EE 224 - Laboratory 4: Using Simulink for Frequency Analysis and Sampling and Reconstruction of Continuous-Time Signals

1 Introduction

In this experiment, we will use Fourier series and Fourier transforms to analyze continuous- time and discrete-time signals and systems. The Fourier representations of signals involve the decomposition of the signal in terms of complex exponential functions. These decompositions are very important in the analysis of linear time-invariant (LTI) systems, due to the property that the response of an LTI system to a complex exponential input is a complex exponential of the same frequency! Only the amplitude and phase of the input signal are changed. Therefore, studying the frequency response of an LTI system gives complete insight into its behavior.

In this experiment and others to follow, we will use the Simulink extension to Matlab. Simulink is an icon-driven dynamic simulation package that allows the user to represent a system or a process by a block diagram. Once the representation is completed, Simulink may be used to digitally simulate the behavior of the continuous or discrete-time system. Simulink inputs can be Matlab variables from the workspace, or waveforms or sequences generated by Simulink itself. These Simulink-generated inputs can represent continuous-time or discrete-time sources. The behavior of the simulated system can be monitored using Simulink's version of common lab instruments, such as scopes, spectrum analyzers and network analyzers.

2. Pre-Lab Getting Started with Simulink

In this section, we will learn the basics of Simulink and build a simple system.

2.1 Help on Simulink

Simulink is an extension of Matlab. It is an icon-driven dynamic simulation package that allows the user to represent a system or a process by a block diagram. To start Simulink and to open the main library block, you need to type the command *simulink;* at the Matlab prompt. To create a new system, first open a window for it by using the New option from the File pull-down menu, and select *Simulink Model*. Then, drag the icons for the components of the system from the libraries into your new system window. You can also create duplicates of your components by using the Copy and Paste options from the Edit pull-down menu. The inputs can be Matlab variables from the workspace, waveforms or sequences generated by continuous-time or discrete-time sources. The components can be linear or nonlinear. You can connect any two components of your system by left-clicking on the output of one and dragging it to the input of the other.

After your system is complete and ready for simulation, start the simulation by using the Start option from the Simulation pull-down menu. Use the Stop option to stop the simulation. You can change the simulation parameters using the Configuration Parameters option. You can monitor the behavior of the simulated system using instruments like scopes, spectrum and network analyzers, and graph windows. The outputs of a system can be connected to one of the monitoring devices listed above, or they may be saved to the Matlab workspace.

The Simulink systems can be saved using the Save or Save As option from the File pull-down menu. Matlab creates a script file with the filename you specified and extension .mdl for each system you save. You can bring up the Simulink systems you formerly created by typing their filenames (without the .mdl extension) at the Matlab prompt.

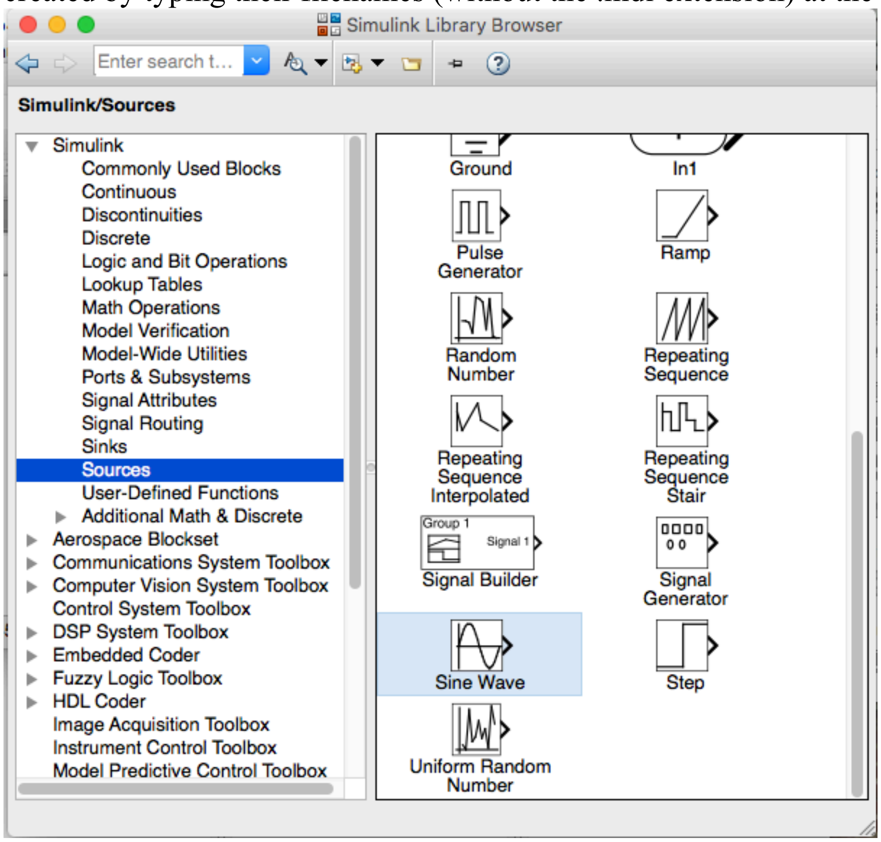


Figure 1. Selecting a block from the Simulink Block Library.

2.2 Introductory Model

In order to familiarize yourself with Simulink, you will first build the system shown in Fig. 2. This system consists of a sine wave generator that feeds a scope and a spectrum analyzer.

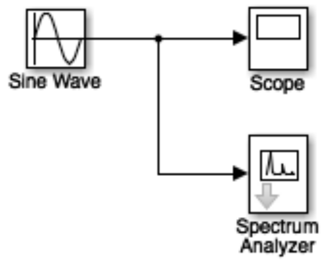


Figure 2: Simulink model for the introductory example.

The blocks for this model are in the basic Simulink and DSP System Toolbox blocksets. Figure 1 shows how to select the sine wave generator. The Scope is under Simulink > Sinks > Scope. The Spectrum Analyzer is a custom block developed for a lab at Purdue and is in the model file SpectrumAnalyzer. It is in the file SpecAnal.zip which you will need to uncompress and store in your active Matlab directory for this lab. Be sure that this directory is in your path.

1. Open a window for a new system by using the New option from the File pull-down menu, and select Model.
2. Drag the Sine Wave, Scope, and Spectrum Analyzer blocks from the Lab3 window into the new window you created.
3. Now you need to connect these three blocks. With the left mouse button, click on the output of the Sine Wave and drag it to the input of the Scope. Now use the right button to click on the line you just created, and drag to the input of the Spectrum Analyzer block. Your system should now look like Fig. 2.
4. Double click on the Scope block to make the plotting window for the scope appear.
5. Set the simulation parameters by selecting Configuration Parameters from the Simulation pull-down menu. Under the Solver tab, set the Stop time to 50, and the Max step size to 0.02. Then select OK. This will allow the Spectrum Analyzer to make a more accurate calculation.
6. Start the simulation by using the Start option from the Simulation pull-down menu. A standard Matlab figure window will pop up showing the output of the Spectrum Analyzer. What frequency is the sine wave at in the spectrum?
7. Change the frequency of the sine wave to 5*pi rad/sec by double clicking on the Sine Wave icon and changing the number in the Frequency field. Restart the simulation. Observe the change in the waveform and its spectrum. What frequency is the peak at?
8. When you are done, close the system window you created by using the Close option from the File pull-down menu.

3. Lab Instructions

Exercise 1:

a. Derive the Fourier Series components for a square wave with a period of T and a duration that is a percentage of time. Show your derivation and your results. What frequencies should the frequency components occur at for fundamental frequencies of 0.25, 2, and 10 seconds?

3.1. Time Scaling Property

Use Simulink to create the model system given below in Figure 3. Write down the time-scaling property of Fourier Series.

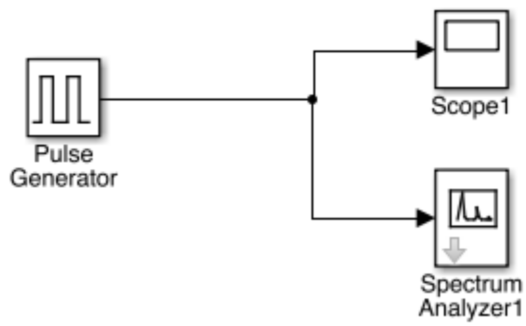


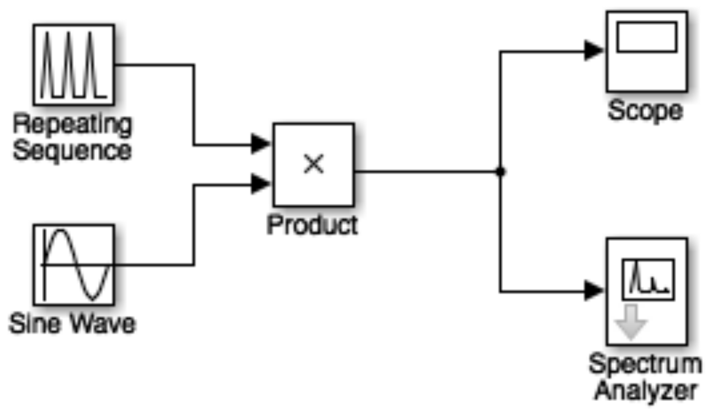
Figure 3. System with a Square Wave Pulse.

1. Set the period of the square wave to 2 and the percentage of the pulse to 25%.
2. Set the Simulation parameters to a length of 50 sec. with a maximum step size of 0.02 sec.
3. Measure the locations of the frequency spikes. Did they occur where you expected them to? Do they look different than the figures seen in class?
4. Repeat the steps above for periods of 0.25 and 10 sec. What happens to the amplitude and locations of the frequency peaks?
5. Do the results that you get correspond to the Time-scaling Property? What is the relationship with the fundamental period in time domain and the locations of the frequency peaks in frequency domain?

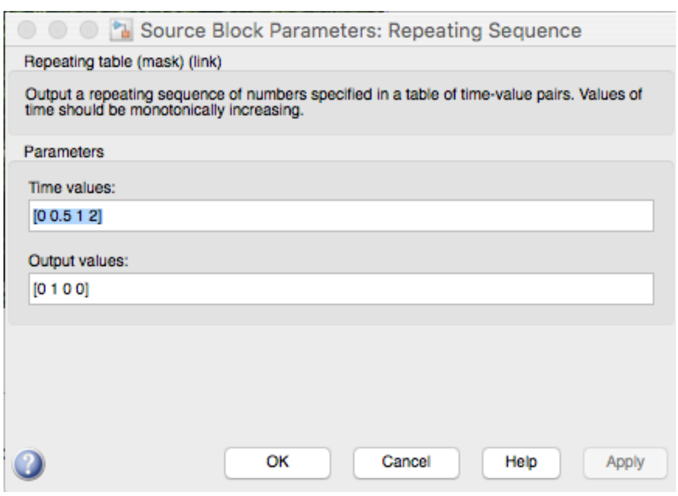
3.2. Modulation Property

The modulation property is used in communication systems to shift a signal in frequency. Modulation occurs when a signal is multiplied by a sinusoid, $m(t) = x(t)\cos(\omega_c t)$. The effect of the modulation is to shift the frequency response of signal x(t) to be centered around frequency ω_c . In this exercise, we will make sure that the fundamental frequencies of the pulse signal and the modulator are harmonically related.

Use Simulink to build the modulation system shown below. This system modulates a triangular pulse signal with a sine wave. You can control the duration and duty cycle of the triangular envelope and the frequency of the modulating sine wave. The system also contains a spectrum analyzer which plots the modulated signal and its spectrum.



Generate the following signals by adjusting the Time values and Output values of the Repeating Sequence block and the Frequency of the Sine Wave. The Time values vector contains entries spanning one period of the repeating signal. The Output values vector contains the values of the repeating signal at the times specified in the Time values vector.



Note: The Repeating Sequence block does NOT create a discrete time signal. It creates a continuous time signal by connecting the output values with line segments. Set the simulation parameters for a duration of 50 sec. with a maximum step size of 0.02 sec. Print the output of the Spectrum Analyzer for each signal.

1. Triangular pulse duration of 1 sec; period of 2 sec; modulating frequency of 10 Hz (initial settings of the experiment).
2. Triangular pulse duration of 1 sec; period of 2 sec; modulating frequency of 15 Hz.

3. Triangular pulse duration of 1 sec; period of 3 sec; modulating frequency of 10 Hz.
4. Triangular pulse duration of 1 sec; period of 6 sec; modulating frequency of 10 Hz.
Notice that the spectrum of the modulated signal consists of a comb of impulses in the frequency domain, arranged around a center frequency.

IN LAB REPORT:

Hand in plots of the output of the Spectrum Analyzer for each signal. Answer the following questions:

- What effect does changing the modulating frequency have on the spectral density?
- Why does the spectrum have a comb structure and what is the spectral distance between impulses? Why?
- What would happen to the spectral density if the period of the triangle pulse were to increase toward infinity? (in the limit)

Extra Credit:
Derive the modulation property for Fourier Series to explain what you saw above.