School of Engineering Science, Simon Fraser University
ENSC327 Communication Systems – Spring 2018
Assignment 2

Important Notes:

1) Please upload ONE PDF document containing the solutions on Coursys only. Please write clearly; if it is not clear and can't be marked, it will not be marked. No other format for submission (hard copy, email, etc…) is accepted.

2) Please do not share your solutions with your friends. Any solutions found to be the same will be marked zero. The goal is to get everyone to do the work themselves so that each individual benefits by testing their understanding with these questions.

Goals:
1. Understand the theory of AM, DSB, and Superhet receiver.
2. Be introduced to and experience with Simulink.

1. You are asked to design a DSB-SC modulator to generate a modulated signal $km(t)\cos(w_c t + \theta)$ where m(t) is a signal bandlimited to B Hz. Figure 1 shows a DSB-SC modulator available in the stockroom. The carrier generator available generates not $\cos(w_c t)$ but $\cos^3(w_c t)$ Explain whether you would be able to generate the design using only this equipment. You may use any kind of filter you like.

   (CEAB indicators: 1.1: Mathematical Knowledge, 1.4: Discipline - specific Knowledge, 2.1: Problem Identification, 2.2: Problem Formulation, 2.3: Problem Solution)

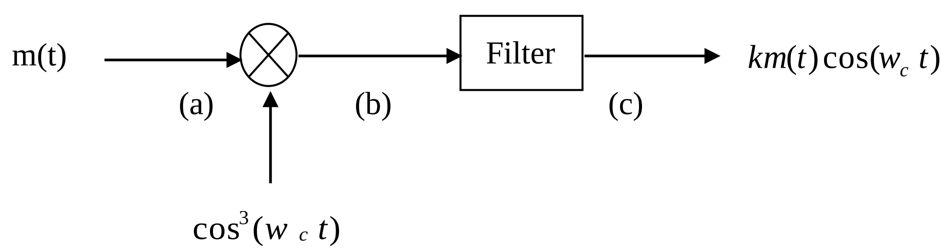


Figure 1 (a).

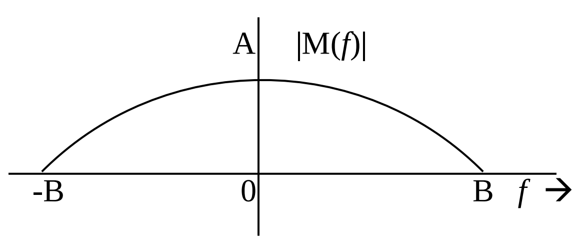
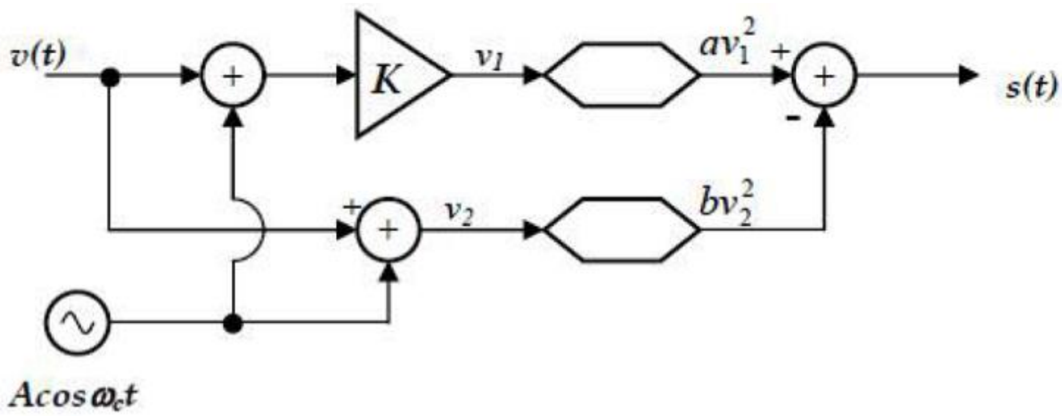


Figure 1 (b).

a). What kind of filter is required in Fig. 1?
b). Determine the signal spectra at points b and c, and indicate the frequency bands occupied by these spectra.
c). What is the minimum usable value of $w_c t$?
d). Would this scheme work in the carrier generator output were $\sin^3(w_c t)$? Explain.
e). Would this scheme work in the carrier generator output were $\cos^n(w_c t)$ for any integer $n \geq 2$?

