ENSC-327-Communication Systems

Instructor: Atousa Hajshirmohammadi

Lab #1: Oscilloscope, Spectrum Analyzer

1. Introduction

The objectives of this lab are:

1. To learn the basic operations of oscilloscope and spectrum analyzer.
2. To understand the concept of dBm and dBV.

2. Matlab at SFU

The steps involved to use a **Named User License** are as follow:

1. Create an account with Mathworks (Matlab's parent company)
2. (possibly) install Java (most machines have it already)
3. Download the software
4. Activate the software

Next Step

Click here to start the process, acquiring an activation key. You will be directed to a MathWorks site where you will be asked to validate your SFU ID and password, as you would with any other SFU site. Your ID will be evaluated for eligibility.

If you are here because you need to **renew an existing license** for Matlab, the above link is for you as well.

Once you get your MathWorks account (above) you can use Matlab online anytime. Please see: https://www.mathworks.com/products/matlab-online.html

What about license renewals?

If you need a new/updated activation key, simply do the following:

Log into the link provided, that goes to the vendors site. Please click on "**License Center**".

From there you can re-activate the copy you are already using on the "**Install and Activate**" tab.
See the topic on the right for, "Activate to Retrieve License File"

3. Oscilloscope

Oscilloscope software, referred to as Time Scope in DSP System Toolbox™, performs time-domain analysis of multichannel signals and data tasks that are traditionally handled by hardware instruments.

Using oscilloscope software (Time Scope in DSP System Toolbox) with MATLAB® and Simulink®, engineers can visualize, measure, and analyze the time-domain characteristics of evolving multichannel signals or streaming data in real-time or during simulation.

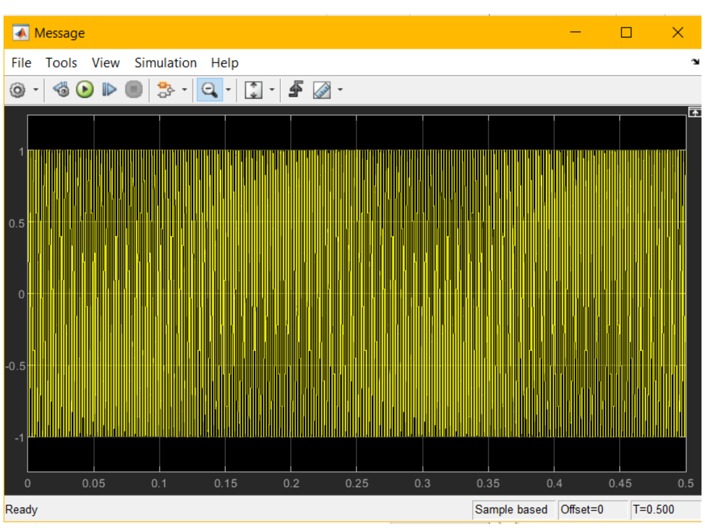
Time Scope displays signal characteristics and measurements such as signal amplitude, cycles, and time between events in the waveform for a variety of signals and data types that include audio, bio signals, communication signals, speech, and streaming data acquired from sensors and instruments. The user interface, features, and measurements are consistent with conventional hardware-based oscilloscopes.

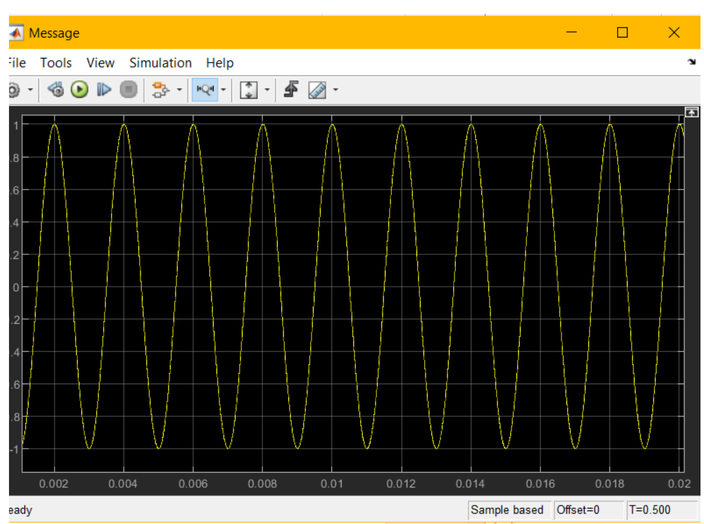
The oscilloscope software in DSP System Toolbox includes the following signal measurements:

- Cursor measurements
- Peak finder
- Triggers
- Signal statistics such as maximum, minimum, mean, median, and RMS values
- Bilevel measurements such as transitions, overshoot or undershoot, and cycles

Oscilloscope (scope) Settings

To view signals on the corresponding oscilloscope (scope) modules, you need to make sure your display settings such that signals can be viewed properly and as intended, just as you would with a physical oscilloscope in the lab. For example, the left and right images below are both for the same sinusoidal signal at 500Hz, but obviously the right image is the one that displays the message in a useful way.

