ECSE 308:
Introduction to
Communication Systems and Networks

L1: Signals and Noise

Part 0: Tutorial on Simulink
Part 1: Presentation of signals and noise
Part 2: Power, bandwidth & SNR

CEAB Data – Engineering tools

- The following engineering tools are used in this laboratory:
  - MATLAB and SIMULINK
  - Software signal generator
  - Power Spectrum analyzer
  - Autocorrelation function

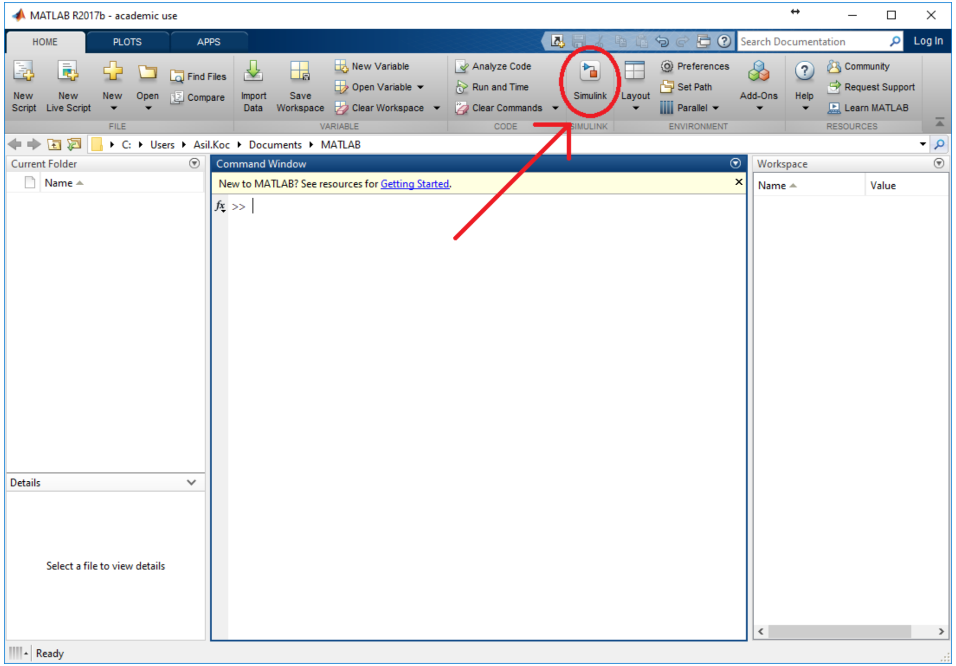
Part 0: Tutorial on Simulink

I. Introduction

- Simulink is an environment for simulation and model-based design for dynamic and embedded systems.
- It provides an interactive graphical environment and a customizable set of block libraries that let you design, simulate, implement, and test a variety of time-varying systems, including communications, controls, signal processing, video processing, and image processing.
- Simulink offers:
  - A quick way of develop your model in contrast to text based-programming language such as e.g., C.
  - Simulink has integrated solvers. In text based-programming language you need to write your own solver.

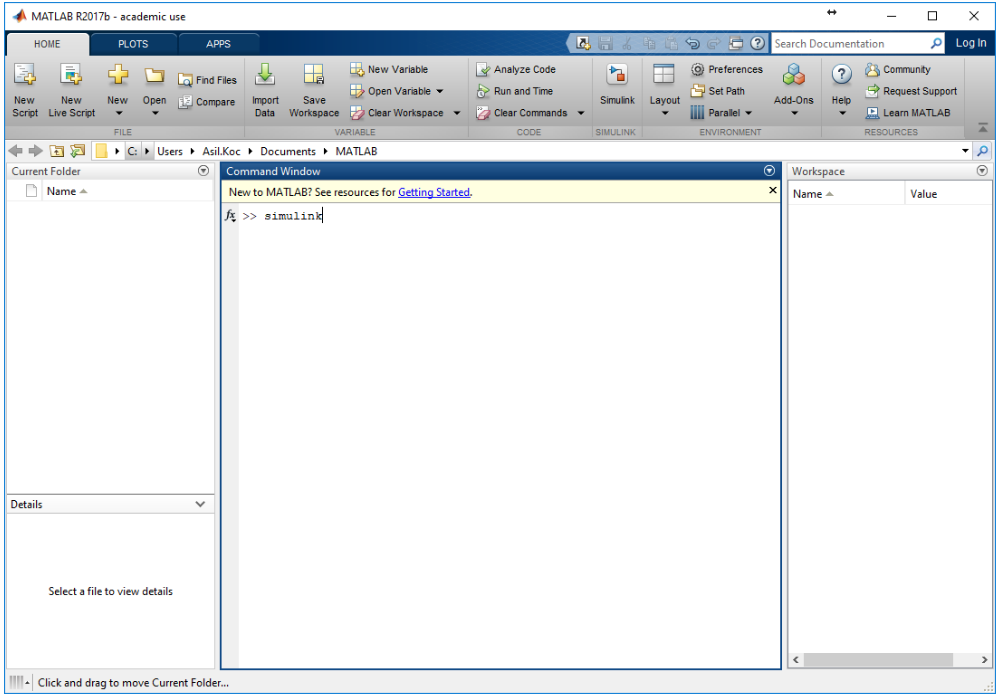
II. How to Open Simulink

- Open MATLAB and select the Simulink icon in the Toolbar:



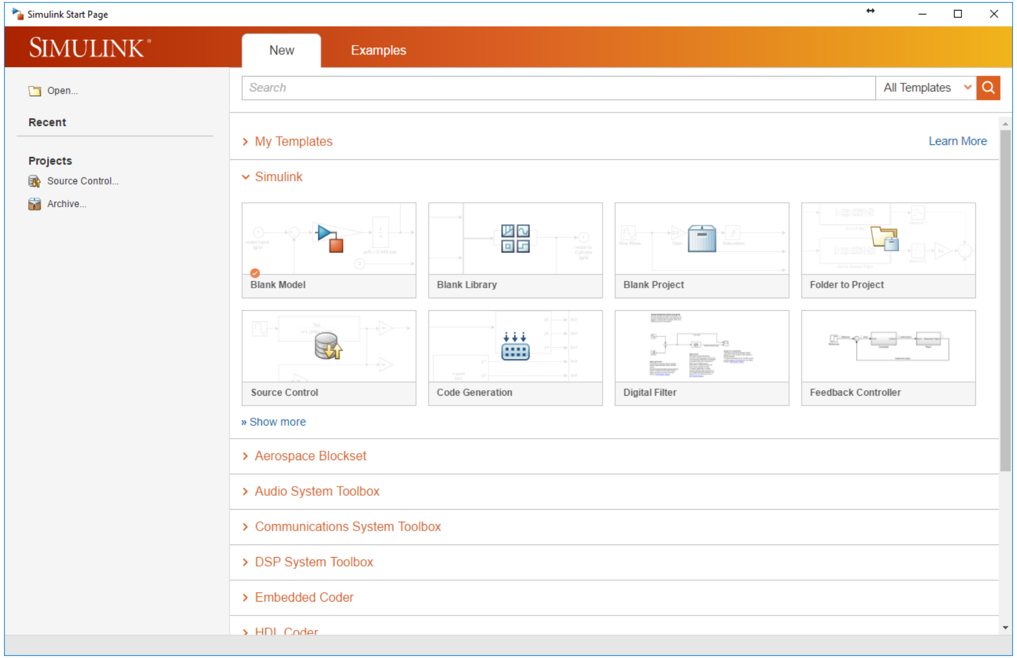
II. How to Open Simulink

- Or type “simulink” in the Command window, like this:



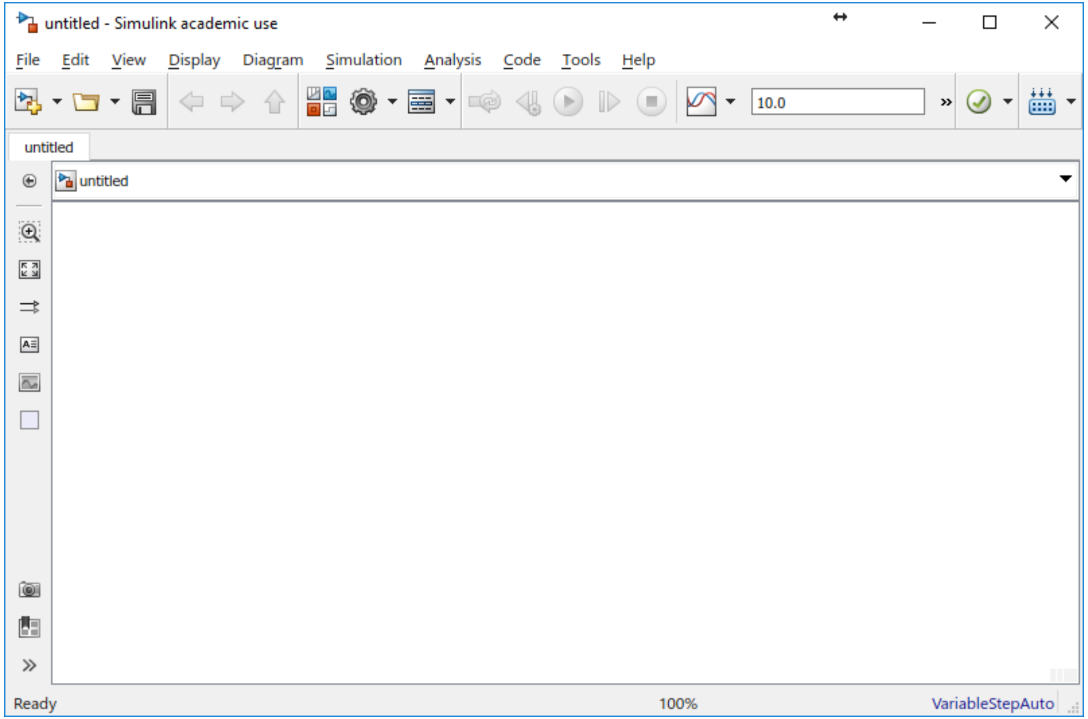
II. How to Open Simulink

- Then, the following window appears (Simulink Start Page):



II. How to Open Simulink

- Select "Blank Model" to open the following Simulink Editor window.

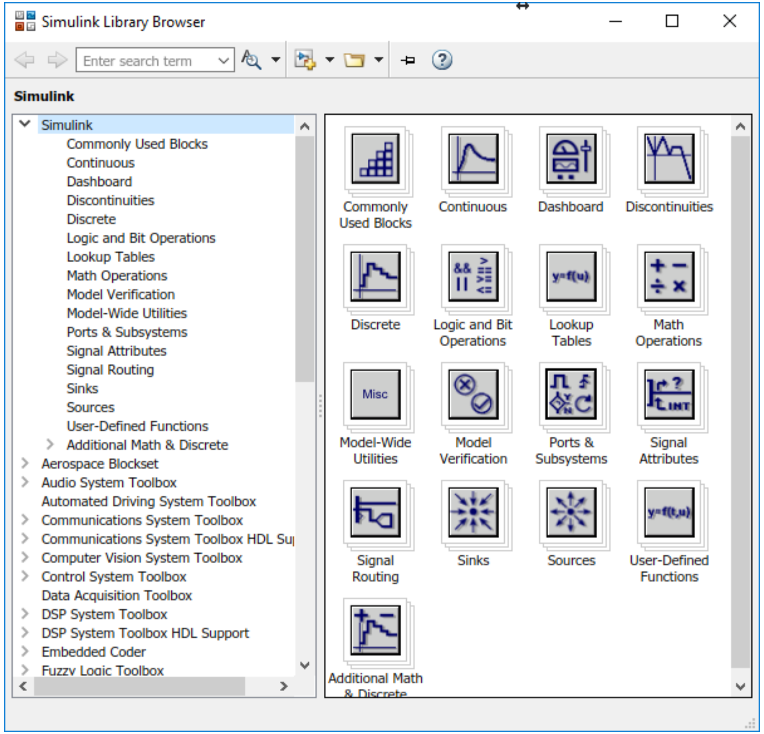


III. Create a Simple Model

- You can use Simulink to model a system and then simulate the dynamic behavior of that system.
- In this part, a simple model is created, which integrates a sine wave signal to a cosine signal and then displays the result, along with the original signal, in a scope window.
- The basic techniques used to create the simple model in this tutorial are the same techniques for more complex models.
- To create this simple model, you need four Simulink blocks:
  - **Sine Wave** — Generates an input signal for the model.
  - **Integrator** — Processes the input signal.
  - **Bus Creator** — Combines the input signal and processed signal into one signal.
  - **Scope** — Visualizes the signals.

