1

GETTING STARTED WITH SIMULINK

1.1 INTRODUCTION

This chapter describes the basic steps to be followed in building a Simulink model. The model presented in this chapter is a simple one: generation of a sinusoidal signal. This presentation uses MATLAB 2014a[1], incorporating Simulink and blocks extracted from the Simulink Library Browser. Subsequent chapters will utilize blocks from the Simulink Library Browser including the Communications System Toolbox and the DSP System Toolbox. The Communications System Toolbox provides a collection of MATLAB functions and simulation blocks that can be utilized for a wide range of digital communications systems simulation models. While MATLAB and Simulink are available for a variety of operating systems, all of the descriptions and examples presented in this book are implemented on Windows-based computers.

A comprehensive presentation of MATLAB® and Simulink® by The MathWorks™ is available at http://www.mathworks.com/help/. Product descriptions from this documentation are provided as follows:

[1]Every model presented in this book executes successfully in MATLAB 2014a. In some instances, a notice is generated indicating that the model was developed in an earlier release.

Modeling of Digital Communication Systems Using SIMULINK®, First Edition.
Arthur A. Giordano and Allen H. Levesque.

Companion Website: www.wiley.com/go/simulink

MATLAB®

MATLAB® is a high-level language and interactive environment for numerical computation, visualization, and programming. Using MATLAB, you can analyze data, develop algorithms, and create models and applications. The language, tools, and built-in math functions enable you to explore multiple approaches and reach a solution faster than with spreadsheets or traditional programming languages, such as C/C++ or Java®. MATLAB is an abbreviation for "matrix laboratory." While other programming languages mostly work with numbers one at a time, MATLAB is designed to operate primarily on whole matrices and arrays.

Simulink®

Simulink® is a block diagram environment for multidomain simulation and Model-Based Design. It supports system-level design, simulation, automatic code generation, and continuous test and verification of embedded systems. Simulink provides a graphical editor, customizable block libraries, and solvers for modeling and simulating dynamic systems. It is integrated with MATLAB, enabling you to incorporate MATLAB algorithms into models and export simulation results to MATLAB for further analysis.

Upon gaining familiarity with Simulink, the user will discover that multiple paths can be followed in developing a Simulink model. The choice of the path is left to the user but each path will lead to the same solution.

The topics covered in this chapter are:

- Starting a MATLAB session
- Viewing Simulink block libraries
- Building a new Simulink model
  - Setting simulation parameters
  - Setting and using Scopes
- Executing the model
- Sending data to Workspace
- Using the Model Explorer
- Selecting Model Configuration Parameters

1.2 STARTING A MATLAB SESSION

To start a MATLAB session on a Windows machine, simply double-click on the MATLAB icon; this will open the MATLAB desktop, shown in Figure 1.1.

The default desktop view shown in this text includes four panels. (Other desktop views may be selected under the Home menu; see the Layout tabs on the toolbar.) The left-hand panel displays the Current Folder, which can contain MATLAB or Simulink models in addition to user-developed figures and user files resident in the folder. The center panel is the Command Window, where the user inserts MATLAB commands, assigns values to model parameters, and performs calculations using MATLAB mathematical functions. The upper right-hand panel is the Workspace, where variables defined in the Command Window are displayed, and the lower right-hand panel is the Command History, where the user can view or rerun commands entered at the command line. In Figure 1.1 the bar above the Command Window is labeled here as ▶ C: ▶ Users ▶ Default User

The path can be changed to a different folder. At the end of this bar the symbol ▼ also allows the path to be changed to a previous selection.

1.3 SIMULINK BLOCK LIBRARIES

Building a Simulink model consists of selecting individual blocks contained in libraries and joining them in a block diagram of the system to be simulated.

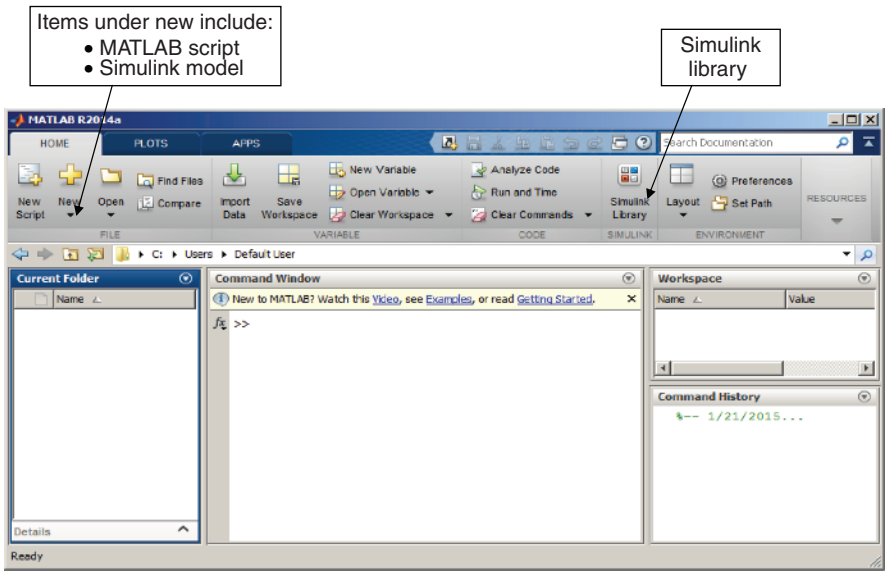

Figure 1.1 MATLAB Default Desktop View. Reprinted with permissions from The MathWorks™, Inc[2].

[2]This Simulink parameter window and similar figures appearing throughout the book are all reprinted with permissions from The MathWorks™, Inc.

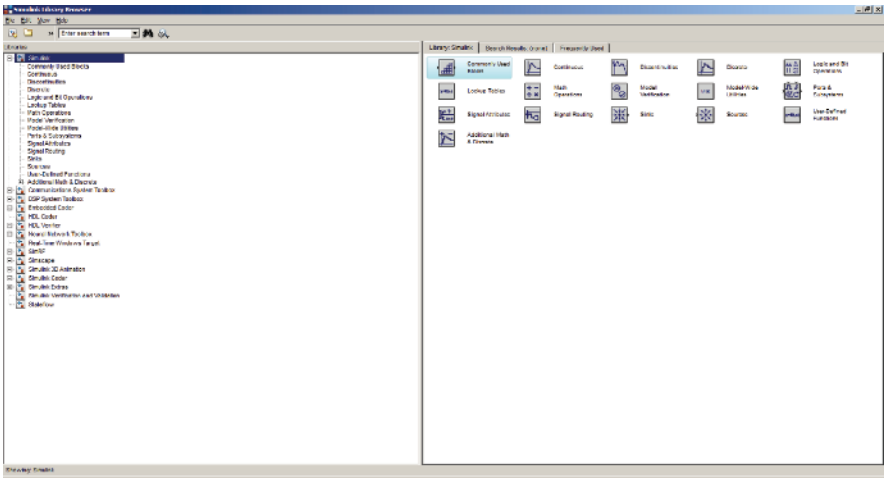

Figure 1.2 Simulink Block Library.

To view available blocks, select Simulink Library on the MATLAB toolbar. This opens the window shown in Figure 1.2.

The Simulink Library Browser shows a listing of available Simulink blocks. The focus in this book is on modeling digital communication systems, and the blocks you will find most useful are contained in the basic Simulink block library as well as in the Communications System Toolbox and the DSP System Toolbox.

Simulink library blocks used throughout this book are listed here.

