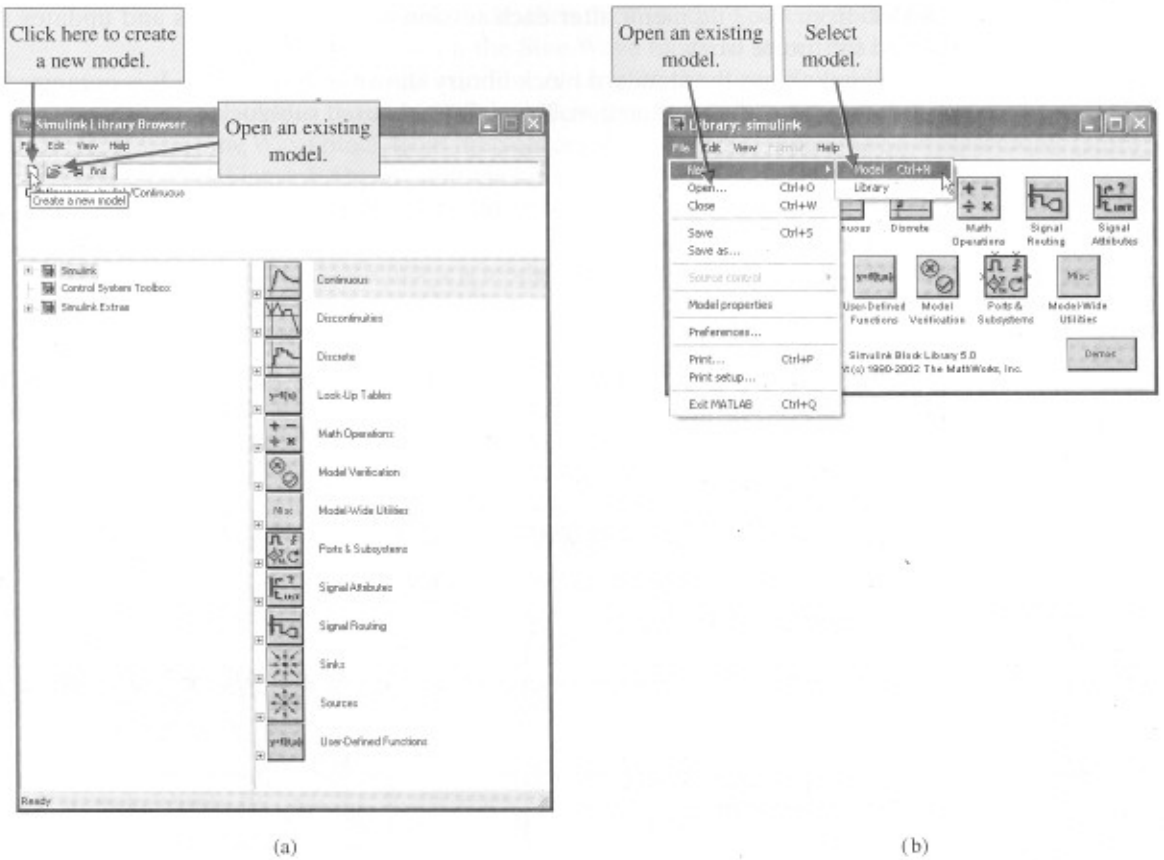

FIGURE B.3 Creating a new model. (a) From the Windows Library Browser. (b) From the library window.

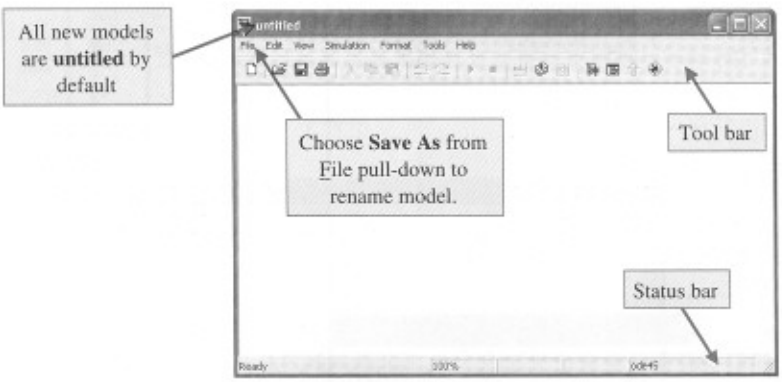

FIGURE B.4
The new model window.