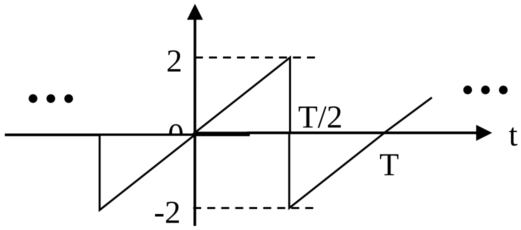
2. Consider the following circuit with an input signal, **v(t)**, an amplifier with gain **K**, a sinusoidal carrier signal at w_c, and two nonlinear devices:



a) Express the output signal, **s(t)** in terms of **v(t)**, **A**, **K**, $\cos(w_c t)$, **a** and **b**.
b) Determine the choice of the gain K so that the above circuit performs as a DSB modulator without output filtering.

(CEAB indicators: 1.1: Mathematical Knowledge, 1.4: Discipline - specific Knowledge, 2.1: Problem Identification, 2.2: Problem Formulation, 2.3: Problem Solution, 2.4: Solution Verification and Evaluation, 3.2: Investigation Design, 4.1: Requirement and Constraint Identification, 4.3: Design Candidate Generation, 4.5: Detailed Design)

3. **(AM Power efficiency)** The following periodic message signal m(t) with period T is applied to an AM modulator. Find out its power efficiency when amplitude sensitivity k_a = 0.3, 0.5, and 1, respectively.



(CEAB indicators: 1.1: Mathematical Knowledge, 1.4: Discipline-specific Knowledge, 2.1: Problem Identification, 2.2: Problem Formulation, 2.3: Problem Solution, 3.3: Data Collection, 3.4: Data Analysis and Integration)

4. Simulink Assignment:

(CEAB indicators: 1.4: Discipline - specific Knowledge, 2.3: Problem Solution, 2.4: Solution Verification and Evaluation, 4.1: Requirement and Constraint Identification, 4.3: Design Candidate Generation, 4.5: Detailed Design)

(Note: Simulink is available at ENSC undergraduate computer lab. Please don't wait until last minute to work on your assignment, as there are only 20 licenses for the Signal Processing Block set to be used simultaneously.)

A simple tutorial of basic Simulink operations can be found at the end of this document, which shows you how to build a system in Simulink. Please read the tutorial first, then build an AM modulator in the Matlab Simulink that can generate the output signal

$$s(t) = A_c\left[1 + k_a m(t)\right]\cos(2\pi f_c t).$$

Set the following parameters in your model:

- Message: 500 Hz cosine wave with unit amplitude.
- Carrier: 2500Hz cosine wave.
- Sample time of the message and carrier: 1e-5 sec.
- Gain Ac: 10.
- Simulation stop time: 0.025 second
- Amplitude sensitivity k_a : test with 0.25,0.75, 1, and 2.0.

The following building boxes are required for this assignment:

- From the Simulink/Sources sublibrary: Constant, Sine Wave.
- From the Simulink/Math Operations sublibrary: Add, Product, Gain.
- From the Simulink/Sinks sublibrary: Scope.
- From the Signal Processing Blockset/Signal Processing Sinks sublibrary: Spectrum Scope.

You can put additional scopes or spectrum scopes to measure the signal at different points of the system.

In your assignment include the following:

1- Your system layout. 2- Your message signal. 3- The modulated signals for each k_a, in both time domain and frequency domain. 4- Briefly explain the spectrum of the modulated signals. **<u>You do not need to submit your Simulink files, but your TA has the right to request the files if necessary.</u>**

Below you can find more details about the steps you need to take for your Simulink assignment, followed by a brief tutorial on Simulink. Note that the following information is based on Simulink 2010. There might be minor differences between this and the more recent versions.

Continuous state vs discrete state:

This assignment is essentially a discrete time simulation of the AM system, so we need to tell Simulink to use discrete time simulator. This can be done by opening the “Simulation / Configuration parameters…” window in the system layout window. Click on “Solver” in the left side of the window, then in the “Solver options” part of the right side, find the name “Solver”, and choose “**discrete (no continuous states)**” from the list next to it. Without this setting, you will get a warning message in the Matlab command window when you run the simulation, although it does not affect the simulation.