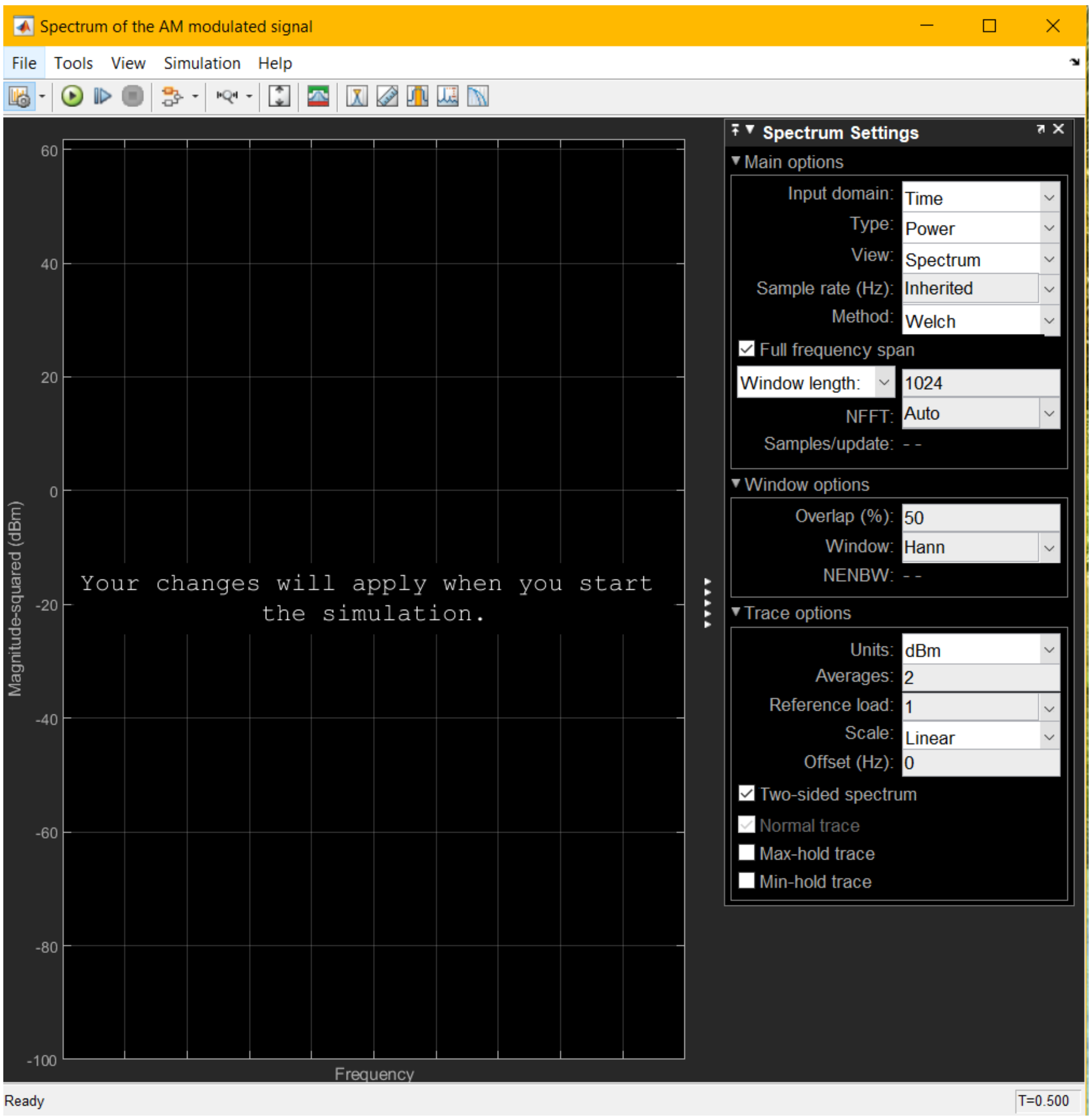




Spectrum analyzers (SA), like oscilloscopes are a basic tool used for observing signals. While oscilloscopes look at signals in the time domain, spectrum analyzers look at signals in the frequency domain.
The Spectrum Analyzer block accepts input signals with discrete sample times and displays frequency spectra of these signals.

How to set the parameter of the spectrum scope

Double click the Spectrum Analyzer block to bring up the Window. In the "View" tab, click on "Spectrum Setting". In "Spectrum Settings" window, check that the parameters are set as below, or modify accordingly.



5. dBm, dBV and their relationship

Many RF equipment provide power ratio measurements in dBm unit. The "dBm" (or dBmW) notation represents a measured power level (in dB) relative to 1mW (milliwatt) with respect to a fixed resistance (For example, 50Ω). In other words, "how much stronger/weaker is this signal than 1mW?"

The dBm of a power P with respect to 1 mW is defined as

$$x = 10\log_{10}(P/(1\ \mathrm{mW}))$$

To convert a dBm value back to the power in Watts, we can use the following formula:

$$P = (1\ \mathrm{mW})10^{(x/10)}$$

For example, if a signal has a power level of 3dBm, then it is 2 times as strong as 1mW, so the signal power level is 2mW. Similarly, 30dBm corresponds to 1W, and -30dBW corresponds to 1E-6 W.

Sometimes it's necessary to define the ratio of two voltages in dB scale. This is called "dBV." For example, the gain of an amplifier can be measured by the ratio of its input and output voltages. This can be defined as

$$x = 10\log_{10}\frac{V_2^{\ 2}}{V_1^2} = 20\log_{10}\frac{V_2}{V_1}.$$

Using dB notation simplifies power calculations in communications systems. For example, if the measured power at the input of an amplifier is 5dBm and the gain of the amplifier is 20dB, then the measured output power after the amplifier should be 5dBm + 20dB = 25dBm. For further details, please refer to

http://en.wikipedia.org/wiki/DBm

6. Deliverables (100 points)

A simple tutorial of basic Simulink operations can be found at the end of this document (Appendix), which shows you how to build a system in Simulink. Please read it carefully before you start the following parts. In these parts, you will try to add and multiply some sinusoidal signals and view them on the Scope and spectrum analyzer.

6.1. Add a 2V DC signal to a 5KHz, 2V p-p sinusoidal signal. Set the simulation stop time to 10(s). Observe the output on the Scope and the Spectrum Analyzer (you can use one constant, one sine wave block, one Sum block, one Scope, and one Spectrum Analyzer). **(30 points)**

a. Include a screenshot of the model you have created in your report. (3 points)

b. Include the figure you observe on the Scope (Make sure that you set the display setting and sample time properly, as explained in Appendix). (5 points)

c. Include a screen shot of what you observe on the Spectrum Analyzer (Make sure that you set the display setting and sample time properly, as explained in Appendix). (5 points)

d. Change the DC signal from 2V to 10V and double the frequency of the Sinusoidal signal to10 KHz. What changes do you observe in the output of the Scope and the output of the Spectrum Analyzer compared to previous figures? Include the figures and explain briefly. (7 points)

e. Report the Average power of the signal in dBm (You can observe it using the Spectrum Analyzer. Go to Tools>>Measurements>>CCDF measurements). Change the simulation stop time to 20(s) and report the Average power again. Did you observe any changes? Why? (5 points)

f. Based on your answer in part **e**, calculate the power of the signal in Watts. (5 points)

6.2. Add a 5KHz, 1V p-p sinusoidal signal to a 100KHz, 2V p-p sinusoidal signal. Set the simulation stop time to 10(s). Observe the output signal on both Scope and the Spectrum Analyzer. In this part, your output signal is in the following form:

Output signal= $A\cos(2\pi f_1 t) + B\cos(2\pi f_2 t)$

Where $A = 0.5, B = 1, f_1 = 5\times10^3, f_2 = 10^5$. **(30 points)**

a. Include a screenshot of the model you have created in your report. (5 points)

b. Include the figure you observe on the Scope (Make sure that you set the display setting and sample time properly, as explained in Appendix). (5 points)

c. Include a screen shot of what you observe on the Spectrum Analyzer (Make sure that you set the display setting and sample time properly, as explained in Appendix). (5 points)

a. What are the maximum and the minimum amplitudes of the output signal on the Scope? (5 points)

b. Report the Average power of the signal in dBm. Also, calculate the power of the signal in Watts. (10 points)

6.3. Multiply a 5KHz, 1V p-p sinusoidal signal by a 100KHz, 2V p-p sinusoidal signal. Observe the output signal on both Scope and the Spectrum Analyzer. Set the simulation stop time to 10(s) (You can use two sine wave blocks, one Product block, one Scope, and one Spectrum Analyzer). **(30 points)**

a. Include a screenshot of the model you have created in your report. (3 points)

b. Include the figure you observe on the Scope (Make sure that you set the display setting and sample time properly, as explained in Appendix). (5 points)

c. Include a screen shot of what you observe on the Spectrum Analyzer (Make sure that you set the display setting and sample time properly, as explained in Appendix). (5 points)

a. Adjust the Spectrum Analyzer so that you can see the peaks clearly. What are the minimum and maximum amplitudes on the Scope? What is the maximum amplitude observed on the spectrum analyzer? (5 points)

b. Report the Average power of the signal in dBm. Also, calculate the power of the signal in Watts. (10 points)

6.4. An amplifier has a gain of 3dB. What would be the ratio of the output and input voltage? (**10 points**)

Appendix: Basic Simulink Operations

Several basic operations of Simulink are listed below. For further information, please refer to Matlab Simulink help, or the following tutorials (these links refer to the latest version of Matlab).

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

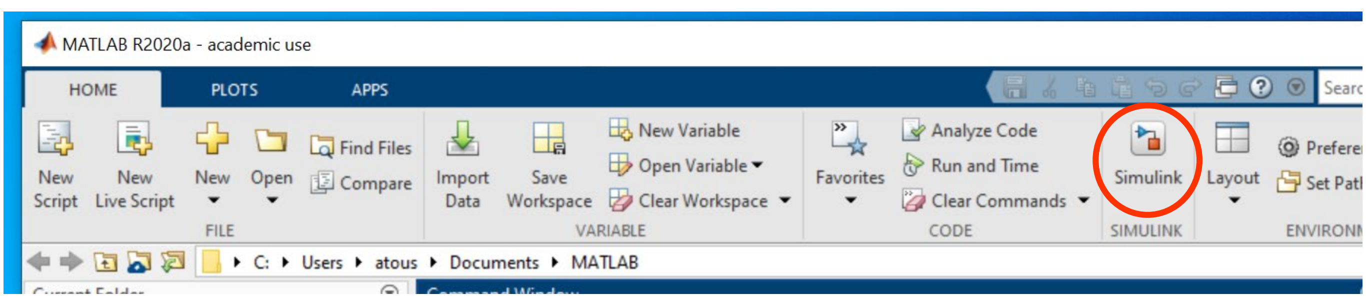
Simulink Components

The following building blocks are required for this assignment:

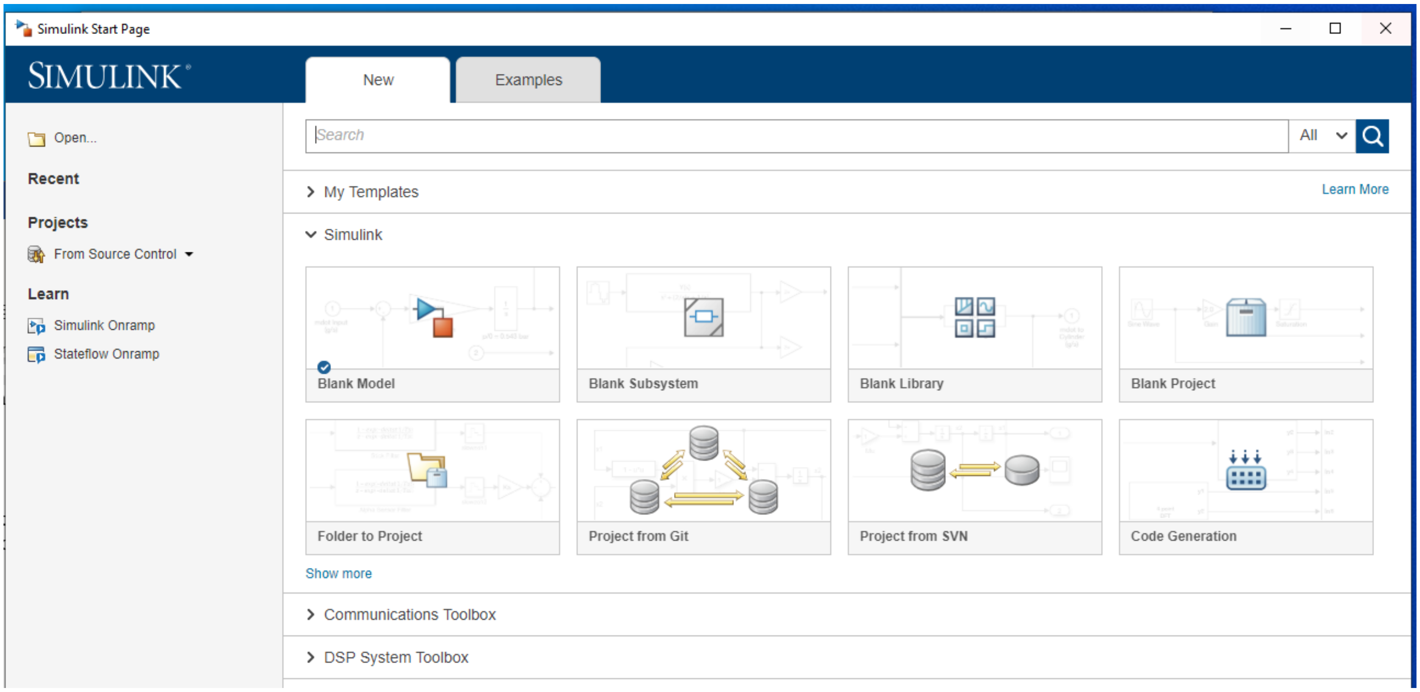
- From the Simulink/Sources sublibrary: Constant, Sine Wave.
- From the Simulink/Commonly Used Blocks sublibrary: Add, Product, Gain, Scope.
- From the Audio Toolbox/Sinks sublibrary: Spectrum Analyzer.

