Network Simulation

Exercise 1

Faculty of Architecture, Computing and Engineering (FACE)
School of Applied Computing
Mount Pleasant Campus
Swansea
SA1 6ED

Module Leader: Dr Carlene Campbell

Introduction to Simulink
Archie Watt

Introduction to Simulink

Getting Started

Begin by opening Matlab; by default this will start with the command window.

Set Simulation Parameters

Type **"simulink"** in the command window and press ENTER to get started.

This will open the **Simulink Library Browser**, as shown in Figure 1.

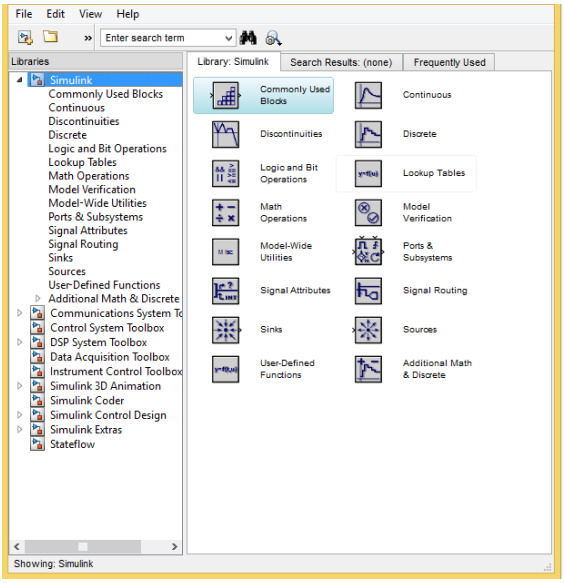

Figure 1 - Simulink Library Browser

This provides access to all the building blocks available for use in Simulink. A Simulink model essentially consists of a collection of connected blocks, which are representative of a system.

Elements in Simulink

Building elements in Simulink can be divided into two major classes: **blocks** and **lines**. Essentially, blocks are used generate, modify, combine, output and display signals, whilst lines are used to transfer signals between blocks.

Blocks

Blocks in Simulink can be categorized in several ways:

- A **source** block is used to generate signals.
- A **sink** is used to output/display signals.
- **Discrete** blocks – discrete time elements.
- **Linear** blocks – continuous time system elements.
- **Nonlinear** blocks – nonlinear operators.
- **Connections** – e.g. multiplex, demultiplex.

Lines

Line elements transmit signals from one block to another, in the direction indicated by the arrow. They must always transmit signals from the output of one block to the input of another. The only exception to this is that a line can "tap off" another line, essentially splitting the signal to two destination blocks. However, lines can never inject into another line.

Building & Simulating a Simple Model

This tutorial will introduce the basic steps involved with building a model in Simulink, and additionally will cover building a simple model to display a sine wave in a scope.

Creating a New Model

You can create a new Simulink model by clicking **New > Simulink Model** in the top menu bar, as illustrated in Figure 2.

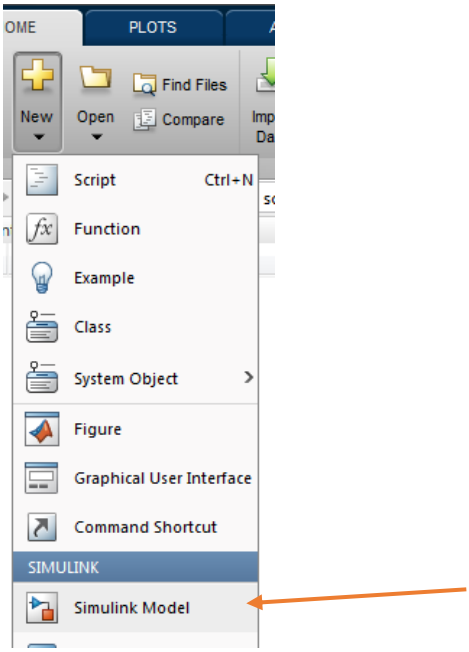

Figure 2 - Create a New Model

Once you've created your Simulink model, there are four basic steps to building your simulation model:

1. Gather the necessary blocks from the Simulink Library Browser (refer to Figure 1)
2. Configure the block (and simulation) parameters as necessary.
3. Connect the blocks together with lines.
4. Simulate the complete system to verify correct operation.