Simulink

- **Commonly used blocks**
  - Constant
  - Delay
  - In1 & Out1
  - Scope
  - Math operations
    - Abs
    - Sum
    - Product
    - Complex to Real-Imag
    - Math Function
  - Model Wide Utilities

    - Model Info
  - Signal Routing
    - From
    - Goto
    - Mux
  - Sinks
    - Display
    - Scope
    - To Workspace
  - Sources
    - Constant
    - From Workspace
    - Random number
    - Sine Wave
  - User-Defined Functions
    - Matlab Function
- **Communications System Toolbox**
  - Channels
    - AWGN
    - Multipath Rayleigh Fading
  - Comm Sources
    - Noise Generators
      - Gaussian Noise Generator
    - Random Data Sources
      - Bernoulli Binary Generator
      - Random Integer Generator
  - Error Detection & Correction
    - Block
    - Convolutional
  - Modulation
    - Digital Baseband Modulation
      - AM (QAM)
      - PM (BPSK,QPSK,M-PSK)
      - FM(M-FSK)
- **DSP System Toolbox**
  - Filtering

  - Adaptive Filters
    - Block LMS Filter
    - Kalman Filter
    - RLS Filter
- Signal Management
  - Buffers
  - Sinks
    - Spectrum Analyzer
    - Time Scope
    - Vector Scope
  - Sources (DSP Sine Wave)
  - Statistics
    - Mean
    - Variance
    - Autocorrelation
    - Correlation
  - Transforms
    - FFT
    - IFFT

Appendix A lists principal Simulink blocks used in Chapters 1–13.

1.4 BUILDING A NEW SIMULINK MODEL

To begin building a new Simulink model, on the MATLAB toolbar, under the **HOME** tab, pull down **New** and select **Simulink Model**. This will open a blank Simulink model window, shown in Figure 1.3. Note that on the title bar at the top of the window, this model is labeled untitled. In the model window, the user may select the duration of the model execution, shown here to be set at 10.0 s. This will fix the duration of each of the simulations to be demonstrated in this chapter.

To rename the model, on the toolbar, select **File or File:save as** and enter the model name **First_Simulink_Model**, then **save**. The renamed model is shown in Figure 1.4.

1.4.1 Inserting Signal Source and Scope

The model can be constructed either by selecting blocks from the Simulink Library Browser or by copying blocks from an existing model. Here, blocks

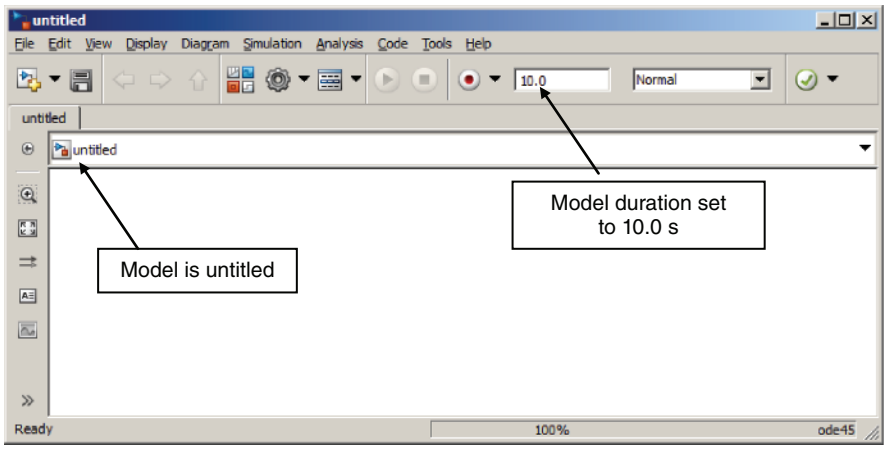

Figure 1.3 Simulink Model Blank Window.

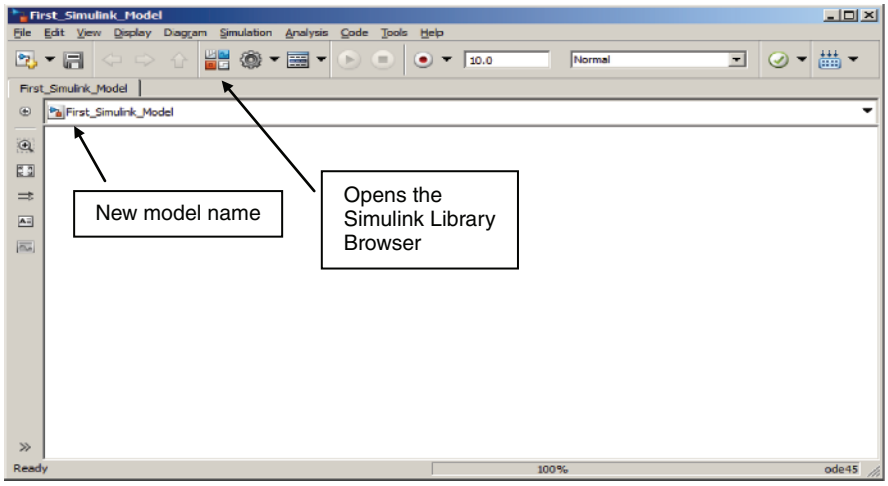

Figure 1.4 Simulink Model Window Renamed First_Simulink_Model.

will be copied from the library, which can be opened from the model window by clicking on the four-symbol icon on the toolbar shown in Figure 1.4. First, in the library window (Figure 1.2), click on **Sources** to open the window shown in Figure 1.5. With both the **First_Simulink_Model** window and the Simulink Library Browser window open, left-click on the **Sine Wave** icon and drag a copy into the model window. Alternatively, you can right-click on the icon in the library and select **Add to First_Simulink_Model**.

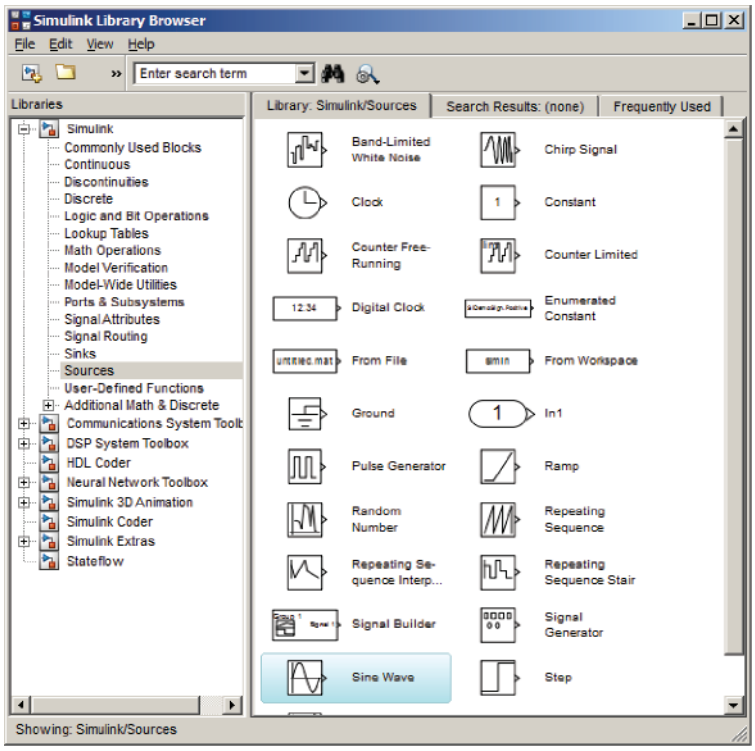

Figure 1.5 Simulink Library Browser with **Sources** Selected.

Next, add a scope to the model by returning to the Simulink Library Browser and clicking on **-Sinks**, selecting **scope**, and dragging a copy into the model window, now shown in Figure 1.6. In the figure, the **Sine Wave** block has been connected to the **Scope** by clicking on the arrowhead at the **Sine Wave** output and dragging a line to the corresponding arrowhead at the input to the **Scope**.

Another block, entitled **Model Info**, is shown in Figure 1.6 and is available in the Simulink library under **Simulink Model-Wide Utilities**. Dragging this block to the **First_Simulink_Model** and double clicking on this block opens a text box. This utility is very useful for conveniently displaying the parameters of each simulation model and identifying pertinent information about the model.