Set the parameters for the sine wave box

Double click the sine wave box to bring up its parameter window, in which you can set the amplitude, bias (DC offset), frequency, phase, and sample time of the sine wave. Please keep the first two setting (“**time-based sine type**”, and “**use simulation time**”).

The phase of the carrier should be set properly in order to get a cosine wave instead of a sine wave. This is also necessary if your message is a cosine signal. Note that the frequency and phase units used by Simulink are rad/sec and rad, respectively. Therefore, we need to set the frequency as “2 * pi * 500” rad/sec if we want to use a frequency of 500 Hz. Similarly, we need to use “pi/2” to represent a 90^{o} phase shift.

The sample time should be small enough so that we get a smooth waveform. A value of 1e-5 sec (corresponding to a sample frequency of 100 kHz) is sufficient for this lab. The same sample time should be used for the message and the carrier.

How to set the parameter of the spectrum scope

Double click the Spectrum Scope box to bring up the Parameter Window. In the “Scope Properties” tab, choose “dBm” for “Spectrum units”, and “Two sided ((-Fs/2…Fs/2))” for “Spectrum type”. Select the “Buffer Input” check box, then set the “buffer size” to 1024 and “buffer overlap” to 512. Keep other default options.

Save the scope output data to Matlab workspace

Double click a scope to open its display window before running the simulation. The spectrum scope window will be opened automatically once you start the simulation.

You may need to adjust the axis scales of the scope by right-clicking the displayed waveform on the scope output window, choosing “Autoscale”, then using the “Zoom X-axis” icon in the menu bar to display a suitable amount of data.

Alternatively, you can easily export the scope output result to Matlab workspace. To do this, double click the Scope box to bring up the display window. Click the second icon on the menu bar to open the Parameters window (next to the printer). Click the Data History tag, and then click the “Save data to workspace” check box. Type in a name for the displayed data in the input box next to the “Variable name:” prompt. For example, you can type in **`myoutput`**.

The exported data can have one of three possible formats:

- Structure with time: If this format is chosen, the exported variable is a Matlab structure. We can easily plot a figure similar to the scope output (with black background) by the command "**`simplot(myoutput)`**". The data sequence can be accessed by **`myoutput.signals.values.`**
- Structure: Same as "structure with time", except that the time information is not exported. This format is not very useful.
- Array: In this format, the exported variable is a 2-D array, with the time information in the first column and the variable in the second column. We can plot the data by the command "**`plot(myoutput(:,1), myoutput(:,2))`**". The advantage of this format over the first one is that the figure has white background, as regular Matlab figures. It is also useful if you only want to display a portion of the data in order to observe the data more clearly. For example: "**`plot(myoutput(1:100,1), myoutput(1:100,2))`**".

After this, run the simulation. When the simulation stops, go to the Matlab command window, type "**`whos`**", you will find your variable from the list of all current variables in the workspace. You can then play around the data in Matlab.

Change number of displayed samples of the Scope

The default setup of the scope only displays the last 5000 samples. The setup can be changed by opening the Parameters window of the scope, then in the "Data History" tab, uncheck the "Limit data points to last 5000" option, or changing the number 5000.

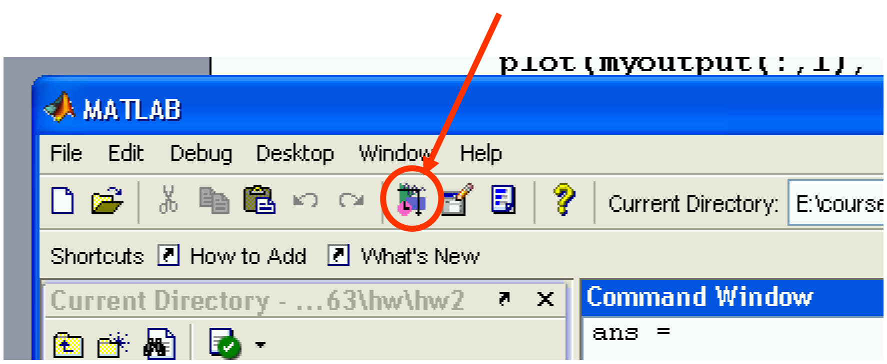
Appendix: Basic Simulink Operations

Several basic operations of simulink are listed below. For futher information, please refer to Matlab Simulink help, or the following tutorials:

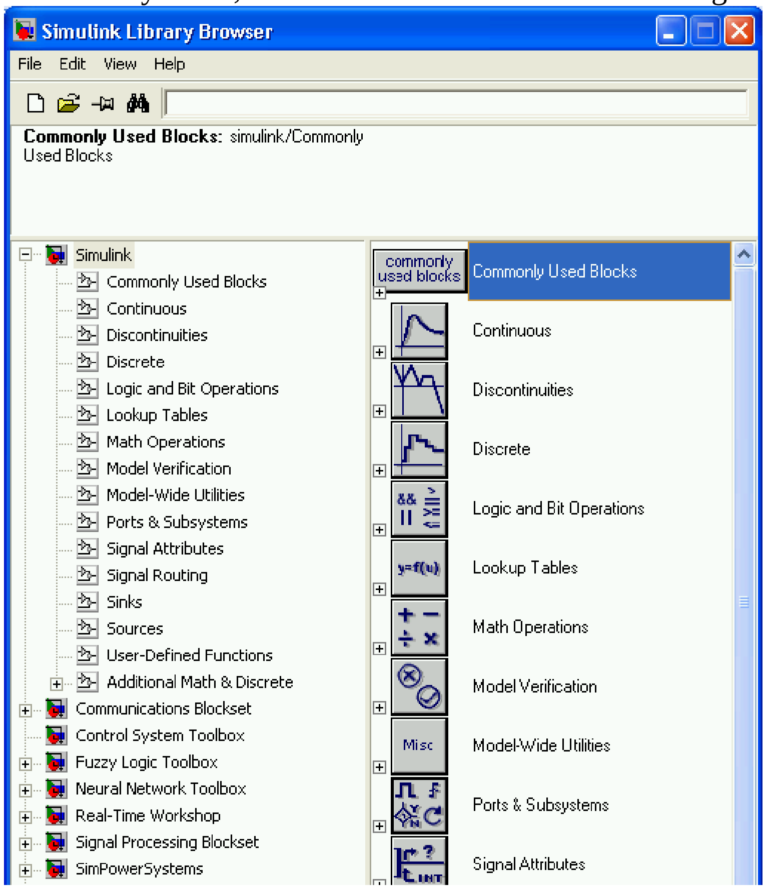
https://www.google.ca/?gfe_rd=cr&ei=2L6XVr7pBcL8iALA17iwCQ&gws_rd=ssl#q=simulink+tutorial+for+beginners
http://www.mathworks.com/help/pdf_doc/simulink/sl_gs.pdf
http://www.mathworks.com/academia/student_center/tutorials/slregister.html

1. Start Simulink:

Type simulink from Matlab command window, or click the Simulink icon in the Matlab menu bar, as shown below.

