Staffordshire
UNIVERSITY

Matlab, simulink

Building a Simple Model

Satellite Communications
CE74027-3
Tutor: A. El-HELW
Email: elhelw@staffs.ac.uk

Building a Simple Model

This section explains how to build a simple Simulink model that displays a sine wave in a scope. For more detailed information on building models, see the Simulink documentation.

The Basic Steps

This section describes the basic steps in building a model. It explains how to

- Set simulation parameters with commstartup
- Open a new model window
- Open block libraries
- Move blocks into a model window
- Connect the blocks
- Set block parameters
- Set simulation parameters
- Run the model

Building a model usually involves several iterations, as you decide which blocks to include and what parameter settings to make. In the example in this section, you will refine the model by adding noise. The section explains how to

- Add noise to the model
- Save the model

The section also explains

- Frames and frame-based processing
- Discrete signals and sample times
- Continuous signals

Using commstartup to Set Simulation Parameters

Before starting to build the model, enter

```
commstartup
```

at the MATLAB prompt.

This

- Sets the Simulink **Boolean logic signals** parameter to `Off`
- Sets default simulation parameters that are optimal for communications models

The Communications Blockset does not support signals with `boolean` data types. If you want to use Simulink blocks that output `boolean` data types, such as the Logical Operator block, in a model with blocks from the Communications Blockset, enter `commstartup` before building the model. The `commstartup` settings apply to any models you create during the current MATLAB session. You must enter `commstartup` at the beginning of each MATLAB session to establish these settings.

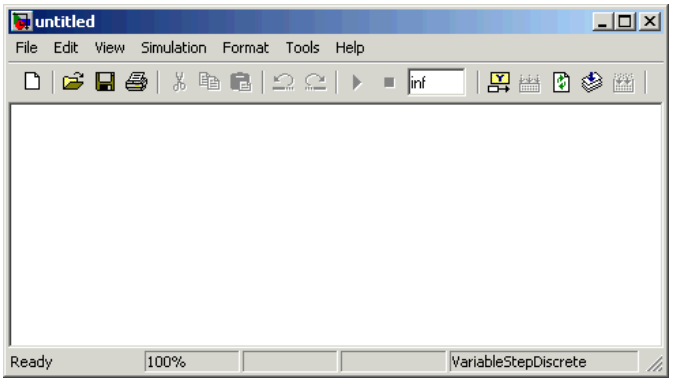
If you build a model without entering `commstartup` and subsequently decide to use Simulink blocks that output signals with `boolean` data types, turn off the model's **Boolean logic signals** parameter by entering

```
set_param('my_model', 'BooleanDataType', 'off')
```

where `my_model.mdl` is the name of the model.

Opening a New Model Window

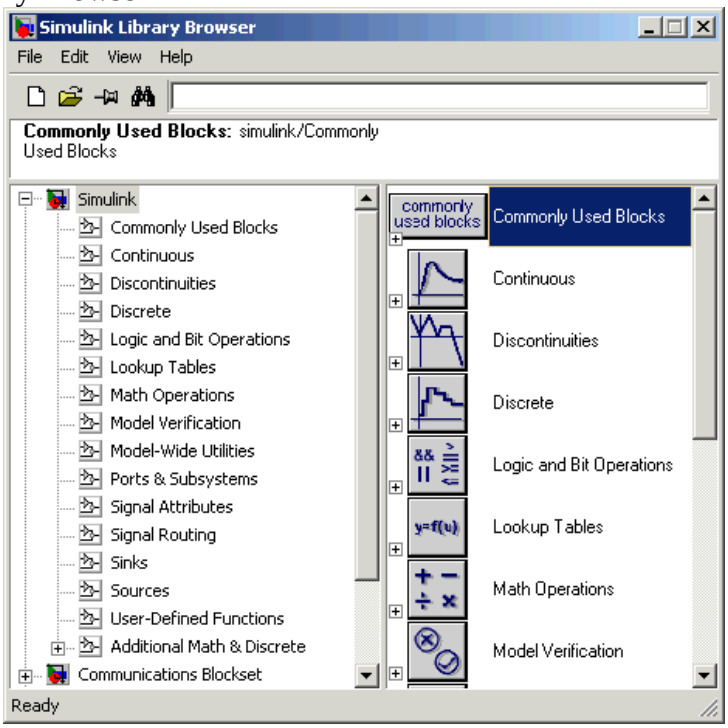
The first step in building a model is to open a new model window. To do so, select **New** from the **File** menu, and then select **Model**. This opens an empty model window, as shown in the following figure.