Insert Blocks into Model

The Simulink Library contains all the blocks that are available for building your simulation models. These are organized by the "toolboxes" that are available, and therefore these may vary depending on your license/installation. For communication system modelling, the libraries we need are **Simulink, Communications, and DSP** blocksets. To find the blocks you need, you can either look through the contents of the library or search by keyword.

For this initial simulation, we need two blocks: a sine wave, and a vector scope, both of which can be found in the DSP blockset library. The sine wave is located in **DSP Sources**, the vector scope is located in **DSP Sinks**.

To insert a block into a model window, simply click & hold the mouse button on the appropriate block, drag it into the model window, and then release the button.

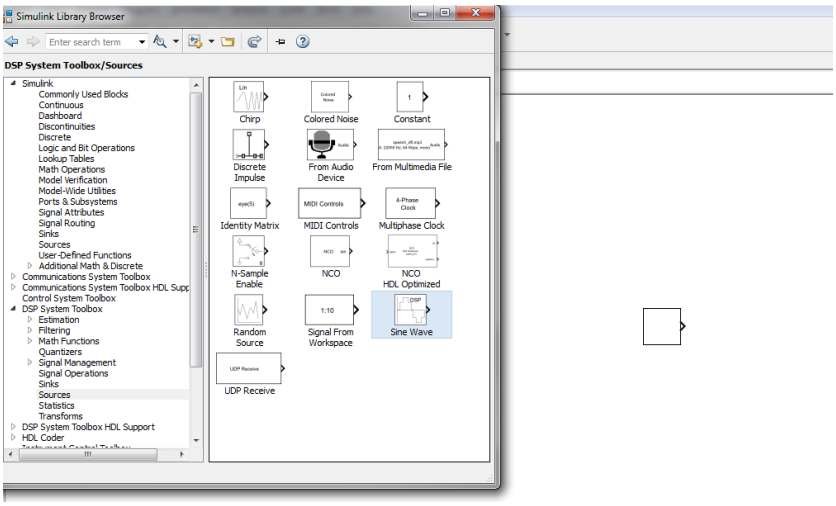

Figure 3 - Adding a Block

Your model window should now reflect Figure 4.

Figure 4 - Sine Wave & Vector Scope

Setting Block Parameters

Block parameters can be set by double clicking on the block in question and entering the desired values in the appropriate data fields.

Change the following parameters for the sine wave block and then click **OK**:

- Amplitude: **5**
- Frequency: **30**
- Samples per frame: **100**

Connecting Blocks

Source blocks have a small arrowhead pointing outwards to the right; this is an output port for the data generated by the block. Sink blocks, on the other hand, will have input ports that are on the left of the block and point inwards. To connect the sine wave with the vector scope, click and hold the mouse on the output port of the sine wave block and drag the line to the input port of the vector scope. Your model should now reflect Figure 5.

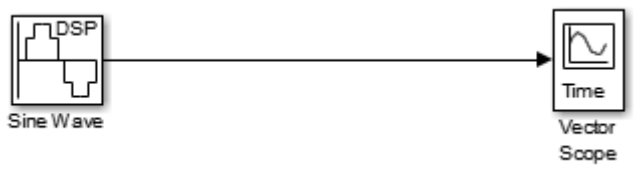

Figure 5 - Connected Blocks

Setting Simulation Parameters

Simulation parameters relates to the settings that control the whole simulation, for example the simulation run time, rather than individual blocks. The optimal settings for communication systems modelling can be set by issuing the **commstartup** command immediately after launching Matlab.

The simulation parameters can be manually altered by selecting **Simulation > Model Configuration Parameters** or by issuing the **CTRL & E** command from the Simulink model window.

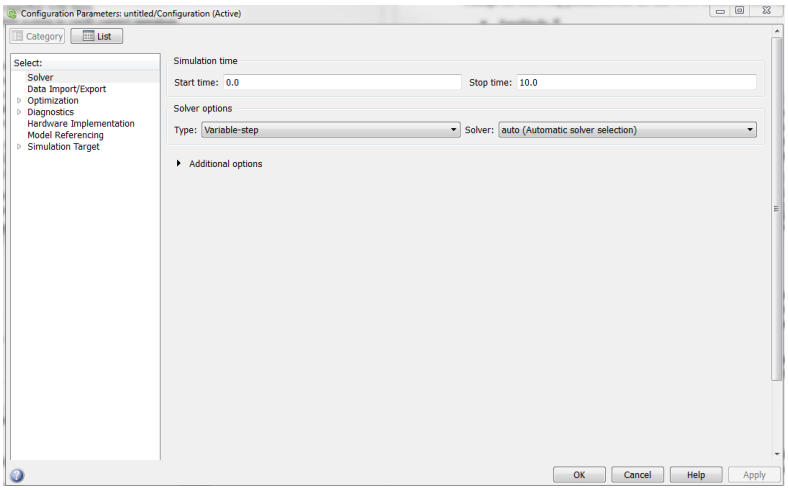

Figure 6 - Simulation Parameters

Running the Simulation

Once you have configured all the necessary parameters, you can run the simulation to view the results. To do this, click the **Play** icon in the top taskbar (see Figure 7).

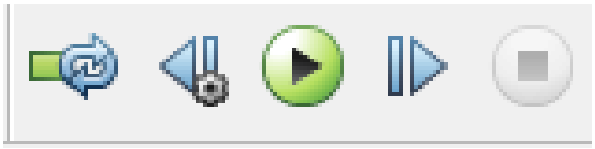

Figure 7 - Run Simulation

Depending on the simulation parameters you have configured (for example, if you have configured the simulation to run infinitely), you may have to manually stop the simulation.

Once the simulation is complete you should be able to see your sine wave displayed in the vector scope, as shown in Figure 8.

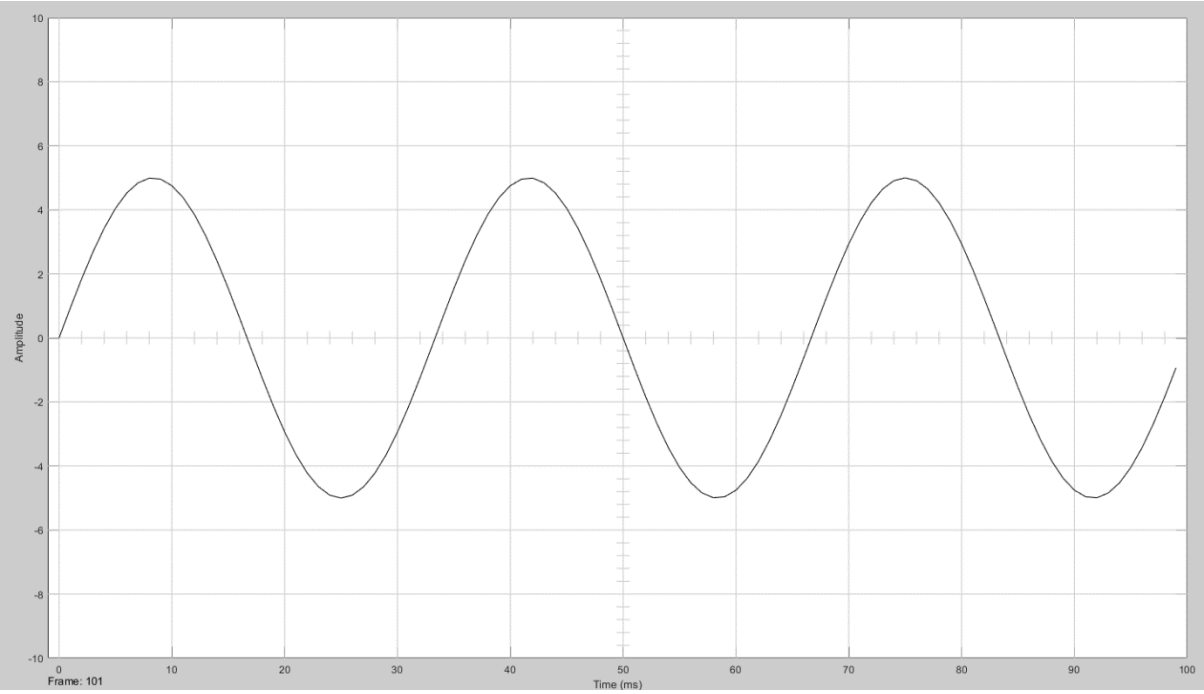

Figure 8 - Sine Wave

Note: if you do not see the sine wave displayed, double check that you have set the 'Samples per frame' value correctly.

Building a simulation model generally involves multiple iterations; the results that you generate should indicate how you can refine the parameters you have configured. In this instance, we will add Additive White Gaussian Noise (AWGN) to the sine wave to observe the effects. You are further encouraged to change parameters (such as amplitude, frequency) on your sine wave block and observe the effects.

Adding Noise to a Model

Noise can be simply defined as an undesired disturbance of a signal that contains useful information. In this case, we will use Additive White Gaussian Noise (AWGN). This can be done by adding the "AWGN Channel" block in between the already inserted sine wave and vector scope blocks, as shown below.

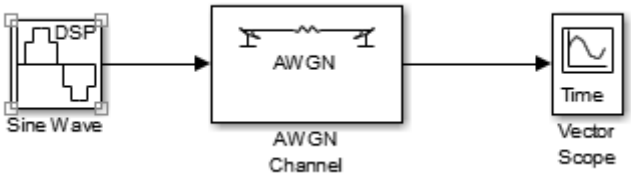

Figure 9 - Simulink Model with AWGN Block

Double click on the AWGN block and, using the drop down menu next to **Mode**, select **Signal to noise ratio (SNR),** as shown in Figure 10, and then click **OK**.

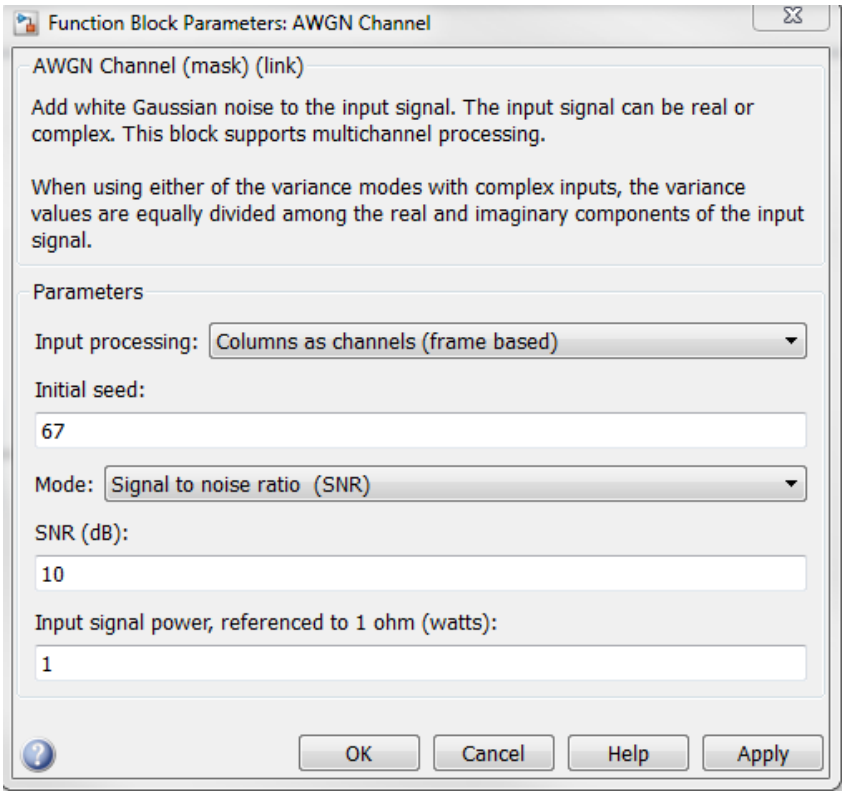

Figure 10 - AWGN Parameters

Now run the simulation again, and observe the changes in the sine wave that the scope shows. You are encouraged to experiment further by introducing varying levels of noise by modifying the signal to noise ratio (SNR) value in the AWGN block parameters, and observing the effects that are displayed. You may need to conduct some research on noise in communication systems to explain these effects.

Saving a Model

You can save your model the standard way by using **File > Save**.