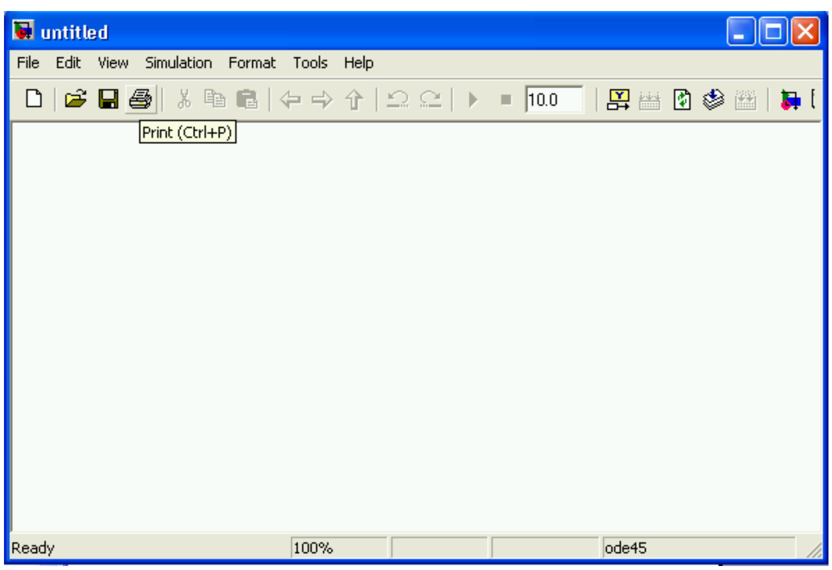


The following Simulink Library Browser window will be opened, which lists the basic simulink library and additional toolboxes and blocksets. You can click a library name in the left to expand it and show the sublibraries in it. Click on a sublibrary name, the boxes in it will be listed in the right hand side.



2. Create a new model:

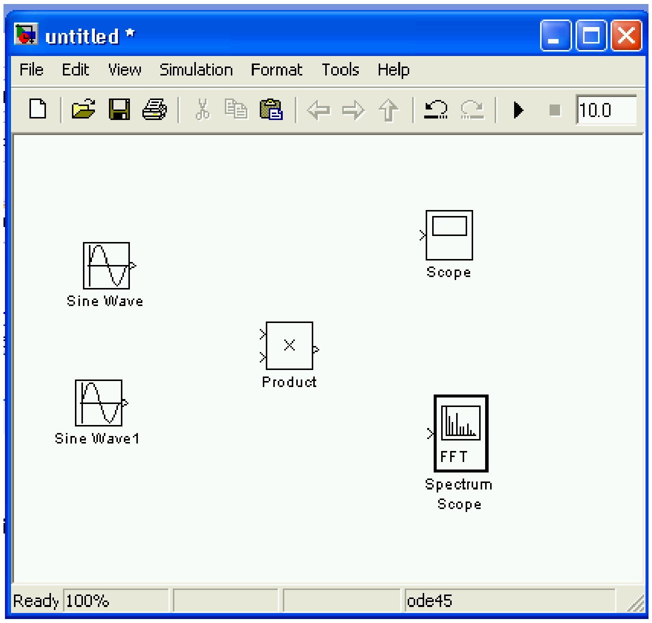
In the Simulink Library Browser window, click the "Create a new model" menu icon, or choose File/New/Model menu option. A blank model window will be opened. This is where we can build our system.



3. Add building boxes to the model:

To add a box to our system, simply click the box in the Simulink Library Browser window, and drag and drop it into the model window.

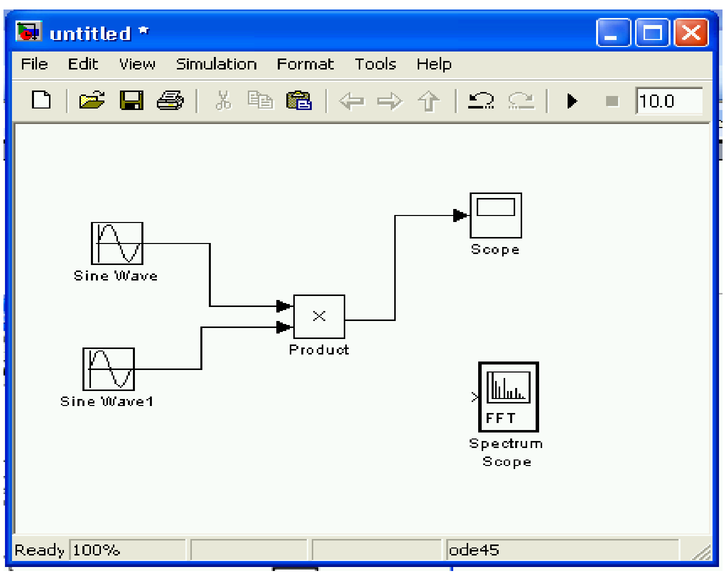
As a simple example, suppose we want to multiply two sine waves. We can add two sine wave boxes (in the Simulink/Sources sublibrary), one Product box (in the Simulink/Math Operations sublibrary), one Scope (Simulink/Sinks sublibrary), one Spectrum Scope (Signal Processing Blockset/Signal Processing Sinks sublibrary) to our new model window. The following is what we have after this.