Opening Block Libraries

The next step is to select the blocks for the model. These blocks are contained in libraries. To view the libraries for the products you have installed, type `simulink` at the MATLAB prompt (or, on Microsoft Windows, click the Simulink button on the MATLAB toolbar). If you are using Microsoft Windows, this displays the Simulink Library Browser, as shown below.

Simulink Library Browser



The left pane displays the installed products, each of which has its own library of blocks. To open a library, click the + sign next to the name of the blockset in the left pane. This displays the contents of the library in the right-hand pane.

You can find the blocks you need to build models of communication systems in the libraries of the Communications Blockset, the Signal Processing Blockset, and Simulink.

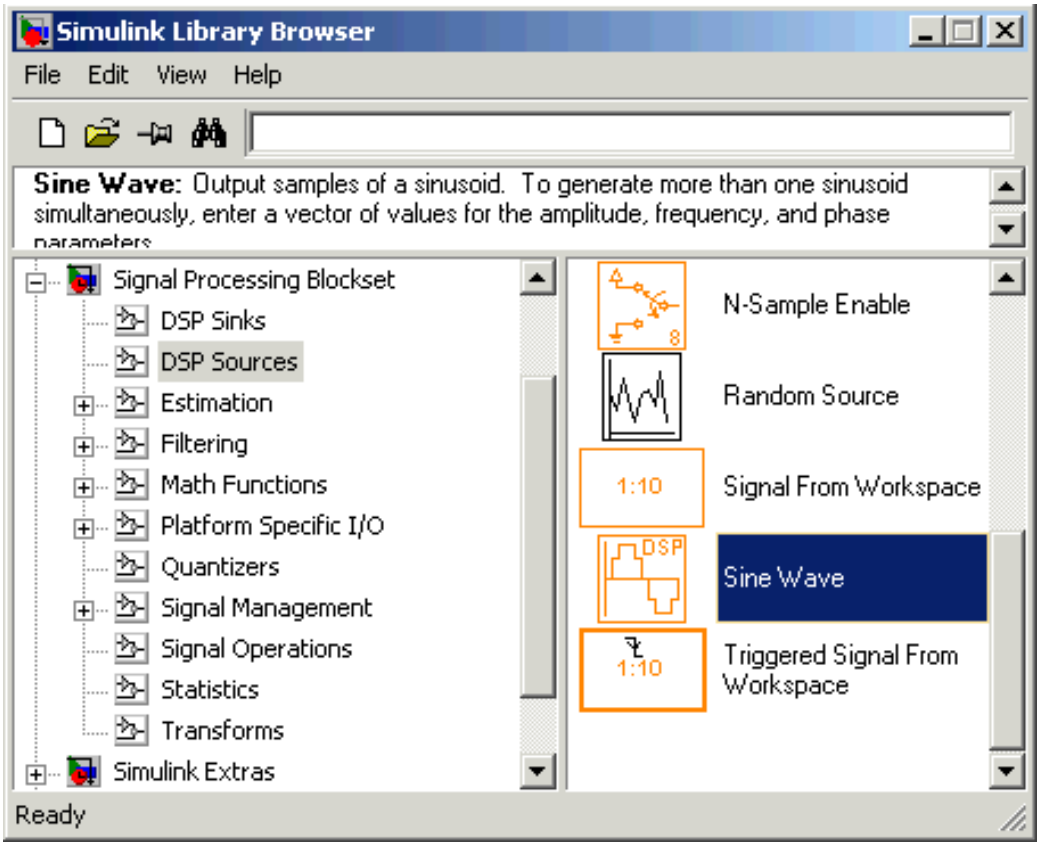
Moving Blocks into the Model Window

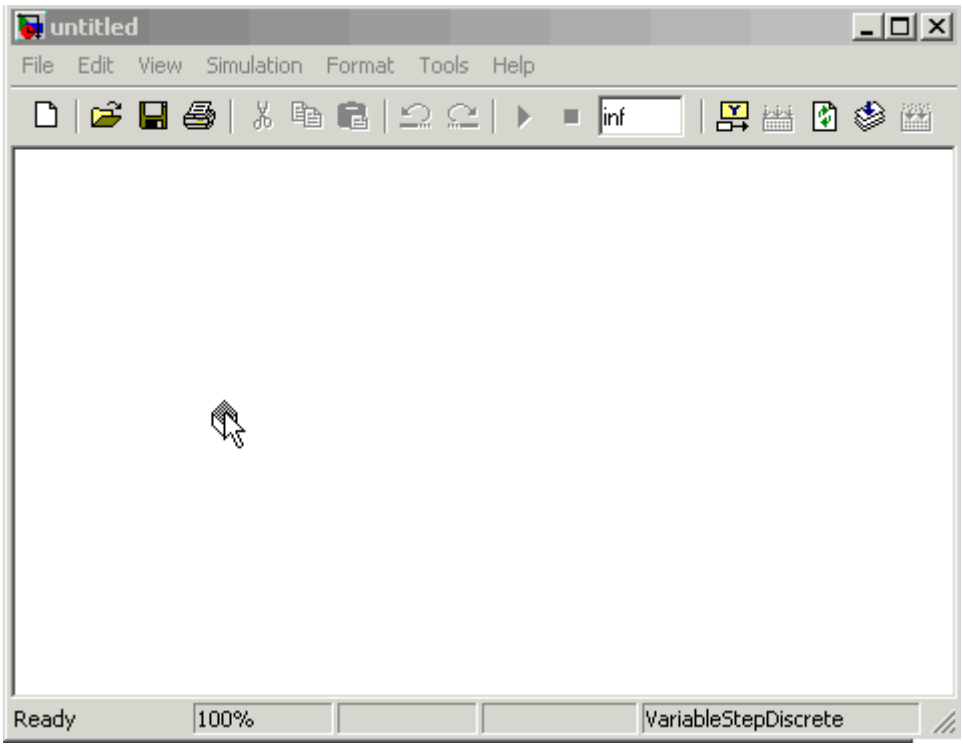
The next step in building the model is to move blocks from the Simulink Library Browser into the model window. To do so,

1. Click the **+** sign next to **Signal Processing Blockset** in the left pane of the Library Browser. This displays a list of the Signal Processing Blockset libraries.
2. Click **DSP Sources** in the left pane. This displays a list of the DSP Sources library blocks in the right pane. If you do not see the Sine Wave block, scroll down the list until it is visible.