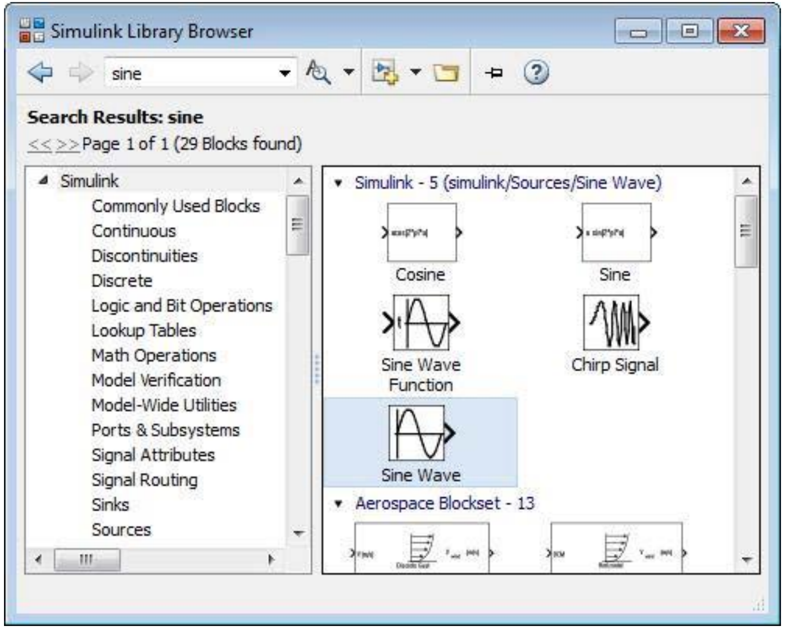
III. a. Open the Simulink Library Browser

- In “**Blank Model**” window, click the **Library Browser** (icon) button. A short delay occurs for the first time opening the Simulink Library Browser.
- In order to design the Simulink models, you may now drag the blocks you want to use from the Simulink Library Browser to the model surface.



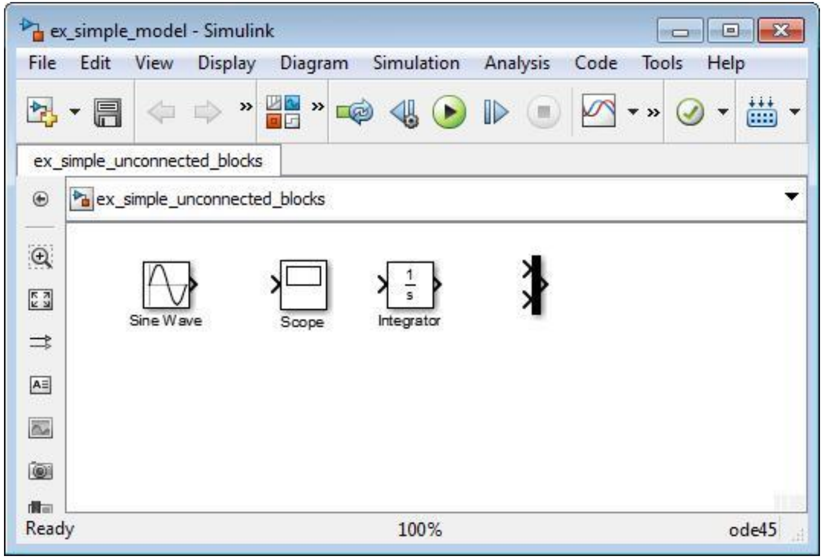
III. b. Search for Specific Blocks

- You can search all the available block libraries.
  - Search for a Sine Wave block. In the search box on the browser toolbar, enter **sine**, and then press the Enter key. Simulink searches the libraries for blocks with **sine** in their name, and then displays the blocks.
  - To get detailed information about a block, right-click a block and then select **Help for the <block name>**. The Help browser opens with the reference page for the block.
  - To view block parameters, right-click a block and then select **Block Parameters**. The block parameters dialog box opens.



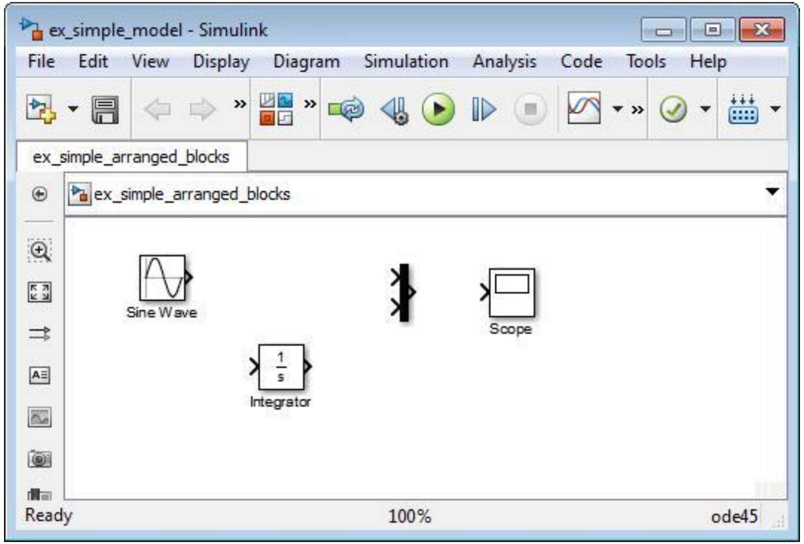
III. c. Add Blocks to a Model

- Add “**Integrator**” and “**Bus Creator**” blocks to your model, using the same approach that you used to add the Sine Wave and Scope blocks.
- Your model should now have the blocks you need for the simple model.



III. d. Move and Resize Blocks

- Before you connect the blocks in your model, arrange them logically to make the signal connections as straightforward as possible.
- Move the Scope block after the Bus block output. You can either:
  - Click and drag a block.
  - Select the block, and then press the arrow keys on your keyboard.
- Move the blocks until your model looks similar to the following figure.

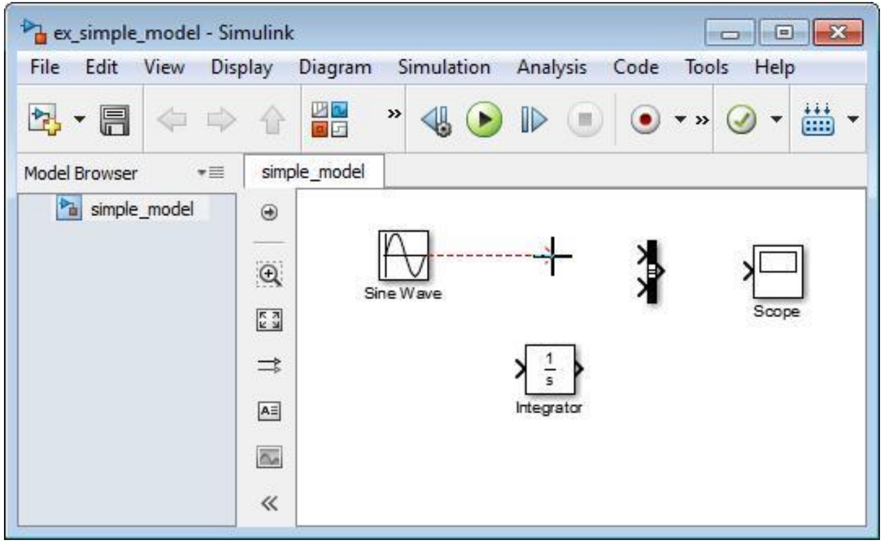


III. e. Simulink Block Connections

- After you add blocks to your model, you can connect them with lines. The connecting lines represent signals within your model.
- Most blocks have angle brackets on one or both sides. These angle brackets represent input and output ports:
  - The > symbol pointing into a block is an input port.
  - The > symbol pointing out of a block is an output port.

