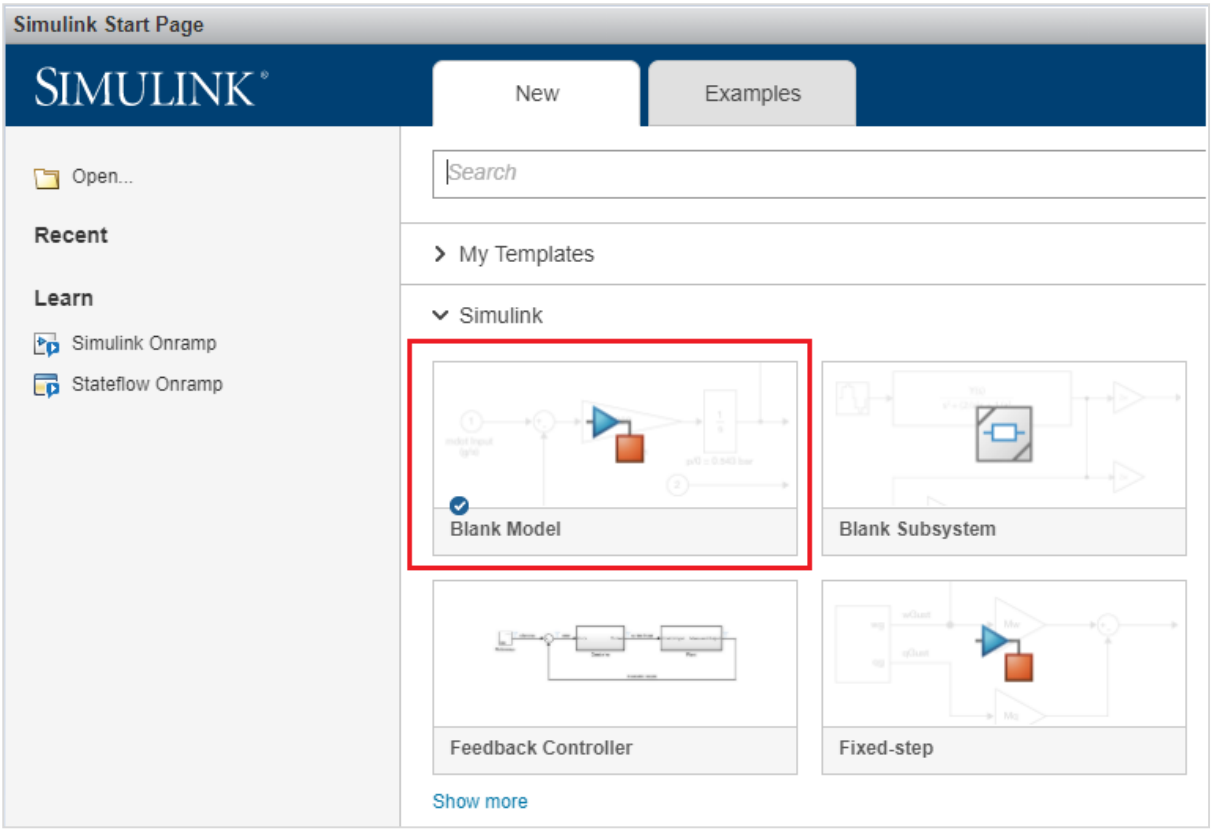
5. MATLAB Simulink — Lines

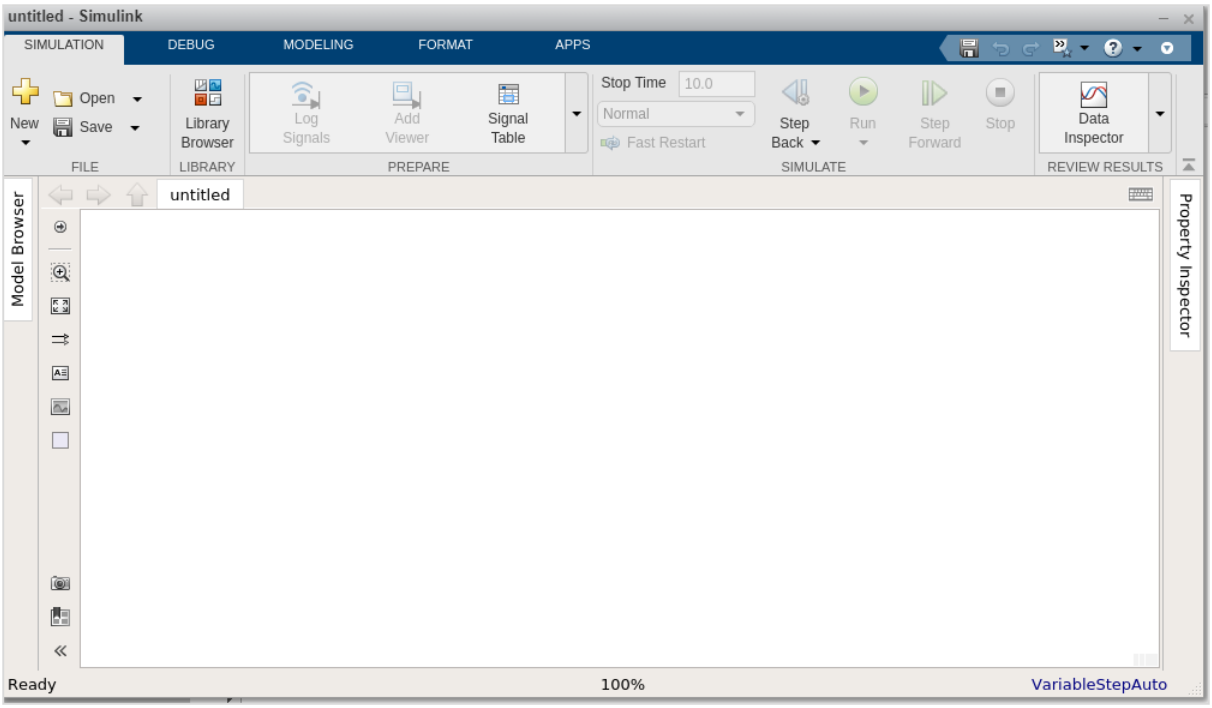
In the previous chapter, we learnt about the different types of blocks which are available with Simulink library. In this chapter, we are going to understand about lines.

Lines are used to connect the blocks with an arrow. Each block will have its own input and output connector. The communication between the blocks will take place with the help of lines.

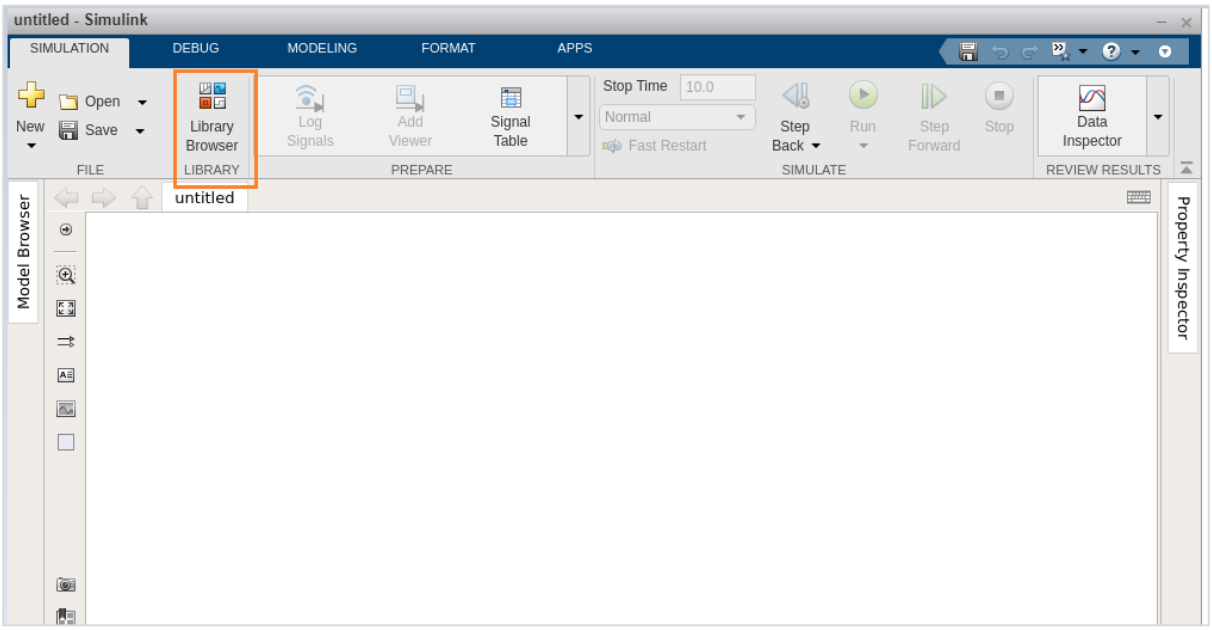
Let us understand the same with an example. Select a blank model from Simulink page as shown below:



It will open a blank model workspace as shown below:



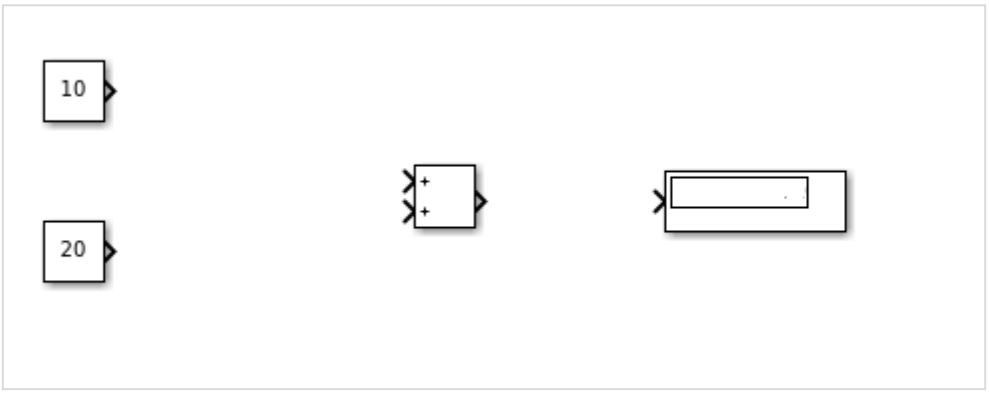
Click on Simulink Library browser to drag some blocks in the model workspace.



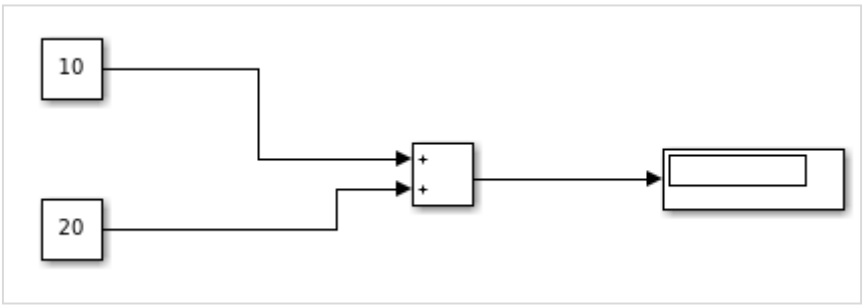
Consider that in the model, we want to add two given numbers. So let us pick the Add block, the display block and the constant block.