3. Click the Sine Wave block and drag it into the model window.
4. Click **DSP Sinks** in the left pane of the Library Browser.
5. Scroll down in the right pane of the Library Browser until you see the Vector Scope block, and drag the block into the model window to the right of the Sine Wave block.

Once a block is in the model window, you can move it to another position by dragging the block with the mouse.

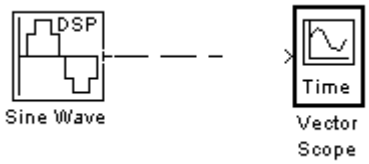
Dragging a Block into a Model Window



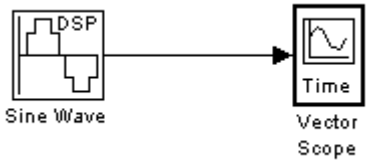


Connecting Blocks

The small arrowhead pointing outward from the right side of the Sine Wave block is an output port for the data the block generates. The arrowhead pointing inward on the Vector Scope block is an input port. To connect the two blocks, click the output port of the Sine Wave block and drag the mouse toward the input port of the Vector Scope block, as shown in the following figure.



When the pointer is on the input port of the Vector Scope block, release the mouse button. You should see a solid arrow appear, as in the following figure.



If you do not see a solid arrowhead, you have not made a connection. In this case, click the arrowhead again, drag it all the way to the Vector Scope's input port, and release the mouse button.