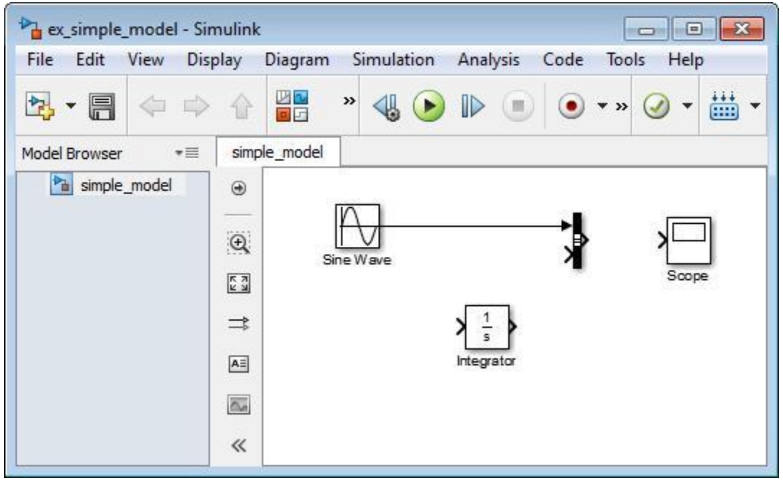
III. f. Draw Signal Lines Between Blocks

- Connect the blocks by drawing lines between output ports and input ports.
- Position the cursor over the output port on the right side of the Sine Wave block. The pointer changes to a cross hair (+) while over the port.
- Click, and then drag a line from the output port to the top input port of the Bus block. While you are holding down the mouse button, the connecting line appears as a red dotted arrow.



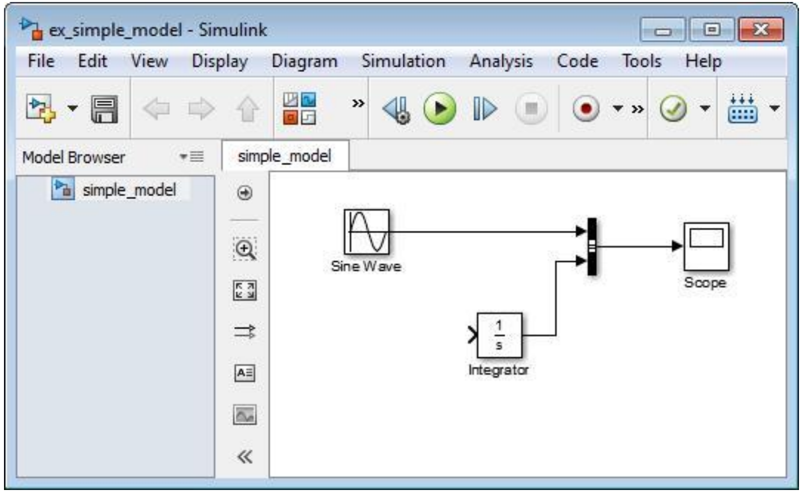
III. f. Draw Signal Lines Between Blocks

- Release the mouse button when the pointer is over the output port. Simulink connects the blocks with a line and an arrow indicating the direction of signal flow.
- Connect the output port of the Integrator block to the bottom input port on the Bus Connector block using this alternative procedure:
  - Select the Integrator block,
  - Press and hold the Ctrl key,
  - Click the Bus Connector block.



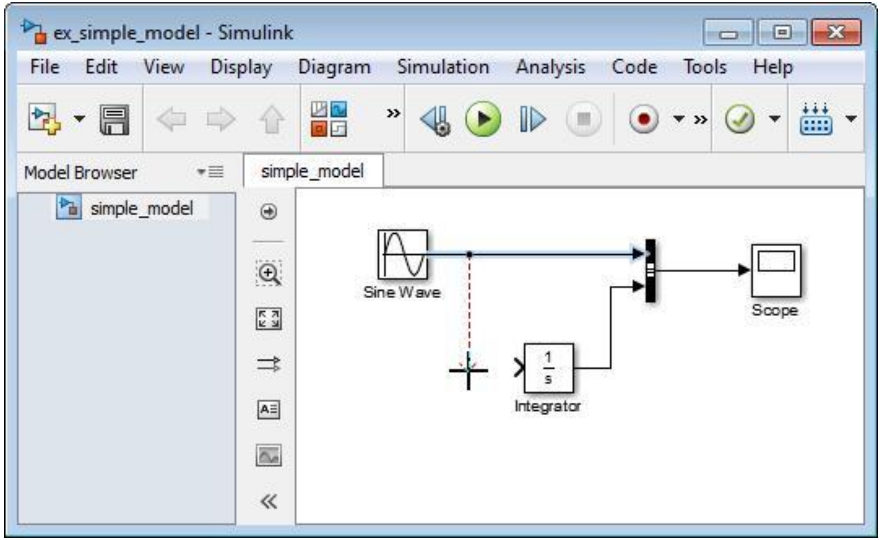
III. f. Draw Signal Lines Between Blocks

- Integrator block connects to the Bus Connector block with a signal line
- Connect the Bus Connector block to the Scope block by aligning ports:
  - Click and drag the Scope block until its input port is aligned with the Bus Connector output port. A light blue line appears between the ports.
  - Release the mouse button, and then click the blue arrow. The blue arrow changes to a black signal line connecting the blocks.