1. Start Simulink:

Start (run) Matlab. Type Simulink on Matlab command window or click the Simulink icon in the Matlab menu bar, as shown below.

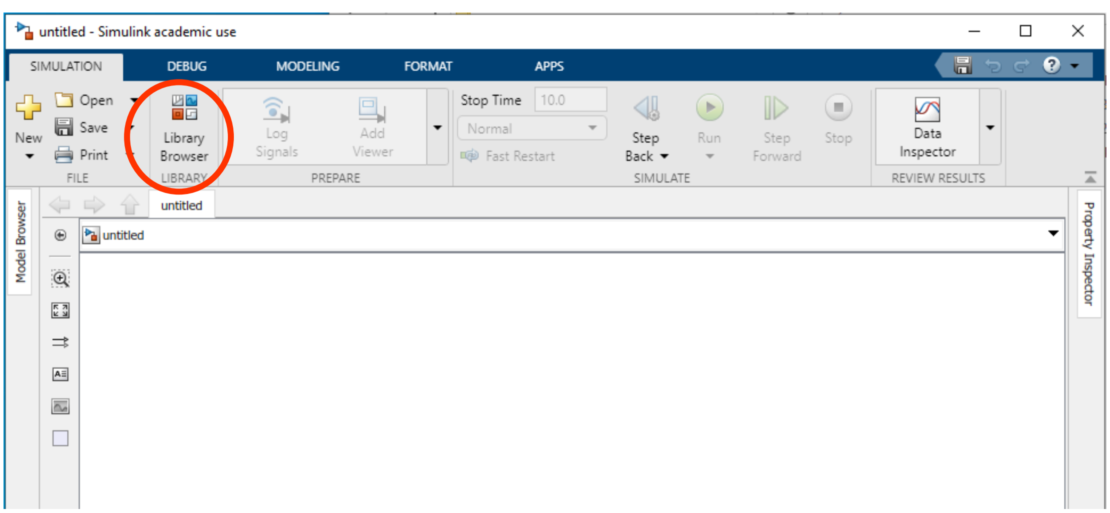


The following Simulink Start Page window will be opened:



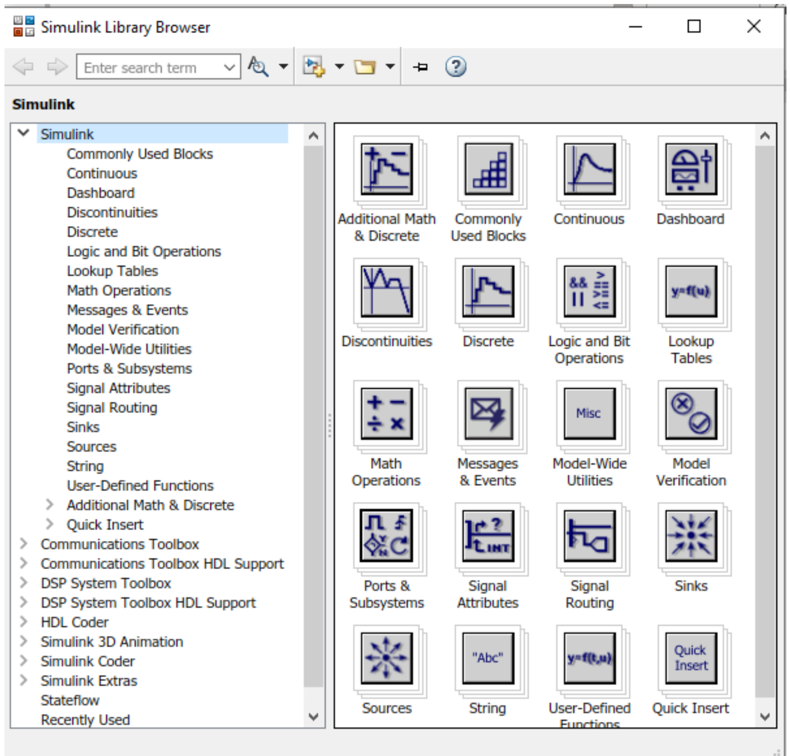
2. Create a new model:

In the Simulink Start Page window, click “Blank Model” to start a new Simulink model. A blank model window will be opened. This is where we can build our system.