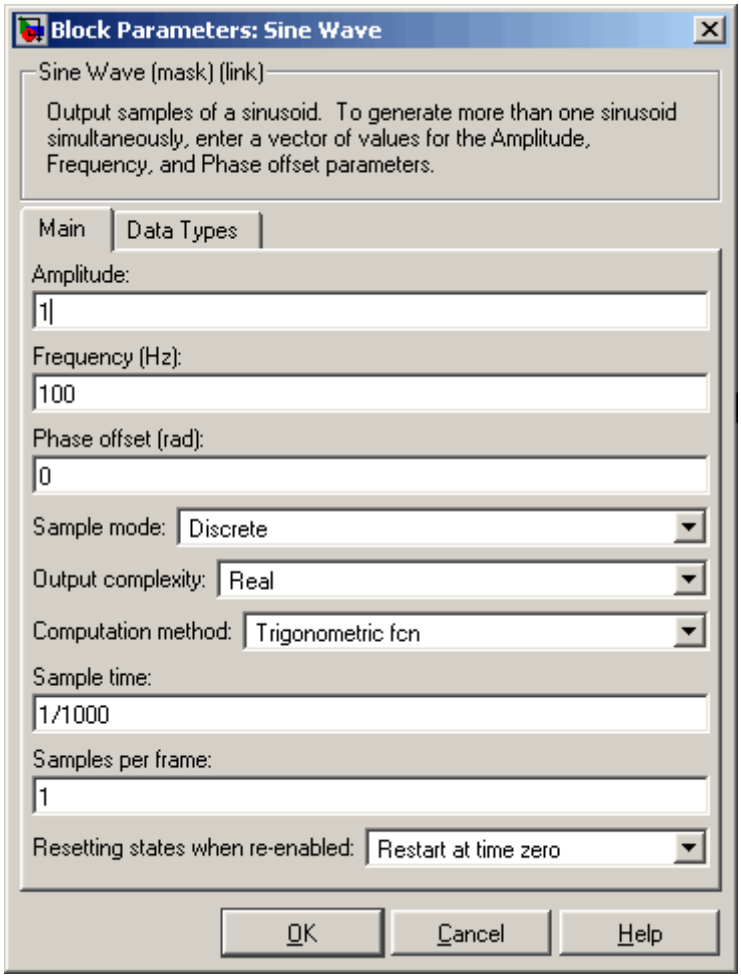
Setting Block Parameters

To set parameters for the Sine Wave block, double-click the block to open its dialog, as shown in the following figure. Change the following parameters by clicking in the field next to the parameter, deleting the default setting, and entering the new setting in its place:

1. Set **Amplitude** to `5`.
2. Set **Frequency** to `30`.
3. Set **Samples per frame** to `100`.
4. Click **OK**.

> **Note** You must set **Samples per frame** to a value larger than `1` to see an image of the sine wave in the Scope block.

