Appendix B

T his appendix is a brief introduction to Simulink. This author feels that we can best intro-
duce Simulink with a few examples. Some familiarity with MATLAB is essential in under-
standing Simulink, and for this purpose, Appendix A is included as an introduction to
MATLAB.

B.1 Simulink and its Relation to MATLAB

The MATLAB® and Simulink® environments are integrated into one entity, and thus we can
analyze, simulate, and revise our models in either environment at any point. We invoke Simulink
from within MATLAB. We will introduce Simulink with a few illustrated examples.

Example B.1

For the circuit of Figure B.1, the initial conditions are $i_L(0^-) = 0$, and $v_c(0^-) = 0.5 \text{ V}$. We will
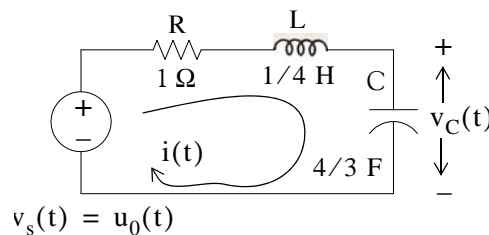compute $v_c(t)$.

Figure B.1. Circuit for Example B.1

For this example,

$$i = i_L = i_C = C\frac{dv_C}{dt} \tag{B.1}$$

and by Kirchoff's voltage law (KVL),

$$Ri_L + L\frac{di_L}{dt} + v_C = u_0(t) \tag{B.2}$$

Substitution of (B.1) into (B.2) yields

$$RC\frac{dv_C}{dt} + LC\frac{d^2v_C}{dt^2} + v_C = u_0(t) \qquad \text{(B.3)}$$

Substituting the values of the circuit constants and rearranging we obtain:

$$\frac{1}{3}\frac{d^2v_C}{dt^2} + \frac{4}{3}\frac{dv_C}{dt} + v_C = u_0(t)$$

$$\frac{d^2v_C}{dt^2} + 4\frac{dv_C}{dt} + 3v_C = 3u_0(t) \qquad \text{(B.4)}$$

$$\frac{d^2v_C}{dt^2} + 4\frac{dv_C}{dt} + 3v_C = 3 \qquad t > 0 \qquad \text{(B.5)}$$

To appreciate Simulink's capabilities, for comparison, three different methods of obtaining the solution are presented, and the solution using Simulink follows.

First Method – Assumed Solution

Equation (B.5) is a second–order, non–homogeneous differential equation with constant coefficients, and thus the complete solution will consist of the sum of the forced response and the natural response. It is obvious that the solution of this equation cannot be a constant since the derivatives of a constant are zero and thus the equation is not satisfied. Also, the solution cannot contain sinusoidal functions (sine and cosine) since the derivatives of these are also sinusoids.

However, decaying exponentials of the form ke^{-at} where k and a are constants, are possible candidates since their derivatives have the same form but alternate in sign.

It can be shown[*] that if $k_1e^{-s_1t}$ and $k_2e^{-s_2t}$ where k_1 and k_2 are constants and s_1 and s_2 are the roots of the characteristic equation of the homogeneous part of the given differential equation, the natural response is the sum of the terms $k_1e^{-s_1t}$ and $k_2e^{-s_2t}$. Therefore, the total solution will be

$$v_c(t) = \text{natural response} + \text{forced response} = v_{cn}(t) + v_{cf}(t) = k_1e^{-s_1t} + k_2e^{-s_2t} + v_{cf}(t) \quad \text{(B.6)}$$

The values of s_1 and s_2 are the roots of the characteristic equation

---

[*] *Please refer to Circuit Analysis II with MATLAB Applications, ISBN 0–9709511–5–9, Appendix B for a thorough discussion.*

$$s^2 + 4s + 3 = 0 \tag{B.7}$$

Solution of (B.7) yields of $s_1 = -1$ and $s_2 = -3$ and with these values (B.6) is written as

$$v_c(t) = k_1 e^{-t} + k_2 e^{-3t} + v_{cf}(t) \tag{B.8}$$

The forced component $v_{cf}(t)$ is found from (B.5), i.e.,

$$\frac{d^2 v_C}{dt^2} + 4\frac{dv_C}{dt} + 3v_C = 3 \qquad t > 0 \tag{B.9}$$

Since the right side of (B.9) is a constant, the forced response will also be a constant and we denote it as $v_{Cf} = k_3$. By substitution into (B.9) we obtain

$$0 + 0 + 3k_3 = 3$$

or

$$v_{Cf} = k_3 = 1 \tag{B.10}$$

Substitution of this value into (B.8), yields the total solution as

$$v_C(t) = v_{Cn}(t) + v_{Cf} = k_1 e^{-t} + k_2 e^{-3t} + 1 \tag{B.11}$$

The constants k_1 and k_2 will be evaluated from the initial conditions. First, using $v_C(0) = 0.5$ V and evaluating (B.11) at $t = 0$, we obtain

$$v_C(0) = k_1 e^0 + k_2 e^0 + 1 = 0.5$$

$$k_1 + k_2 = -0.5 \tag{B.12}$$

Also,

$$i_L = i_C = C\frac{dv_C}{dt}, \quad \frac{dv_C}{dt} = \frac{i_L}{C}$$

and

$$\left.\frac{dv_C}{dt}\right|_{t=0} = \frac{i_L(0)}{C} = \frac{0}{C} = 0 \tag{B.13}$$

Next, we differentiate (B.11), we evaluate it at $t = 0$, and equate it with (B.13). Thus,

$$\left.\frac{dv_C}{dt}\right|_{t=0} = -k_1 - 3k_2 \tag{B.14}$$

By equating the right sides of (B.13) and (B.14) we obtain

$$-k_1 - 3k_2 = 0 \tag{B.15}$$

Simultaneous solution of (B.12) and (B.15), gives $k_1 = -0.75$ and $k_2 = 0.25$. By substitution into (B.8), we obtain the total solution as

$$\mathbf{v_C(t) = (-0.75e^{-t} + 0.25e^{-3t} + 1)u_0(t)} \tag{B.16}$$

Check with MATLAB:

```
syms t                              %  Define symbolic variable t
y0=-0.75*exp(-t)+0.25*exp(-3*t)+1;  %  The total solution y(t), for our example, vc(t)
y1=diff(y0)                         %  The first derivative of y(t)

y1 =
3/4*exp(-t)-3/4*exp(-3*t)

y2=diff(y0,2)                       %  The second derivative of y(t)

y2 =
-3/4*exp(-t)+9/4*exp(-3*t)

y=y2+4*y1+3*y0                      %  Summation of y and its derivatives

y =
3
```

Thus, the solution has been verified by MATLAB. Using the expression for $v_C(t)$ in (B.16), we find the expression for the current as

$$\mathbf{i = i_L = i_C = C\frac{dv_C}{dt} = \frac{4}{3}\left(\frac{3}{4}e^{-t} - \frac{3}{4}e^{-3t}\right) = e^{-t} - e^{-3t}\ A} \tag{B.17}$$

Second Method – Using the Laplace Transformation

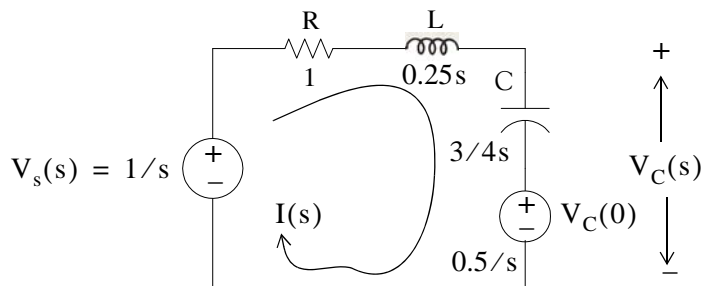The transformed circuit is shown in Figure B.2.

Figure B.2. Transformed Circuit for Example B.1

By the voltage division[*] expression,

$$V_C(s) = \frac{3/4s}{(1 + 0.25s + 3/4s)} \cdot \left(\frac{1}{s} - \frac{0.5}{s}\right) + \frac{0.5}{s} = \frac{1.5}{s(s^2 + 4s + 3)} + \frac{0.5}{s} = \frac{0.5s^2 + 2s + 3}{s(s + 1)(s + 3)}$$

Using partial fraction expansion,[†] we let

$$\frac{0.5s^2 + 2s + 3}{s(s + 1)(s + 3)} = \frac{r_1}{s} + \frac{r_2}{(s + 1)} + \frac{r_3}{(s + 3)} \qquad (B.18)$$

$$r_1 = \left.\frac{0.5s^2 + 2s + 3}{(s + 1)(s + 3)}\right|_{s = 0} = 1$$

$$r_2 = \left.\frac{0.5s^2 + 2s + 3}{s(s + 3)}\right|_{s = -1} = -0.75$$

$$r_3 = \left.\frac{0.5s^2 + 2s + 3}{s(s + 1)}\right|_{s = -3} = 0.25$$

and by substitution into (B.18)

$$V_C(s) = \frac{0.5s^2 + 2s + 3}{s(s + 1)(s + 3)} = \frac{1}{s} + \frac{-0.75}{(s + 1)} + \frac{0.25}{(s + 3)}$$

Taking the Inverse Laplace transform[‡] we find that

$$\mathbf{v_C(t) = 1 - 0.75e^{-t} + 0.25e^{-3t}}$$

Third Method – Using State Variables

$$Ri_L + L\frac{di_L}{dt} + v_C = u_0(t)^{**}$$

---

[*] *For derivation of the voltage division and current division expressions, please refer to Circuit Analysis I with MATLAB Applications, ISBN 0–9709511–2–4.*

[†] *Partial fraction expansion is discussed in Chapter 3, this text.*

[‡] *For an introduction to Laplace Transform and Inverse Laplace Transform, please refer Chapters 2 and 3, this text.*

[**] *Usually, in State–Space and State Variables Analysis, $u(t)$ denotes any input. For distinction, we will denote the Unit Step Function as $u_0(t)$. For a detailed discussion on State–Space and State Variables Analysis, please refer to Chapter 5, this text.*

By substitution of given values and rearranging, we obtain

$$\frac{1}{4}\frac{di_L}{dt} = (-1)i_L - v_C + 1$$

or

$$\frac{di_L}{dt} = -4i_L - 4v_C + 4 \tag{B.19}$$

Next, we define the state variables $x_1 = i_L$ and $x_2 = v_C$. Then,

$$\dot{x}_1 = \frac{di_L}{dt}\text{*} \tag{B.20}$$

and

$$\dot{x}_2 = \frac{dv_C}{dt} \tag{B.21}$$

Also,

$$i_L = C\frac{dv_C}{dt}$$

and thus,

$$x_1 = i_L = C\frac{dv_C}{dt} = C\dot{x}_2 = \frac{4}{3}\dot{x}_2$$

or

$$\dot{x}_2 = \frac{3}{4}x_1 \tag{B.22}$$

Therefore, from (B.19), (B.20), and (B.22), we obtain the state equations

$$\dot{x}_1 = -4x_1 - 4x_2 + 4$$

$$\dot{x}_2 = \frac{3}{4}x_1$$

and in matrix form,

$$\begin{bmatrix} \dot{x}_1 \\ \dot{x}_2 \end{bmatrix} = \begin{bmatrix} -4 & -4 \\ 3/4 & 0 \end{bmatrix}\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + \begin{bmatrix} 4 \\ 0 \end{bmatrix}u_0(t) \tag{B.23}$$

Solution[†] of (B.23) yields

---

* *The notation* $\dot{x}$ *(x dot) is often used to denote the first derivative of the function* x, *that is,* $\dot{x} = dx/dt$.

† *The detailed solution of (B.23) is given in Chapter 5, Example 5.10, Page 5–23, this text.*

$$\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} = \begin{bmatrix} e^{-t} - e^{-3t} \\ 1 - 0.75e^{-t} + 0.25e^{-3t} \end{bmatrix}$$

Then,

$$x_1 = i_L = e^{-t} - e^{-3t} \tag{B.24}$$

and

$$x_2 = v_C = 1 - 0.75e^{-t} + 0.25e^{-3t} \tag{B.25}$$

Modeling the Differential Equation of Example B.1 with Simulink

To run Simulink, we must first invoke MATLAB. Make sure that Simulink is installed in your system. In the MATLAB Command prompt, we type:

simulink

Alternately, we can click on the Simulink icon shown in Figure B.3. It appears on the top bar on MATLAB's Command prompt.

Figure B.3. The Simulink icon

Upon execution of the Simulink command, the **Commonly Used Blocks** appear as shown in Figure B.4.

In Figure B.4, the left side is referred to as the **Tree Pane** and displays all Simulink libraries installed. The right side is referred to as the **Contents Pane** and displays the blocks that reside in the library currently selected in the Tree Pane.

Let us express the differential equation of Example B.1 as

$$\frac{d^2 v_C}{dt^2} = -4\frac{dv_C}{dt} - 3v_C + 3u_0(t) \tag{B.26}$$

A block diagram representing relation (B.26) above is shown in Figure B.5. We will use Simulink to draw a similar block diagram.[*]

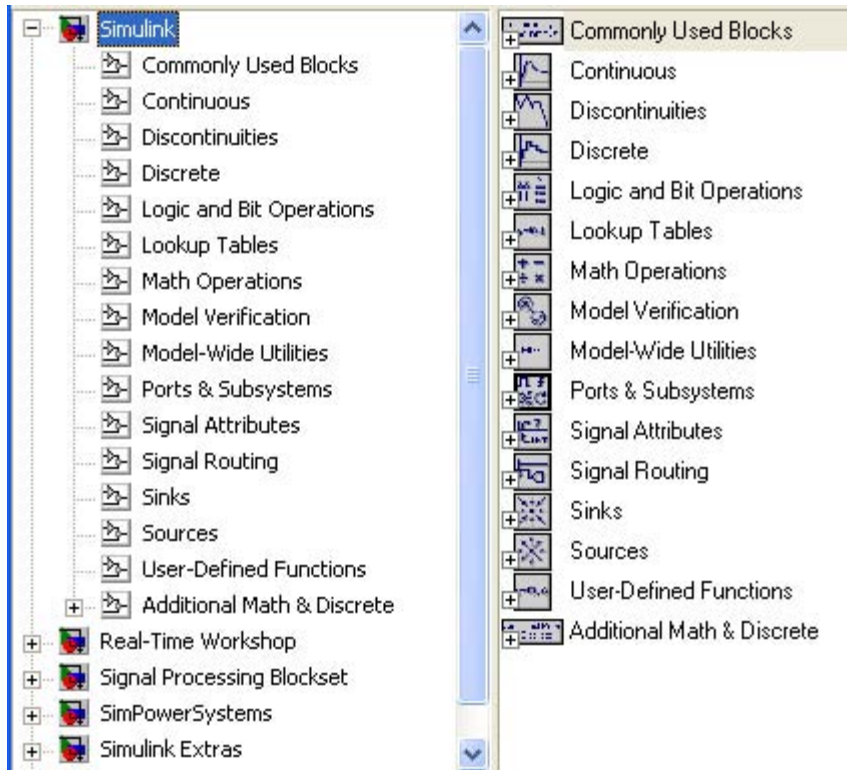[*] *Henceforth, all Simulink block diagrams will be referred to as models.*
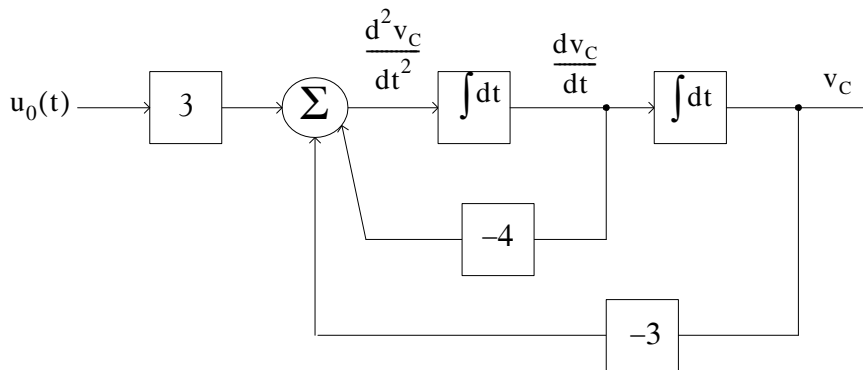
Figure B.4. The Simulink Library Browser

Figure B.5. Block diagram for equation (B.26)

To model the differential equation (B.26) using Simulink, we perform the following steps:

1. On the **Simulink Library Browser**, we click on the leftmost icon shown as a blank page on the top title bar. A new model window named **untitled** will appear as shown in Figure B.6.
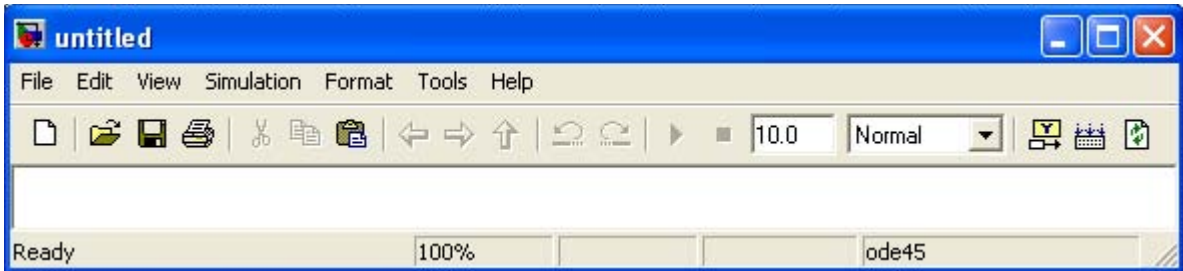
Figure B.6. The Untitled model window in Simulink.

The window of Figure B.6 is the model window where we enter our blocks to form a block diagram. We save this as model file name Equation_1_26. This is done from the File drop menu of Figure B.6 where we choose **Save as** and name the file as Equation_1_26. Simulink will add the extension .mdl. The new model window will now be shown as **Equation_1_26**, and all saved files will have this appearance. See Figure B.7.
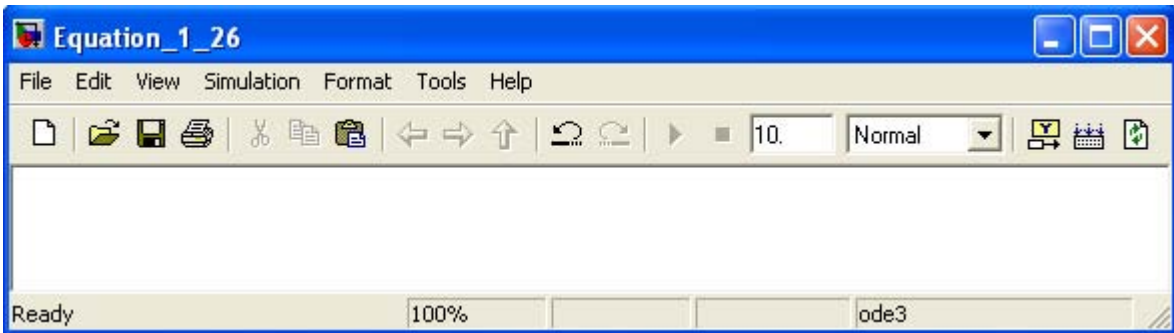
Figure B.7. Model window for Equation_1_26.mdl file

2. With the **Equation_1_26** model window and the **Simulink Library Browser** both visible, we click on the **Sources** appearing on the left side list, and on the right side we scroll down until we see the unit step function shown as **Step**. See Figure B.8. We select it, and we drag it into the **Equation_1_26** model window which now appears as shown in Figure B.8. We save file Equation_1_26 using the File drop menu on the **Equation_1_26** model window (right side of Figure B.8).

3. With reference to block diagram of Figure B.5, we observe that we need to connect an amplifier with Gain 3 to the unit step function block. The gain block in Simulink is under **Commonly Used Blocks** (first item under Simulink on the **Simulink Library Browser**). See Figure B.8. If the **Equation_1_26** model window is no longer visible, it can be recalled by clicking on the white page icon on the top bar of the **Simulink Library Browser**.

4. We choose the gain block and we drag it to the right of the unit step function. The triangle on the right side of the unit step function block and the > symbols on the left and right sides of the gain block are connection points. We point the mouse close to the connection point of the unit step function until is shows as a cross hair, and draw a straight line to connect the two

blocks.[*] We double–click on the gain block and on the **Function Block Parameters**, we change the gain from 1 to 3. See Figure B.9.
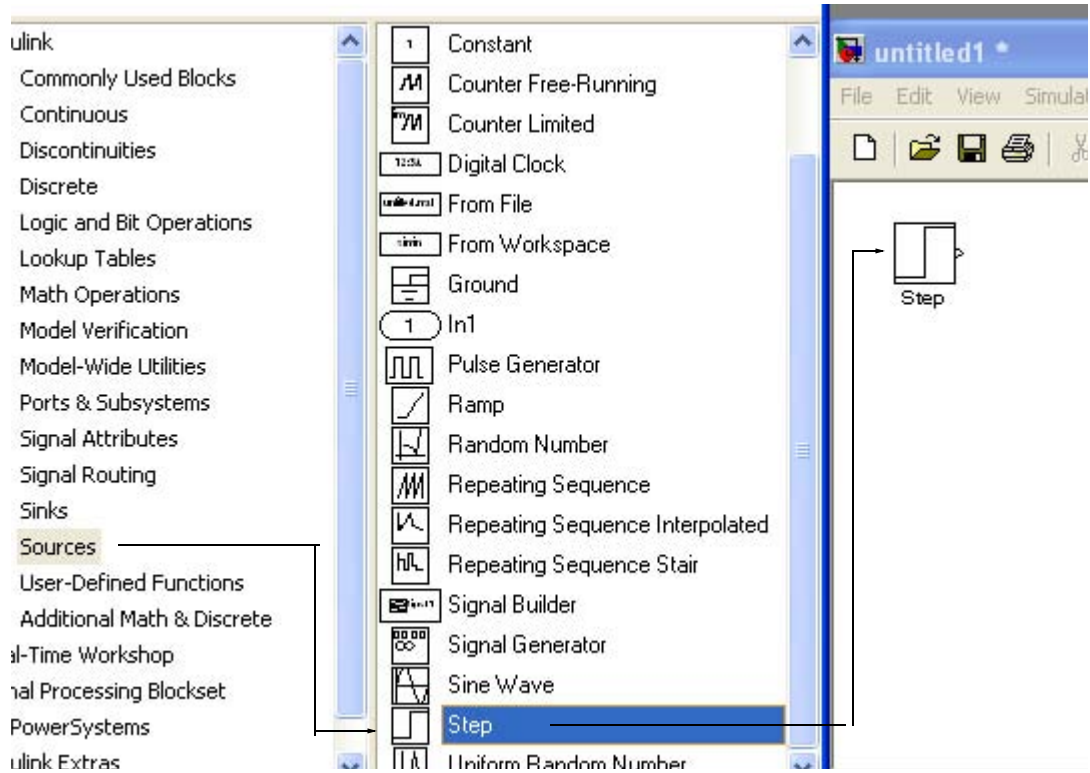

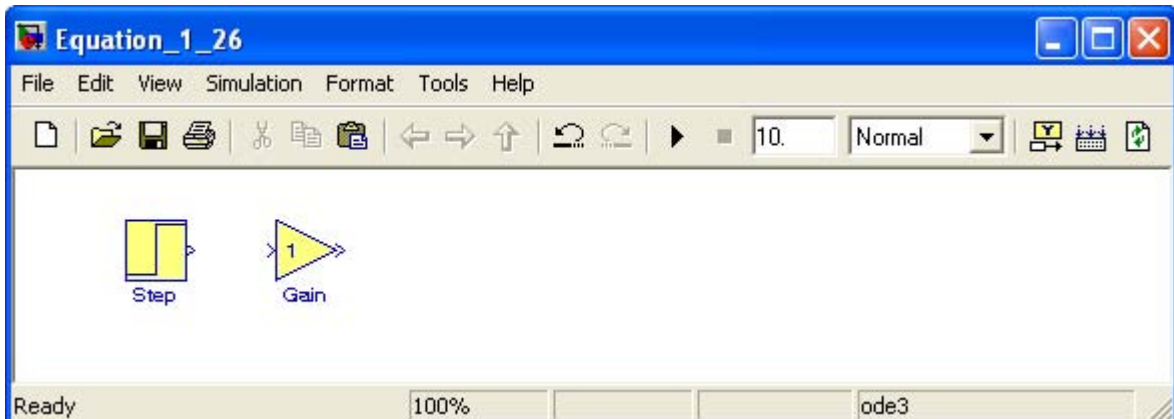Figure B.8. Dragging the unit step function into File Equation_1_26

Figure B.9. File Equation_1_26 with added Step and Gain blocks

---

[*] *An easy method to interconnect two Simulink blocks by clicking on the source block to select it, then hold down the **Ctrl** key and left–click on the destination block.*

5. Next, we need to add a thee–input adder. The adder block appears on the right side of the **Simulink Library Browser** under **Math Operations**. We select it, and we drag it into the Equation_1_26 model window. We double click it, and on the **Function Block Parameters** window which appears, we specify 3 inputs. We then connect the output of the of the gain block to the first input of the adder block as shown in Figure B.10.
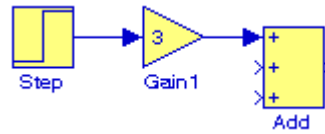

Figure B.10. File Equation_1_26 with added gain block

6. From the **Commonly Used Blocks** of the **Simulink Library Browser**, we choose the **Integrator** block, we drag it into the Equation_1_26 model window, and we connect it to the output of the **Add** block. We repeat this step and to add a second **Integrator** block. We click on the text "Integrator" under the first integrator block, and we change it to Integrator 1. Then, we change the text "Integrator 1" under the second Integrator to "Integrator 2" as shown in Figure B.11.

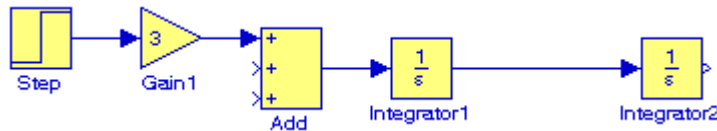

Figure B.11. File Equation_1_26 with the addition of two integrators

7. To complete the block diagram, we add the **Scope** block which is found in the **Commonly Used Blocks** on the **Simulink Library Browser**, we click on the Gain block, and we copy and paste it twice. We flip the pasted Gain blocks by using the **Flip Block** command from the Format drop menu, and we label these as Gain 2 and Gain 3. Finally, we double–click on these gain blocks and on the **Function Block Parameters** window, we change the gains from to –4 and –3 as shown in Figure B.12.
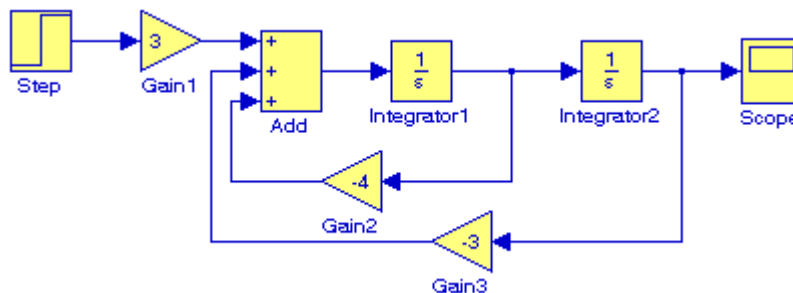

Figure B.12. File Equation_1_26 complete block diagram

8. The initial conditions $i_L(0^-) = C\dfrac{dv_C}{dt}\bigg|_{t=0} = 0$, and $v_c(0^-) = 0.5 \text{ V}$ are entered by double clicking the Integrator blocks and entering the values 0 for the first integrator, and 0.5 for the second integrator. We also need to specify the simulation time. This is done by specifying the simulation time to be 10 seconds on the **Configuration Parameters** from the **Simulation** drop menu. We can start the simulation on **Start** from the **Simulation** drop menu or by clicking on the ▸ icon.

9. To see the output waveform, we double click on the **Scope** block, and then clicking on the Autoscale ⚓ icon, we obtain the waveform shown in Figure B.13.
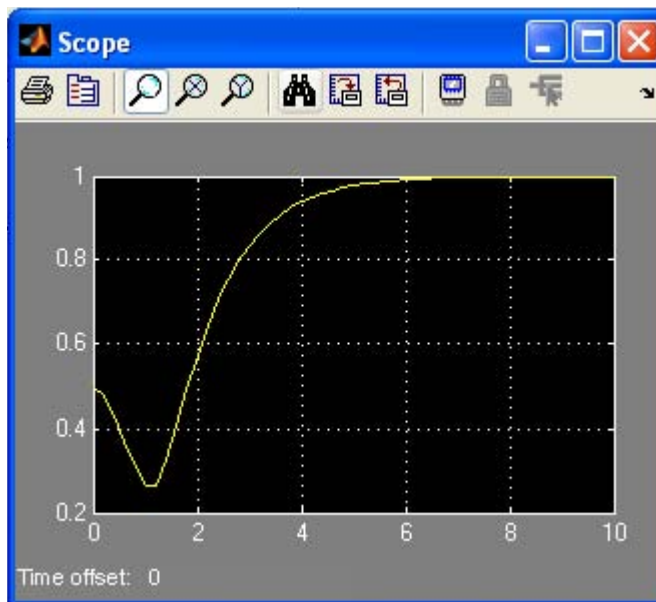

Figure B.13. The waveform for the function $v_C(t)$ for Example B.1

Another easier method to obtain and display the output $v_C(t)$ for Example B.1, is to use **State–Space** block from **Continuous** in the Simulink Library Browser, as shown in Figure B.14.
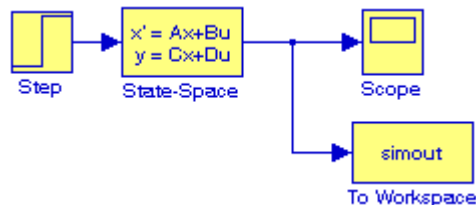

Figure B.14. Obtaining the function $v_C(t)$ for Example B.1 with the State–Space block.

The **simout To Workspace** block shown in Figure B.14 writes its input to the workspace. The data and variables created in the MATLAB Command window, reside in the MATLAB Workspace. This block writes its output to an array or structure that has the name specified by the block's Variable name parameter. This gives us the ability to delete or modify selected variables. We issue the command who to see those variables. From Equation B.23, Page B–6,

$$\begin{bmatrix} \dot{x}_1 \\ \dot{x}_2 \end{bmatrix} = \begin{bmatrix} -4 & -4 \\ 3/4 & 0 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + \begin{bmatrix} 4 \\ 0 \end{bmatrix} u_0(t)$$

The output equation is

$$y = Cx + du$$

or

$$y = \begin{bmatrix} 0 & 1 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + [0]u$$

We double–click on the **State–Space** block, and in the **Functions Block Parameters** window we enter the constants shown in Figure B.15.
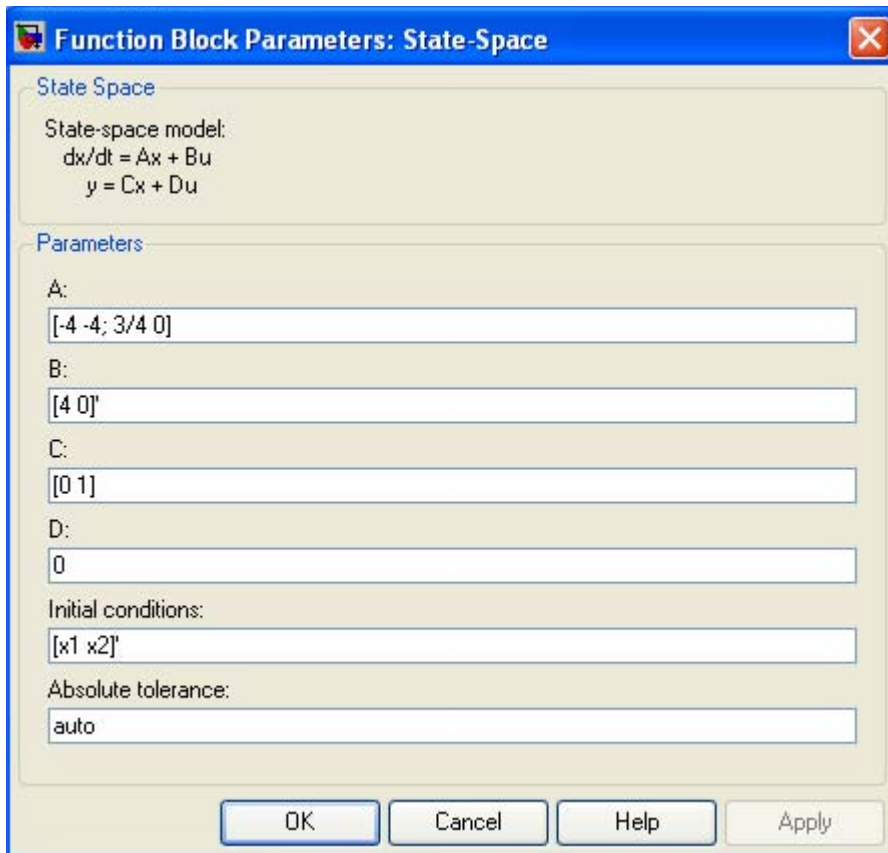

Figure B.15. The Function block parameters for the State–Space block.

The initials conditions [x1 x2]' are specified in MATLAB's Command prompt as

x1=0; x2=0.5;

As before, to start the simulation we click clicking on the ▶ icon, and to see the output wave-

form, we double click on the **Scope** block, and then clicking on the Autoscale 🔍 icon, we obtain the waveform shown in Figure B.16.
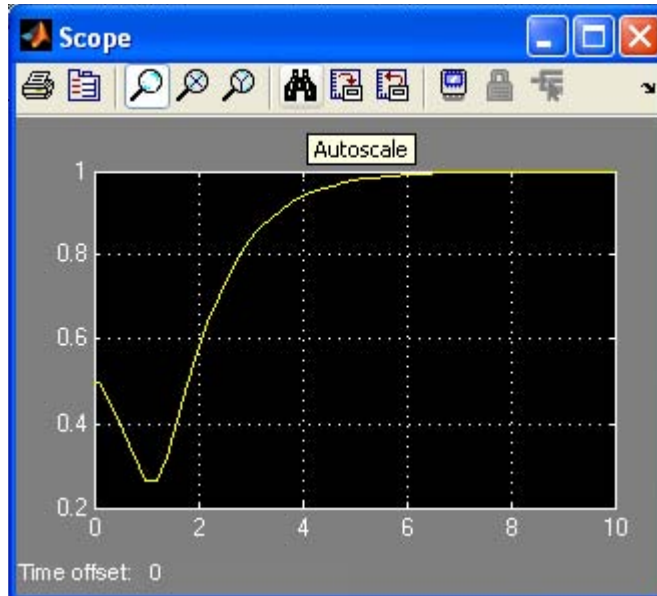

Figure B.16. The waveform for the function $v_C(t)$ for Example B.1 with the State–Space block.

The state–space block is the best choice when we need to display the output waveform of three or more variables as illustrated by the following example.

Example B.2

A fourth–order network is described by the differential equation

$$\frac{d^4y}{dt^4} + a_3\frac{d^3y}{dt^3} + a_2\frac{d^2y}{dt^2} + a_1\frac{dy}{dt} + a_0\,y(t) = u(t) \qquad (B.27)$$

where $y(t)$ is the output representing the voltage or current of the network, and $u(t)$ is any input, and the initial conditions are $y(0) = y'(0) = y''(0) = y'''(0) = 0$.

a. We will express (B.27) as a set of state equations