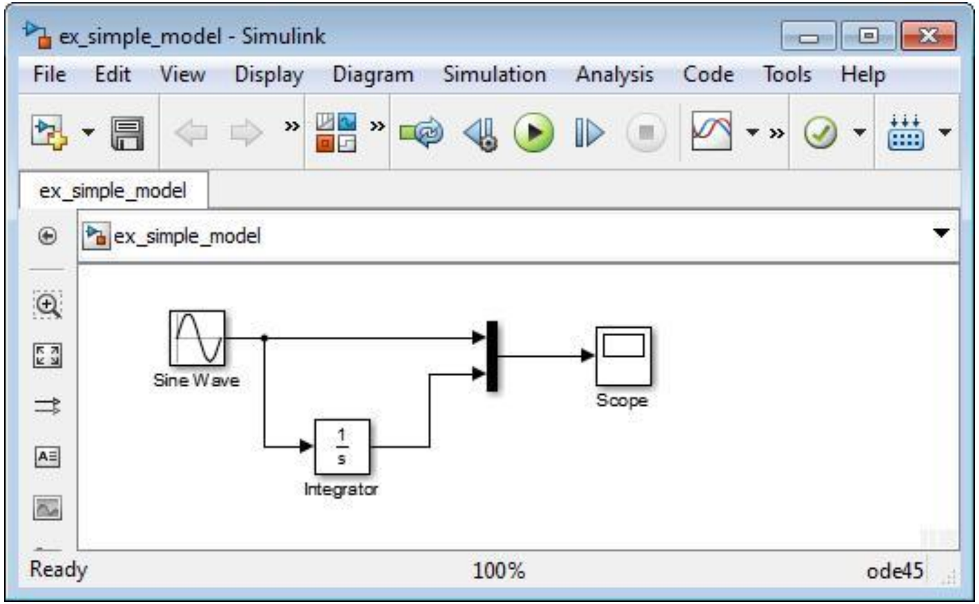
III. g. Draw Branched Signal Lines

- Your simple model is almost complete. To finish the model, connect the Sine Wave block to the Integrator block. This connection is different from the other connections, which all connect output ports to input ports.
- Hold down the **Ctrl** key.
- Position the cursor where you want to start a branch line. Click, and then drag the cursor away from the line to form a dotted-red line segment.



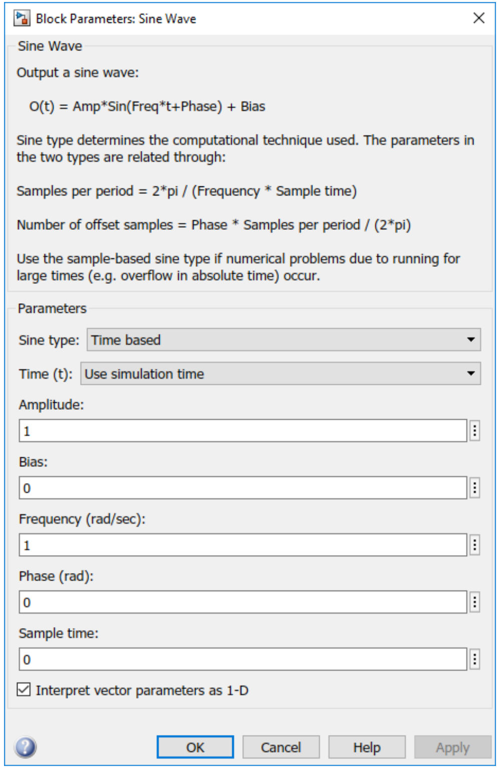
III. g. Draw Branched Signal Lines

- Drag the cursor to the Integrator input port, and then release the mouse button. The new line, called a branch line, carries the same signal that passes from the Sine Wave block to the Bus block.
- Drag line segments to straighten and align with blocks. Your model is now complete.



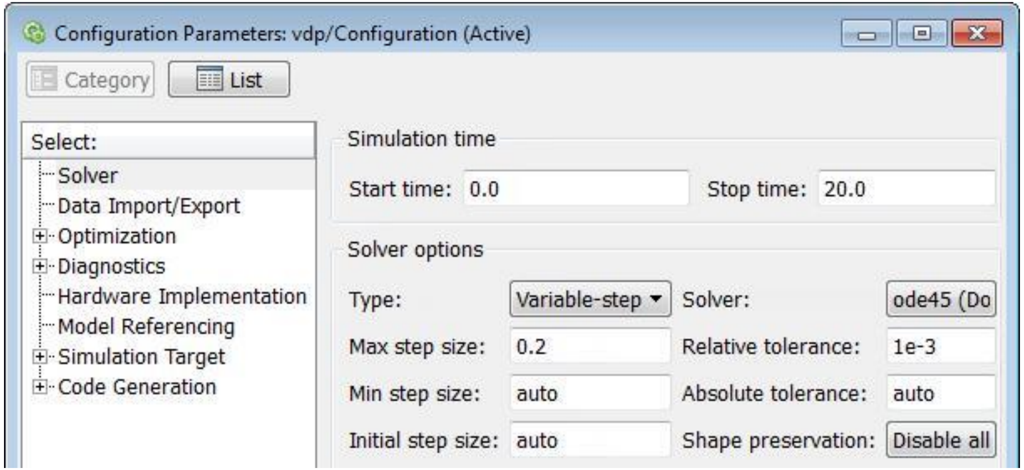
III. h. Define Simulation and Block Parameters

- Before you simulate the behavior of a model, define the simulation parameters.
- Simulation parameters include the type of numerical solver, start, and stop times, and maximum step size.
- Double-click Sine Wave block to open Block Parameters window.
- The parameters of sine wave (e.g., amplitude, frequency, phase, etc.) can be changed.
- For the simple model, the default parameters are used for the simulation



III. h. Define Simulation and Block Parameters

- From the Simulink Editor menu, select **Simulation > Model Configuration Parameters**. The Configuration Parameters dialog box opens to the Solver pane.
- In the Stop time field, enter **20**. In the **Max step size** field, enter **0.2**.
- Click **OK**.



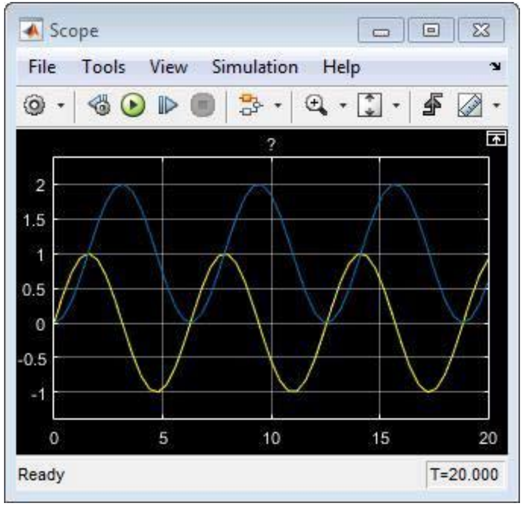
III. i. Run Simulation

- After you define the Model Configuration Parameters, you are ready to simulate your model.
- From the Simulink Editor menu bar, select **Simulation > Run**. The simulation runs, and then stops when it reaches the stop time specified in the Model Configuration Parameters dialog box.
- Alternatively, you can control a simulation by clicking
  - **Run** simulation button 

  and
  - **Pause** simulation button  on the Simulink Editor toolbar.