The constant block has one output connector, the Add block has two input connectors and the display block has one input connector respectively. You can drag the link from one output to another input as shown below.

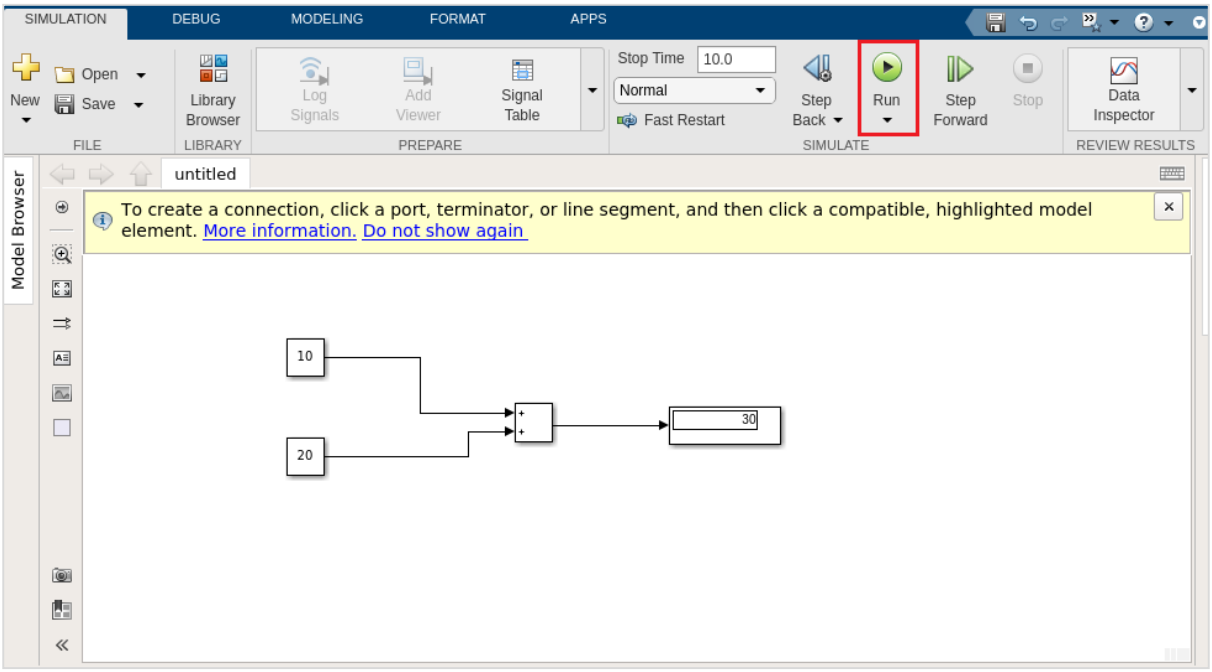
Here, we have two constants with values 10 and 20. They are connected to the add block with lines. The add block is connected to display with a line.



When the lines are connected, the display is as follows:



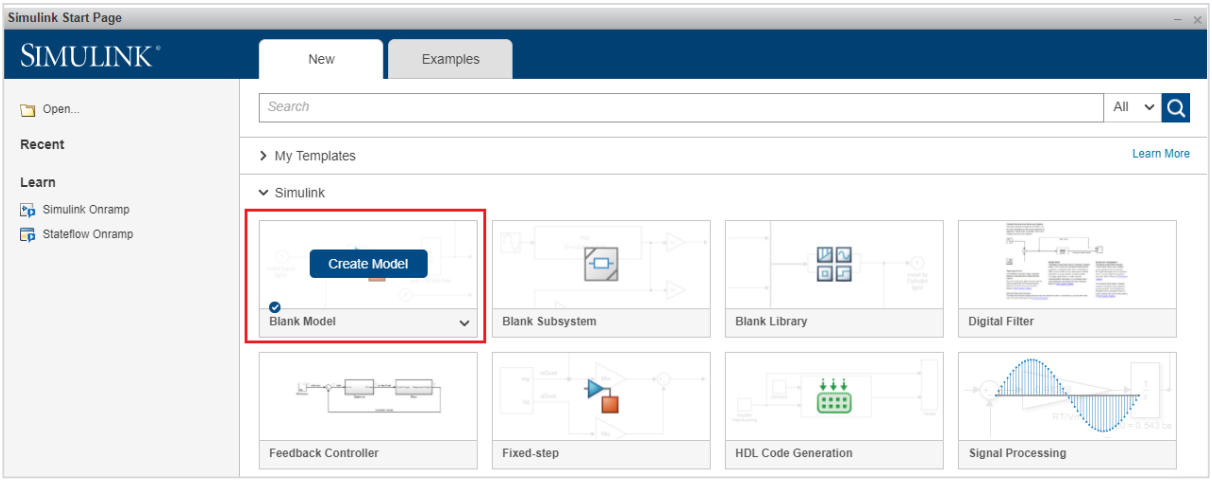
Now click on Run to see the result in the display block. It will add 10 + 20 to give the result as 30 in the display block.



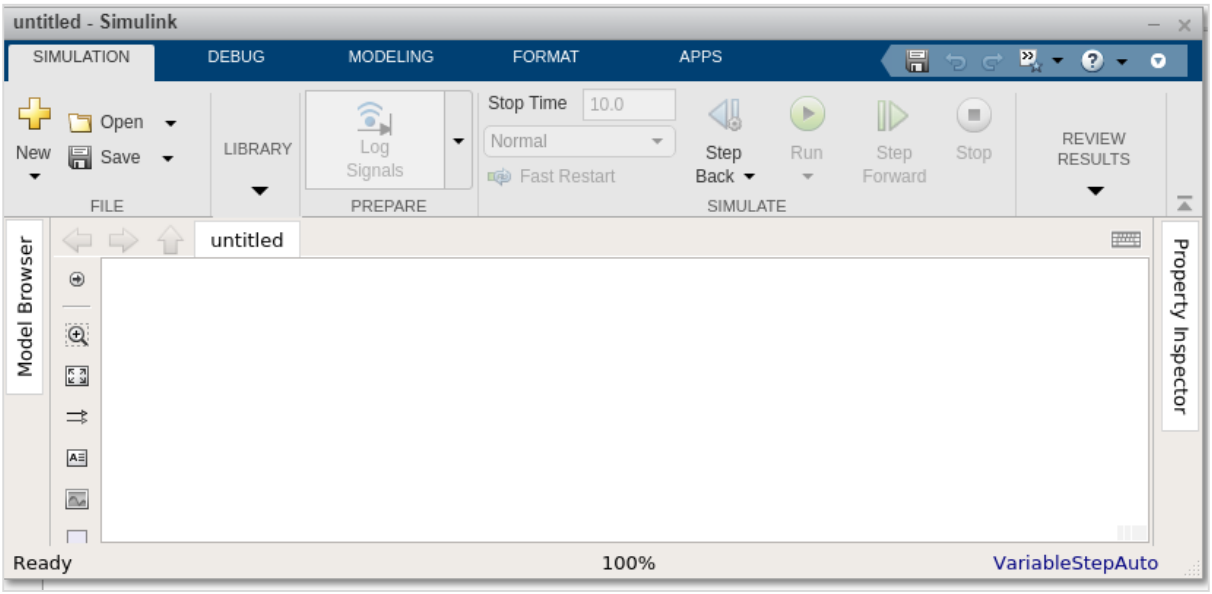
6. MATLAB Simulink — Build & Simulate Model

We have seen the Simulink library browser and the blocks available in the library list. In this chapter, we are going to use the blocks to build a simple sine wave model.

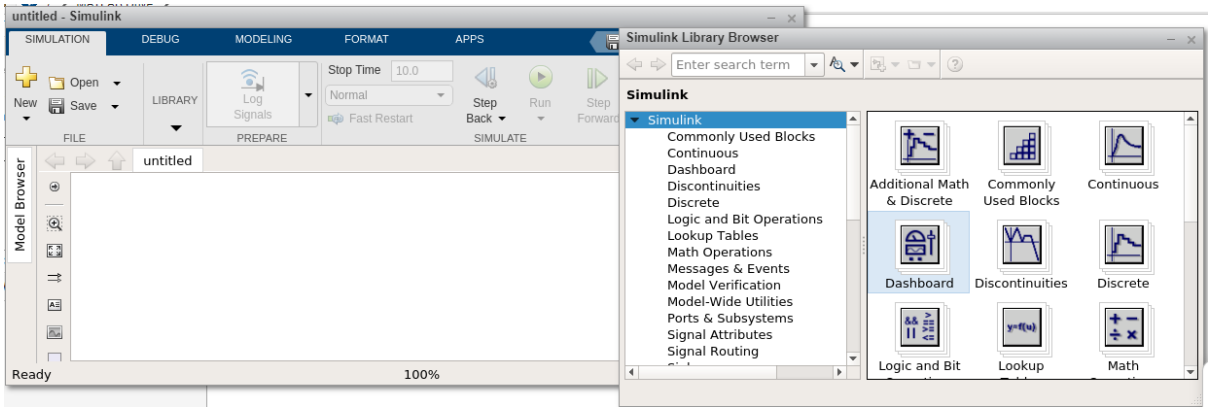
Open Simulink and click on blank model as shown below:



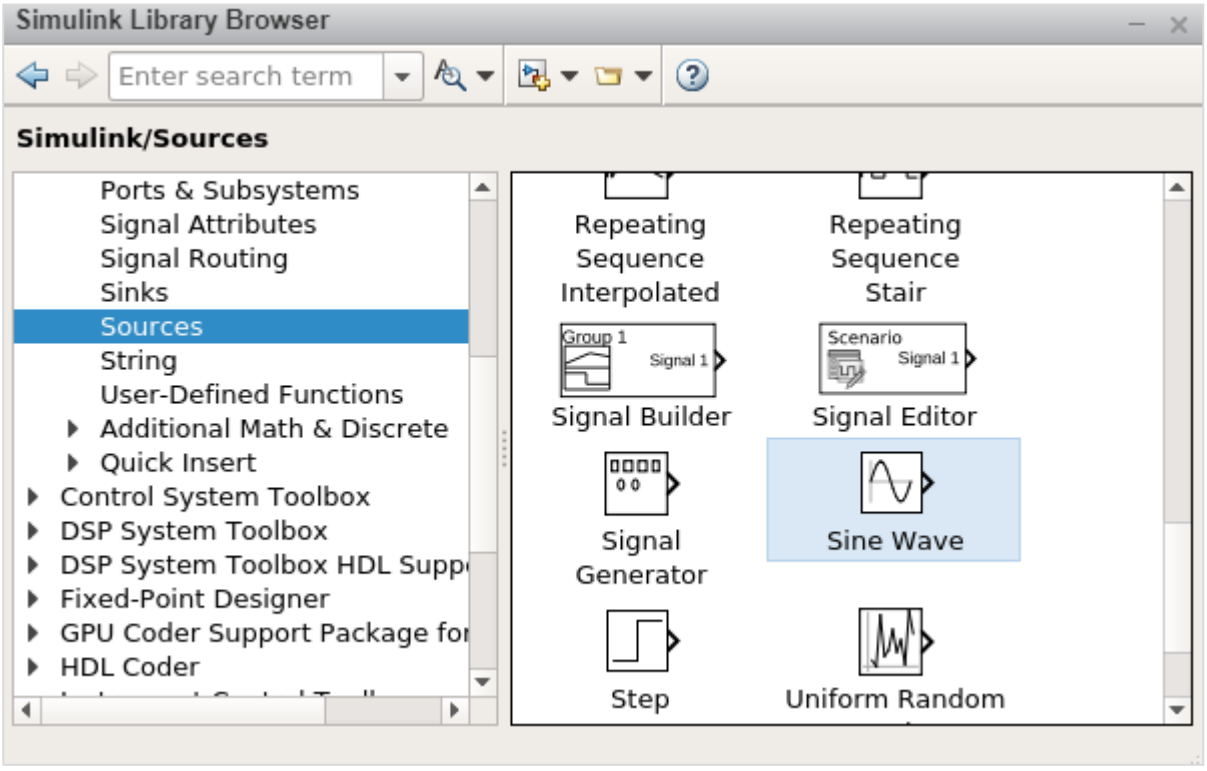
The blank model will open a blank popup window as shown below:



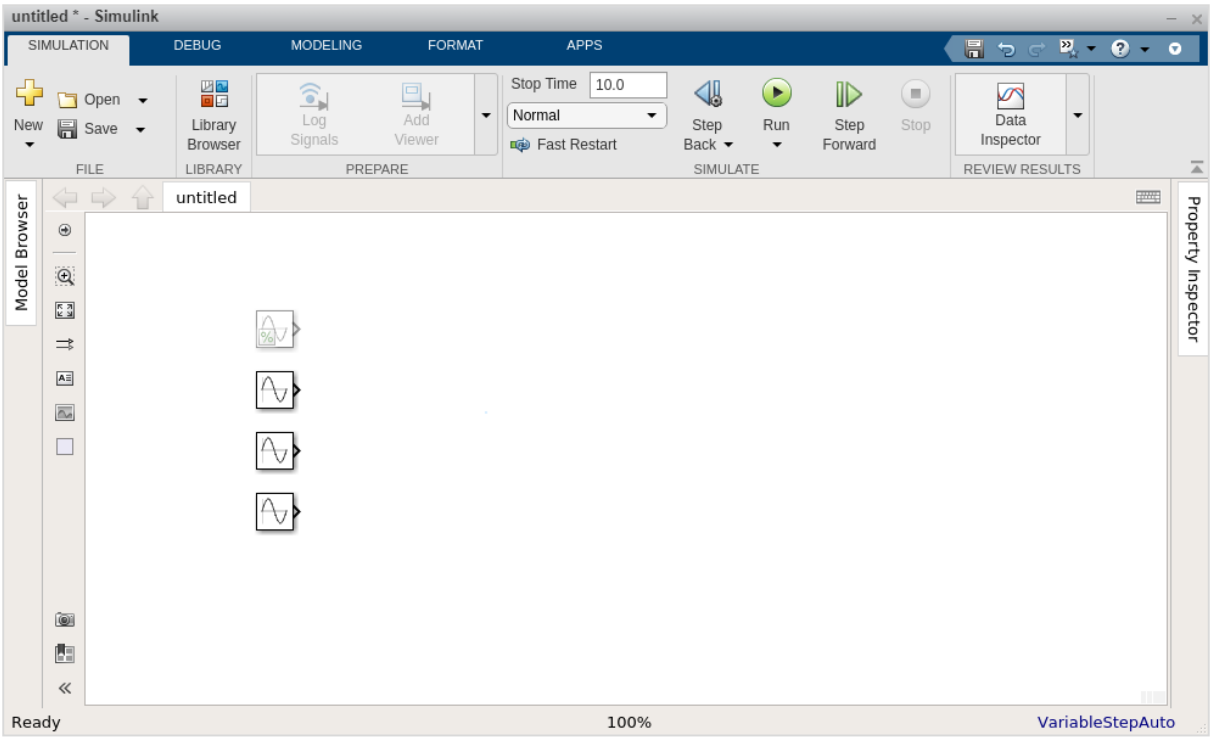
Now, open the Simulink Library browser so that we can select the blocks.



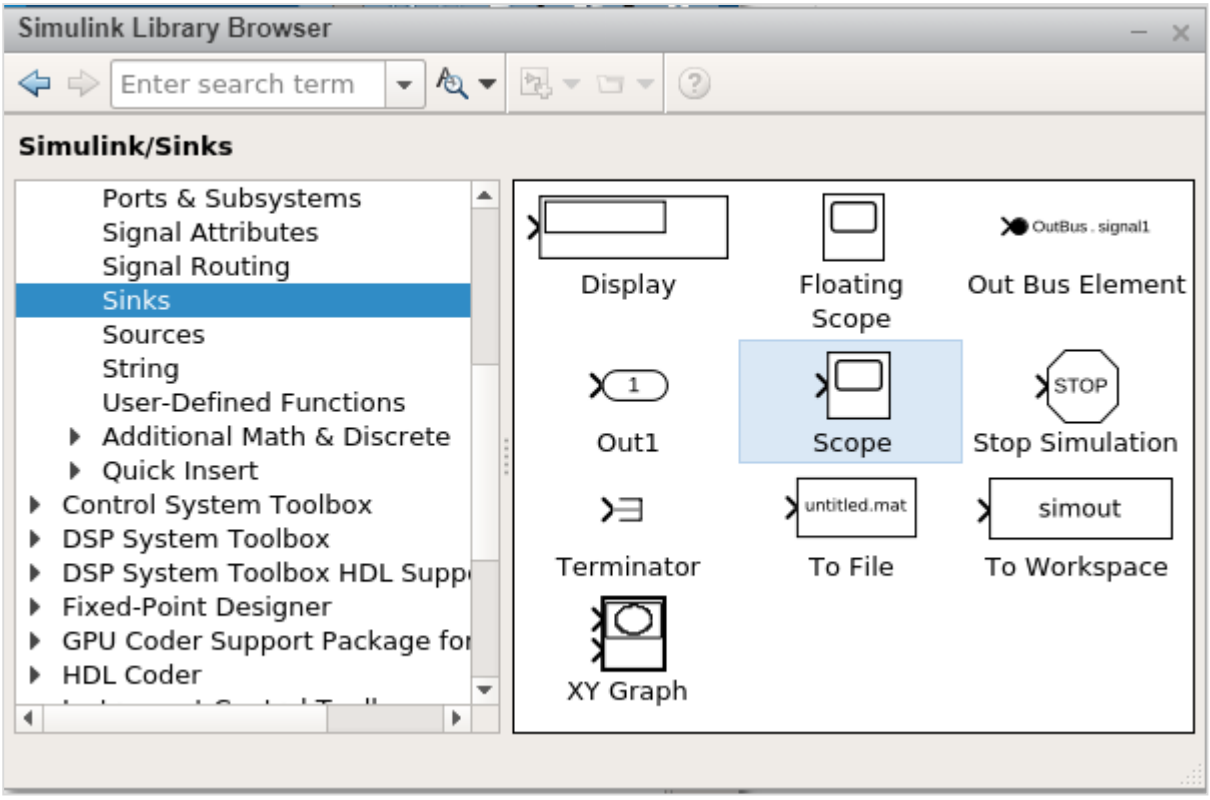
The following screen will appear on your computer:



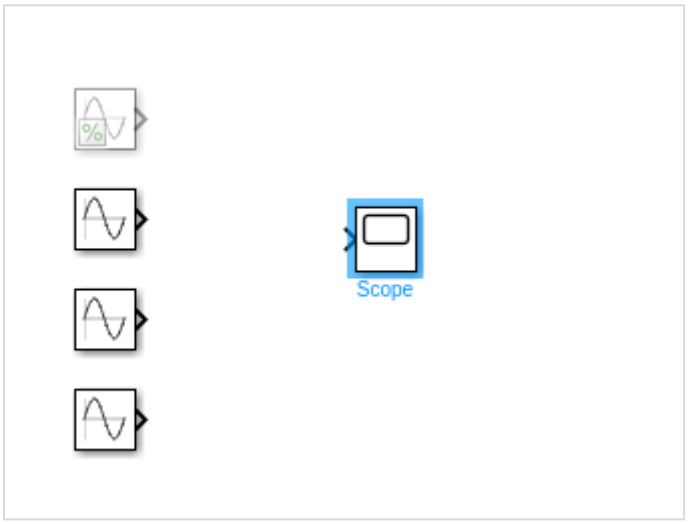
To move the sine wave to your model, select the block and drag it inside your model workspace. We want to display the sine wave and here, we have taken four sign wave in our model workspace as shown below:



Now we want to display the output of the signal, so let us use a Scope block from sinks library as shown below:

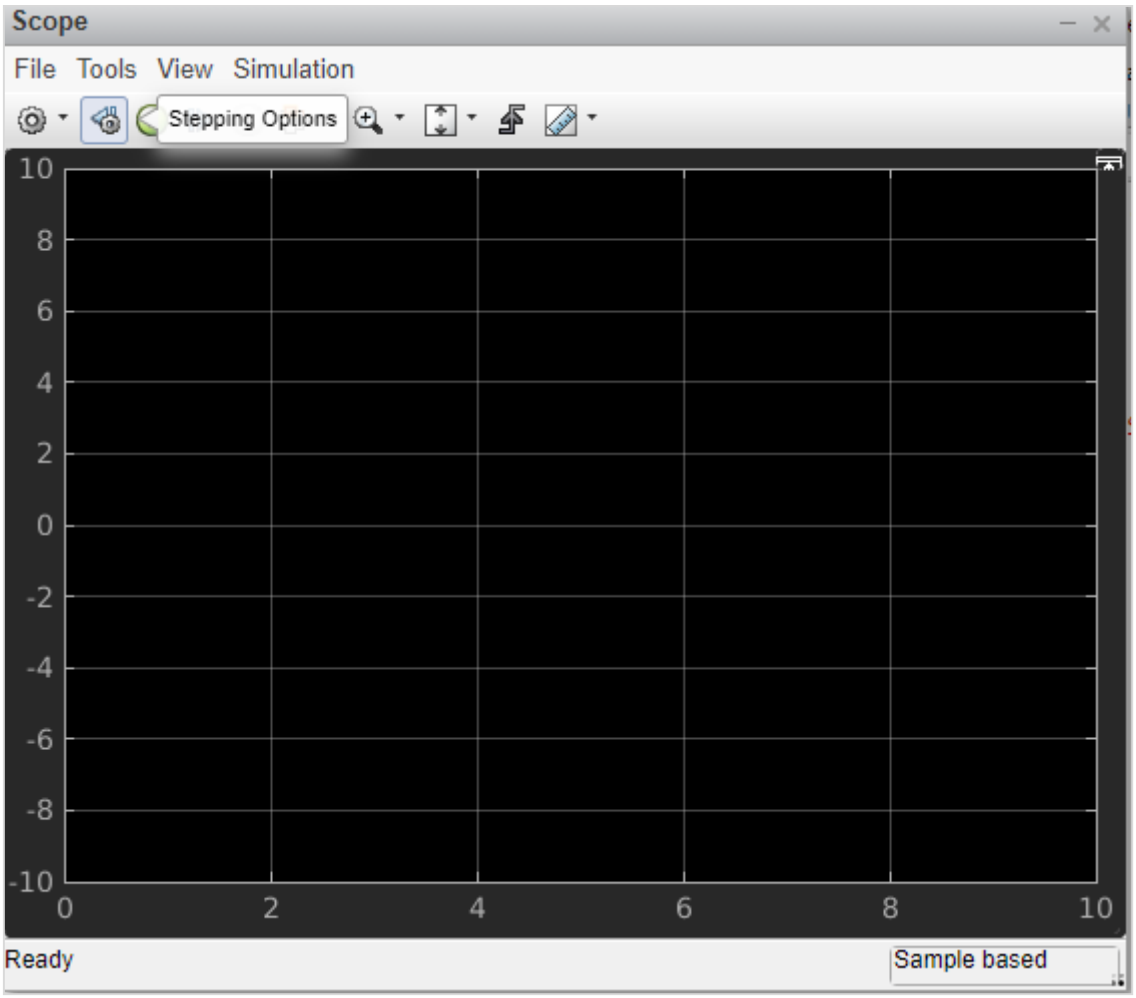


Now select and drag the Scope block inside your model workspace.

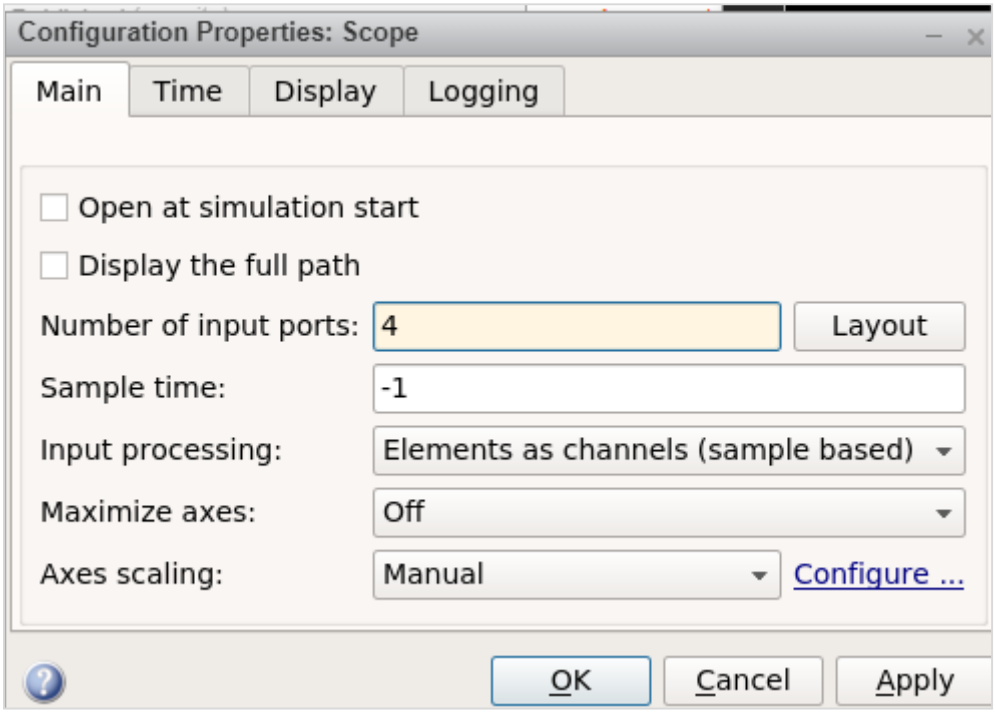


The sine wave has one output and the scope block has one input. We have four sine waves displayed. We have to change the parameters of scope block to take four inputs.

Right click on scope block and click on Block Parameters and it will display the screen as shown below:



Go to settings icon and change the input parameter from 1 to 4 as shown below:

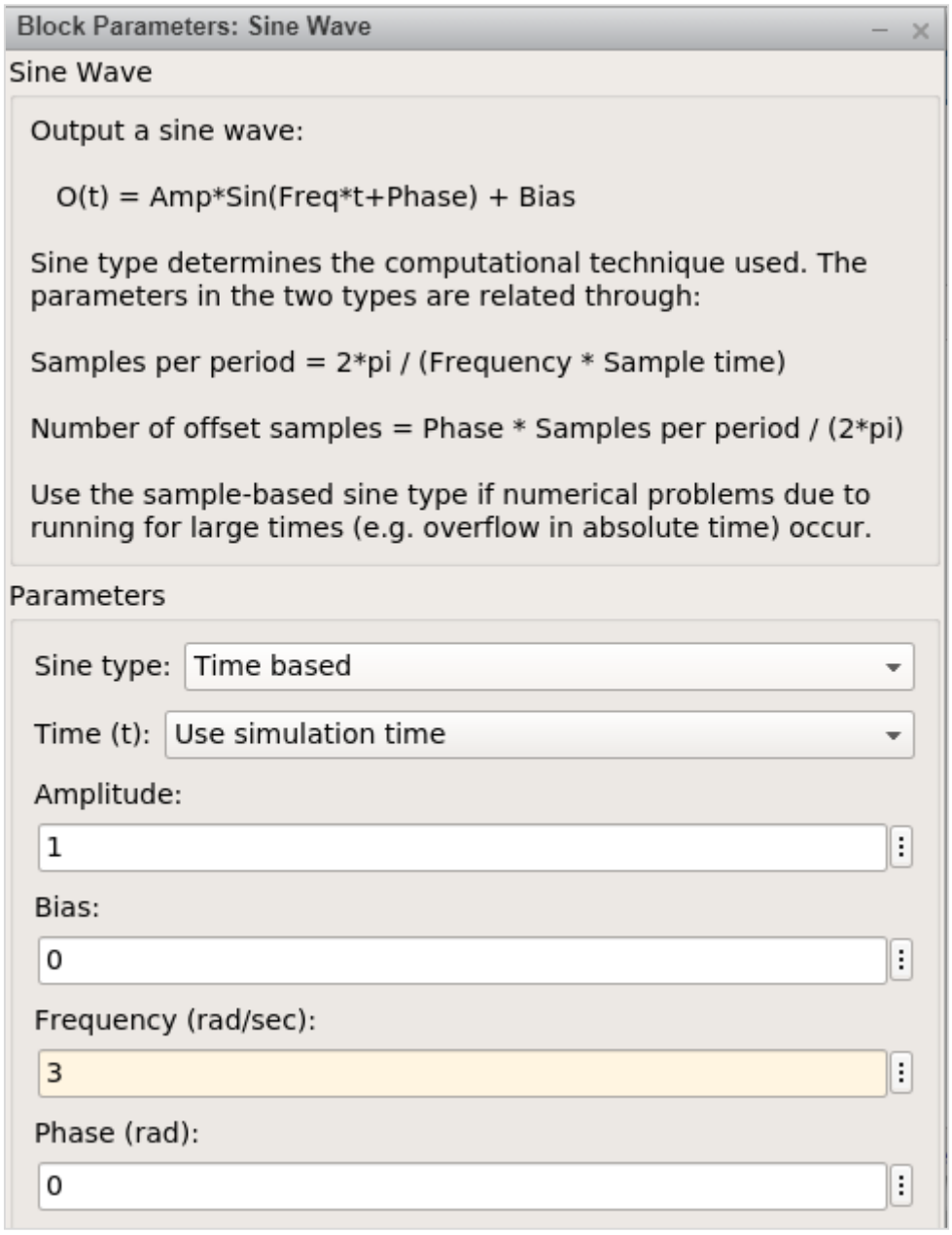


Click on Apply to save the changes.

Let us now connect the sine wave to the scope block with arrows as shown below:

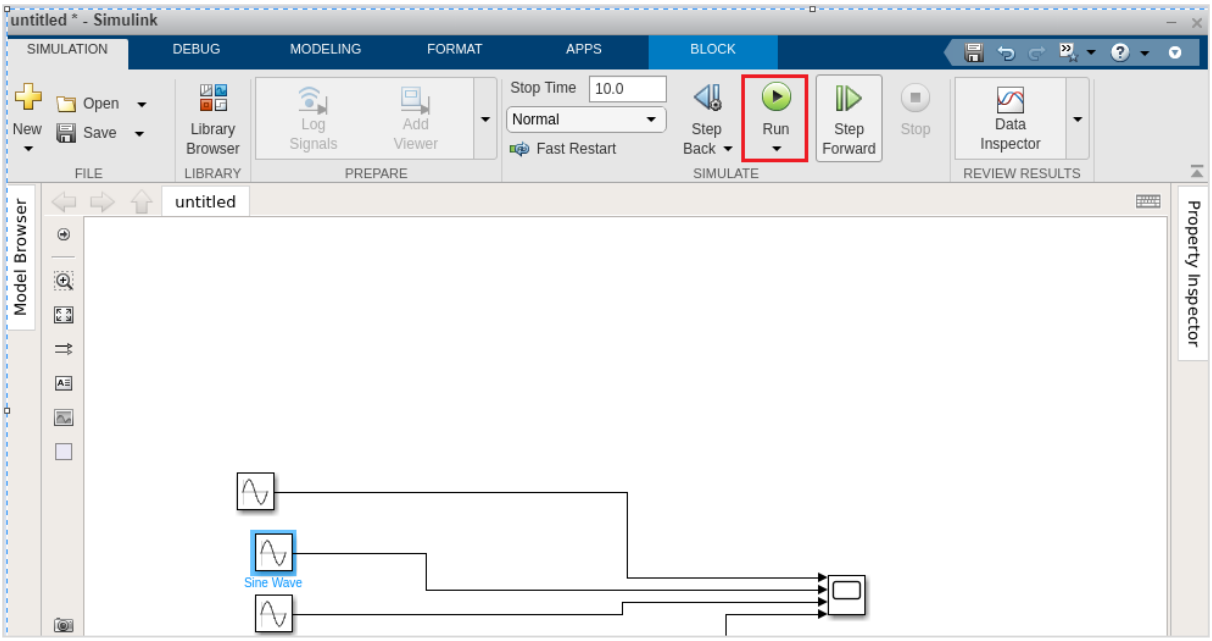
We would like to change the frequency of each sine wave to a different one, so that we get a signal graph of different frequencies.