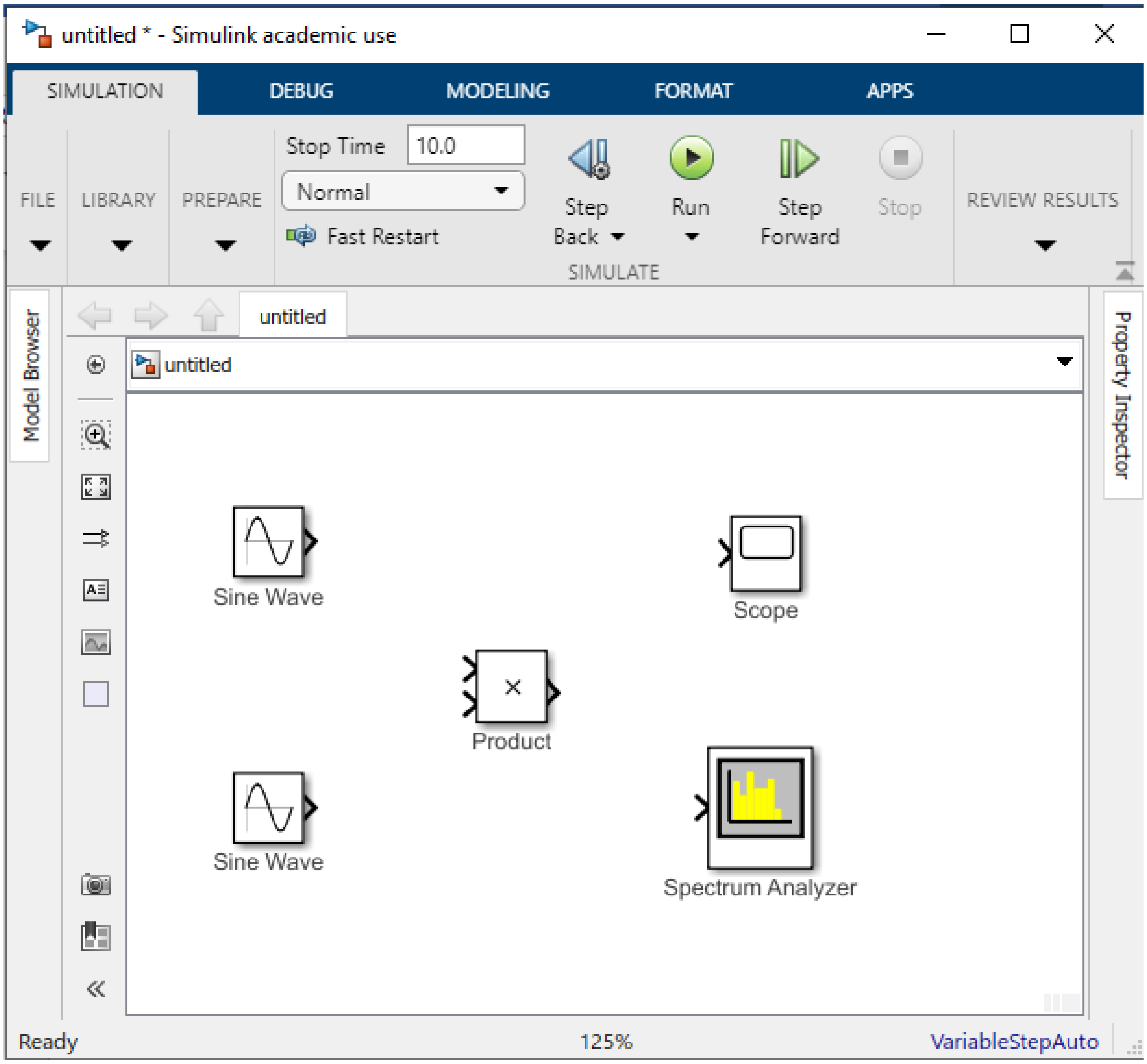


3. Add building blocks to the model:

Open the Library Browser in the above window. The library will open. From there you can select and drag and drop blocks into the model window.

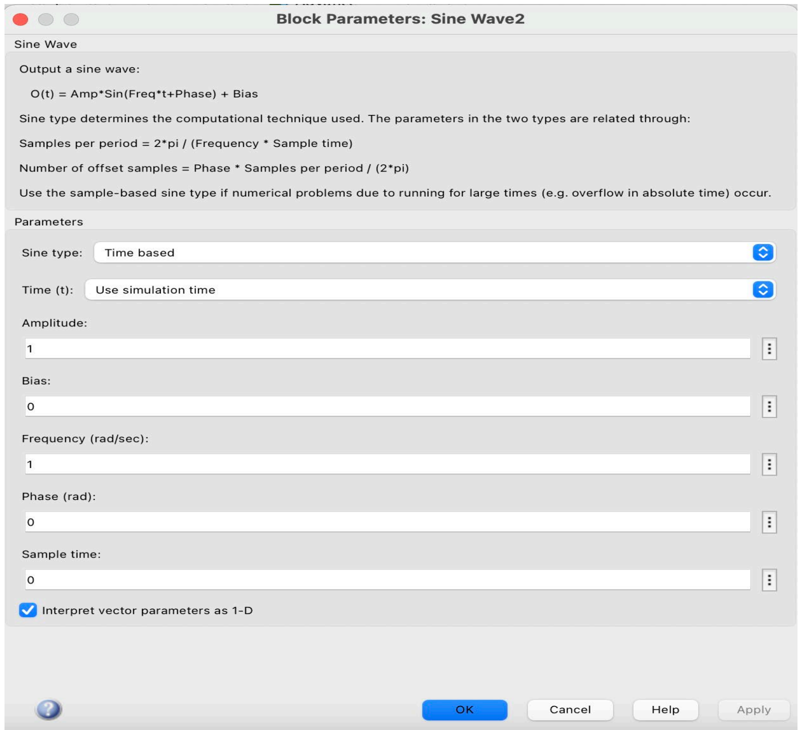


As a simple example, suppose we want to multiply two sine waves. We can add two sine wave blocks (in the Simulink/Sources sublibrary), one Product block (in the Simulink/Commonly Used Blocks sublibrary), one Scope (in the Simulink/ Commonly Used Blocks sublibrary), one Spectrum Analyzer (in the Adio Toolbox/Sinks sublibrary) to our new model window. The following is what we have after this.

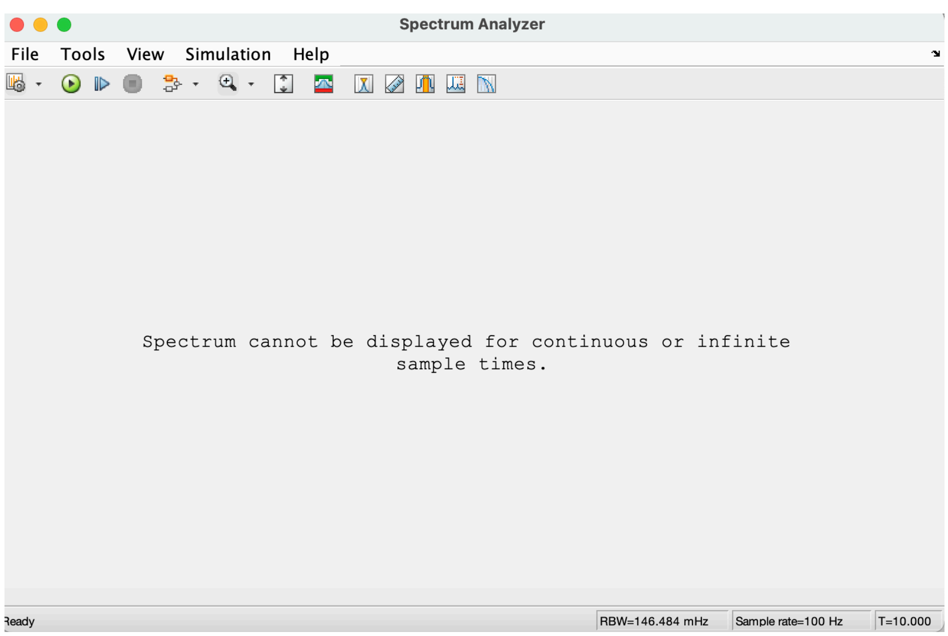


4. Sine Wave Block parameters setup

To setup Sine Wave block parameters, double-click on the Sine Wave block and you have this:



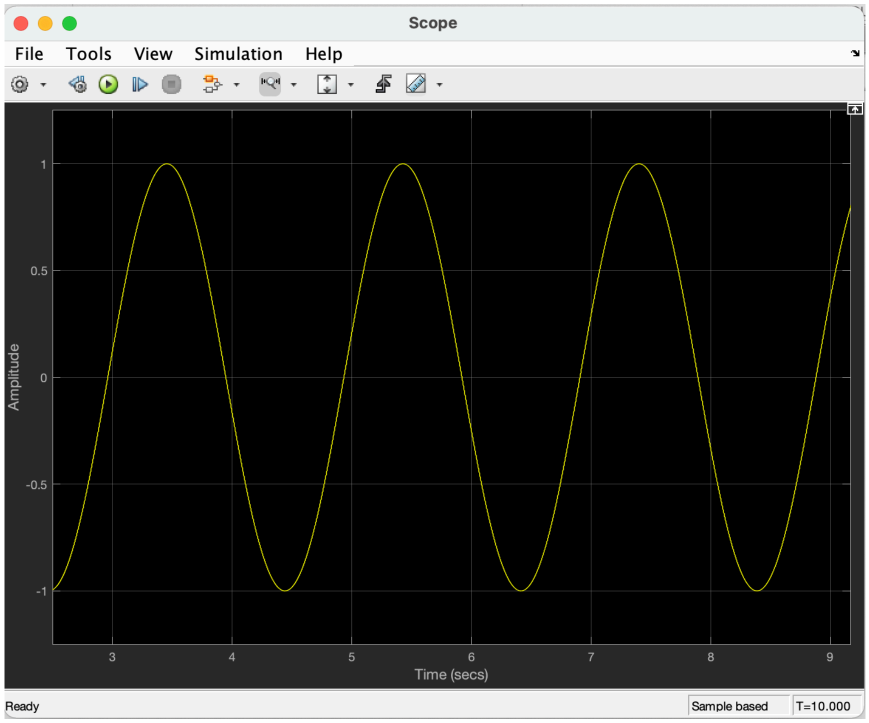
The Sine Wave parameters are explained in the above figure. After setting up the parameters, make sure that you have set a proper sample time for the signal, Otherwise, you might get an error as shown in the figure below, or even if there is no error message, the signal might be displayed completely wrong, as explained in the following section. Also, please note that the frequency is in (rad/sec), and you may need to convert (Hz) to (rad/sec). Moreover, if your sinusoidal signal is a Cosine, your phase would be $\frac{\pi}{2}$.

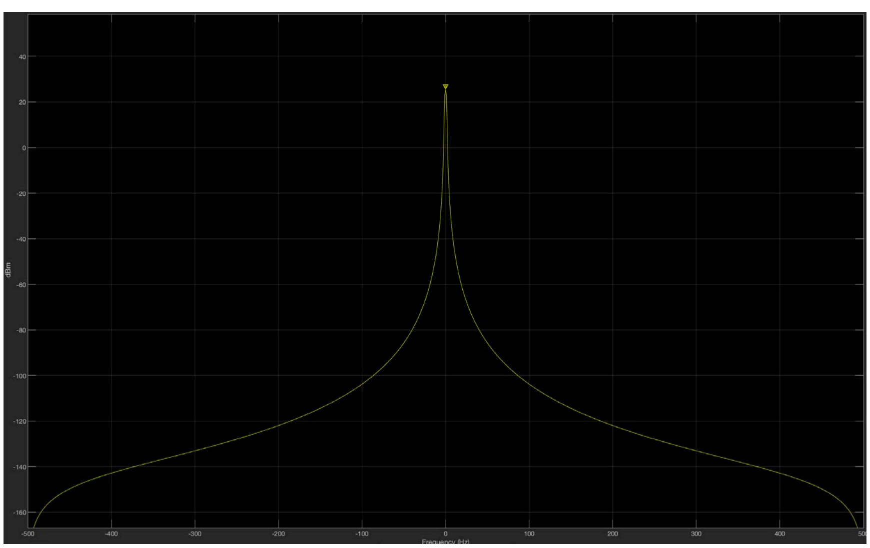


5. Choosing a proper sample time

Choosing a proper sample time is important for a smooth output on the Scope and useful information on the spectrum analyzer. Let's assume we have a 1KHz, 2V p-p Sine signal. The following results are the output of the Scope and Spectrum Analyzer for different sample times:

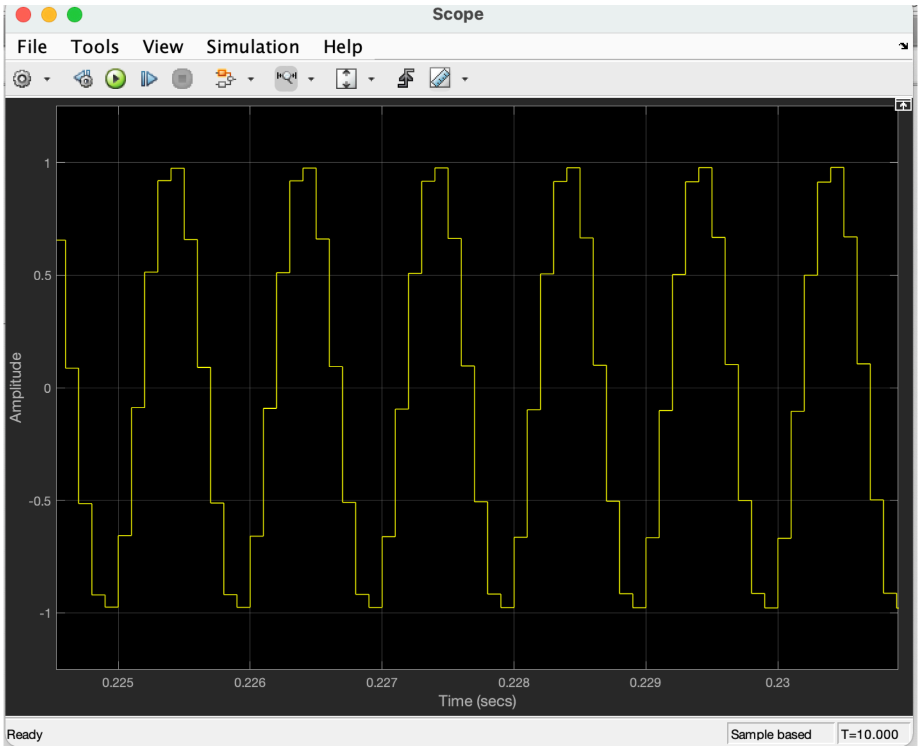
a. Sample time=0.001s:

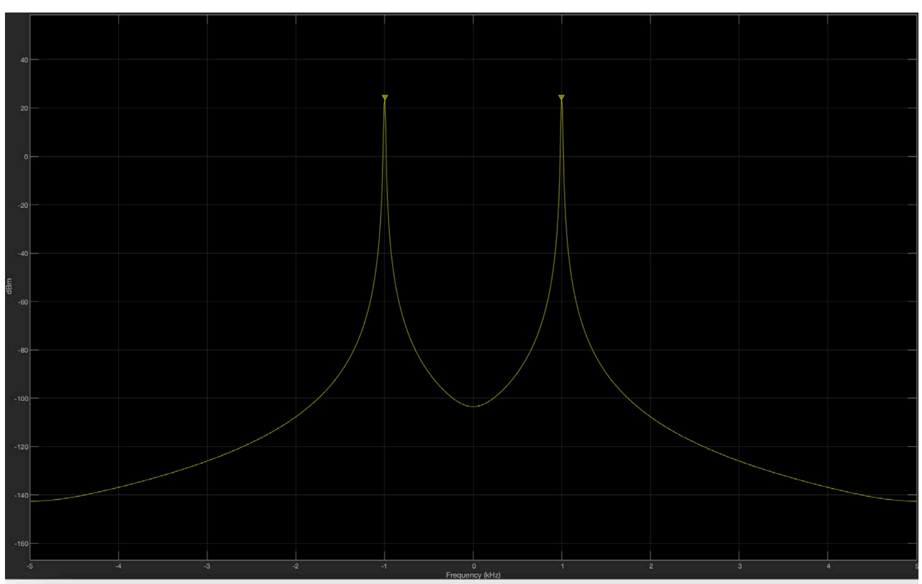




Based on the Scope and Spectrum Analyzer output, we can see that the information they provide us is not valid, i.e. the period and the frequency.

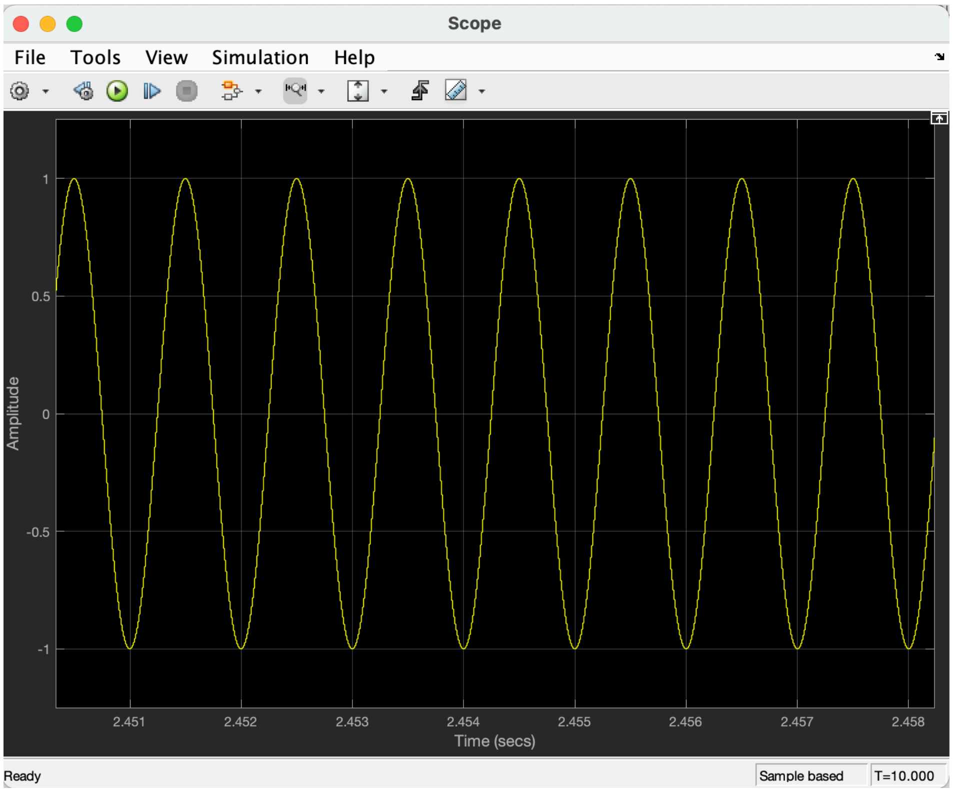
b. Sample time=0.0001s:

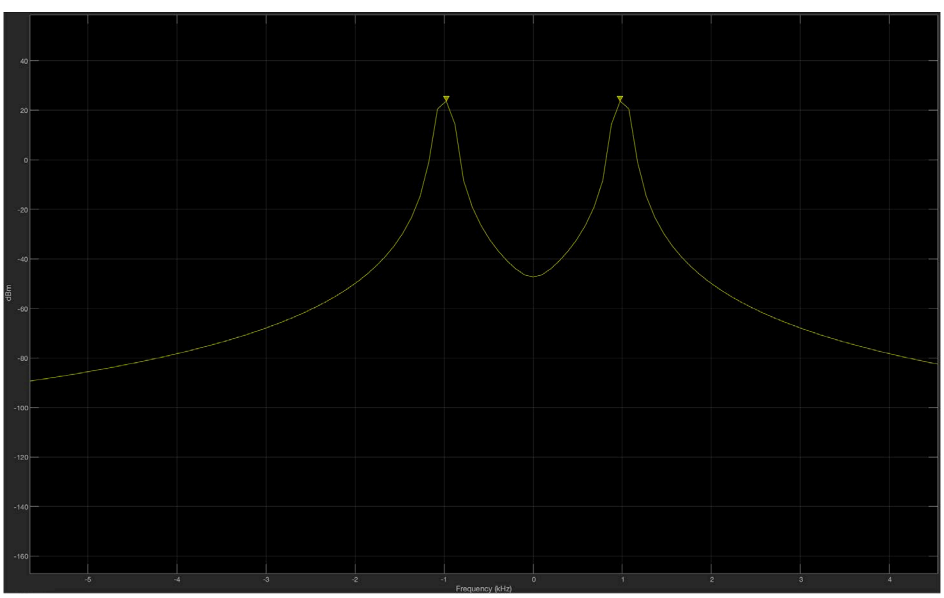




Based on the scope output, the signal period is resolved; however, we have an Aliasing effect in the Since Wave since we have captured less information. But, it seems the Spectrum Analyzer shows the valid information we have expected.

c. Sample time=0.00001s:



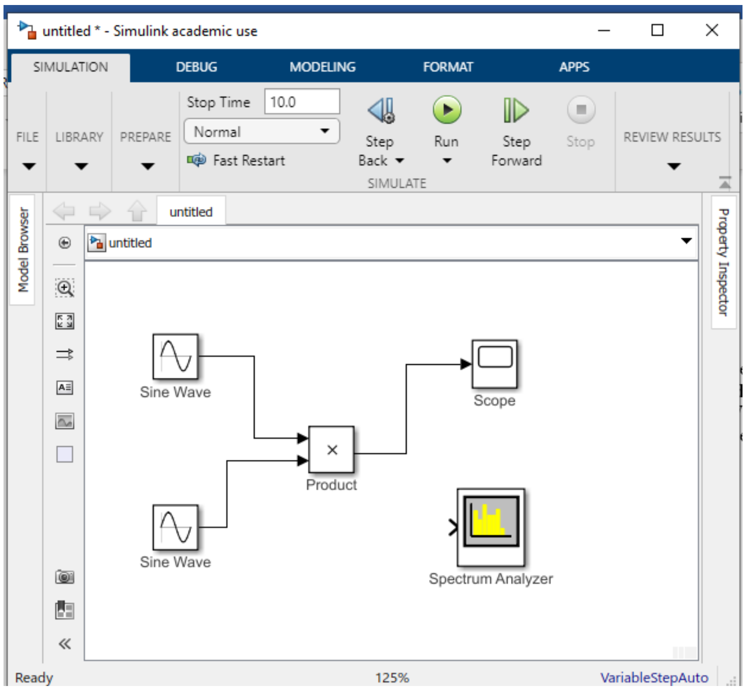


As you can see, the Scope and Spectrum Analyzer shows us the correct information about our signal.

6. Connecting two blocks:

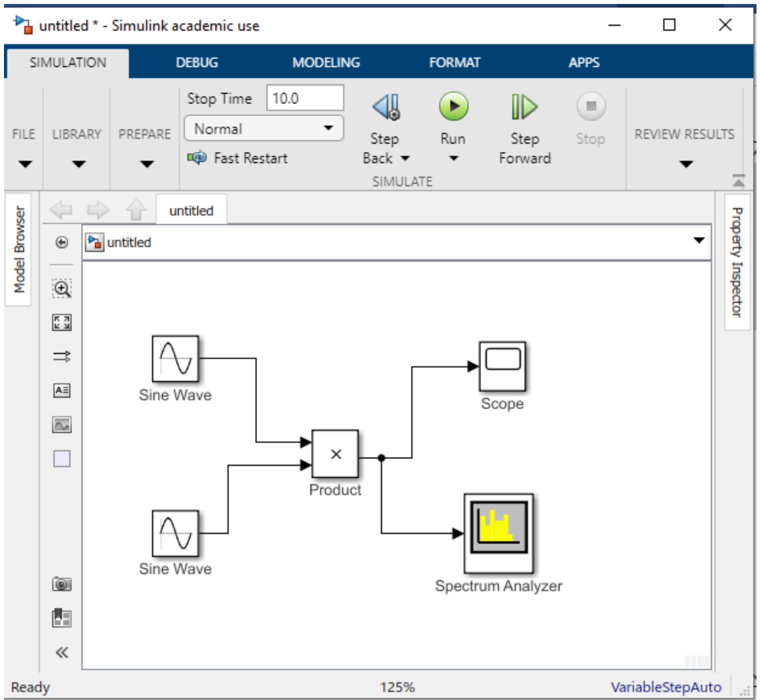
The next step is to connect the system with signal lines to complete the model. Place the cursor on the output port of the sine wave block (the > symbol on the right edge). The cursor will change to a cross-hair shape. Drag the cursor from the output of one block to the > symbol at the input of another block. 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 block, then connect the output of the product to the scope, as shown below.



7. Adding a branch to a line:

Next, we want to connect the spectrum Analyzer 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 Analyzer, as shown below.

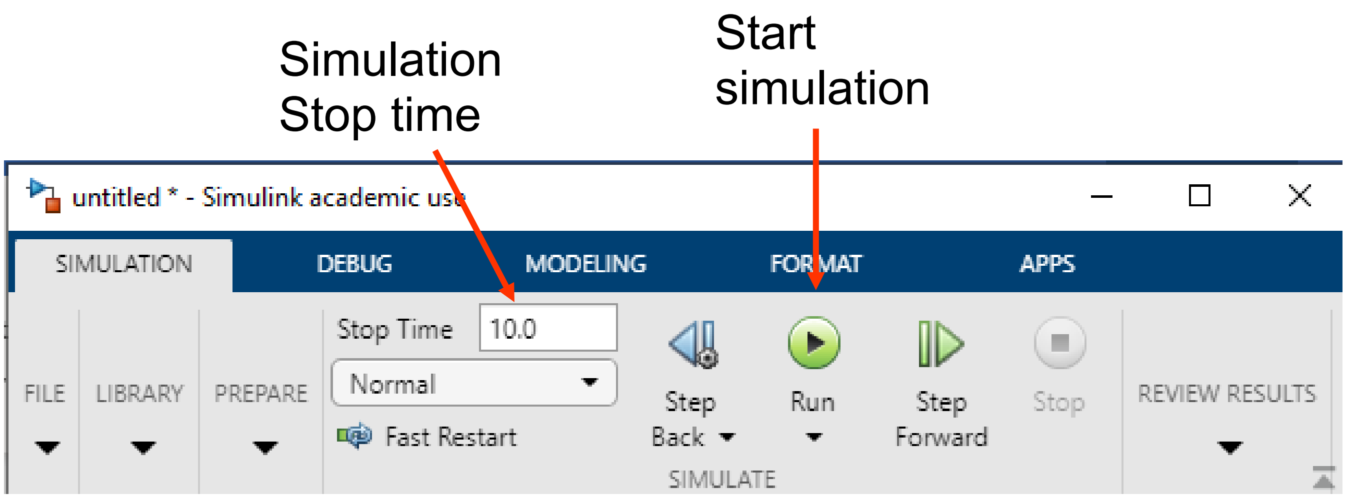


8. Setting the parameter of each block:

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

9. Setting up simulation stop time and running the simulation.

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



10. Saving the model:

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