Dialog for the Sine Wave Block

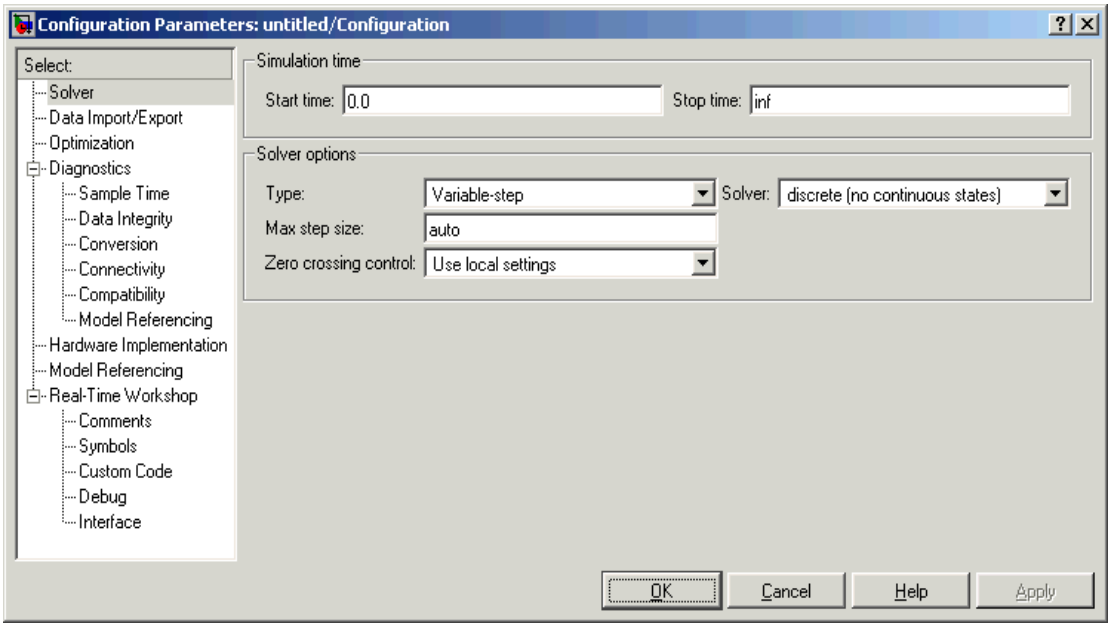


Setting Simulation Parameters

Besides individual block parameters, the model also has overall simulation parameters. To view the current settings,

1. Select the **Simulation** menu at the top of the model window.
2. Select **Configuration parameters** to open the **Configuration Parameters** dialog box, as shown in the following figure.

Configuration Parameters Dialog Box



If you typed commstartup before creating the model, the **Stop time** should be set to inf. The **Stop time** determines the time at which the simulation ends. Setting **Stop time** to inf causes the simulation to run indefinitely, until you stop it by selecting **Stop** from the **Simulation** menu.

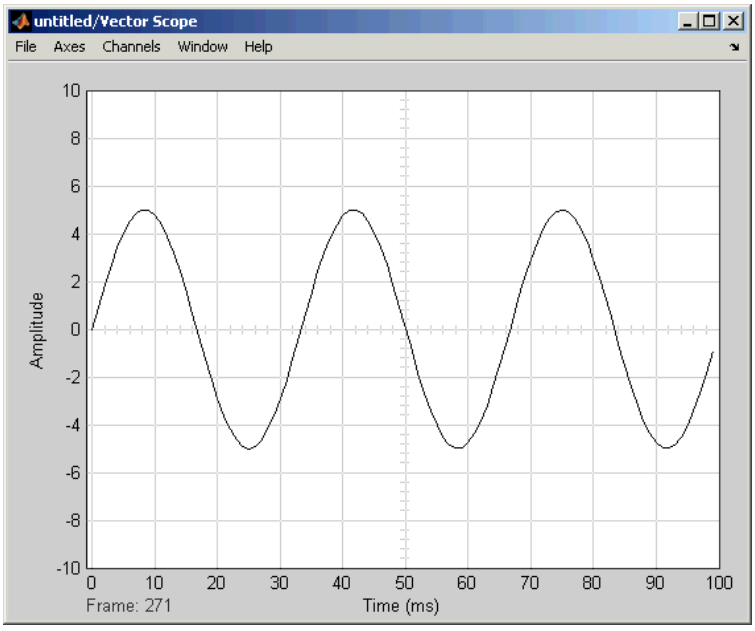
The **Stop time** is not the actual time it takes to run a simulation. The actual run-time for a simulation depends on factors such as the model's complexity and your computer's clock speed.

The settings in the **Configuration Parameters** dialog box affect only the parameters of the current model.

Running the Model

Run the model by selecting **Start** from the **Simulation** menu. When you do so, a scope window appears, displaying a sine wave as shown in the following figure.

Sine Wave Displayed in a Scope

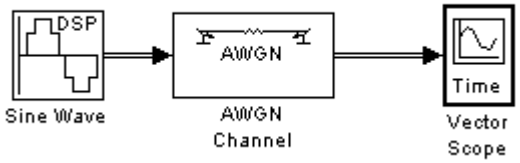


Adding Noise to the Model

You can add noise to the model using the AWGN Channel block, from the Channels library of the Communications Blockset. The block adds white Gaussian noise to the sine wave. Move the block from the Simulink Library Browser into the model window, as described in Moving Blocks into the Model Window. You can add the block to the model as follows:

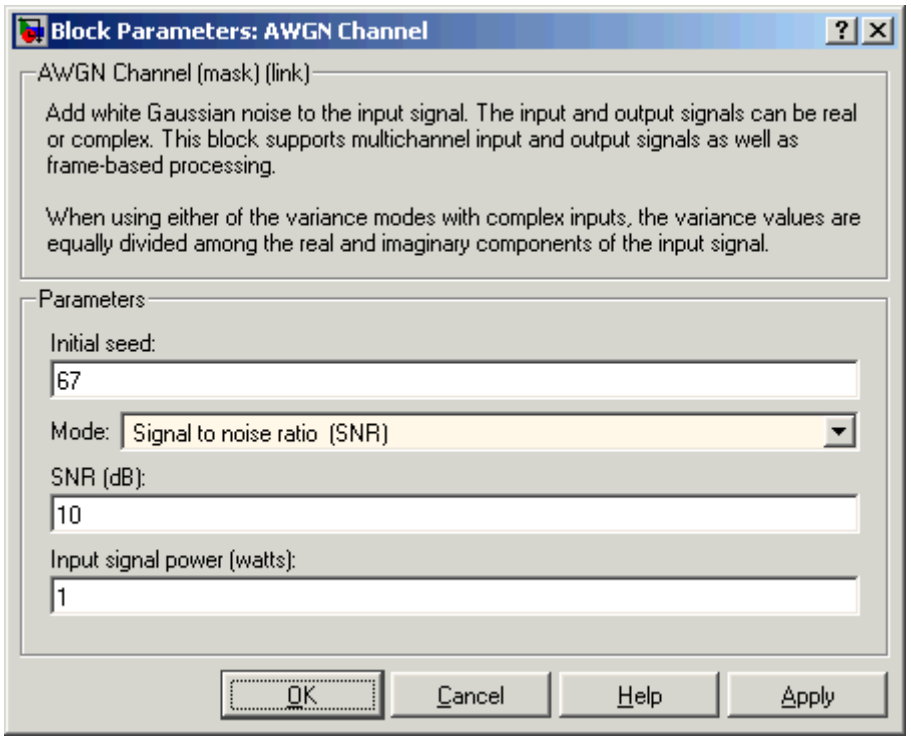
1. Extend the line between the Sine Wave block and the Vector Scope block by dragging the Vector Scope block to the right, to make room for the AWGN Channel block.
2. Click the AWGN block and drag it onto the line. This automatically connects the Sine Wave block and the Vector Scope block to the AWGN Channel block.

   Sine Wave Plus Noise



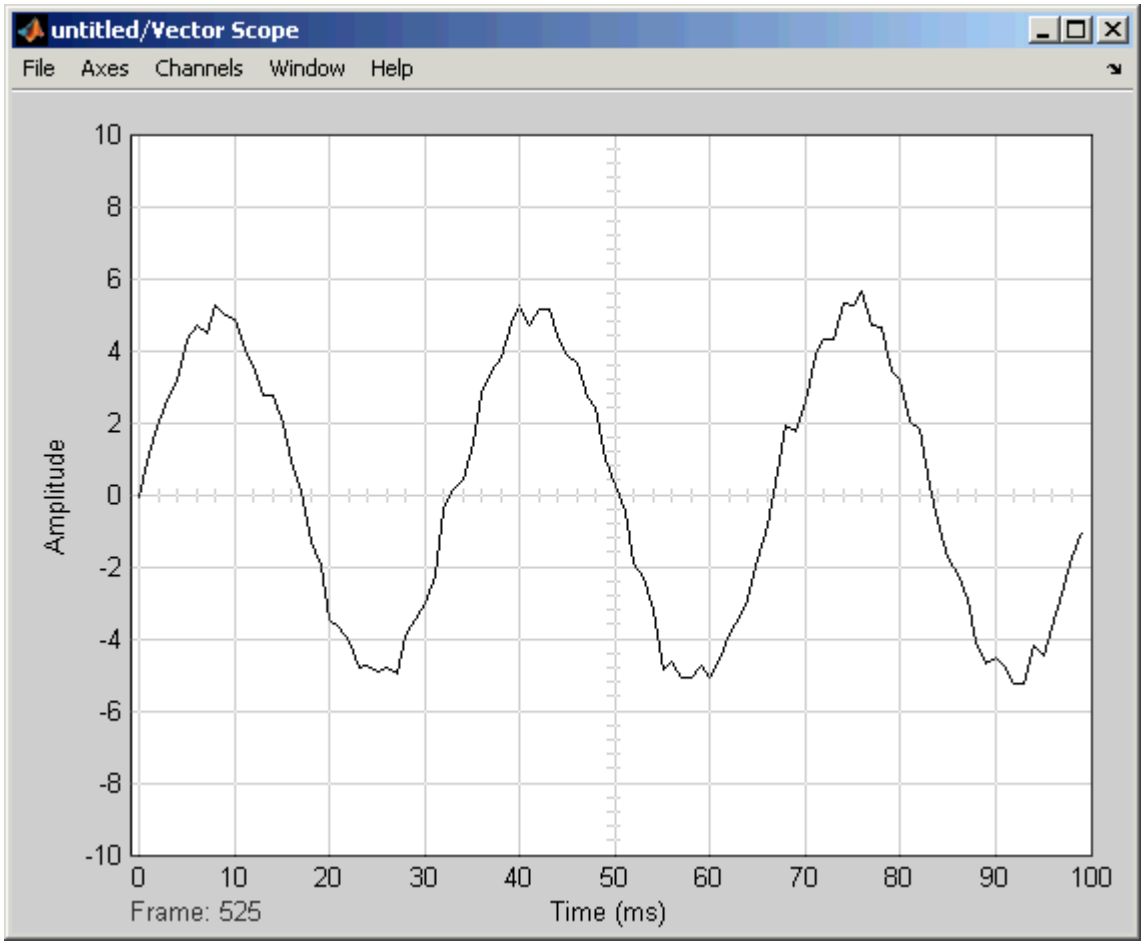
Double-click the AWGN Channel block to open its dialog, as shown in the following figure. Then, click the down arrow in the **Mode** field and select `Signal to noise ratio (SNR)`.

Dialog for the AWGN Channel Block



Now when you run the model, the scope clearly shows the added noise.

Sine Wave with Noise Added



When you are finished observing the simulation, stop the model by selecting **Stop** from the **Simulation** menu.

Saving a Model

To save your model for future use, select **Save** from the **File** menu. The first time you save the model, this displays the **Save As** dialog box. In the **Save in** field, select the directory where you want to save the model. It is best to keep your work files in a separate directory from the files shipped with the product. In the **File name** field, enter a name for the model, such as `sine.mdl`, and click **Save**.

To load the model in a future MATLAB session, first change your working directory to the one where you saved the file. You can do this by selecting the directory in the **Current Directory** field on the MATLAB toolbar. Then enter `sine` in the MATLAB Command Window.

Frames and Frame-Based Processing

A *frame* is a sequence of samples combined into a single vector. By setting **Samples per frame** to `100` in the Sine Wave block, you set the frame size to 100, so that each frame contains 100 samples. This enables the Vector Scope block to display enough data for a good picture of the sine wave.

Another important reason to set the frame size is that many Communications Blockset blocks require their inputs to be vectors of specific sizes. If you connect a source block, such as the

Sine Wave block, to one of these blocks, you can set the input size correctly by setting **Samples per frame** to the required value. The model described in Reducing the Error Rate Using a Hamming Code shows how to do this.

In *frame-based processing* all the samples in a frame are processed simultaneously. In *sample-based processing*, on the other hand, samples are processed one at a time. The advantage of frame-based processing is that it can greatly increase the speed of a simulation. If you see double lines between blocks, the model uses frame-based processing.

Discrete Signals and Sample Times

The Sine Wave block in the Signal Processing Blockset generates a *discrete* signal. This means that it updates the signal at integer multiples of a fixed time interval, called the *sample time*. You can set the length of this time interval in the **Sample time** parameter in the block's dialog. In the example described in Building a Simple Model, the **Sample time** has the default value of `1/1000`. All the sources in the Communications Blockset and the Signal Processing Blockset generate discrete signals exclusively. These sources are primarily designed for modeling digital communication systems.

Continuous Signals

The Simulink libraries also contain blocks that generate *continuous* signals. This means that they update the signal at variable time intervals, whose length is determined by the numerical solver the simulation uses. For example, the Sine Wave block in the Simulink Sources library can generate a continuous sine wave.

Now try to construct this model