III. j. Observe Simulation Results

- After simulating the simple model, you can view the simulation results in a Scope window.
- Double-click the Scope block.
- The Scope window opens and displays the simulation results. The plot shows a sine wave signal with the resulting integrated sine wave signal.



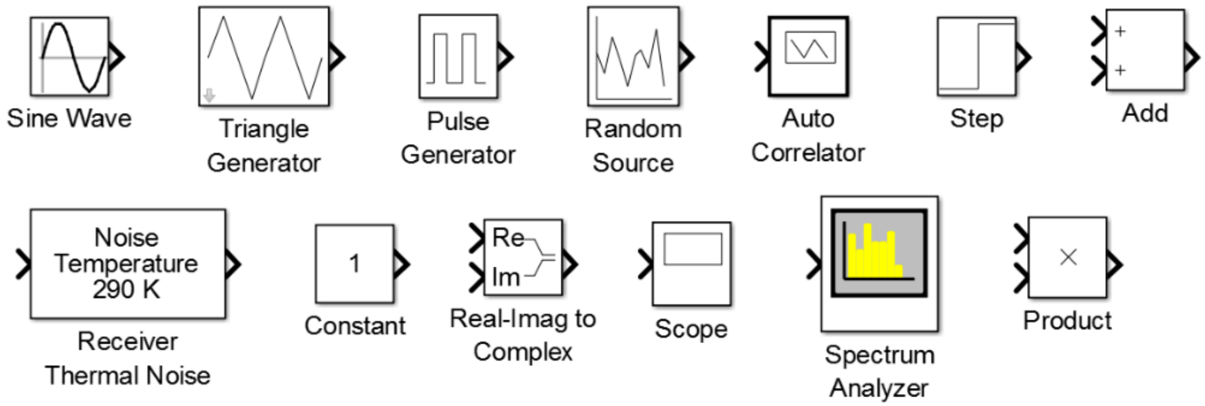
Part 1: Presentation of Signals and Noise

Objectives:

- To understand basic signal and noise concepts:
  - periodic/non-periodic signals,
  - deterministic/random signals,
  - Gaussian/thermal noise.
- To understand time-domain and frequency-domain analysis techniques

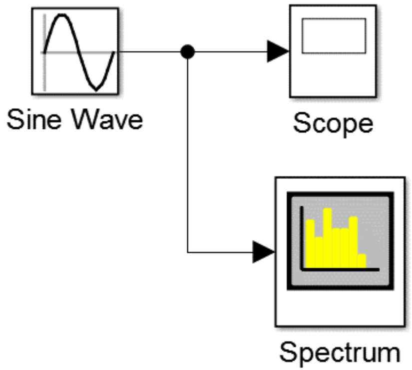
Preparation

- For this lab, the following Simulink blocks will be used.



Periodic signals

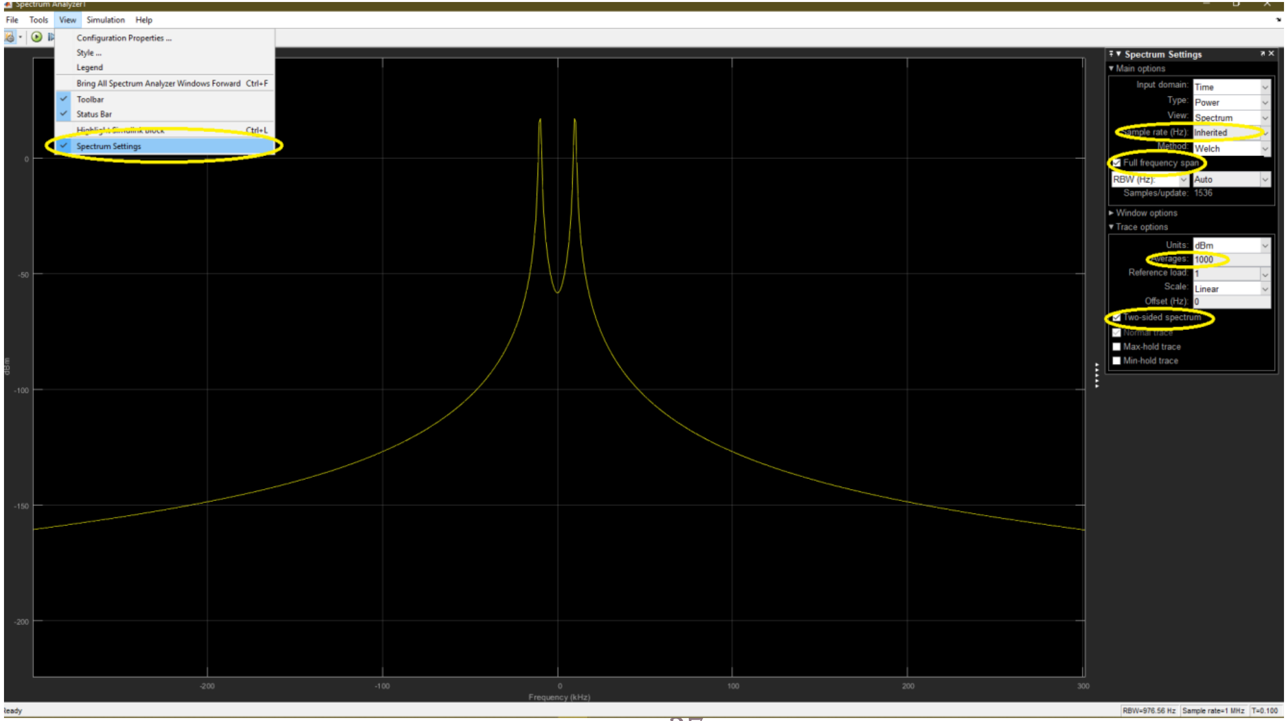
- Connect the blocks as illustrated.



- Parameter setup:
  - **Stop Time**: 1
  - **Sine Wave Sine type**: Sample based | Amplitude: 1 | Samples per period: 100 | Sample time: 1e-6
  - **Triangle Generator**: Frequency (Hz): 1e4 | Sample time: 1e-6
  - **Pulse Generator**: Pulse type: Sample based | Period (number of samples): 100 | Sample time: 1e-6

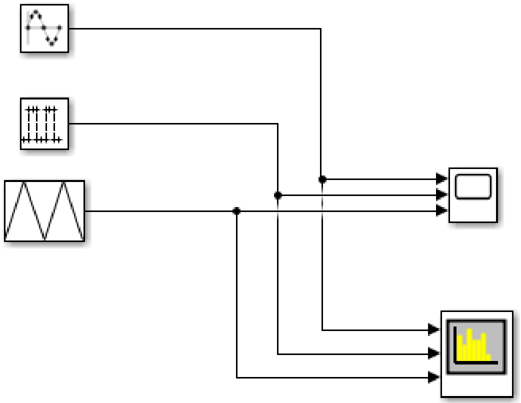
Spectrum Analyzer Configuration

- To obtain a more accurate spectrum, open the view/spectrum setting, in the Trace option window, input 1000 in front of Averages. Check the boxes for Full frequency span and two sided spectrum. Select Inherited for sample rate.