So right click on sine wave and open the Sine wave block parameters as shown below:



We are going to keep the amplitude as 1 for all sine waves and the frequency of the first sine wave is 1, second one is 3, third one is 6 and the last one is 10.

Click on the Run button as shown below to see the sine wave.

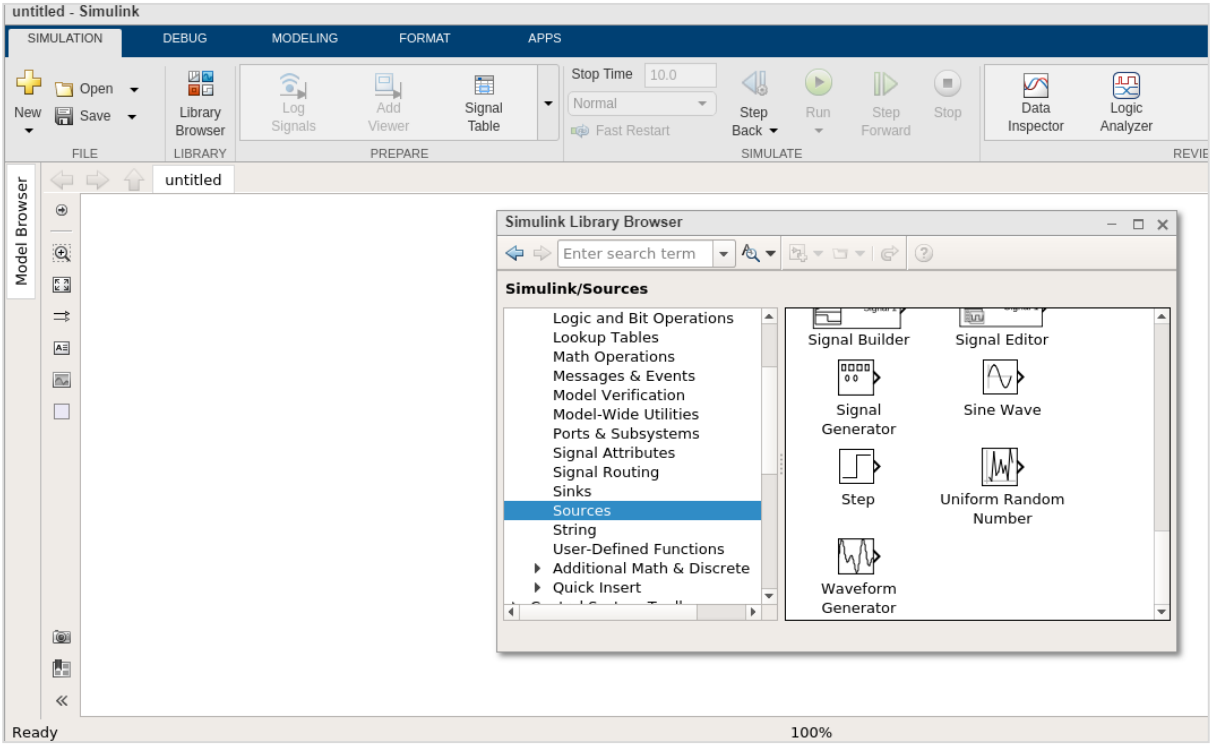


Open the scope block parameters to see the sine wave as shown below:

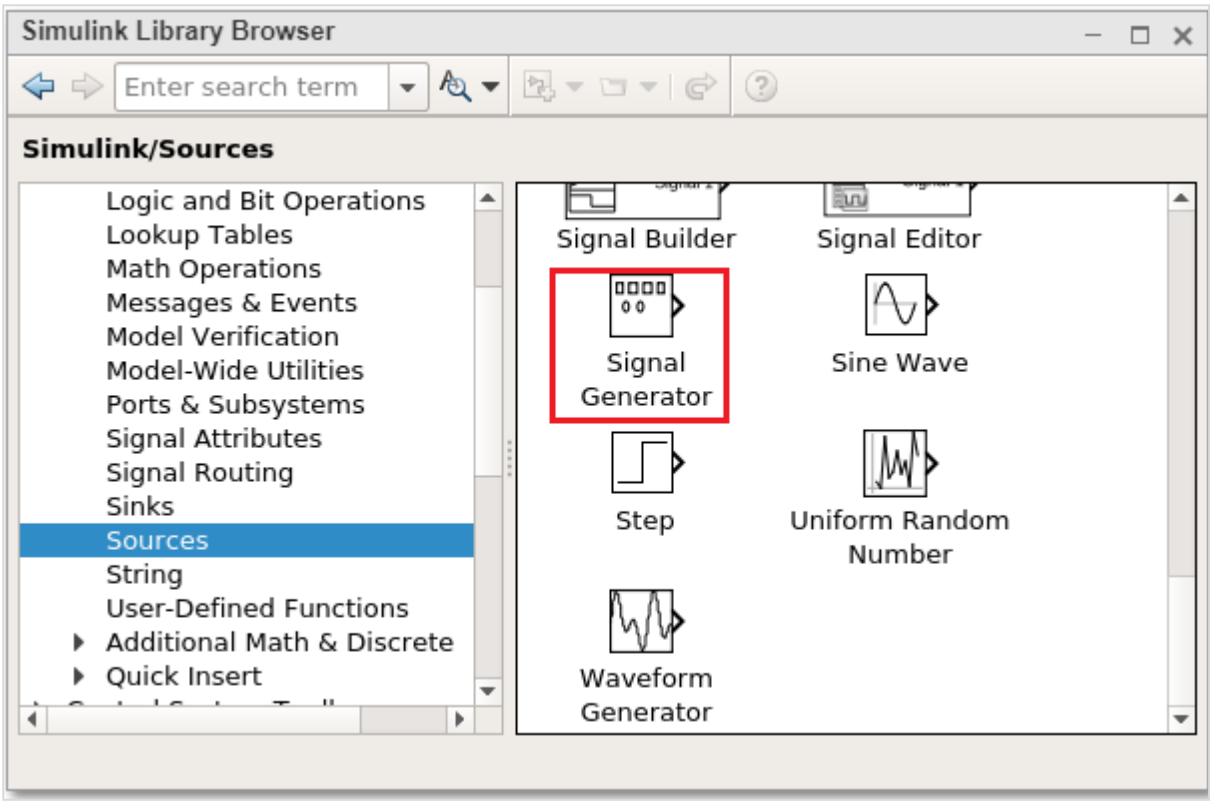
7. MATLAB Simulink — Signals Processing

In this chapter, we will understand the signals generation in Simulink.

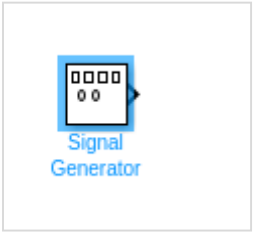
To start with, select a blank model from Simulink page and open Simulink browser library as shown below:



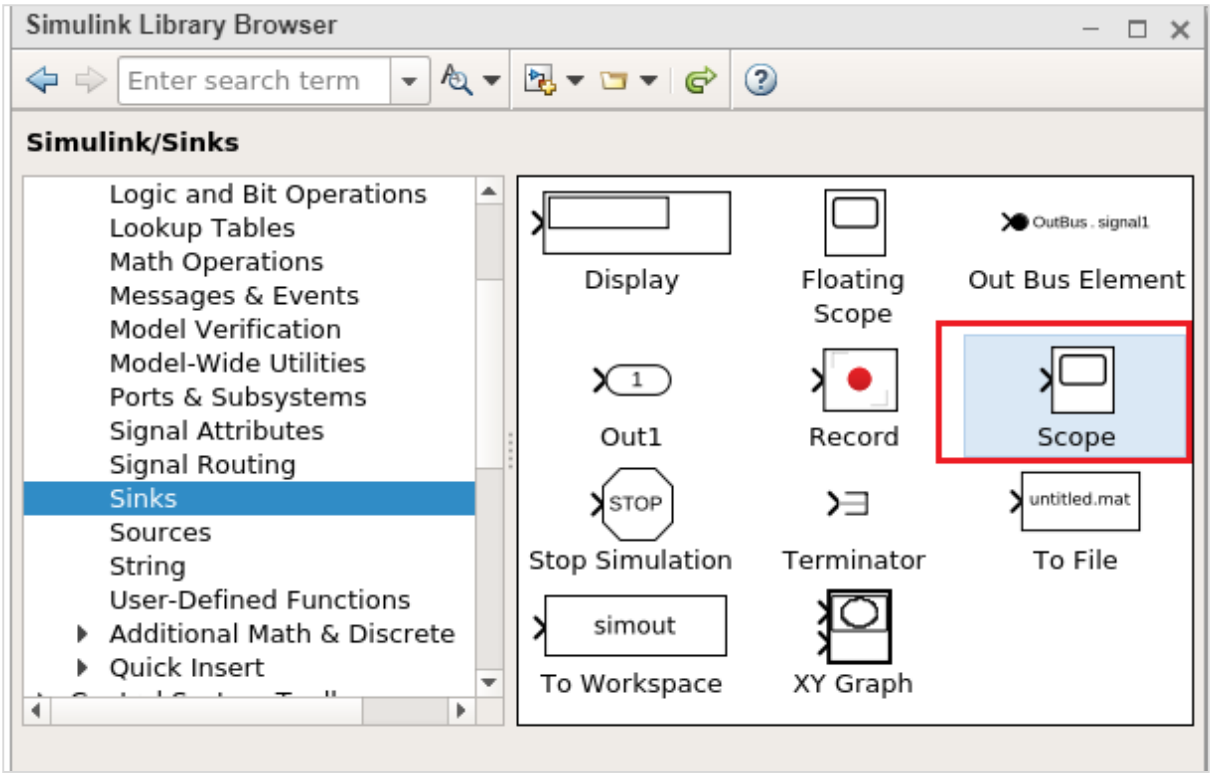
In sources library, you will get a signal generator symbol. It will help us to create different types of signals.