1.4.2 Setting the Source Block Parameters

In the model window, double-click on the **Sine Wave** icon; this opens an information window for the **Sine Wave** block, shown in Figure 1.7. In the

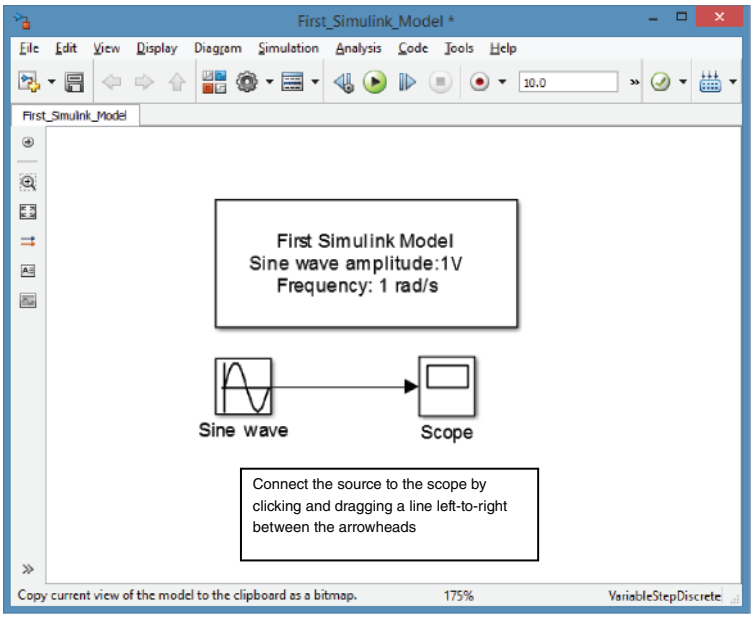

Figure 1.6 Simulink Model With Sine Wave Source, Scope, and Model Info Blocks.

window, the **Amplitude** and **Bias** of the sinusoidal source are selected as 1 and 0, respectively. The **Sine** type and **Time (t)** selections shown in the figure will be suitable for most simulations. The **Sine Wave** block parameters window can also be opened by selecting the block in the model window and right-clicking the mouse, which displays a list of options, and selecting **Block Parameters (Sin)**. This same pull-down menu provides options to manipulate and/or format the selected block, which the user will find helpful in structuring block diagrams in Simulink models.

The user will normally find options that can be selected for each block in addition to entering parameter values. As an example, in this block the user has a choice under **Sine** type to select either **Time-based** or **Sample-based** computation; under **Time (t)** the user can select **Use simulation time** or **Use external signal**. To provide a stream-lined introduction to model building, the presentation in this text will omit detailed discussions of many options. The Simulink documentation, accessible by clicking on the **Help** button, provides more extensive information on the optional selections.

1.4.3 Setting Scope Parameters

In the model window, double-click on the **Scope** icon, opening the **Scope** display, shown in Figure 1.8. At this point, the display is blank, since no

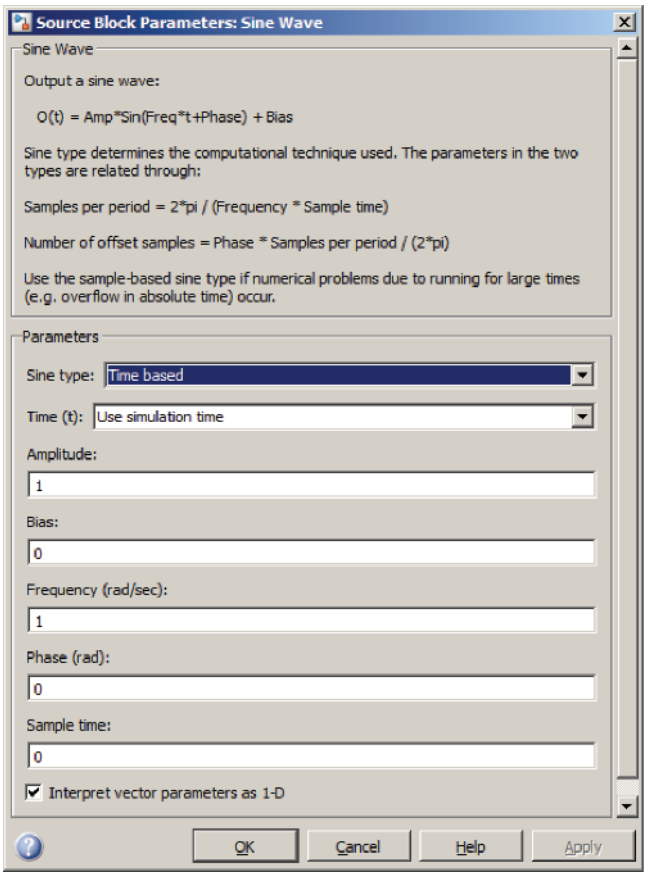

Figure 1.7 Information Window for the Sine Wave Source Block.

simulation has been started with this model. The gear-like icon on the toolbar opens the **Scope Parameters** window, which has three pages. On the **General** page, shown in Figure 1.9, the user is able to set the number of axes to be displayed in the **Scope** display, the simulation time range to be displayed, and to specify where **Tick labels** are to be applied in the display. Since the scope can be set to display multiple axes (for multiple inputs), the time ticks might be applied to all axes (select **all**), or to **none**, or to the **bottom axis only**.

The **History** page of the **Scope Parameters** window is shown in Figure 1.10. Here the user can specify the number of simulation data points to be displayed on the scope and can elect to have the data stored to **workspace**. On the **Style** page of the **Scope Parameters** window, shown in Figure 1.11, the user has various options as to colors, line styles and choice of markers to be used in the scope display.

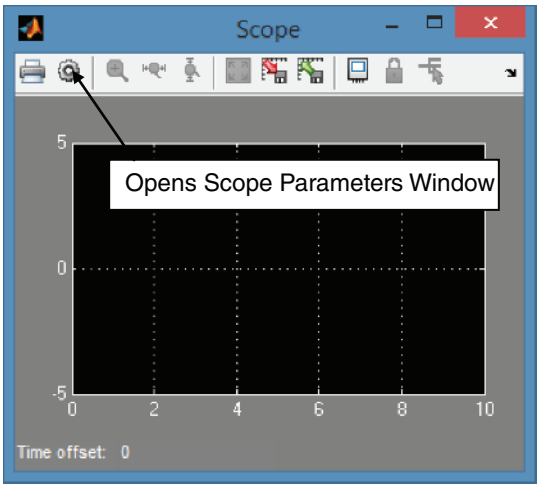

Figure 1.8 Scope Display.

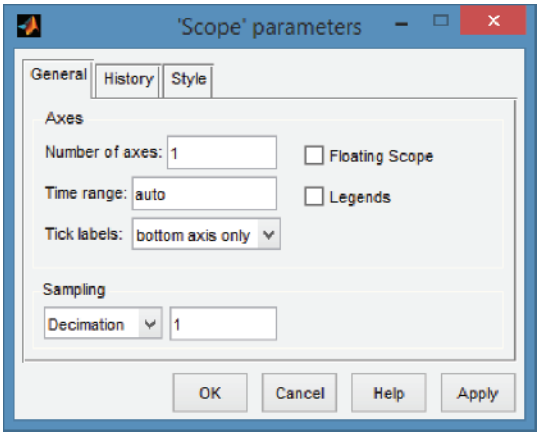

Figure 1.9 Scope Parameters Window: General Page.

1.5 EXECUTING THE SIMULINK MODEL

To execute the simple **Sine Wave** model, in the model window, click on the dark arrow button near the center of the toolbar, as shown in Figure 1.12. Double-clicking on the **Scope** block opens the **Scope** display window, shown in Figure 1.13. Examining the signal trace on the **Scope** display confirms the signal source settings made in Figure 1.7: **Amplitude** = **1**, and **Frequency** = **1 rad/s**.

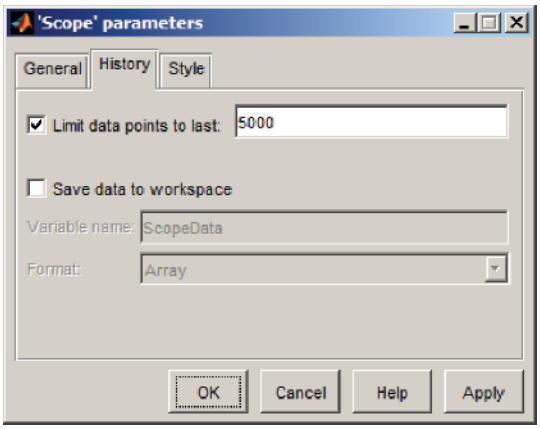

Figure 1.10 Scope Parameters Window: History Page.

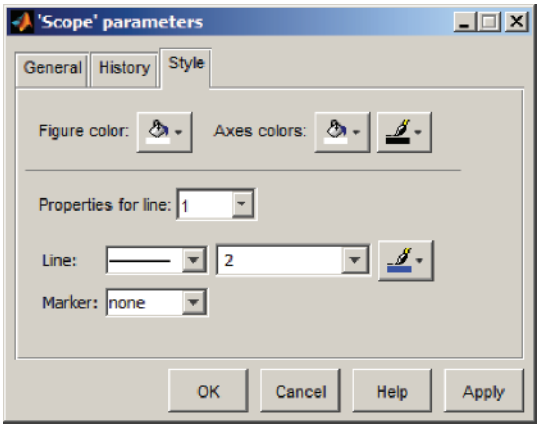

Figure 1.11 Scope Parameters Window: Style Page.

In this model, under the **Display** tab, selecting **Sample Time** and then all adds a data label **Cont** indicating that continuous time has been chosen, a direct result of choosing **Sine type** as **Time based** in Figure 1.7. The model colors are now changed, as seen in Figure 1.14, where the **Sine Wave** and **Scope** blocks are black and the **Model Info** block is magenta.

A **Sample Time Legend** is also displayed, as shown in Figure 1.15, indicating the block colors, the data type and the data values. In this case, the **Sine Wave** block is black, labeled **Cont** for continuous with a zero value; the **Model info** block is magenta with data value and type **Inf,** indicating that there is no time associated with this block, that is, it exists as long as the simulation is active.

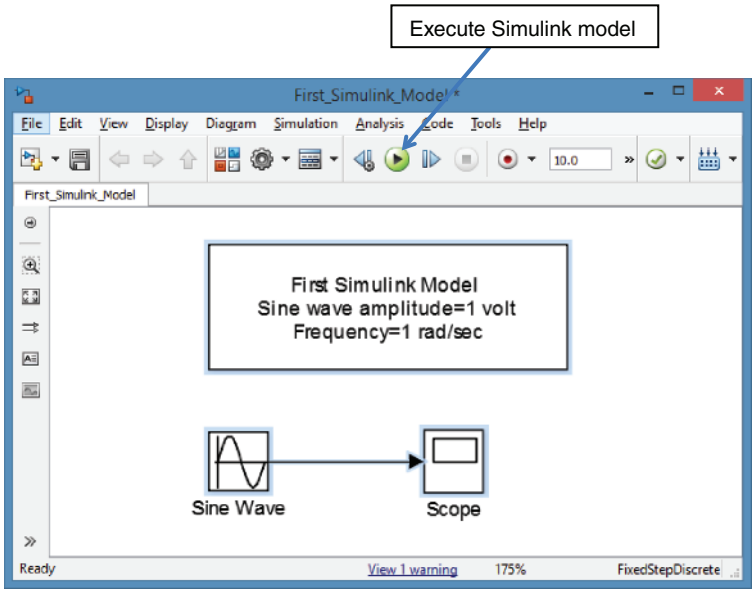

Figure 1.12 Executing the Simulink Model.

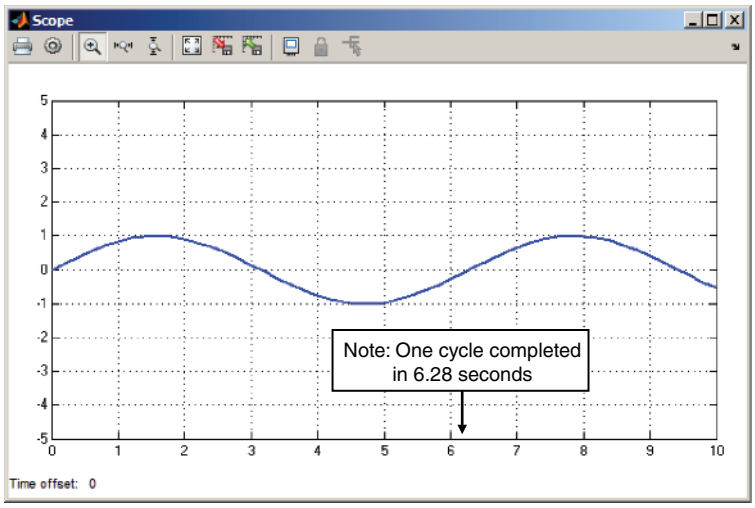

Figure 1.13 Scope Display After Executing the Sine Wave Model.

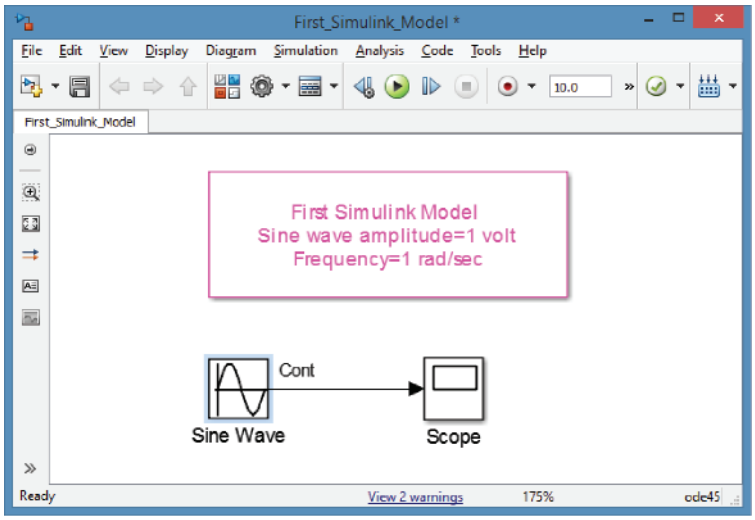

Figure 1.14 Model After Selecting Sample Time Under Display Tab.

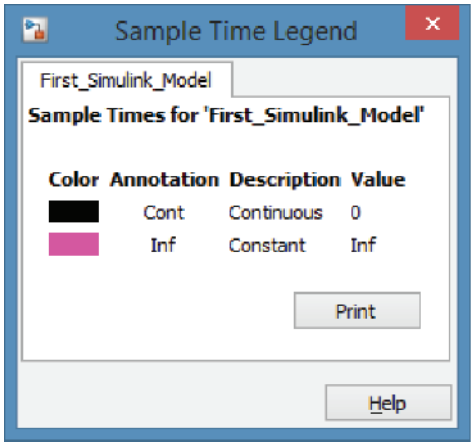

Figure 1.15 Sample Time Legend Indicating Block Characteristics.

1.6 RECONFIGURING THE SIGNAL BLOCK

At this point, another model change will be used to demonstrate an important feature of Simulink: Simulink blocks are designed to accept signals and parameter values as vector inputs. As an example, the time-based **Sine Wave** model in Figure 1.14 can be modified to generate two sinusoids instead of just one. A simple way of doing this is double-clicking on the

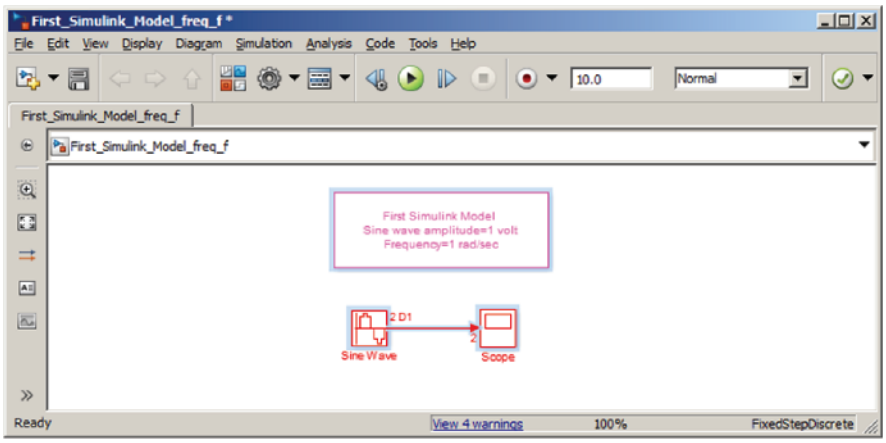

Figure 1.16 Sine Wave Model For Generating Sinusoids at 1 and 10 rad/s.

Sine Wave block, opening the block parameters window, and changing the **frequency (rad/s)** setting by inserting the two-element vector [1 10]. This configures the **Sine Wave** block to generate two sinousoids, one at 1 rad/s, the other at 10 rad/s. This change creates a new model, here renamed **First_Simulink_Model_freq_f**, shown in Figure 1.16. Note in the model window that the **Sine Wave** output is labeled **2D1** and the input to the **Scope** is labeled 2, both resulting from setting the frequency to the vector value [1 10].

After running the simulation, the **Scope** will display the two sampled sinusoids, as seen in Figure 1.17. To generate more frequencies with the **Sine Wave** block, simply add more terms in the vector input for the **frequency (rad/s)** cell in the parameters window.

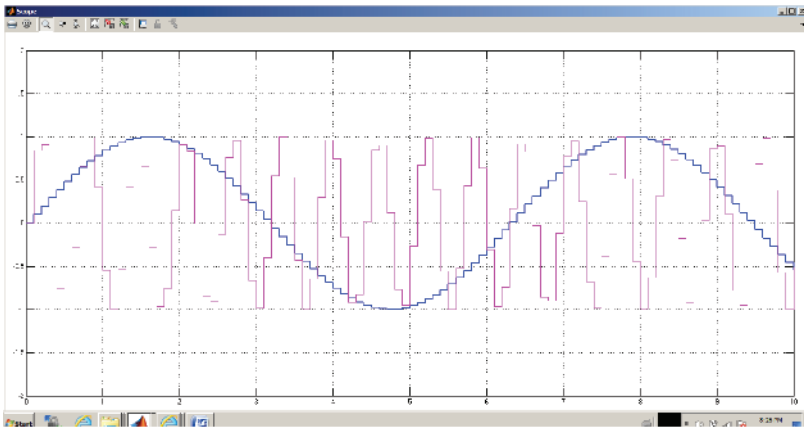

Figure 1.17 Scope Display for First Simulink Model with Frequency Set to [1 10].

Another way of configuring the **Sine Wave** block to generate the two sinusoids is to specify the **frequency (rad/s)** as **f** in the block parameters window, and define the variable **f** by inserting the statement **f = [1 10]** into the MATLAB Command Window. A third way of reconfiguring the **Sine Wave** block is discussed in Section 1.9, where the use of **Model Explorer** is demonstrated.

1.7 SAMPLE-BASED SIGNALS

In Figure 1.18, the simulation model is changed by selecting **Sample based** in the **Source Block Parameters: Sine Wave** window; the **Sample time** is set to 0.1 (where the units are seconds) and the number of **samples**

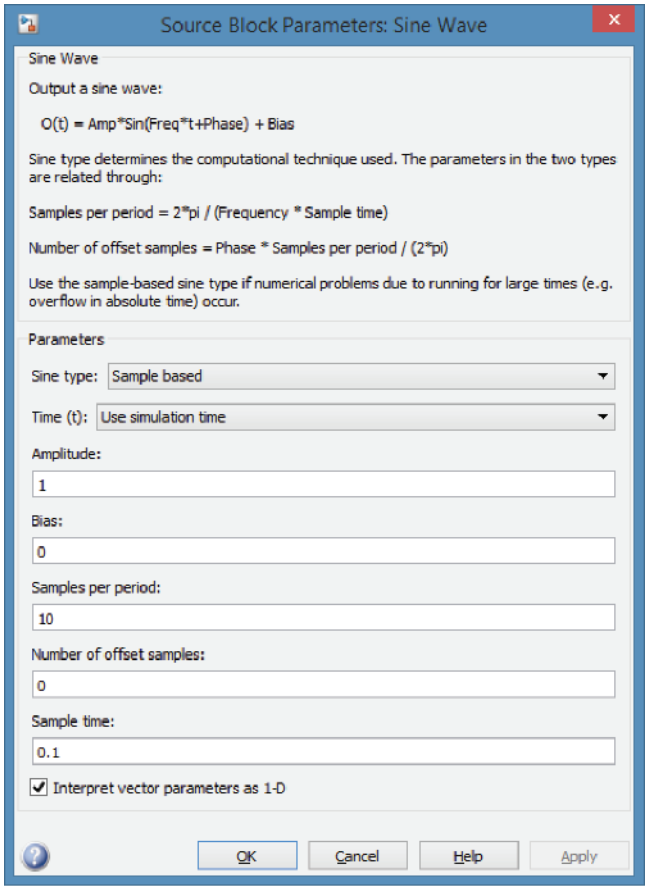

Figure 1.18 Sample Based Selection in Sine Wave Block.

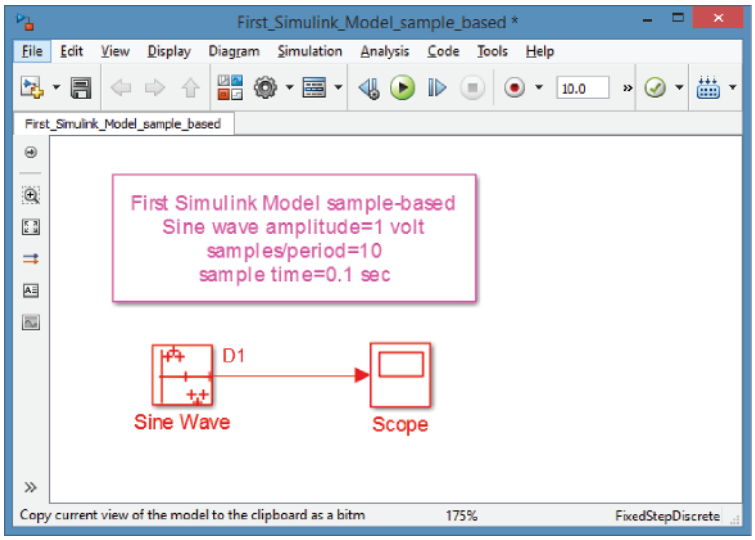

Figure 1.19 First Simulink Model Modified for Sample Based computation.

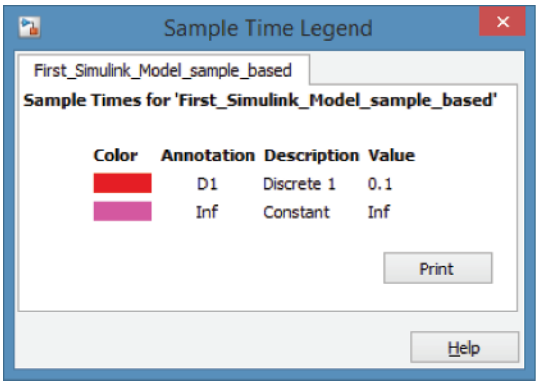

Figure 1.20 Sample Time Legend for Sample Based Computation.

per period is set to 10. Figures 1.19 and 1.20 display, respectively, the sample-based model and the **Sample Time Legend** resulting from the change. The color red in the blocks and data line indicates that the output from the **Sine Wave** block is a discrete vector with 0.1 s time steps and the magenta color shows **inf** corresponding to an infinite associated time.

Upon running the simulation, the scope output, shown in Figure 1.21, displays the resulting sample values as a function of time (Note: labeling of axes is discussed in a later section.)

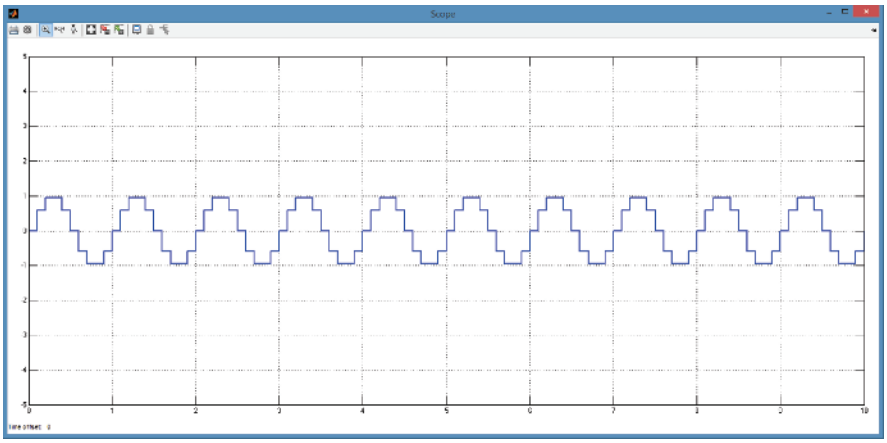

Figure 1.21 Scope Display for Sample-Based Computation.

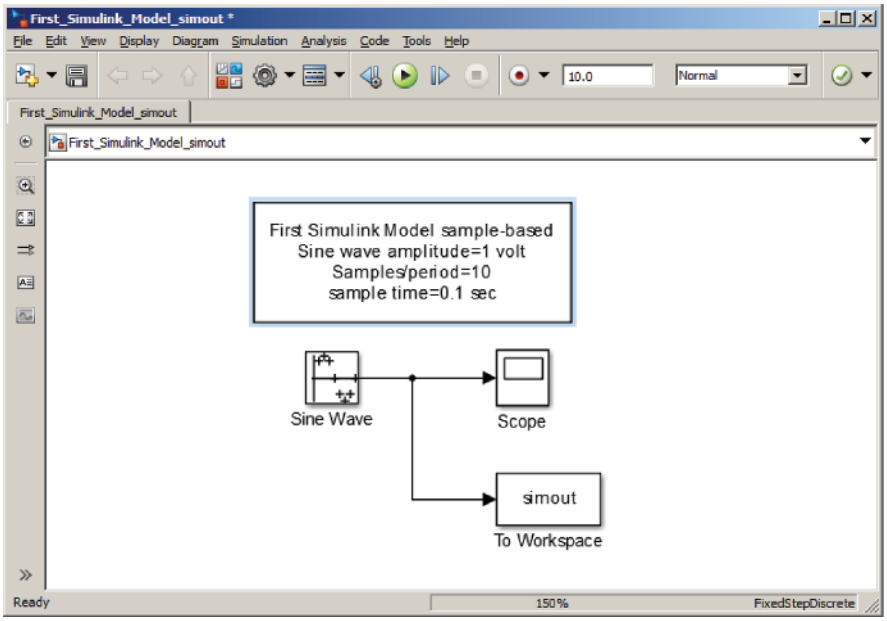

Figure 1.22 Sending Simulation Data to Workspace.

1.8 SENDING DATA TO WORKSPACE

In Figure 1.22, **First Simulink Model-sample-based** has been augmented by adding the **To Workspace** block from **Simulink Sinks** while

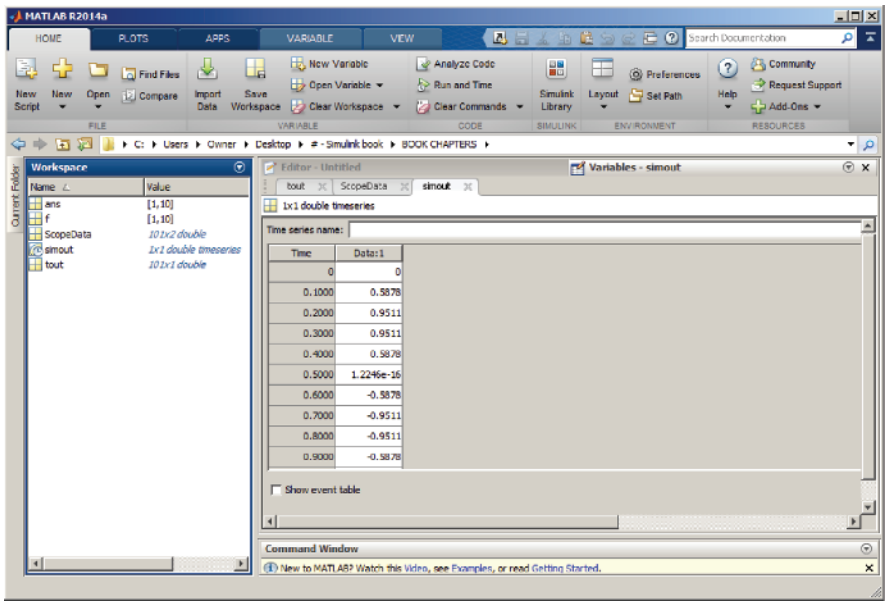

Figure 1.23 Partial Output for the Variable simout.

retaining sample based computation; the model has been renamed by saving it as **First_Simulink_Model_simout**. The **To Workspace** block causes the output data from the **Sine Wave** block to be saved and examined for subsequent use such as plotting. The output block is labeled here as **simout**, a label that can be changed after first double clicking on the block. After running the model, the simulation results are retrieved by clicking on the variable **simout** in the MATLAB Workspace window. Figure 1.23 shows partial output data for the simulation, where both the time increments and sampled sine wave data values are displayed.

1.9 USING MODEL EXPLORER

Model Explorer is a tool available to provide the user with the ability to view, modify or add elements in the Simulink model and **workspace variable**s. Refer to the MathWorks documentation for further information on the use of **Model Explore**r.

To open the **Model Explorer**, select **Model Explorer** under the **View** tab in the Simulink model window. Figure 1.24 shows the window displayed

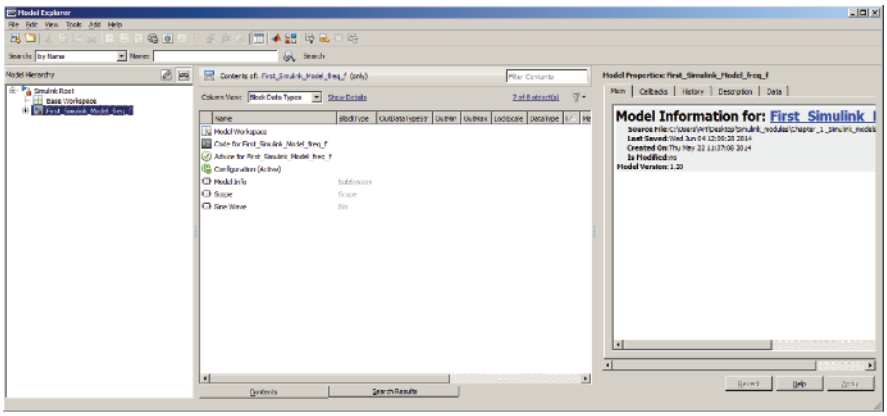

Figure 1.24 Model Explorer Window for First_Simulink_Model_freq_f.

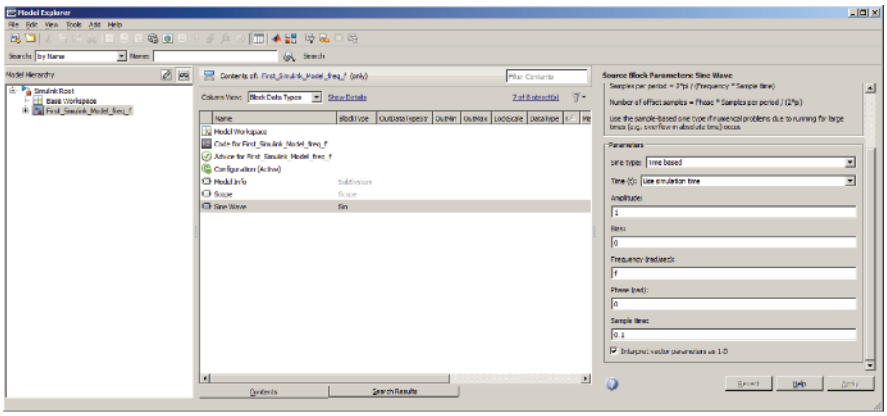

Figure 1.25 Model Explorer Showing Sine Wave Block Information.

after performing this selection from the **First_Simulink_Model_freq_f** model window of Figure 1.16. The menu in the center panel lists all the blocks in the selected model.

By clicking on any model block in the list, the user can display information about that block. For example, Figure 1.25 displays information for the **Sine Wave** block, where the frequency parameter **f** is seen.

This provides another avenue for reconfiguring the **Sine Wave** block to generate multiple frequencies, discussed earlier. That is, the **First_Simulink_Model_freq_f** could have been modified in the **Model Explorer** window by setting **frequency (rad/s)** to **[1 10]**. To accomplish the task of setting the frequency parameter f, open the **Callbacks** tab in the

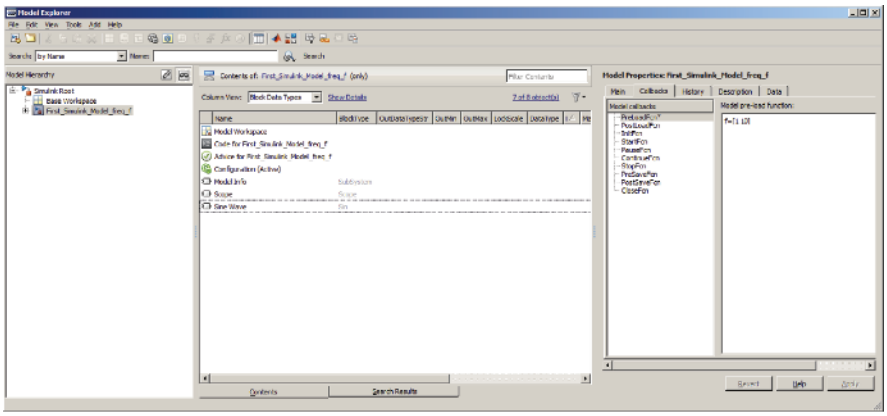

Figure 1.26 Frequency Parameter f set in Model Explorer Callbacks Preloadfcn*.

right-hand panel of the **Model Explorer** window, and in the **PreLoadFcn*** enter **f = [1 10]** as seen in Figure 1.26.

1.10 ADDING LABELS TO FIGURES

The Simulink **scope** block does not support the manipulation of graphics properties in **scope** displays. To add labels to Figure 1.17 the following snippet of Matlab code is used[3]:

```
shh = get(0,'ShowHiddenHandles');
set(0,'ShowHiddenHandles','On');
set(gcf,'menubar','figure');
set(gcf,'CloseRequestFcn','closereq');
set(gcf,'DefaultLineClipping','Off');
set(0,'ShowHiddenHandles',shh);
```

Entering this snippet in the the MATLAB Command Window displays an **Edit** tab in the figure. Selecting **Figure Properties** from the **Edit** tab produces the window shown in Figure 1.27. Changes can now be made to colors, style, axis properties, and labels.

After adding labels, inserting a title, and changing colors, the revised figure is displayed in Figure 1.28.

[3]This snippet of MATLAB code was provided by the MathWorks technical staff.

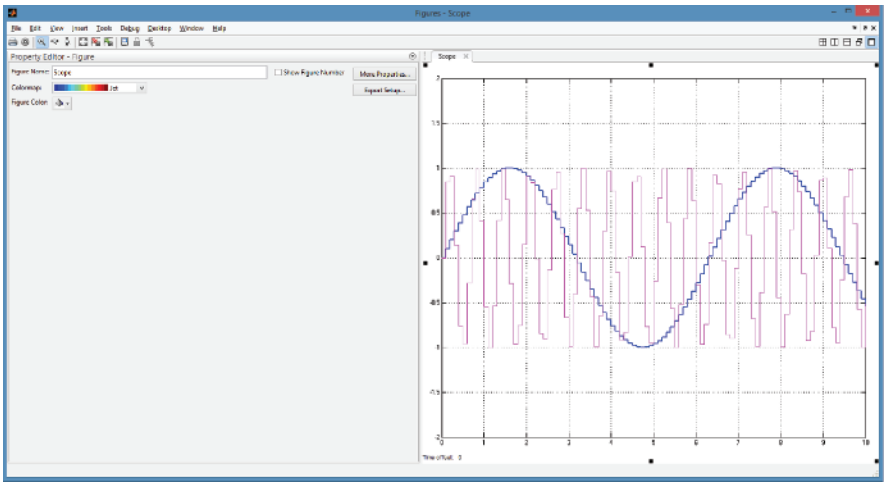

Figure 1.27 Figure Properties Window.

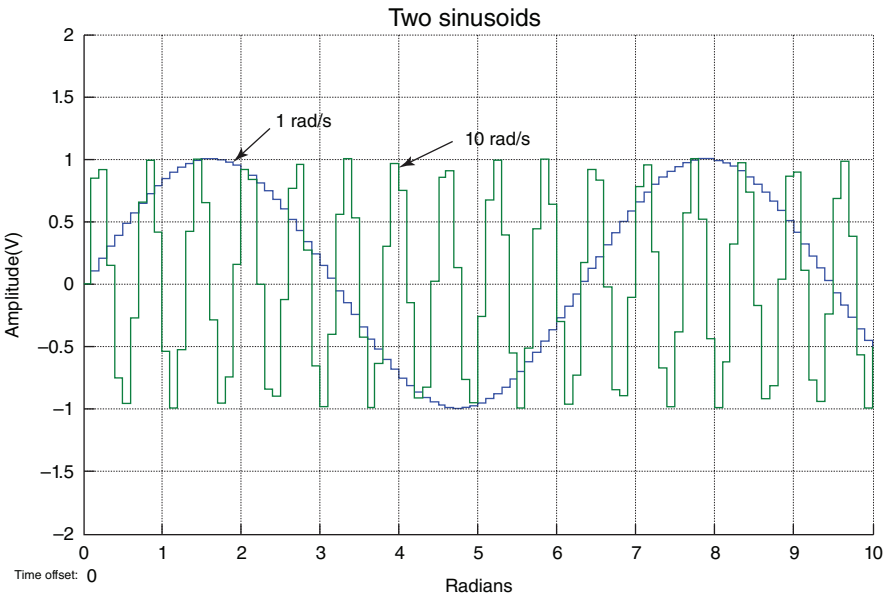

Figure 1.28 Relabeled Scope Output Display.

1.11 SELECTING MODEL CONFIGURATION PARAMETERS

In the Simulink model window, pulling down the **Simulation** tab and selecting **Model Configuration Parameters** opens a window where the user

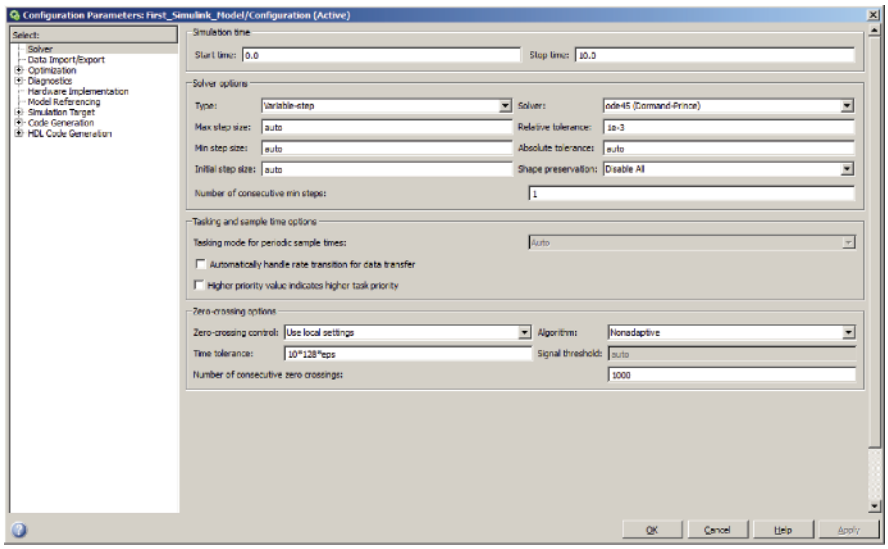

Figure 1.29 Model Configuration Parameters for First Simulink Model.

can specify the simulation start and stop time and choose the solver for the simulation[4]. Figure 1.29 shows the **Model Configuration Parameters** for **First_Simulink_Model** where the **Type** of solver is set to **Variable-step** and the **Solver** is selected as **ode-45(Dormand-Prince**), which, in general is the best first choice as a solver for most Simulink models. By clicking on the **Help** button in the **Model Configuration Parameters** window, the MathWorks documentation describes several aspects of this window including:

- Solver choices
- Simulation and clock time are not the same
- Fixed-step and variable-step size
- Shortened simulation time with variable-step solver

For both fixed-step and variable-step solvers, the next simulation time is the sum of the current simulation time and the step size. Using a fixed-step solver, the step size remains constant throughout the simulation whereas use of a variable-step solver allows the step size to vary from step to step in accordance with the specified error tolerance.

[4]Simulink provides a variety of solvers, each appropriate for running a particular type of simulation model. Detailed discussions of the various solvers can be found in Dabney and Harmon, and in Jamshidi, Farzad and Pedar, cited in Appendix B.

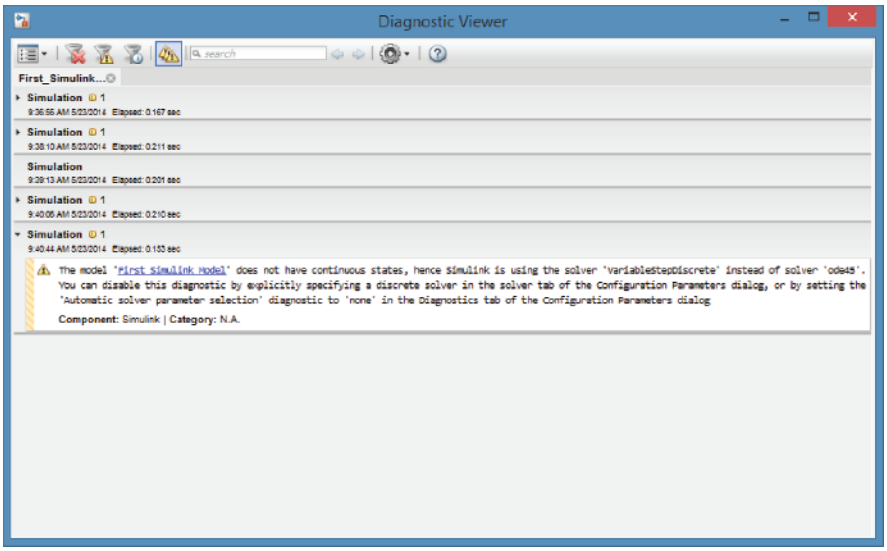

Figure 1.30 Error Diagnostic Caused by Variable-Step Solver.

If the **First_Simulink_Model** is modified where the **Sine Wave** block is chosen to have a **Sample-based** Sine type and a 0.1 s **Sample time** is entered, the model execution will produce a warning message seen at the bottom of the Simulink model. The error diagnostic explains the problem as seen in Figure 1.30.[5]

In Figure 1.31, the **Solver** is changed to **Fixed-step** in the **Solver type** tab and discrete in the **Solver** tab so that no error message is generated.

1.12 SUMMARY DISCUSSION

This chapter has presented a brief introduction to Simulink by demonstrating the basic steps to be taken in constructing a simple simulation model. The MathWorks documentation provides a comprehensive treatment of each block and available tools. The remaining chapters focus on the use of Simulink for modeling digital communications systems, without delving deeply into aspects of the full Simulink capability. Other references that the user is likely to find helpful are listed in Appendix B.

[5]Error diagnostics are often generated when a simulation is first developed. Care must be taken by the user in that the error message identifies a problem but the solution may reside in a block other than the highlighted block. The Simulink toolbar has a check button that can invoke the Model Advisor to help the user correct the problem.

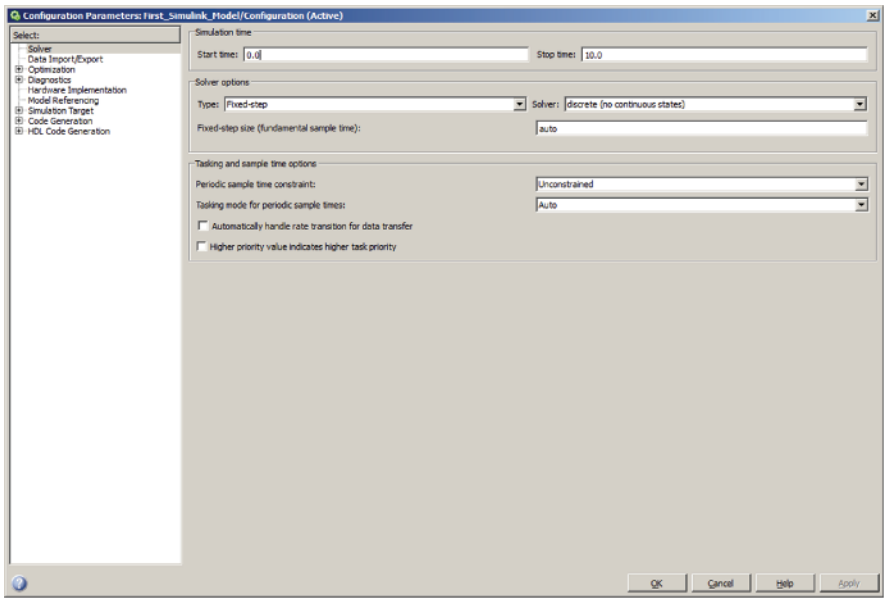

Figure 1.31 Model Configuration Parameters with Solver Changed to Fixed-Step Discrete.

PROBLEMS

1.1 Modify the Simulink model in Figure 1.12 to produce two sinusoidal waves with the following parameters:

frequency = 1 rad/s for both waves
amplitude = 1 V for both waves
phase = 0 for one wave and $\pi/2$ for the second wave
sample time = 0.01 s

a. Show the Simulink Model and include an information block.

b. Display each wave on a separate trace in the scope and label all axes.

Hint: Find the demux block in the Simulink library.

1.2 Let $x(t) = \frac{4}{\pi}\left[\sin(t) + \frac{1}{3}\sin(3t) + \frac{1}{5}\sin(5t)\right]$.

a. Develop a Simulink model for $x(t)$ with an included information block. Assume a 10 s simulation time.

b. Display $x(t)$ in a scope over the range 0 to 2 π with labels.

c. Modify the Simulink model obtained in part a by overlaying a square wave that is +1 between 0 and π and −1 from π to 2π and repeats thereafter.

d. Display the overlay result in a scope over the range 0 to 2 π with labels.

Note that $x(t)$ represents the first three terms of the Fourier series of the square wave.

1.3 Amplitude Modulation (AM) with a tone modulator having a unity modulation index is expressed as

$$x(t) = (1 + \cos(t))\cos(20t)$$

a. Develop a Simulink model for $x(t)$ with an included information block. Use a 10 s simulation time and Goto and From routing blocks from Signal Routing to simplify the model.

b. Display $x(t)$ and $\cos(t)$ on a scope with labeled axes.

c. From the Simulink library, add an AM modulation block to the simulation and form the difference between $x(t)$ and the output of the AM library block.

d. Display $x(t)$, $\cos(t)$, the AM block output and the difference on a scope with four traces. Insert x axis title on bottom trace only; do not label y-axis but add a title to each plot.

1.4 Develop a Simulink model with a sine wave input that feeds both a double-sideband (DSB) AM block and a quantizer followed by a DSB AM block. Assume a 2 s simulation and sine wave block parameters as follows:

Sine wave amplitude = 2, Frequency = 20π rad/s, Sample time = 0.001 s.

For the DSB AM block, assume that the parameters are:

input signal offset = 1, carrier frequency = 100, initial phase = 0

a. Show the model with an included information block.

b. Assume the quantization interval = 0.5 and display the following signals in a scope with 4 traces:

sine wave output, DSB AM output, quantizer/DSB AM output, difference between the DSB AM output and quantizer/DSB AM output.

Provide titles for each trace and label only the x-axis.

c. Repeat part a with a quantization interval = 0.05.