Untitled when first created and can be renamed by selecting **Save As** from the File menu. The Simulink model is created, edited, and simulated in the model window. Saving the model automatically creates a file (with an .mdl extension) containing the information necessary to open the model in future Simulink sessions. Choose

SAVE from the File menu after each session so that all changes and updates are saved for future work.

Simulink has the standard block library shown in Figure B.2b. It is organized in subblocks according to function. The commonly used subblocks are

- Sources
- Sinks
- Discrete
- Continuous
- Math Operations

To create a model, you will drag and drop blocks from the standard block library into the model. In the simple model developed here, you will need to locate and copy a Sine Wave block (from Sources) and a Scope (from Sinks).

You can access the Sine Wave block from either the Library Browser or from the Sources library window. Expanding the Library Browser, as illustrated in Figure B.5a, provides access to the Sine Wave block. To expand the Library Browser, first

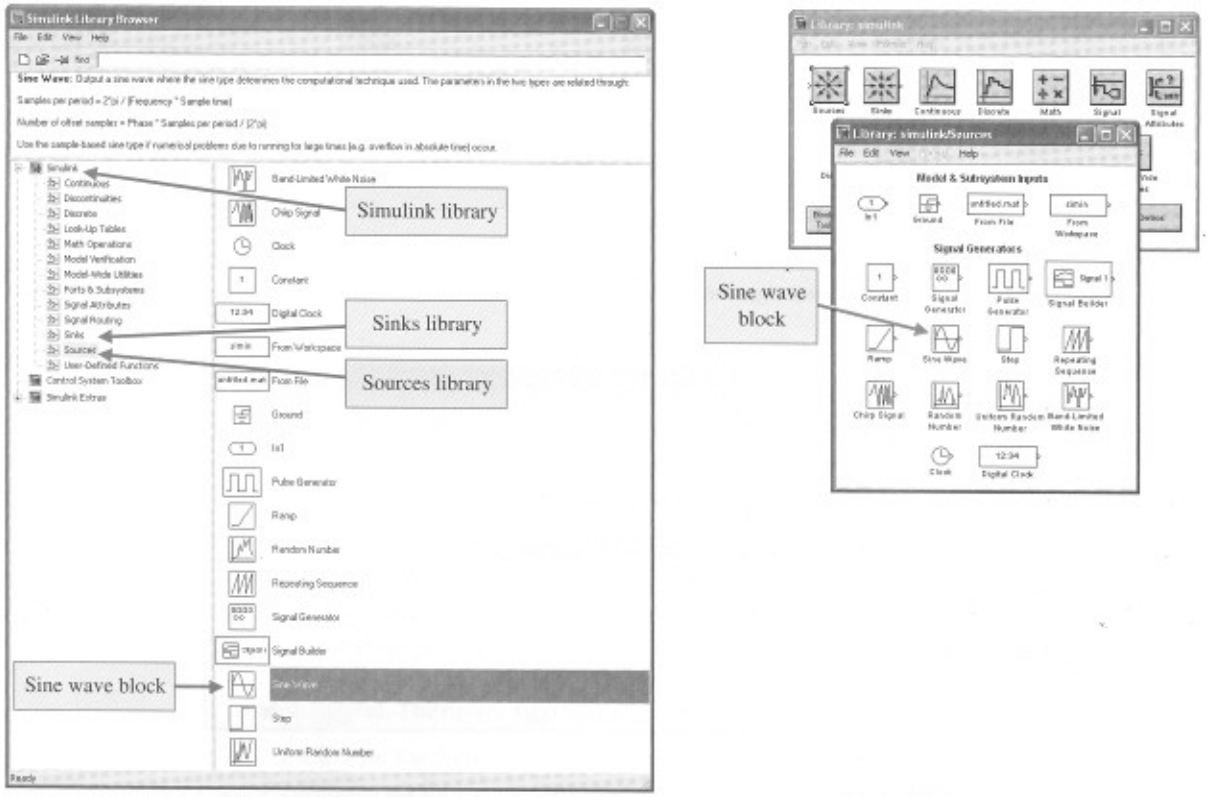

FIGURE B.5 Locating Sources. (a) The Library Browser (Windows). (b) The Sources block window (UNIX and Windows).

click on the Simulink node, then click on the Sources node to display the Sources library blocks. Click on the Sine Wave block to select it, and then drag-and-drop it into the new model window. A copy of the Sine Wave block will be made at the location in the new model window that you dropped the icon. Likewise, to copy the Sine Wave block from the Sources library window, open the Sources window (shown in Figure B.5b) by right-clicking on the Sources icon in the Simulink library window and selecting the **Open the 'Sources' Library** button. (see Figure B.2).