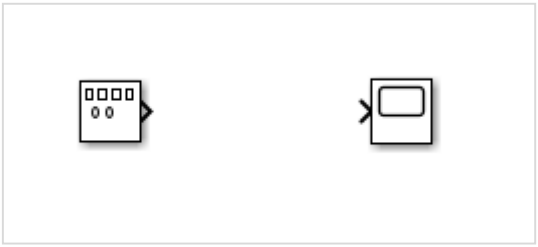
Select the Signal Generator and drag it to get inside the blank model as shown below:



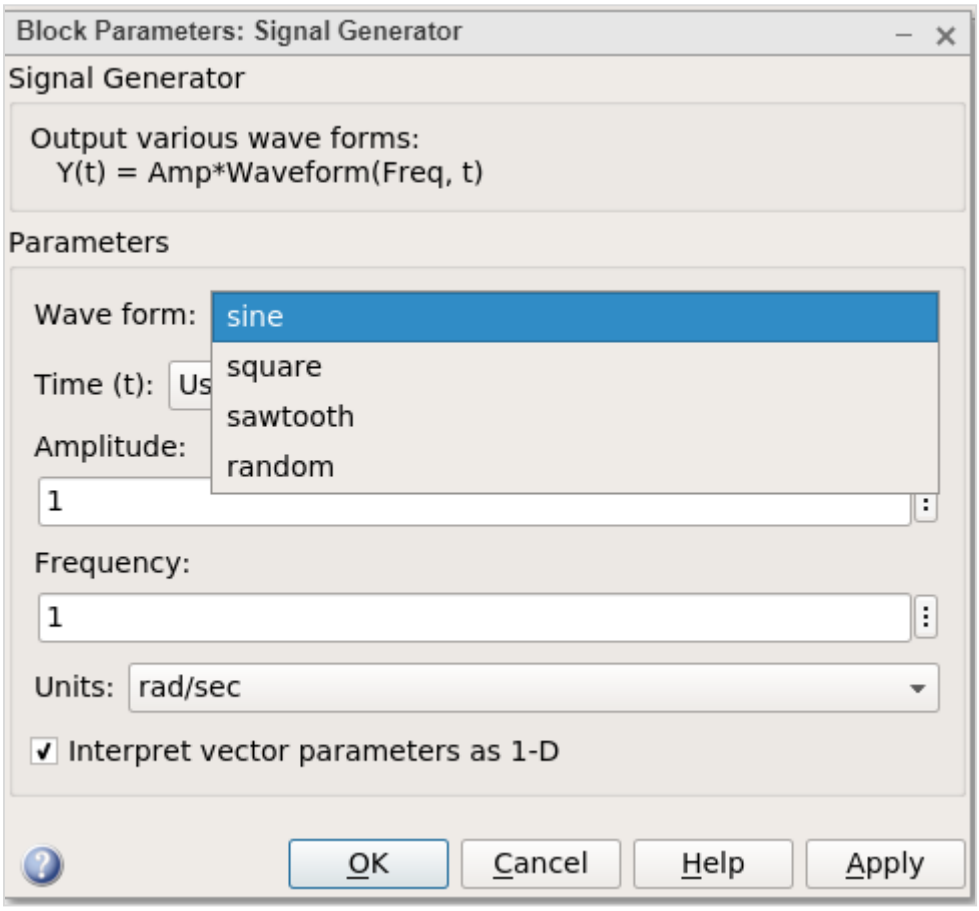
To see the output of the signal generator, we need one more block called scope from sinks library as shown below:



Select the block and drag it to get inside the model.

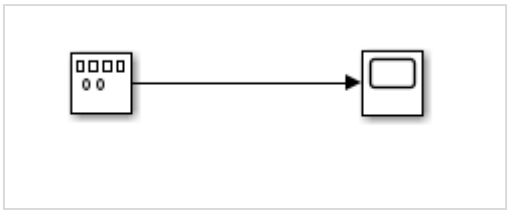


Double click on signal generator or right click and select block parameters and it will display a screen as shown below:

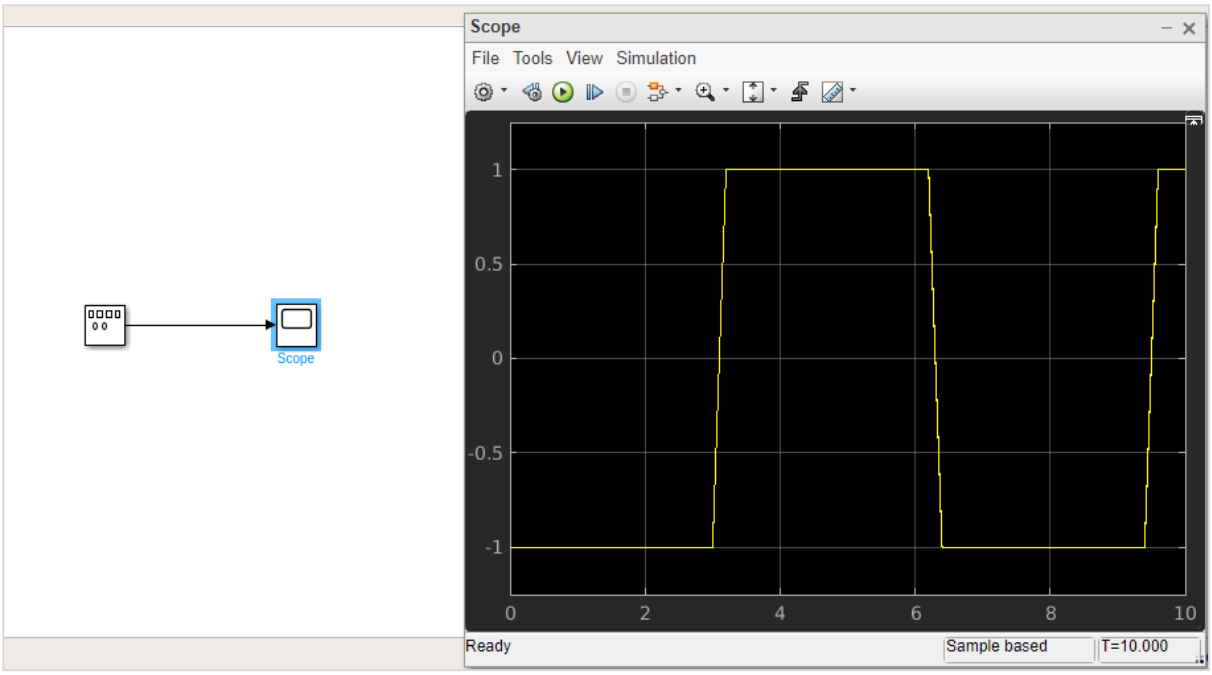


The signal generator can show waveforms like sine, square, sawtooth, random. We will select the square waveform. Let the amplitude and frequency be as 1. Click on OK to update the changes made.

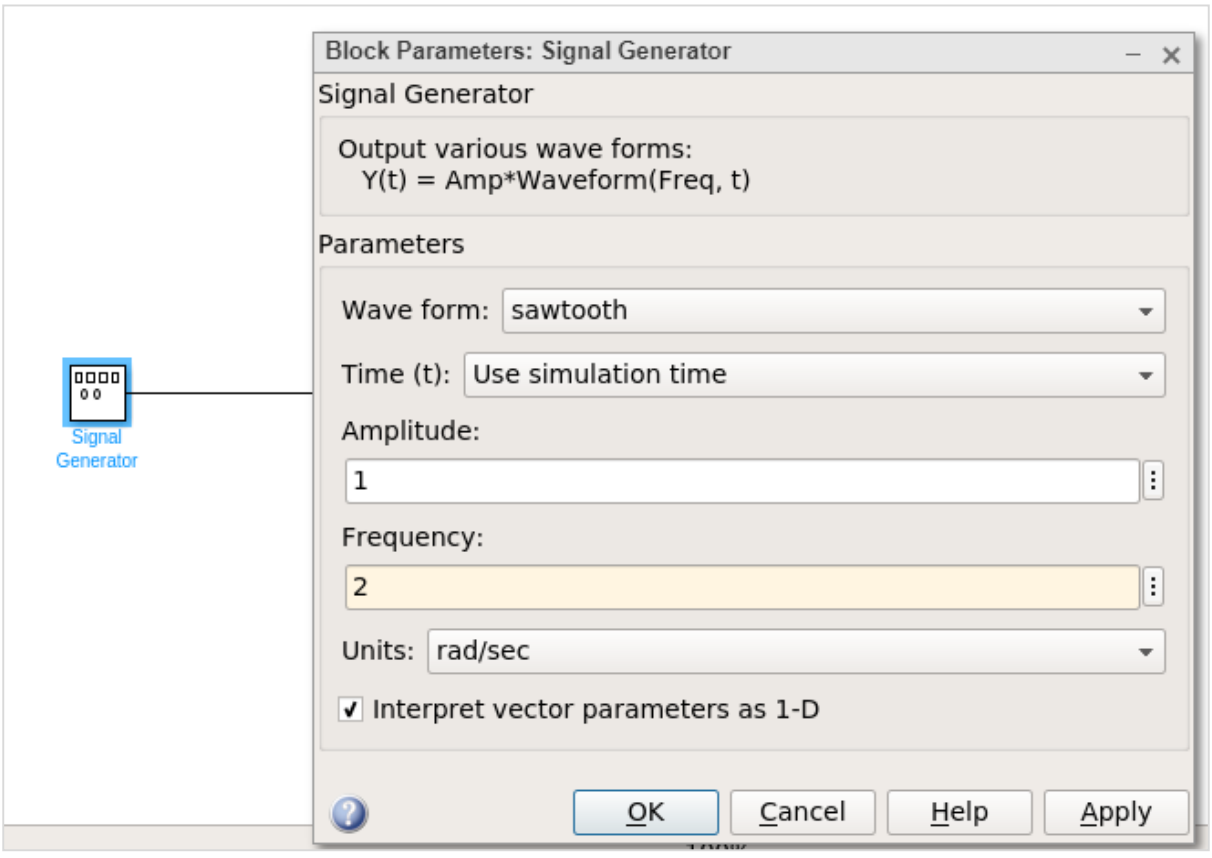
Now, connect the lines between signal generator and scope block as shown below:



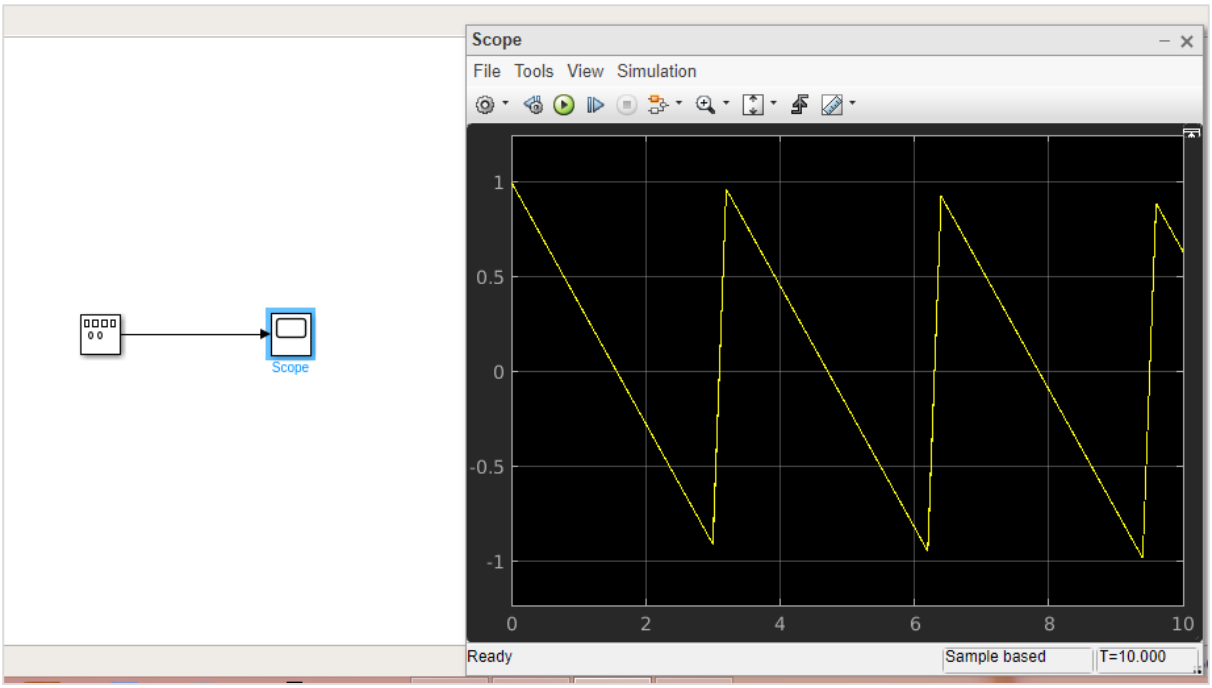
Now click on Run button to see the square waveform as shown below:



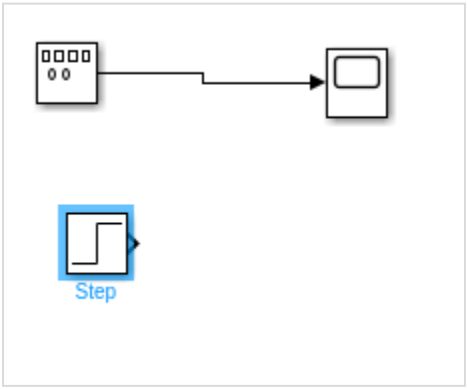
Let us now try the sawtooth wave form. Right click signal generator or double click and change the waveform to sawtooth.



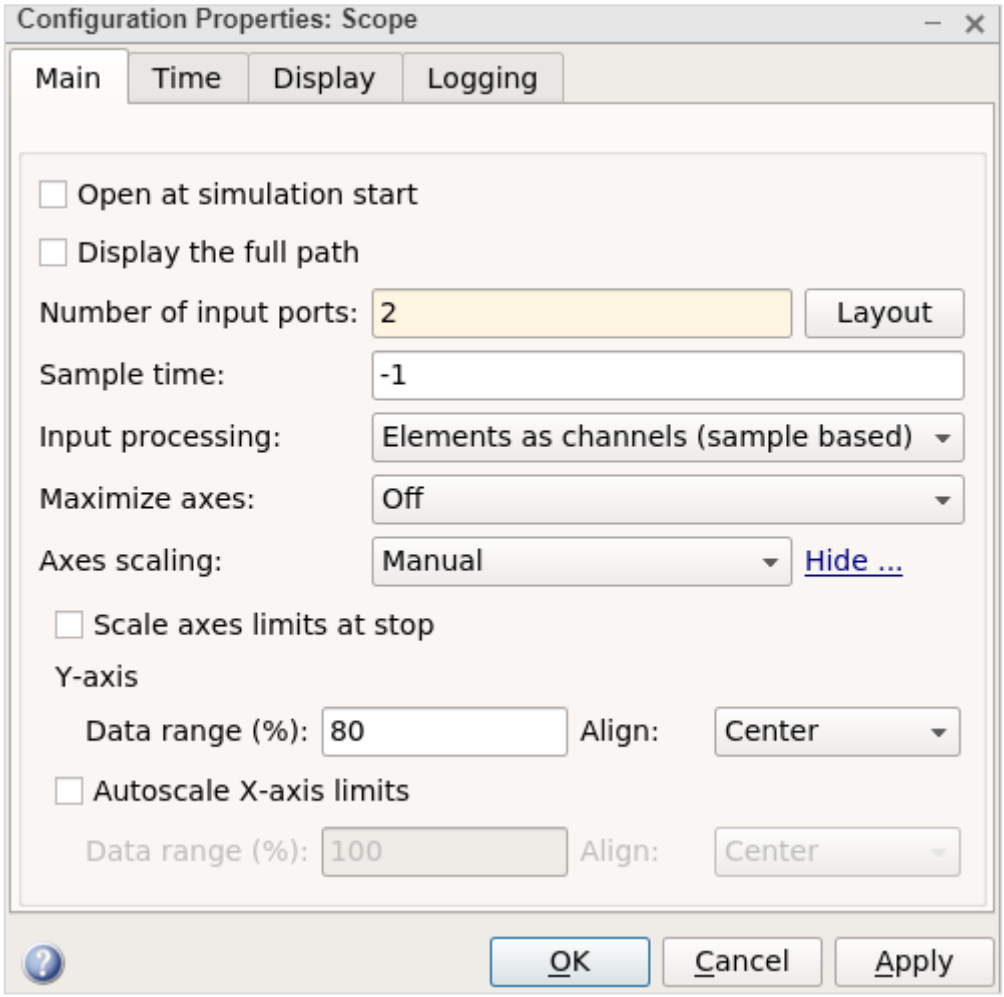
Let us change the frequency to 2. Click on OK to update the changes. Now run the model to see the changes as shown below:



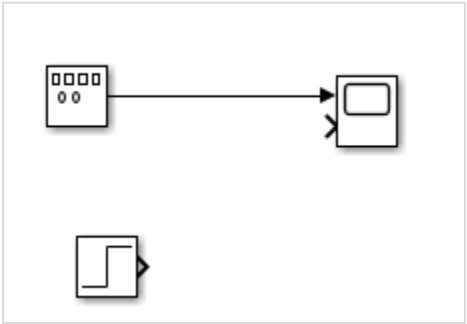
Let us now add some more signals to the above model. We will take the step signal from the sources library as shown below:



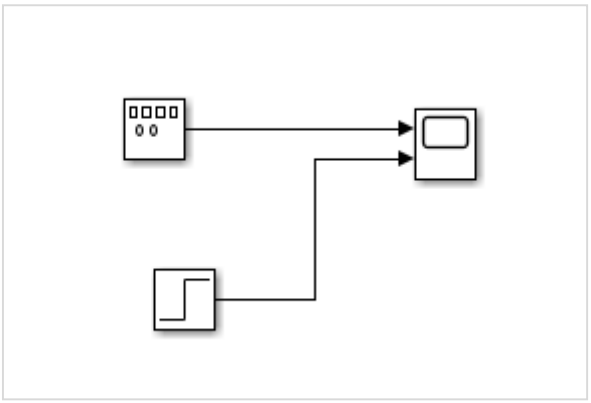
We just have one input for the scope block. Let us increase it to 2 inputs. Right click and open the block parameters.



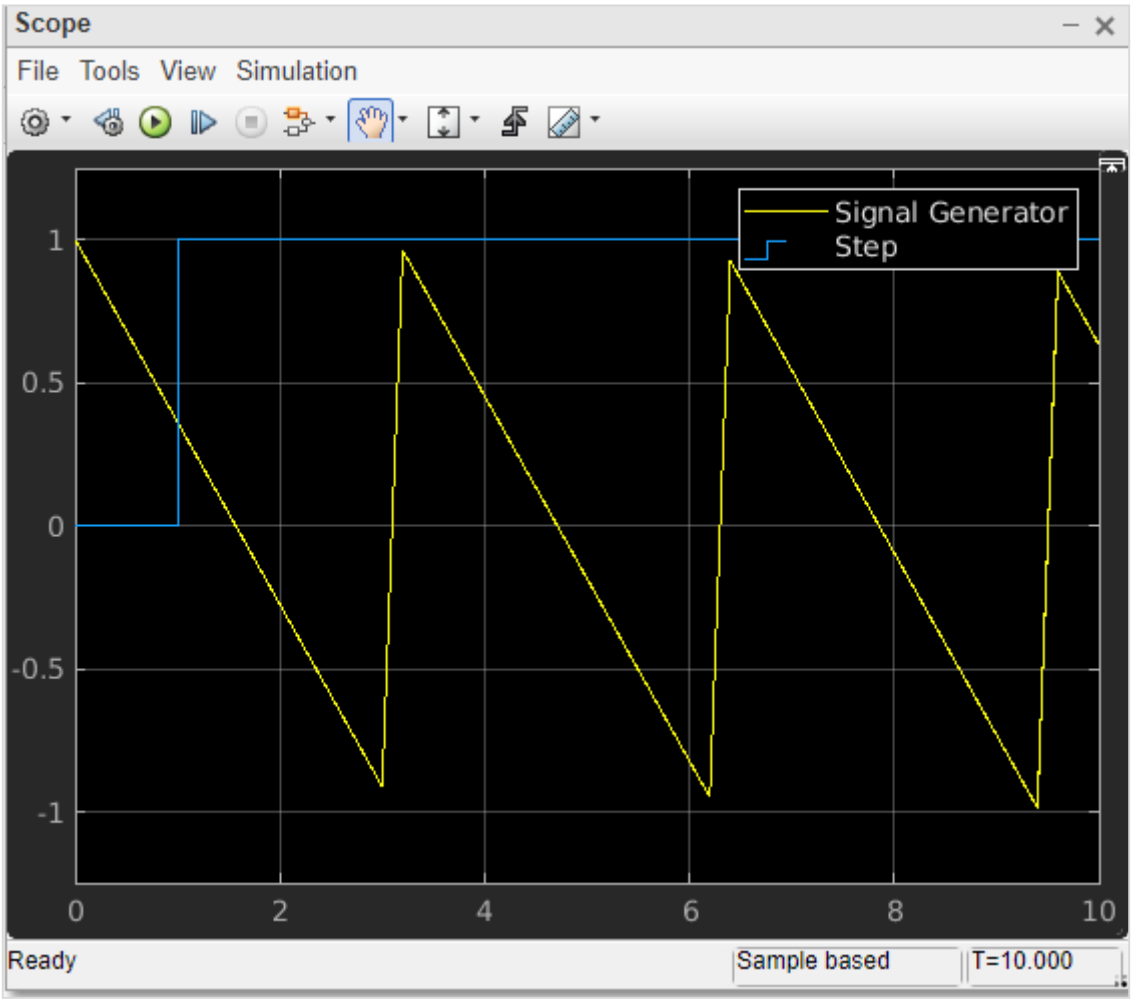
Click on OK button to update the changes. Now, the scope block has 2 inputs as shown below:



Connect the step input arrow with the scope arrow.



Now click on Run button to run the model.

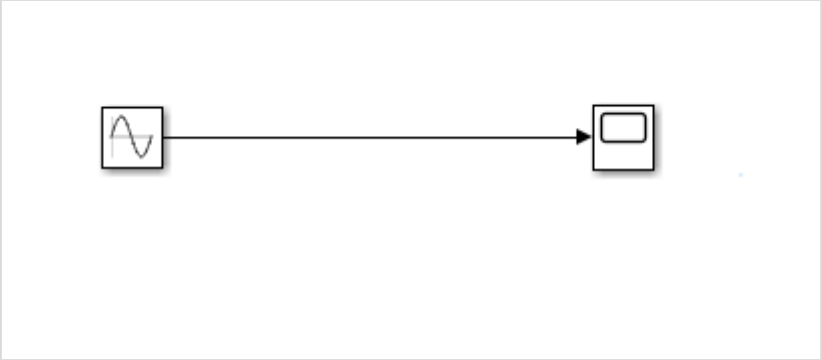


You can add some more signals and test the same.