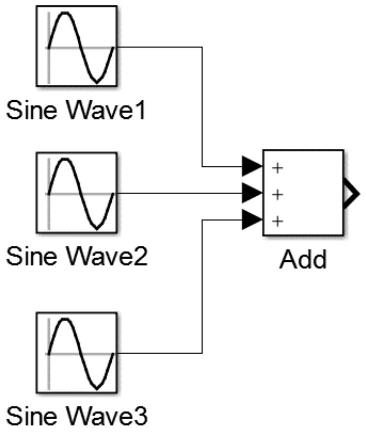
Periodic signals: Experiment

1. Observe the outputs on **Scope** and **Spectrum**. Plot the sine wave over three periods. Indicate the amplitude, the period, and the frequency of the sine wave. What are the fundamental and harmonic components?
2. Repeat Step 1 with a triangular wave generated by **Triangle Generator**.
3. Repeat Step 1 with a 50% duty cycle square wave generated by **Pulse Generator**.
4. Observe the scope and spectrum analyser for the following setup and comment on fundamental and harmonic components of the three signals



Sum of periodic signals

5. Repeat Step 1 with a sum of 3 sine waves as illustrated:

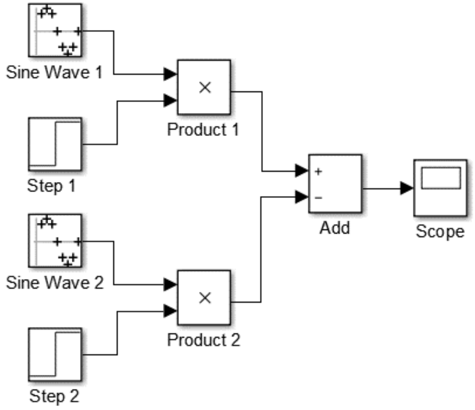


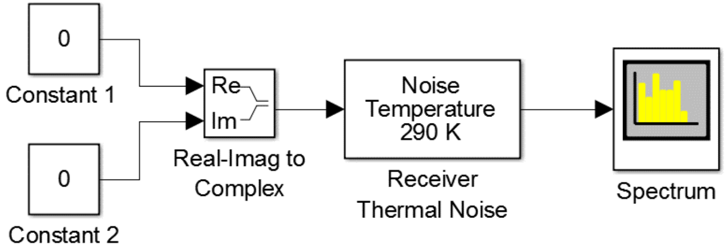
- Parameter setup:
  - **Sine Wave 1**: Sine type: Sample based | Amplitude: 1 | Samples per period: 50 | Sample time: 1e-6
  - **Sine Wave 2**: Sine type: Sample based | Amplitude: 1 | Samples per period: 100 | Sample time: 1e-6
  - **Sine Wave 3**: Sine type: Sample based | Amplitude: 1 | Samples per period: 200 | Sample time: 1e-6

Sum of signals

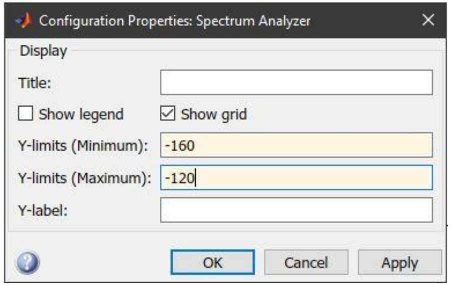
- Connect the blocks as illustrated.
- Parameter setup: **Sine Wave 1**: Amplitude: 0.5 | Samples per period: 1000 | Sample time: 0.001
  - **Step 1**: Step time: 2
  - **Sine Wave 2**: Amplitude: 0.5 | Samples per period: 1000 | Sample time: 0.001
  - **Step 2**: Step time: 4

6. Observe the output on **Scope**. Comment on the periodicity of the sine wave.



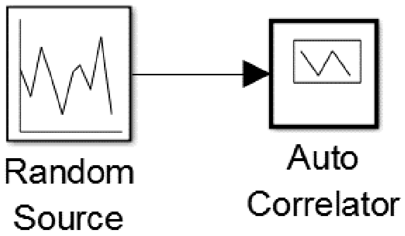


- Connect the blocks as illustrated.
- Parameter setup:
  - **Constant 1:** Constant value: 0 | Sample time: 1e-6
  - **Constant 2:** Constant value: 0 | Sample time: 1e-6
  - **Receiver Thermal Noise:** Specification method: Noise temperature | Noise temperature (K): 290 | Initial seed: randseed
- Spectrum Analyzer:
  - From view menu open: Configuration properties and set up as shown



7. What is the bandwidth and the power spectral density of thermal noise? To obtain a more accurate spectrum, open the view/spectrum setting, in the Trace option window, input 1000 in front of Averages.

- Connect the blocks as illustrated.



- Parameter setup:
  - **Random Source:** Source type: Gaussian | Sample time: 0.001
  - **Auto Correlator:** Length of buffer: 1024 | Sample time: 0.001

8. Observe the output on **Auto Correlator**. Vary the variance of the source. Explain how the peak value of the output on **Auto Correlator** is related to the variance, and thus the noise power.
9. Explain the difference between random signals and deterministic signals such as sine waves, triangular waves, etc. in terms of mathematical characterization.

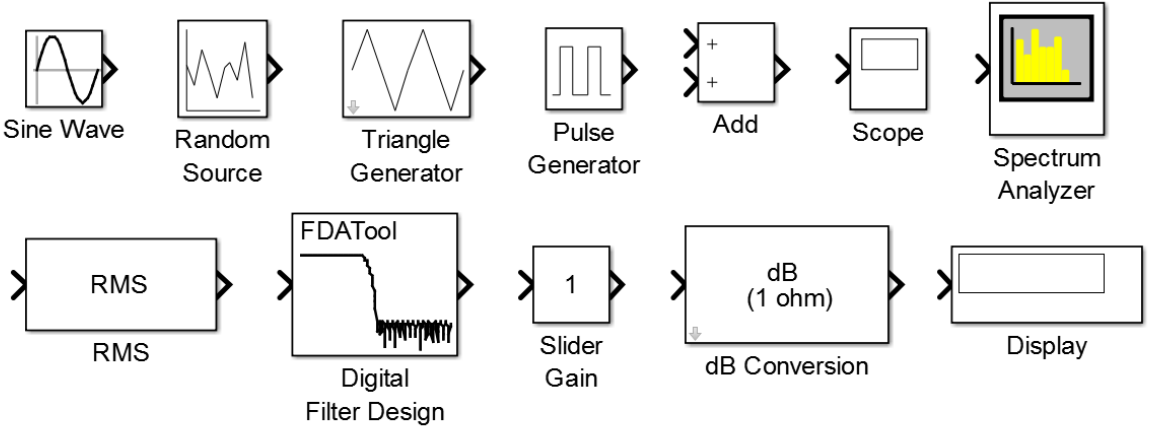
Part 2: Power, Bandwidth & SNR

Objectives:

- To understand the power and the bandwidth of a deterministic/random signal.
- To understand signal-to-noise power ratio (SNR), and filtering.

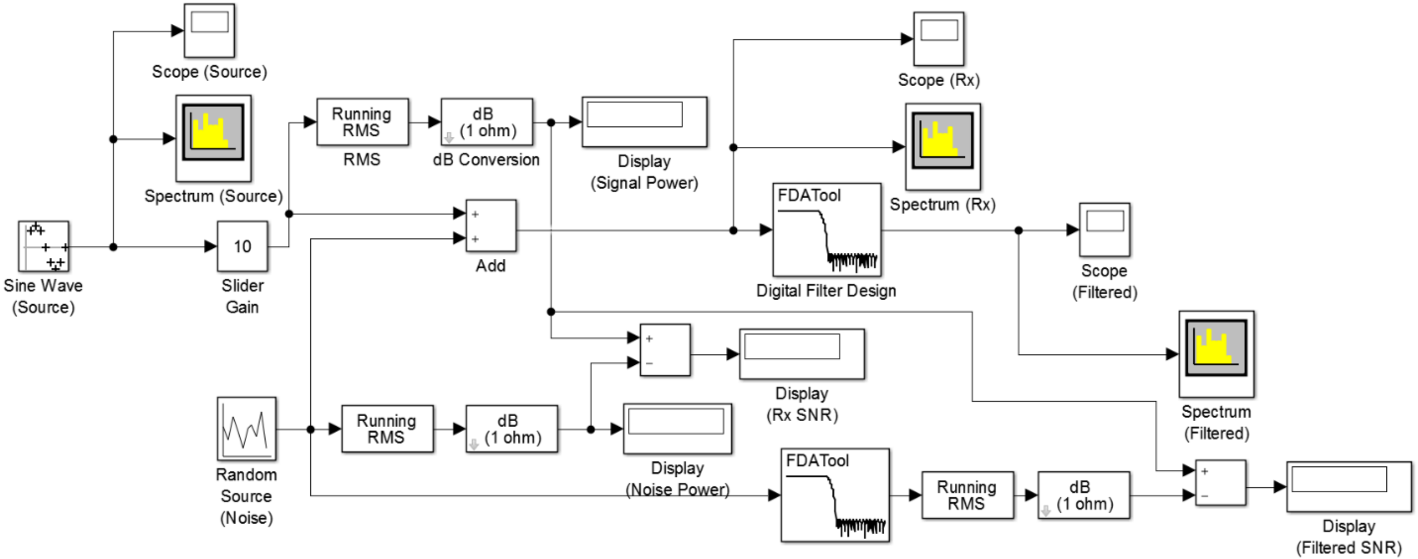
Preparation

- For this lab, the following Simulink blocks will be used.



SNR Measurement

- Connect the blocks as illustrated.



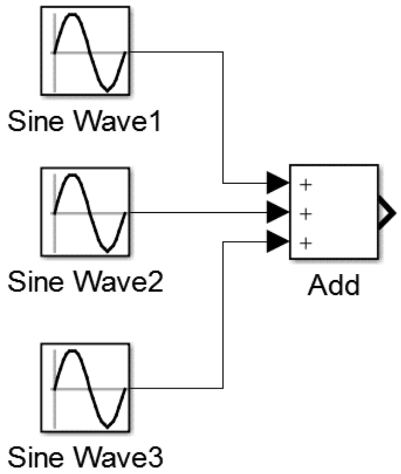
SNR Measurement: Experiment (1/3)

- Parameter setup:
  - **Sine Wave:** Sample based | Amplitude: 1 | Samples per period: 100 | Sample time: 1e-6
  - **Triangular Generator:** Frequency (Hz): 1e4 | Sample time: 1e-6
  - **Pulse Generator:** Pulse type: Sample based | Period (number of samples): 100 | Sample time: 1e-6
  - **Random Source:** Source type: Gaussian | Sample time: 1e-6
  - **Digital Filter Design:** Response Type: Lowpass | Design Method: FIR → Window |
    Filter Order: Specify order → 100 | Options: Scaled Passband, Window → Kaiser |
    Frequency Specifications: Units → Hz, Fs → 1e6 , Fc → 5e4
  - **RMS:** Running RMS

1. Observe the output on **Spectrum (Source)**. What are the power and the bandwidth of the sine wave?
2. Repeat step 1 with a triangular wave generated by **Triangular Generator**.
3. Repeat step 1 with a 50% duty cycle square wave by **Pulse Generator**.
4. Repeat step 1 with a 80% duty cycle square wave by **Pulse Generator**.

SNR Measurement: Experiment (2/3)

5. Repeat step 1 with a sum of 3 sine waves as illustrated:



- Parameter setup:
  - **Sine Wave 1**: Sine type: Sample based | Amplitude: 1 | Samples per period: 50 | Sample time: 1e-6
  - **Sine Wave 2**: Sine type: Sample based | Amplitude: 1 | Samples per period: 100 | Sample time: 1e-6
  - **Sine Wave 3**: Sine type: Sample based | Amplitude: 1 | Samples per period: 200 | Sample time: 1e-6

SNR Measurement: experiment (3/3)

Answer the following questions for the setup of question 5.

6. Observe the outputs on **Scope (Rx)** and **Spectrum (Rx).** Comment on the effect of noise on the signal in the time domain and the frequency domain.
7. Compare the outputs on **Scope (Filtered)** and **Spectrum (Filtered)** with those on **Scope (Rx)** and **Spectrum (Rx),** respectively. Comment on the effect of filtering.
8. Vary **Slider Gain** from small to large. Observe the outputs on **Scope (Filtered)** and **Spectrum (Filtered).** Comment on how the effect of noise varies in accordance with the SNR at the filter output. Repeat for a varied cut-off frequency Fc in **Digital Filter Design**.