Once the Sources window is open, you can select the Sine Wave block and drag-and-drop it into the new model window. At this point in the model development, the model should resemble Figure B.6a.

You need to have a way to observe the sine wave signal generated by the signal source. The Sinks library window provides a collection of display instruments for observing the signal, including Scope, XY Graph, Display, To Workspace, and To File, as shown in Figure B.7. You can use the To Workspace and To File to write data to the workspace or to a file, respectively, so that the data can be made available in the workspace after the Simulink simulation is finished. In your model, you will use the Scope for displaying the sine wave signal.

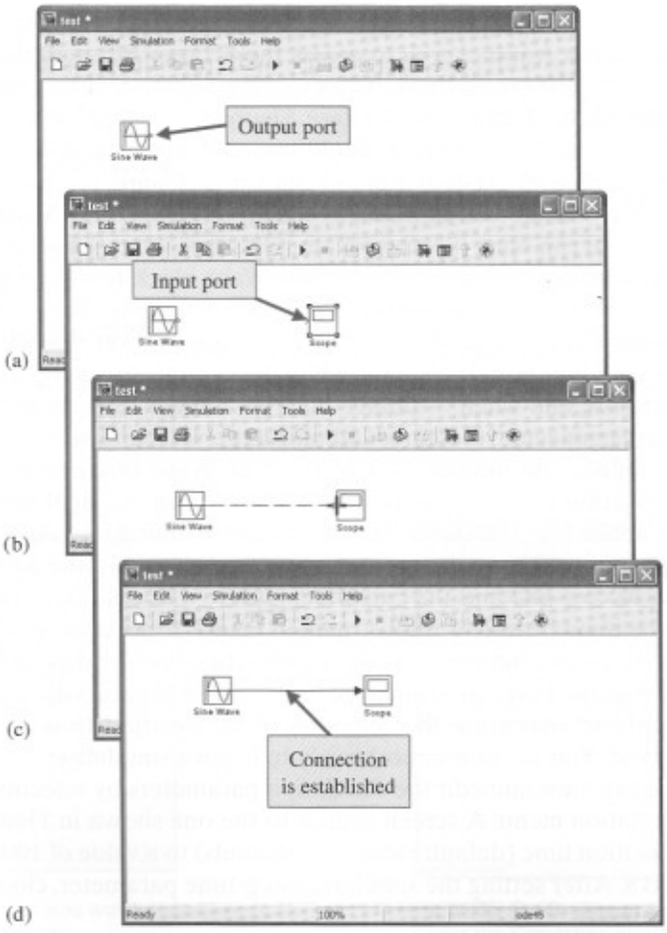

FIGURE B.6
A simple system with the Sine Wave signal source and a Scope for display.

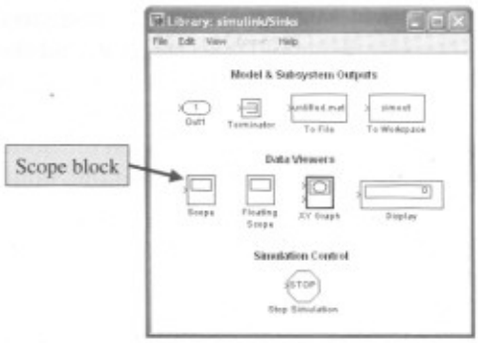

FIGURE B.7
The Sinks block window.

You can copy the Scope block from either the Library Browser or from the Sinks library window. As with the Sources block library, expanding the Library Browser provides access to the Scope block. To expand the Library Browser, first click on the Simulink node, then click on the Sinks node to display the Sinks library blocks. Click on the Scope block to select it and then drag-and-drop it into the new model window. Another way to access the Scope is through the Sinks library window by right-clicking on the Sinks icon in the Simulink library window and selecting **Open the 'sinks' Library** button (see Figure B.2). After copying the Scope to the new model window, the model should resemble Figure B.6b.

Notice that the system window shown in Figure B.6 has been renamed **test** (formerly it was labeled **untitled**, as shown in Figure B.4). Make sure to save your model so you can open it and work on it in subsequent Simulink sessions.

The next task is to connect the Sine Wave block to the Scope block. The angle bracket (>) that appears on the right side of the Sine Wave source is the **output port**. Similarly, the angle bracket (>) that appears on the left side of the Scope block is the **input port**. In general, when the > angle bracket points into a block, it is an input port, and when > points out from a block, it is an output port. When the blocks are connected, the input and output ports disappear.

To connect the output port of the Sine Wave block to the input port of the Scope, position the cursor over the output port on the right side of the Sine Wave block—notice that the cursor changes to a crosshair. Hold down the mouse button and move the cursor to the input port on the left side of the Scope. The connecting line is a dashed line and the cursor changes to a double crosshair as you approach the Scope input port. This is illustrated in Figure B.6c. After reaching the other port, release the mouse button to create a connecting line (or data link) between the two ports. When the ports are connected, the > angle brackets disappear and a line with an arrowhead indicating the direction of information flow appears, as shown in Figure B.6d. You are now almost ready to begin a simulation.

You can view and edit the simulation parameters by selecting **Parameters** from the Simulation menu. A screen similar to the one shown in Figure B.8 appears. Set the simulation time (default value is 10 seconds) to a value of 1000.0, as illustrated in Figure B.8. After setting the simulation stop time parameter, close the dialog box by clicking on **OK**.

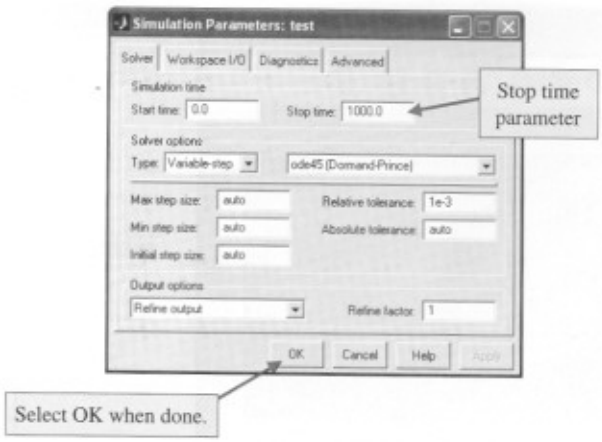

FIGURE B.8 Initializing the simulation parameters.

You can also change the parameters of the sine wave (such as frequency) by double-clicking on the Sine Wave icon in the model window and modifying the desired parameters. This is illustrated in Figure B.9. In this case, the frequency of the sine wave is set to 0.05 rad/sec.

Before starting the simulation, open the Scope by double-clicking on the Scope icon in the model window. To start the simulation, select Start from the Simulation menu, as shown in Figure B.10. The simulation begins and the Scope produces the

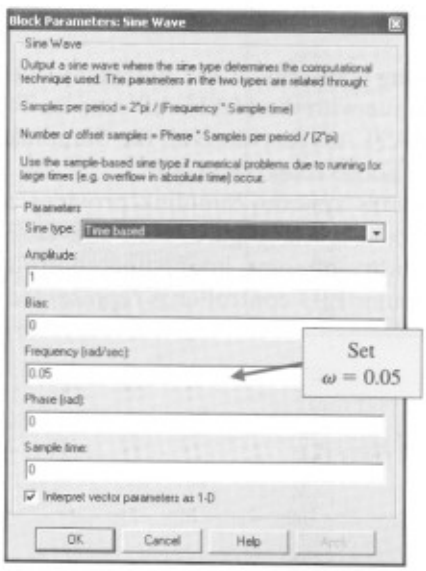

FIGURE B.9 Setting the sine wave frequency to $\omega = 0.05$ rad/sec.

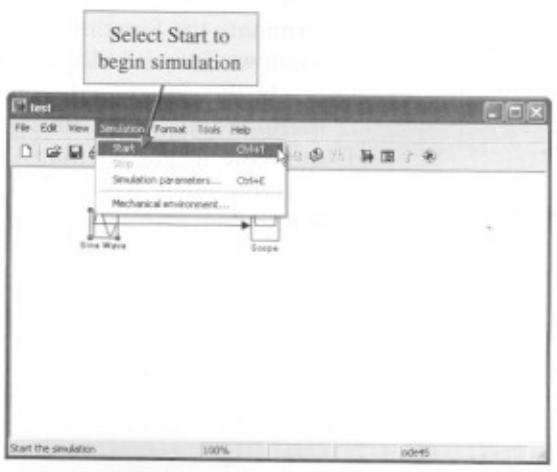

FIGURE B.10 Starting a simulation by choosing Start under the Simulation menu.

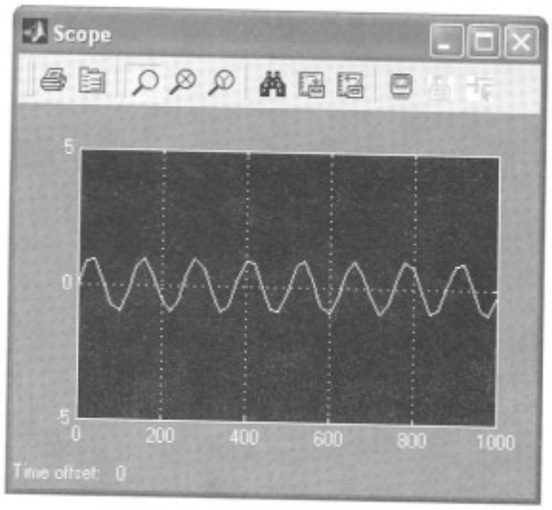

FIGURE B.11
The simulation session with the simple simulation system consisting of a Sine Wave signal source and a Scope for recording the signal. The frequency of the sine wave is $\omega = 0.05$ rad/sec.

graph shown in Figure B.11. The graph is updated dynamically as the data is generated by the Sine Wave source. The plot shown in Figure B.11 is the final plot after the simulation stop time has been reached.

You have now finished constructing the simple model and made a single simulation. Remember to choose **Save** from the File menu before quitting the Simulink session so that all changes and updates to the simulation model are saved for future work.

You can quit the Simulink session by choosing **Close** under the File menu and return to a MATLAB command prompt to continue with the MATLAB session. To terminate the Simulink session, but not leave MATLAB, just close all the Simulink windows. To quit both MATLAB and Simulink, choose **Exit MATLAB**.

In the design and analysis of feedback controls systems, Simulink provides a useful set of transfer function and state-space model blocks. Figures B.12 and B.13 show the location within the Simulink block library of some interesting controls related models. Notice, for example, that the popular PID controller is represented in **Simulink Extras**.

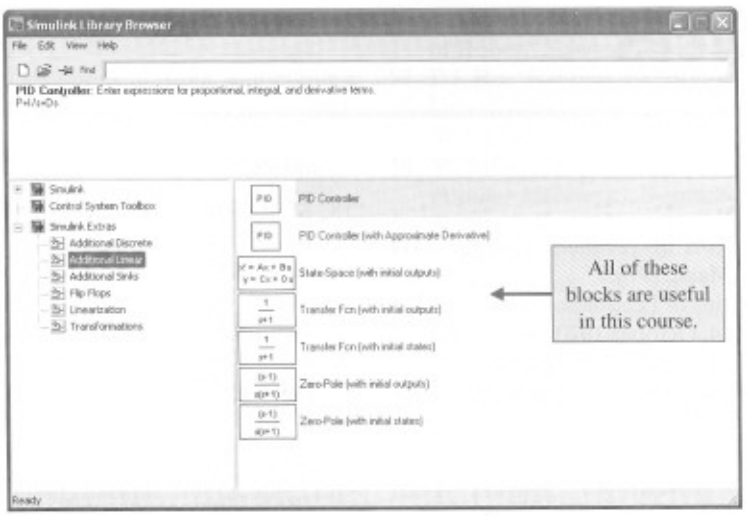

FIGURE B.12
Interesting controls related blocks in the Simulink **Extras** library.

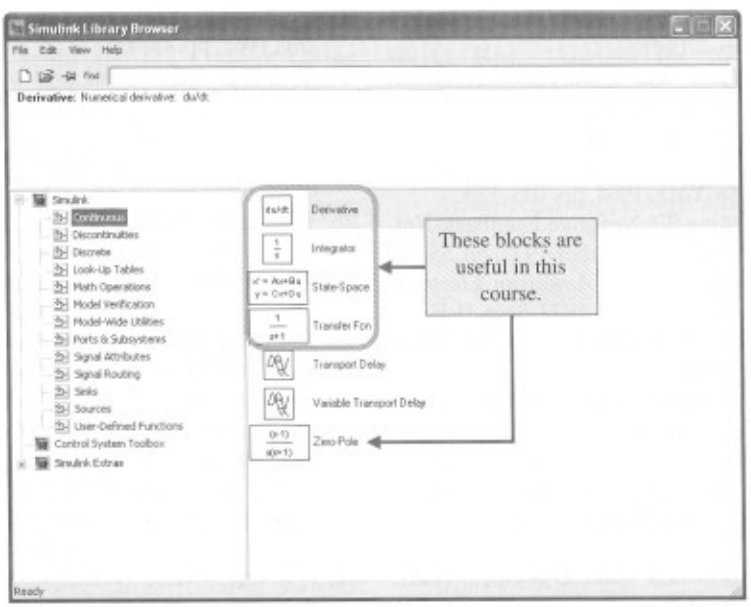

FIGURE B.13
Interesting controls related items in the **Continuous** block library.