4. Connect two boxes by signal line:

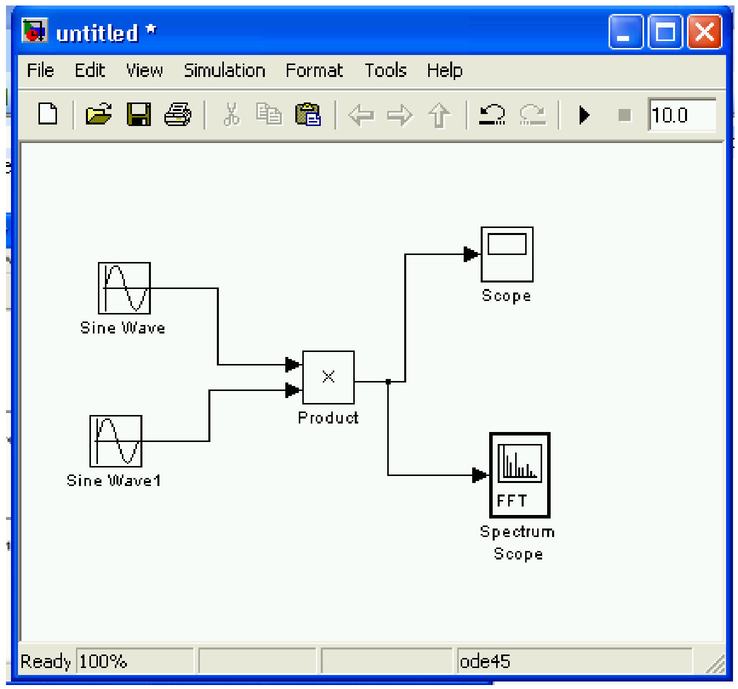
The next step is to connect the system with signal lines to complete the model. Place the cursor on the output port of the sine wave box (the > symbol on the right edge). The cursor will change to a cross-hair shape. Drag the cursor from the output of one box to the > symbol at the input of another box. Release the mouse when the cursor changes to double lined cross-hair. A line will be created between the two ports.

In our simple example, we can connect the two sine waves to the product box, then connect the output of the product to the scope, as shown below.



5. Add branch to a line:

Next we want to connect the spectrum scope to the product output. To do this, we need to branch from the signal line connecting the product to the scope. This can be achieved by **pressing and holding the Ctrl key and clicking on any point of the signal line**. The cursor will become a cross-hair. We can then move the cursor to the destination port. In this example, we connect a line to the spectrum scope, as shown below.

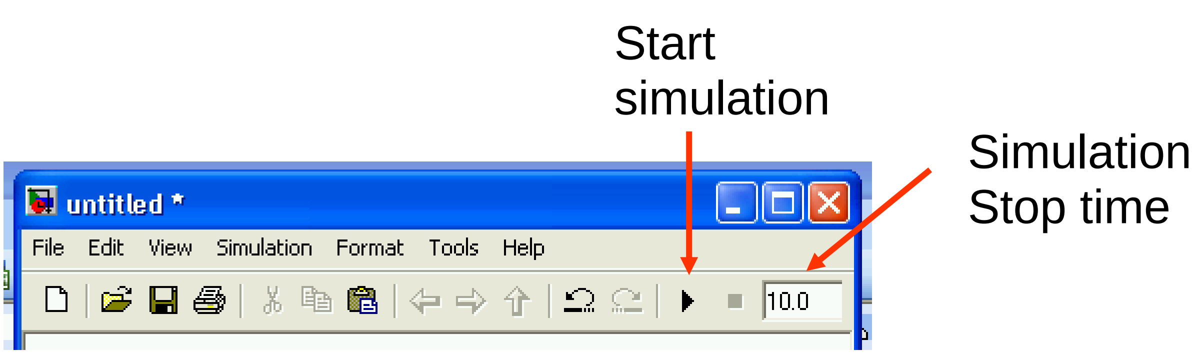


6. Set the parameter of each box:

Double click each box will bring up its parameter window. Please refer to the assignment for details.

7. Set up simulation stop time, and run the simulation.

The simulation stop time can be changed from the input box on the menu bar of the mode window, as shown below. The default value is 10 seconds.



8. Save the system:

The system layout and parameters can be saved as a ".mdl" file, so that you can open and edit it later.

5.