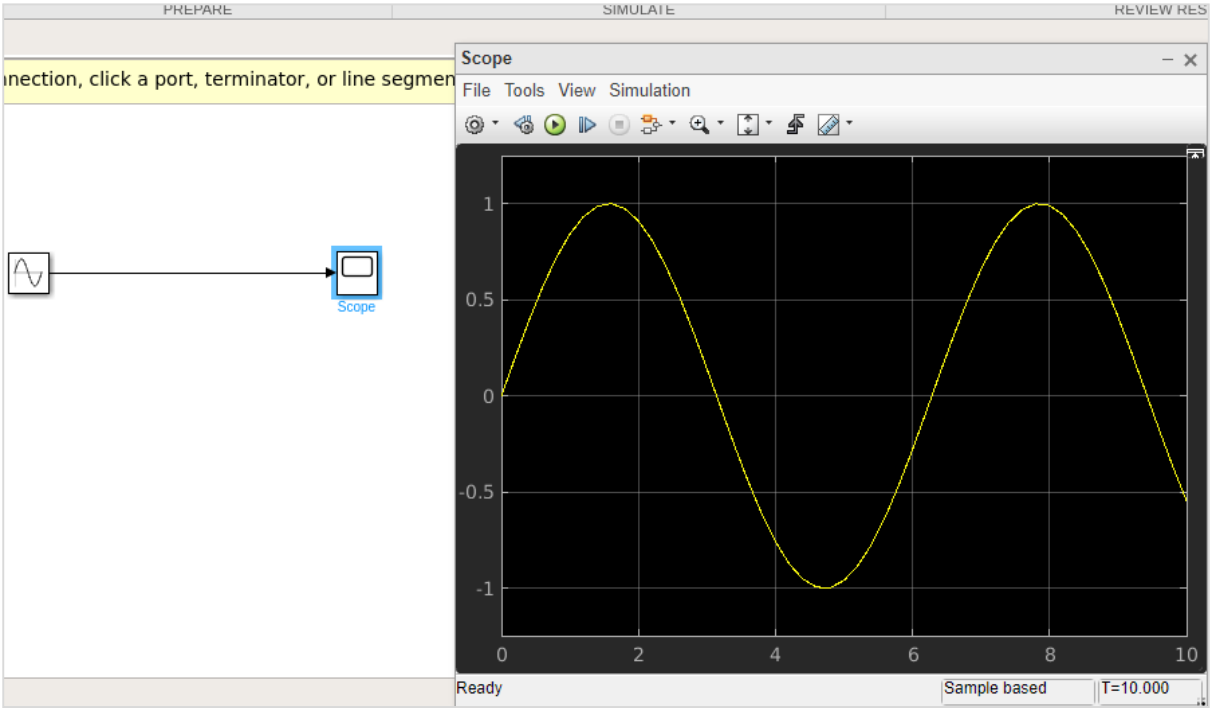
8. MATLAB Simulink — Adding Delay to Signals

We have learnt in the previous chapter about the different signal simulations. In this chapter, we will learn how to add delay to the signals.

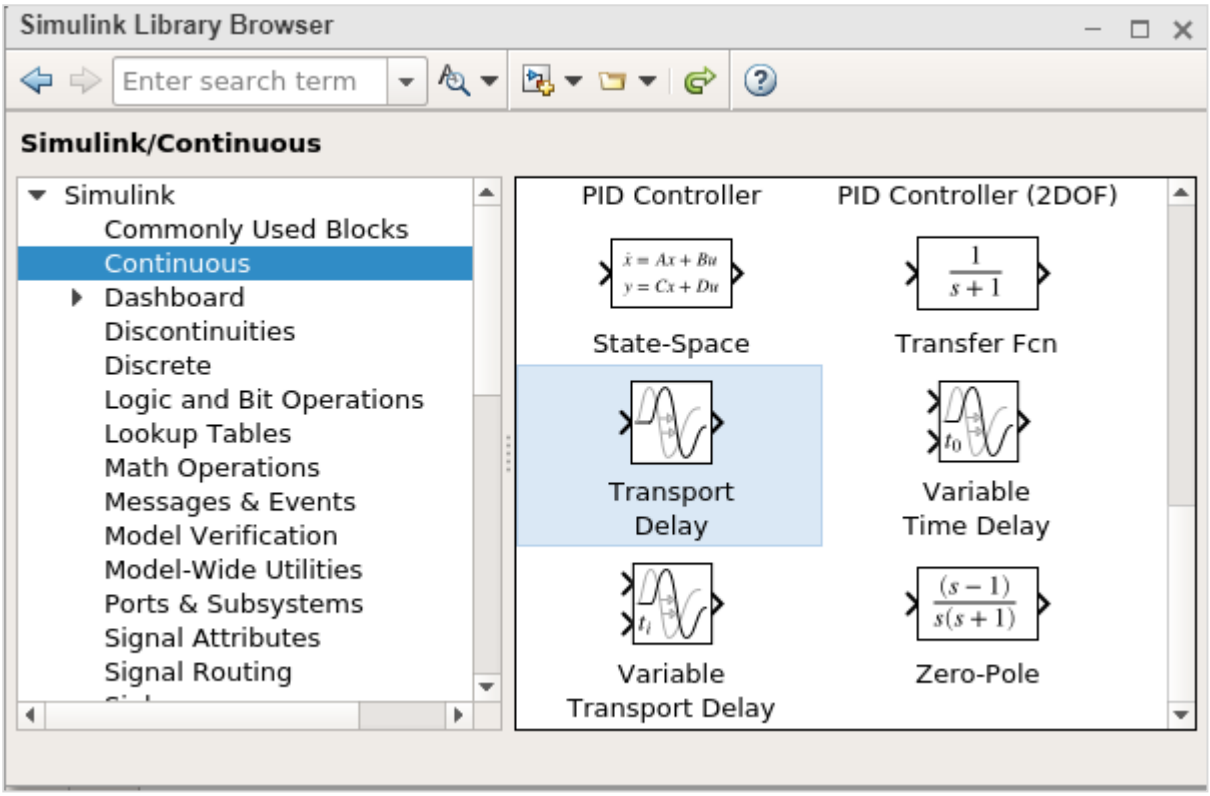
Let us take a blank model and add sine wave and scope block to it as shown below:



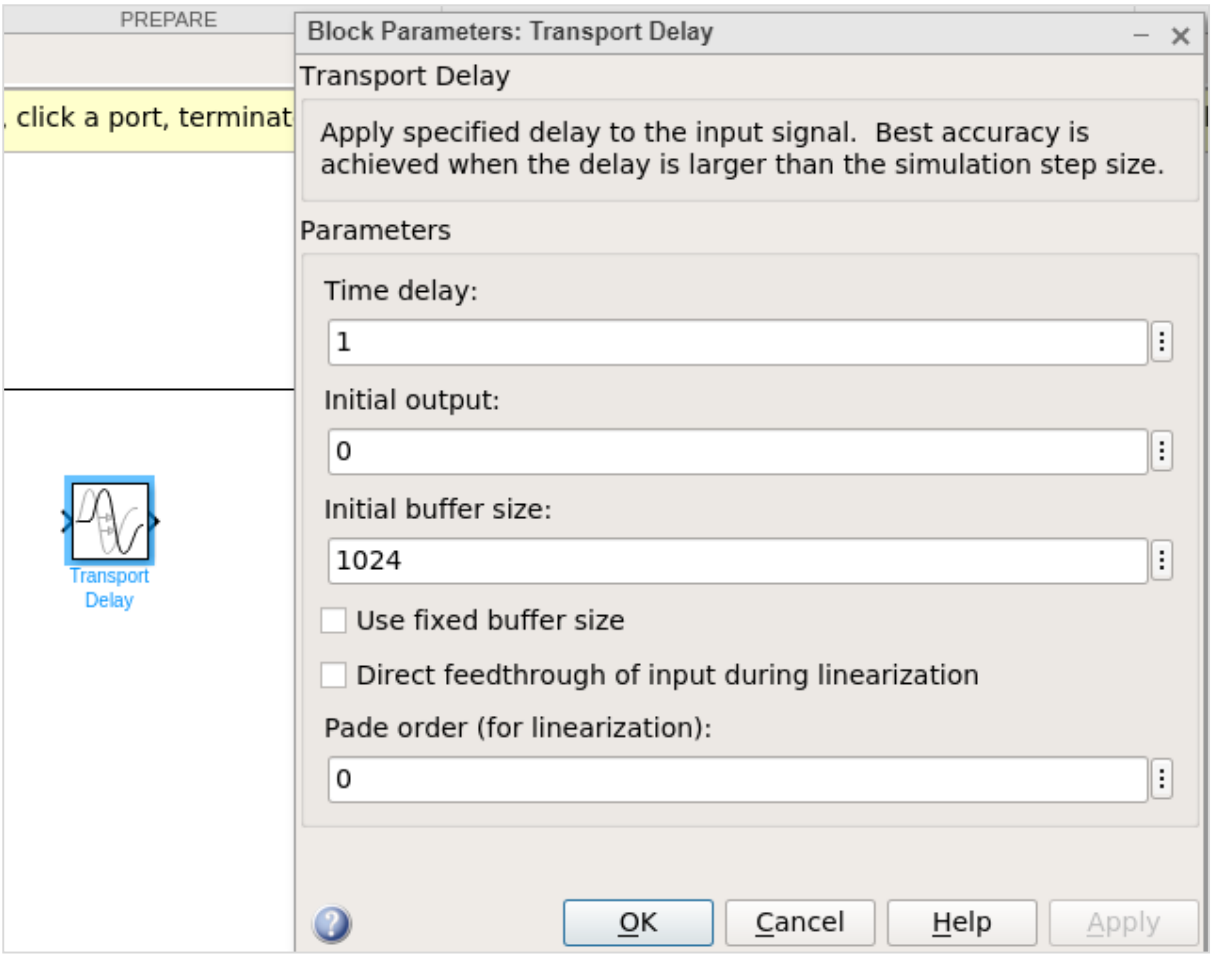
Let us now run the model to see the simulation in scope block. The sine wave is as shown below:



Let us now add delay for the sine wave. We will make use of transport delay block from continuous library as shown below:



Select the block and drag it in your model canvas. Now that we have the Transport delay in our model, right click on it and open block parameters as shown below:



Let us change the time delay from 1 to 3. Make the changes and click on OK button.

Now add one more input port to scope block. Right click on scope block and select the signals and ports. Select 2 for number of input ports as shown below: