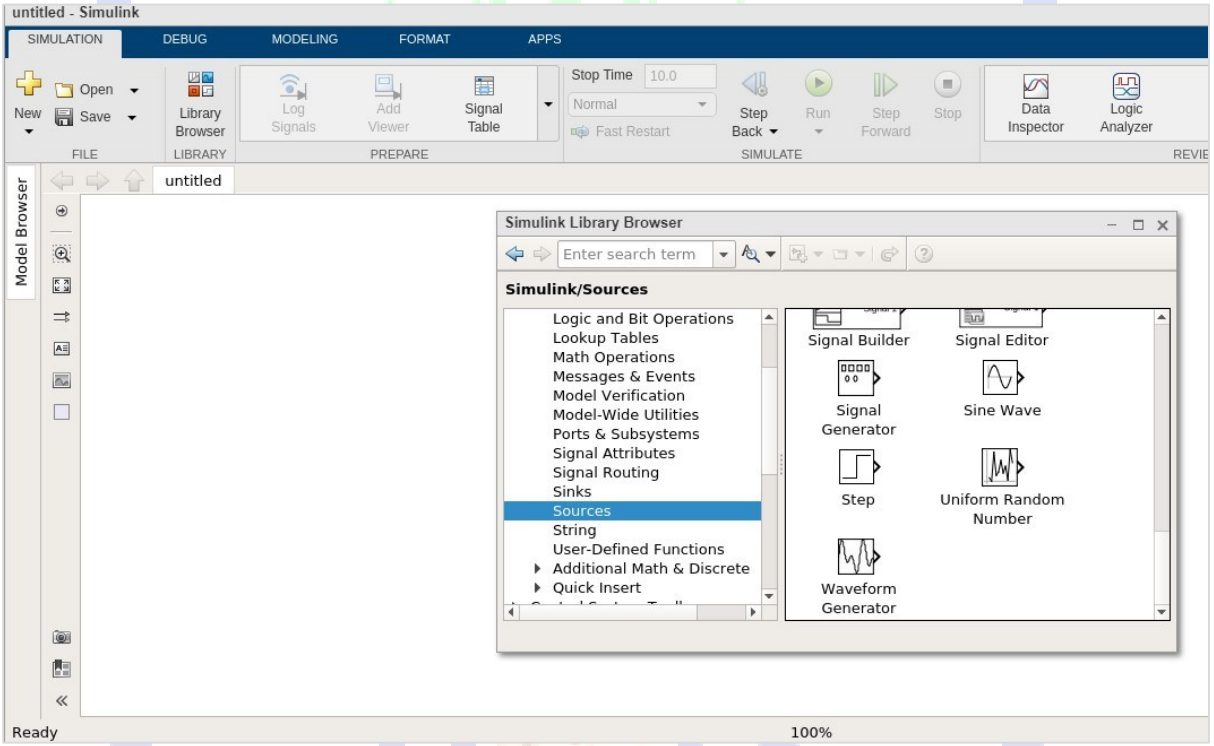
الكلية	الهندسة
القسم	الكهرباء
المادة باللغة الانجليزية	MATLAB Simulink
المادة باللغة العربية	ماتلاب
المرحلة الدراسية	الثالثة
اسم التدريسي	قصي حاتم عيسى
عنوان المحاضرة باللغة الانجليزية	Signals Processing
عنوان المحاضرة باللغة العربية	معالجة الإشارات
رقم المحاضرة	7
المصادر والمراجع	https://www.mathworks.com/help/simulink/getting-started-with-simulink.html

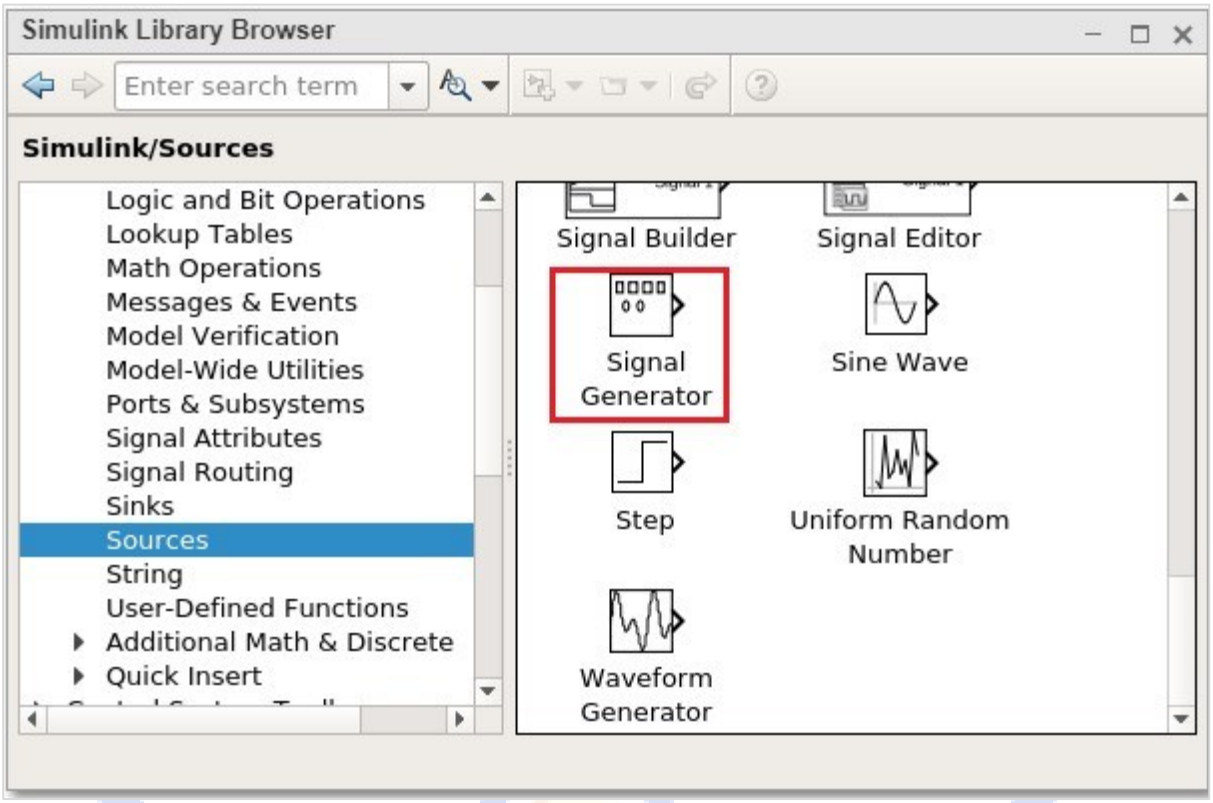
7. MATLAB Simulink — Signals Processing

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

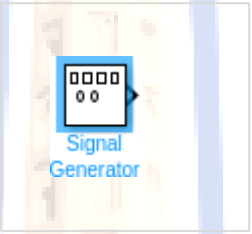
To start with, select a blank model from Simulink page and open Simulink browser library as shown below:



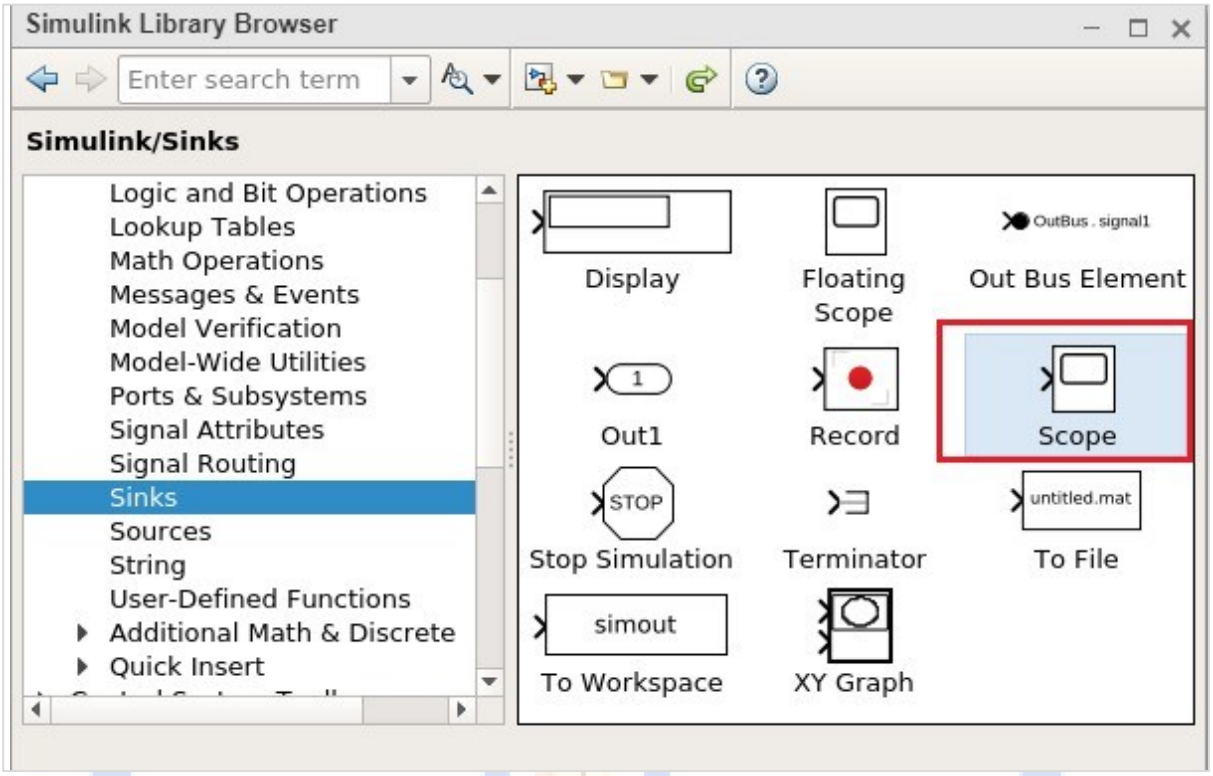
In sources library, you will get a signal generator symbol. It will help us to create different types of signals.



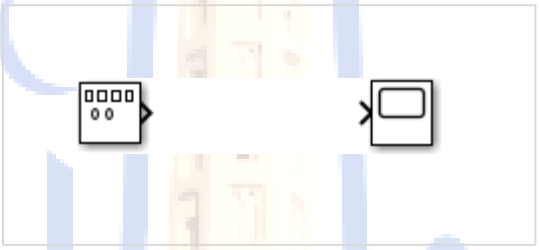
Select the Signal Generator and drag it to get inside the blank model as shown below:



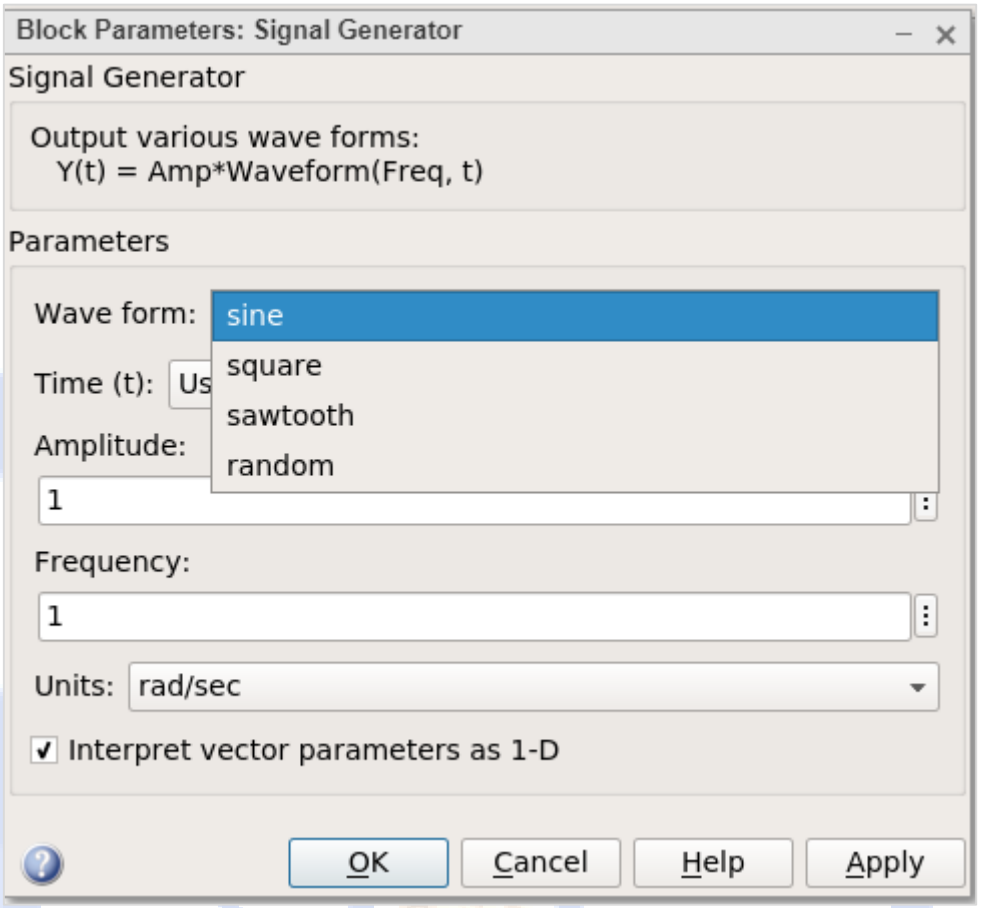
To see the output of the signal generator, we need one more block called scope from sinks library as shown below:



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

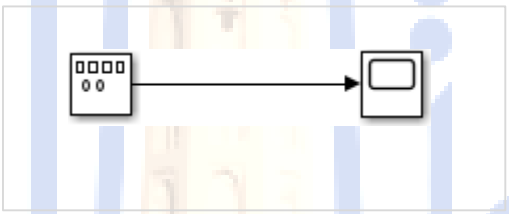


Double click on signal generator or right click and select block parameters and it will display a screen as shown below:

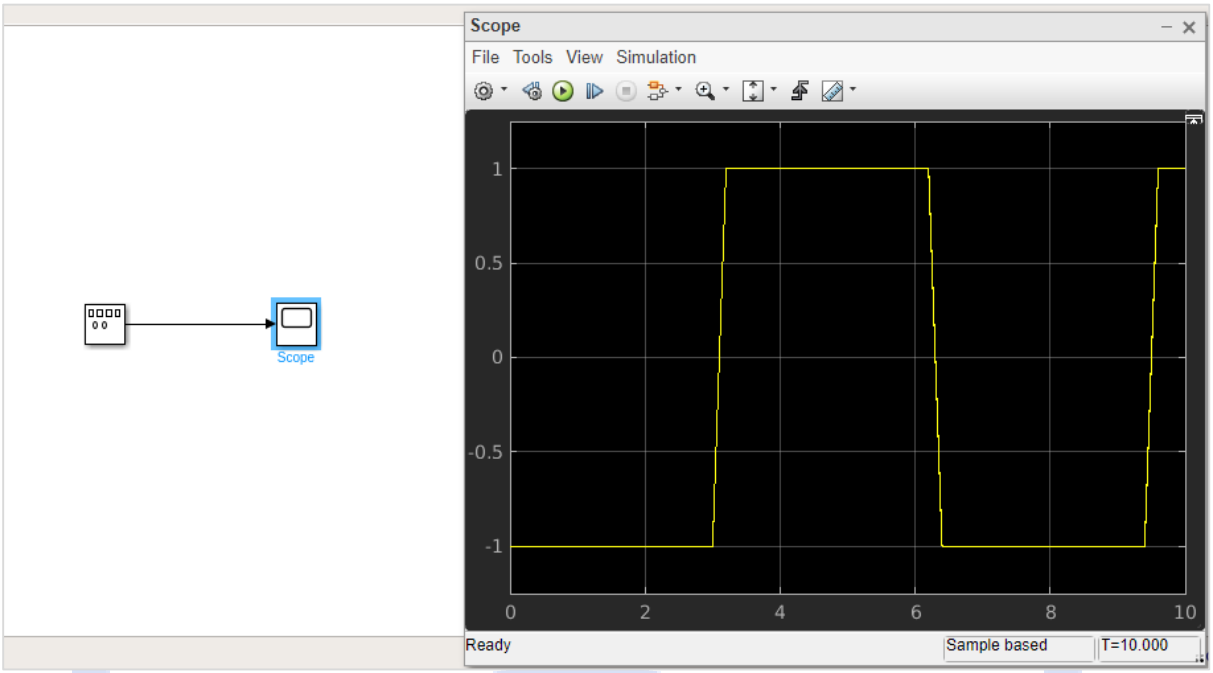


The signal generator can show waveforms like sine, square, sawtooth, random. We will select the square waveform. Let the amplitude and frequency be as 1. Click on OK to update the changes made.

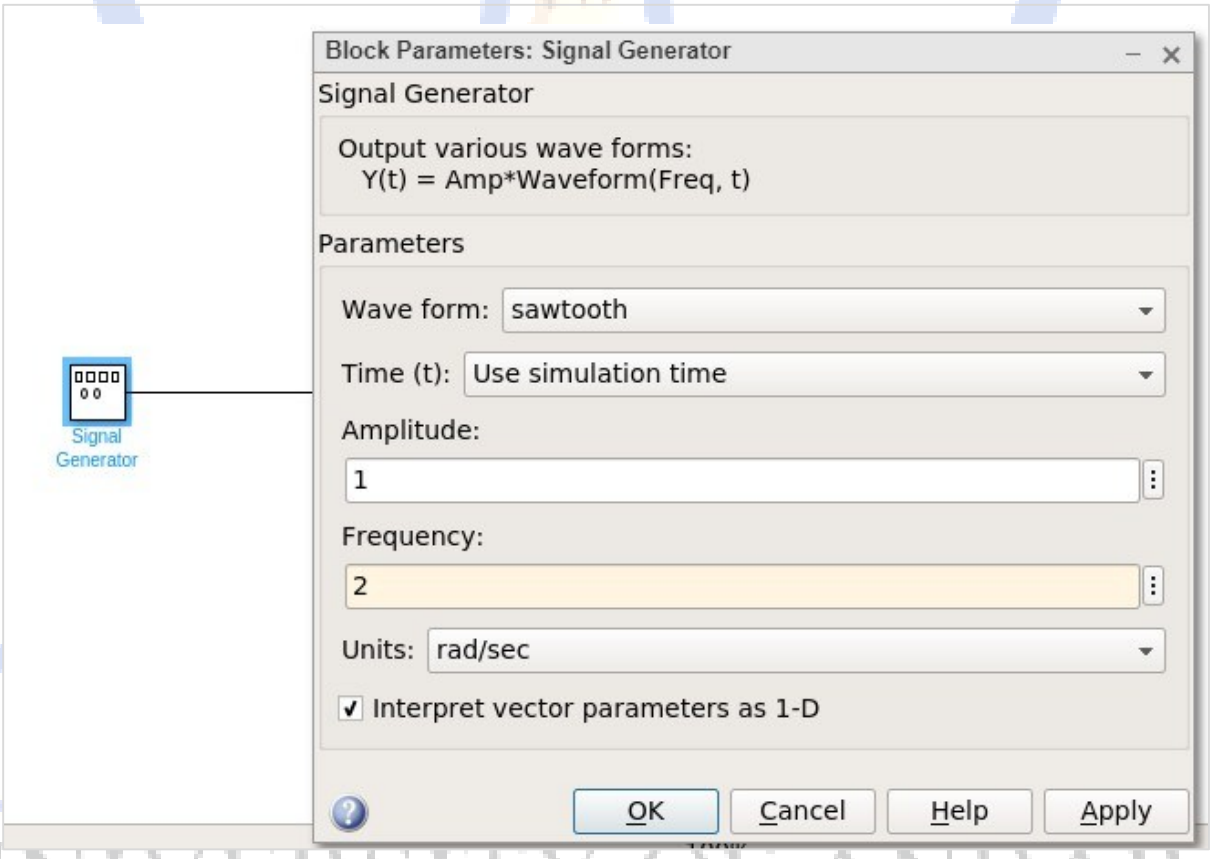
Now, connect the lines between signal generator and scope block as shown below:



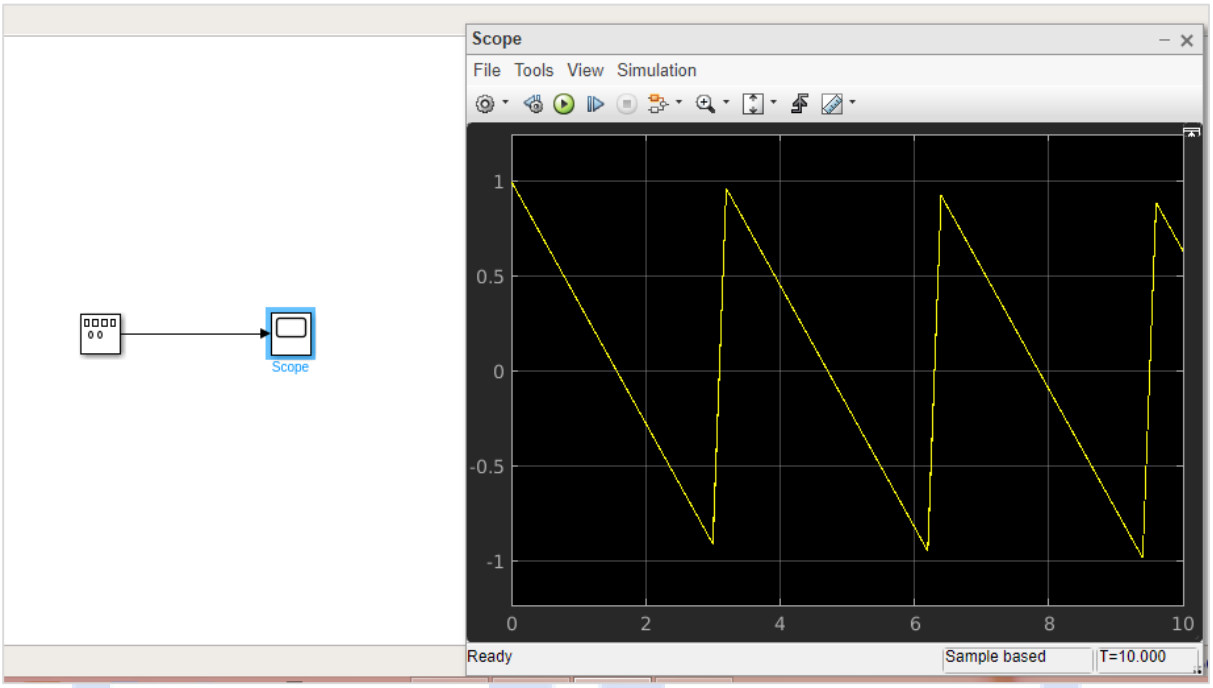
Now click on Run button to see the square waveform as shown below:



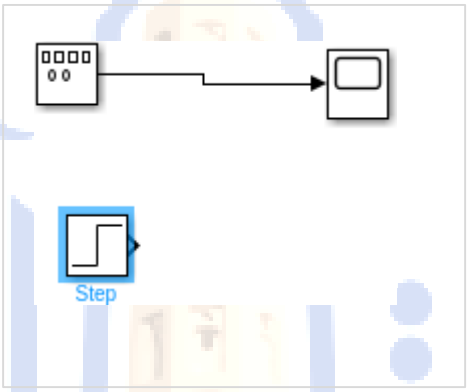
Let us now try the sawtooth wave form. Right click signal generator or double click and change the waveform to sawtooth.



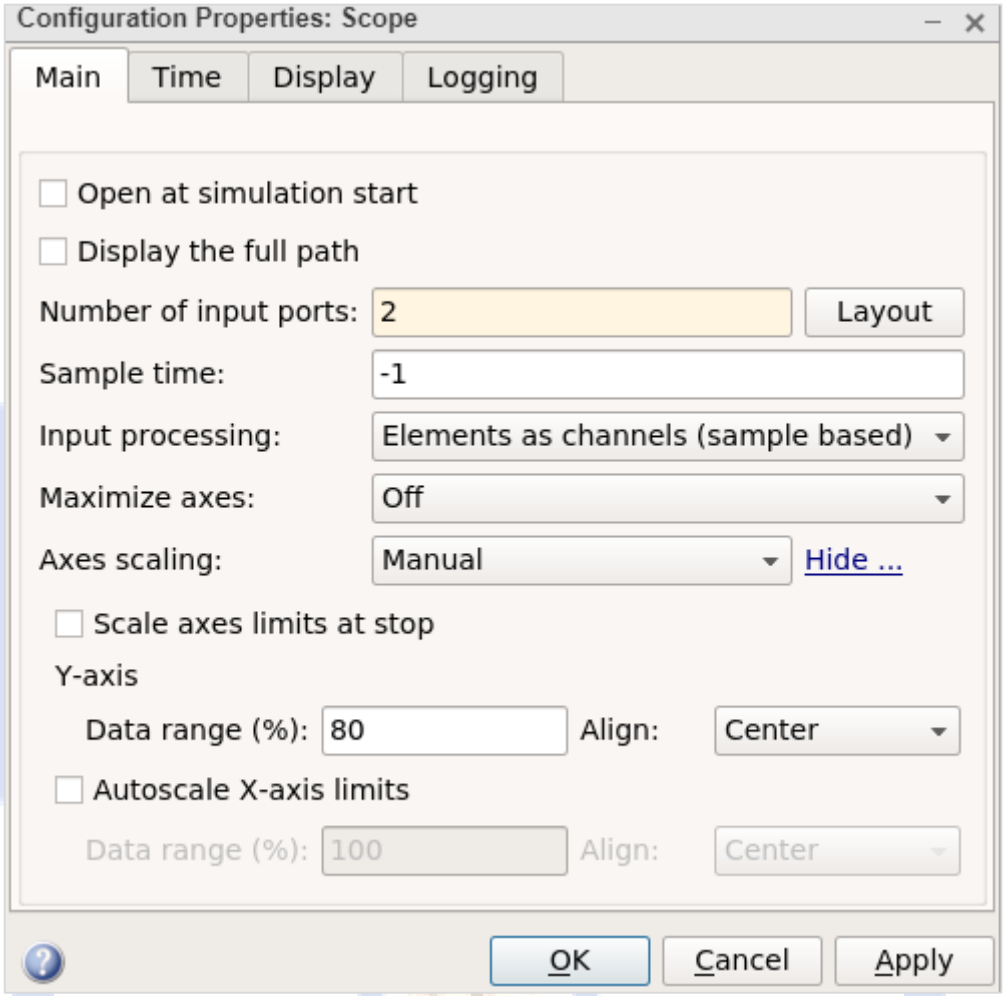
Let us change the frequency to 2. Click on OK to update the changes. Now run the model to see the changes as shown below:



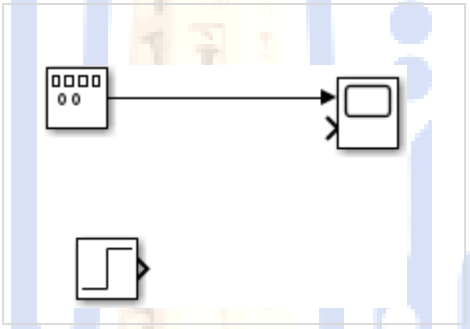
Let us now add some more signals to the above model. We will take the step signal from the sources library as shown below:



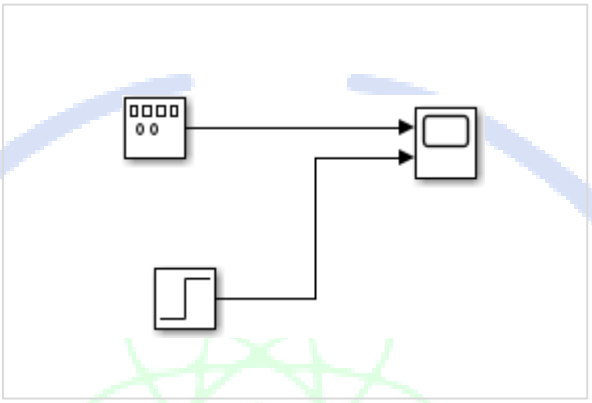
We just have one input for the scope block. Let us increase it to 2 inputs. Right click and open the block parameters.



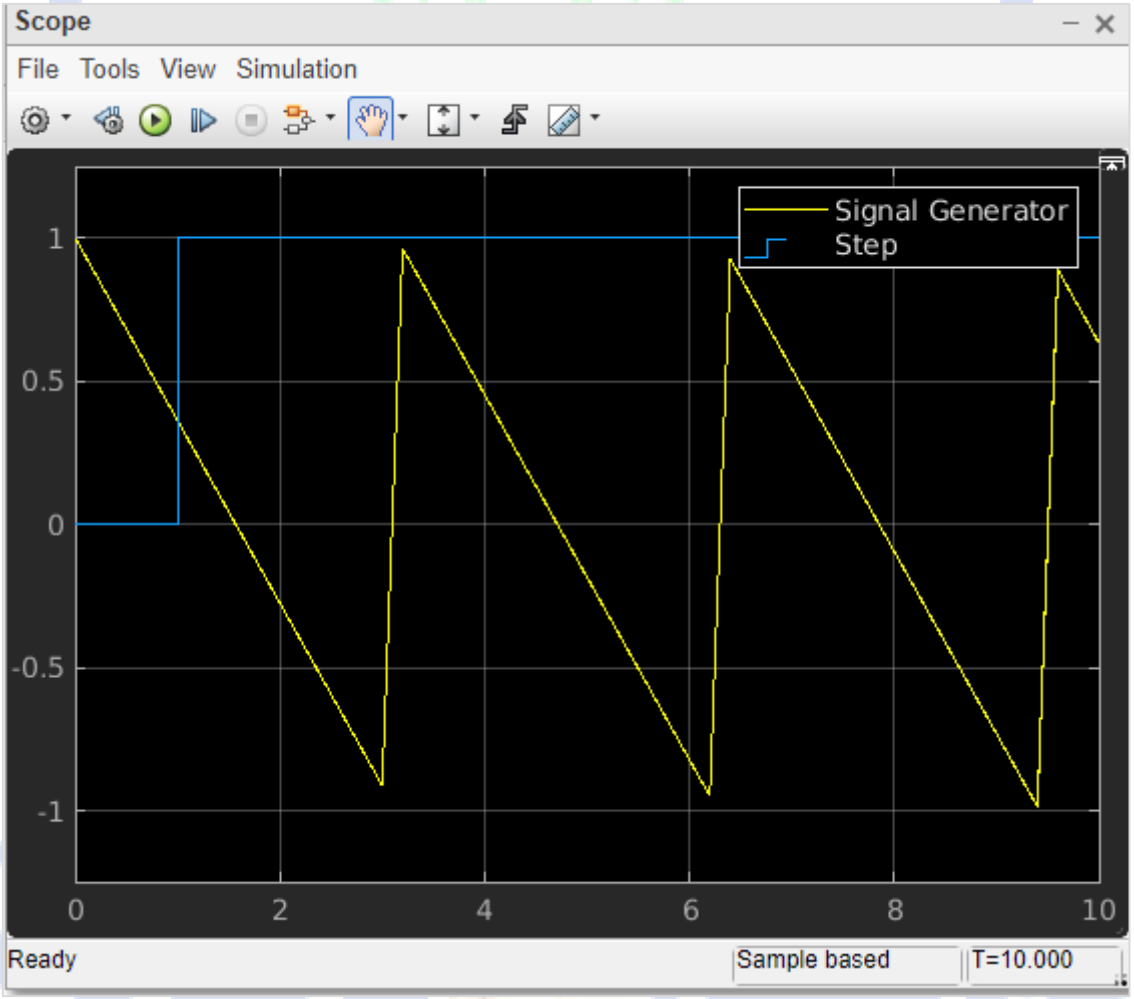
Click on OK button to update the changes. Now, the scope block has 2 inputs as shown below:



Connect the step input arrow with the scope arrow.



Now click on Run button to run the model.

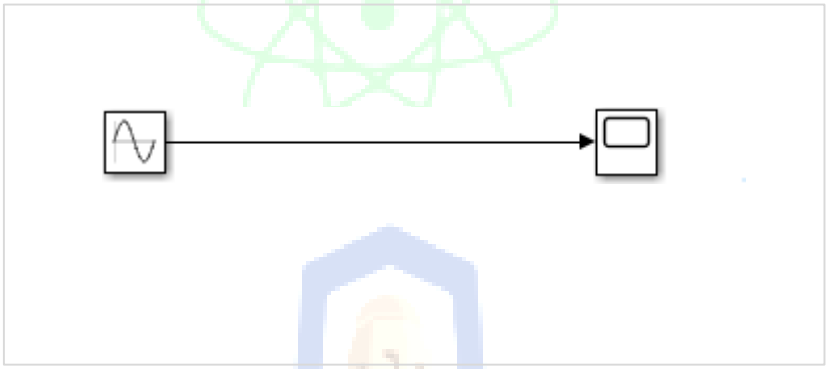


You can add some more signals and test the same.

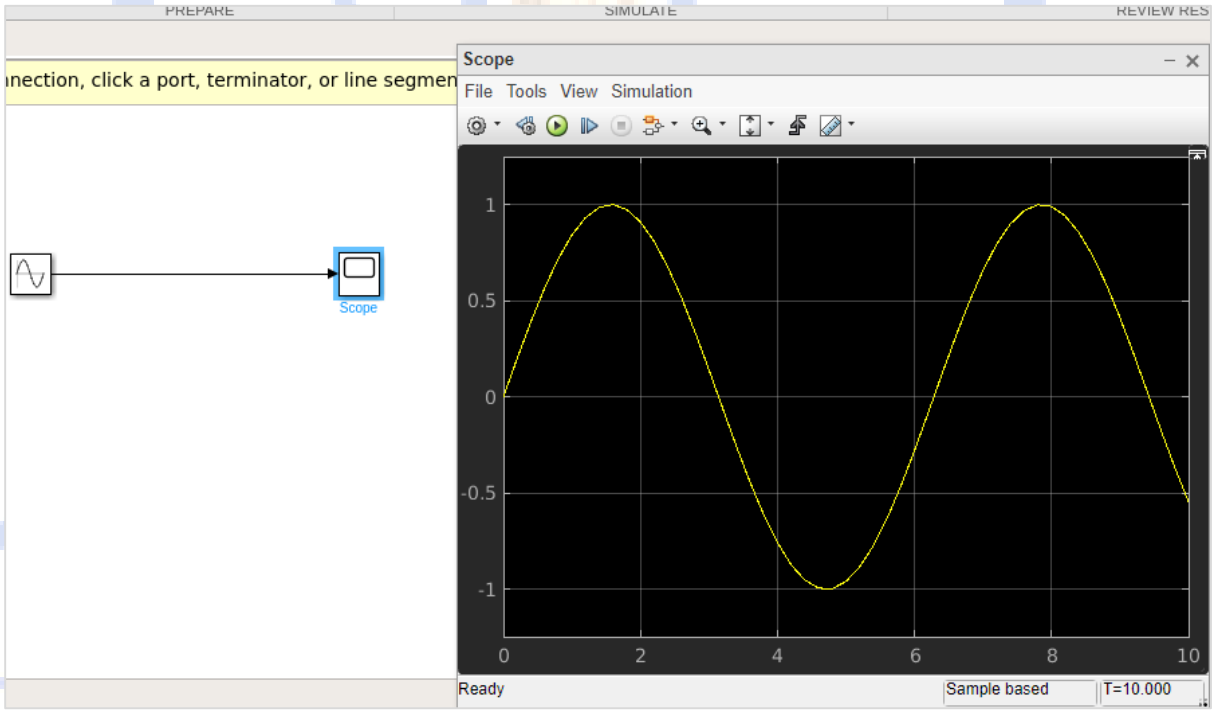
8. MATLAB Simulink — Adding Delay to Signals

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.

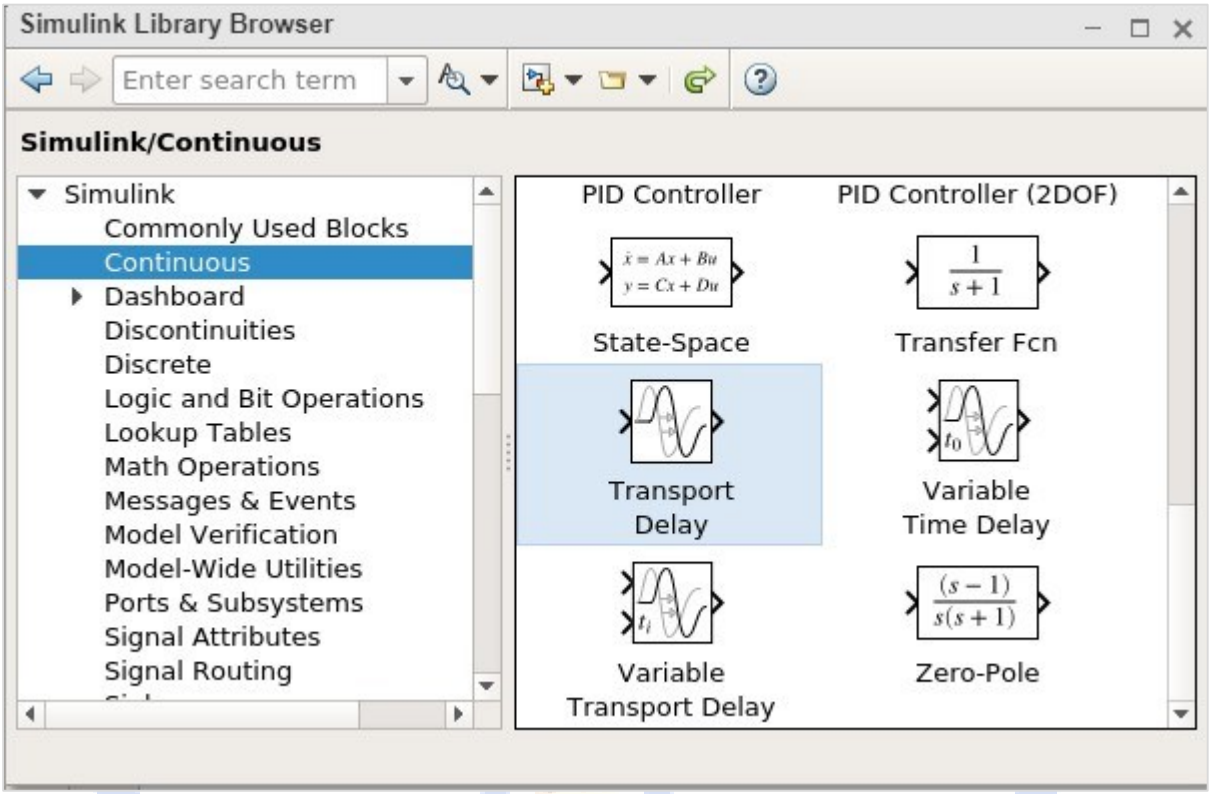
Let us take a blank model and add sine wave and scope block to it as shown below:



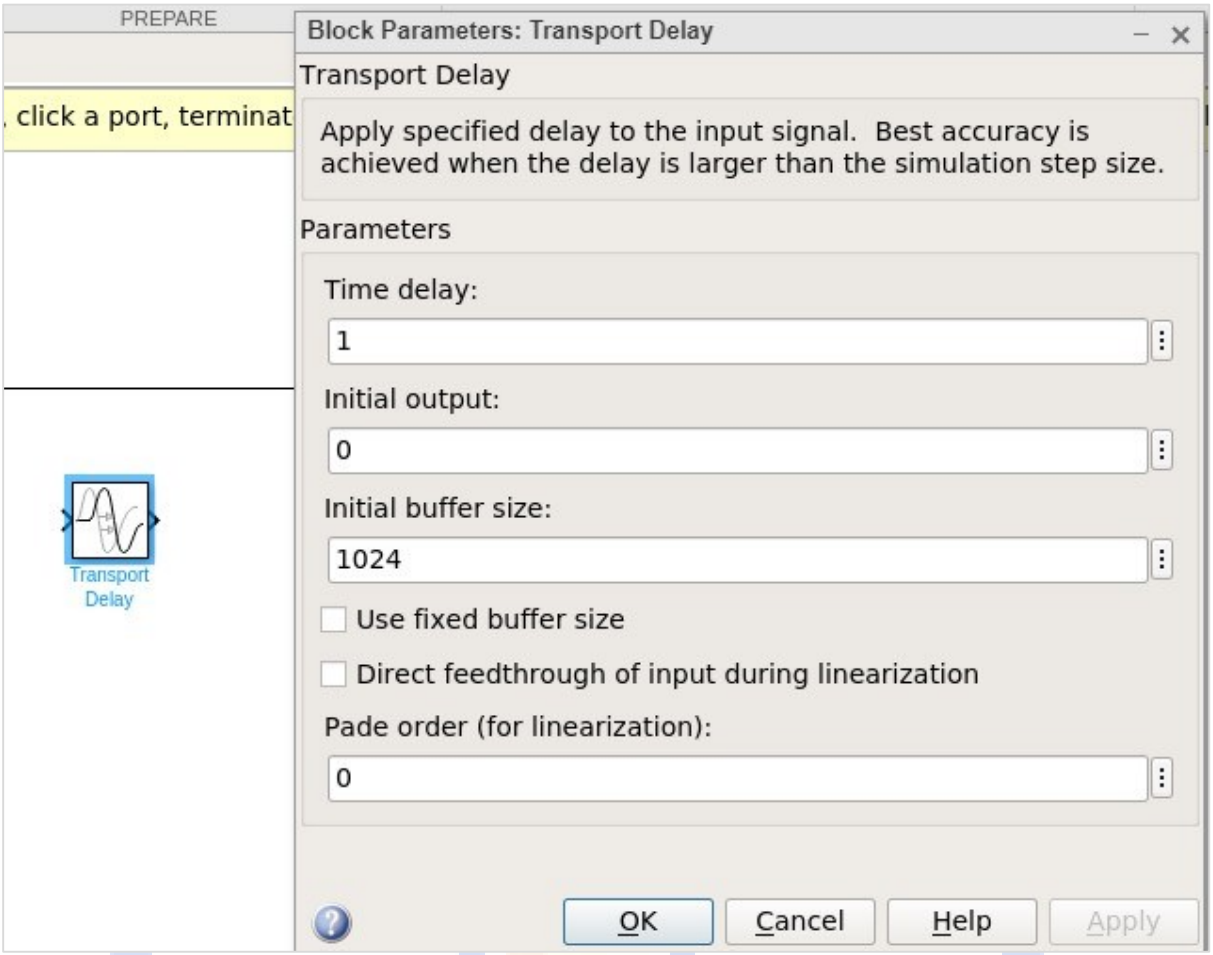
Let us now run the model to see the simulation in scope block. The sine wave is as shown below:



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

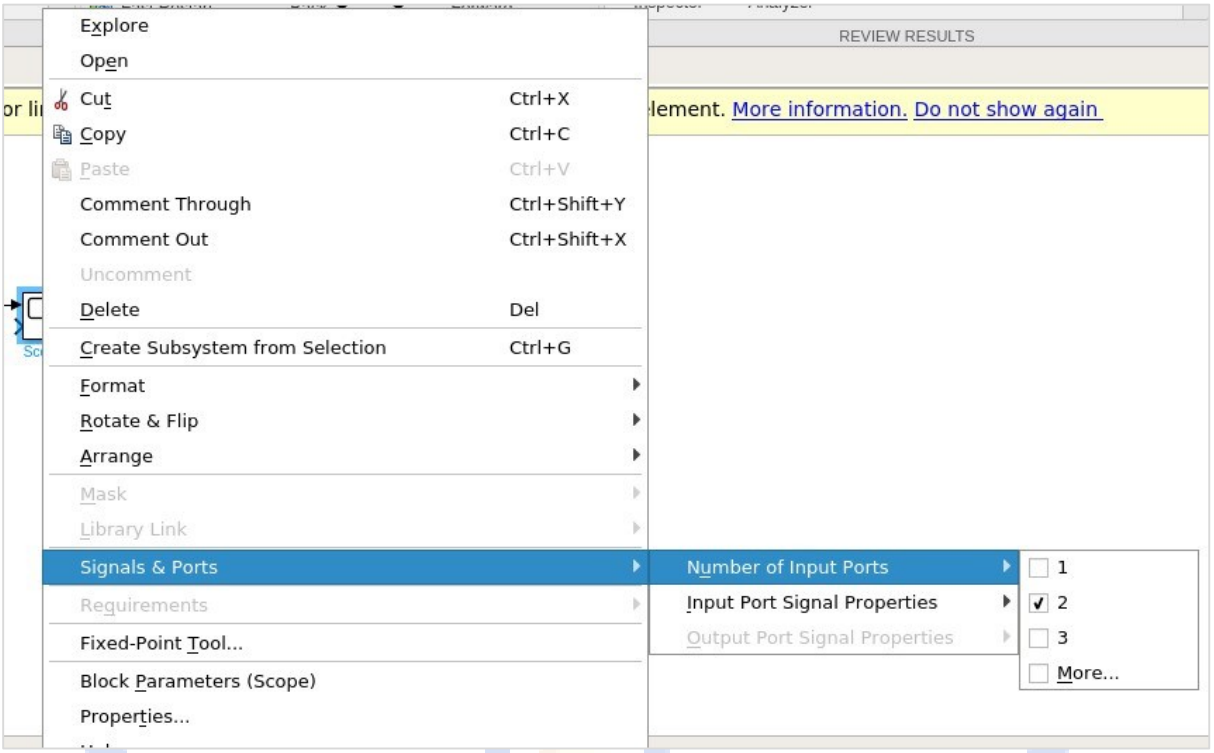


Select the block and drag it in your model canvas. Now that we have the Transport delay in our model, right click on it and open block parameters as shown below:

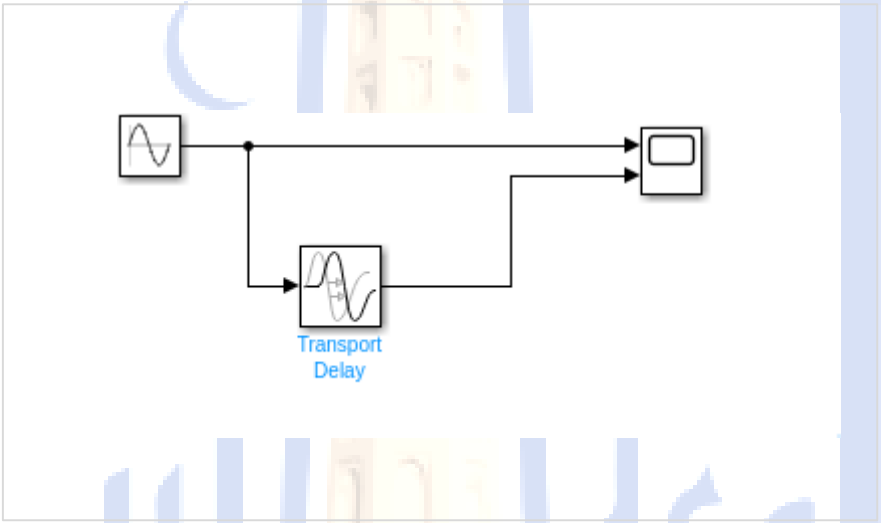


Let us change the time delay from 1 to 3. Make the changes and click on OK button.

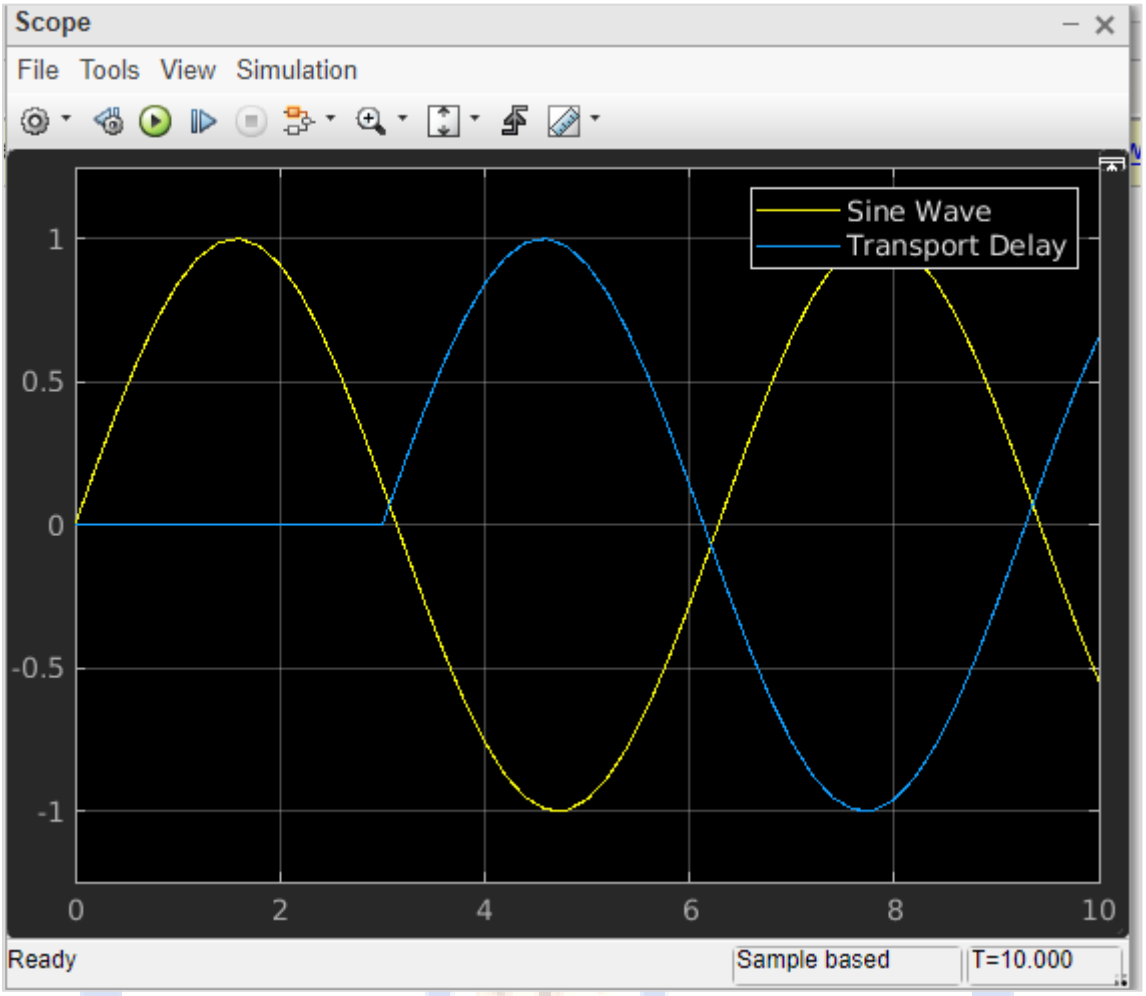
Now add one more input port to scope block. Right click on scope block and select the signals and ports. Select 2 for number of input ports as shown below:



Now connect the transport delay to sine wave and to scope as shown below:



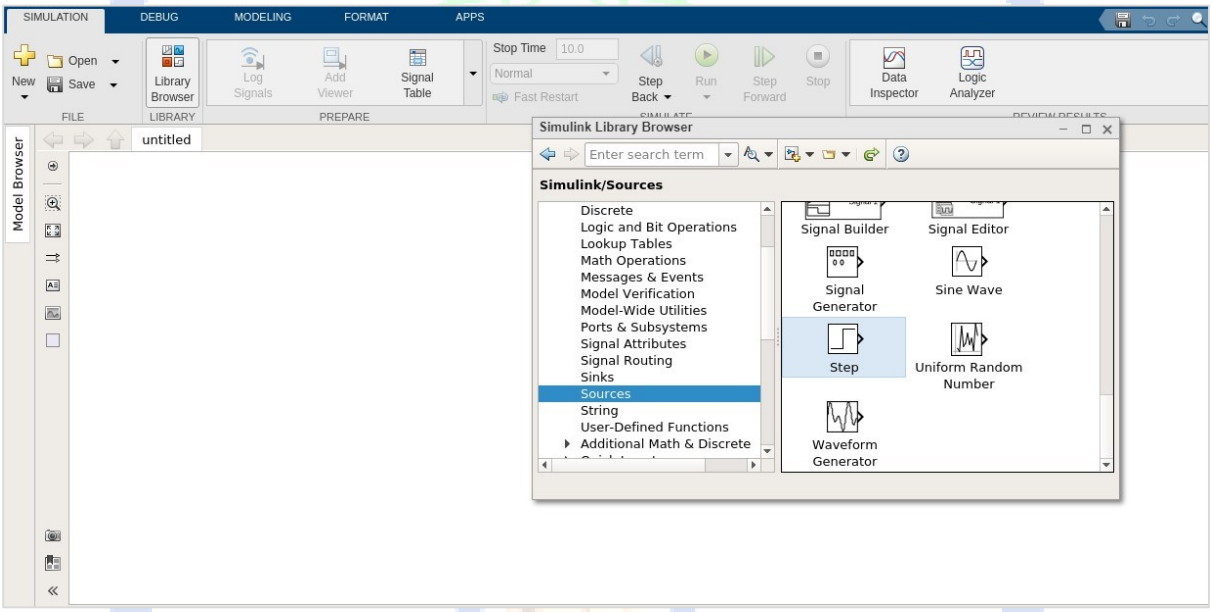
Now run the simulation to see a delay of 3 seconds to the sign wave. Right click scope block and select block parameters to see the display.



9. MATLAB Simulink — Mathematical Library

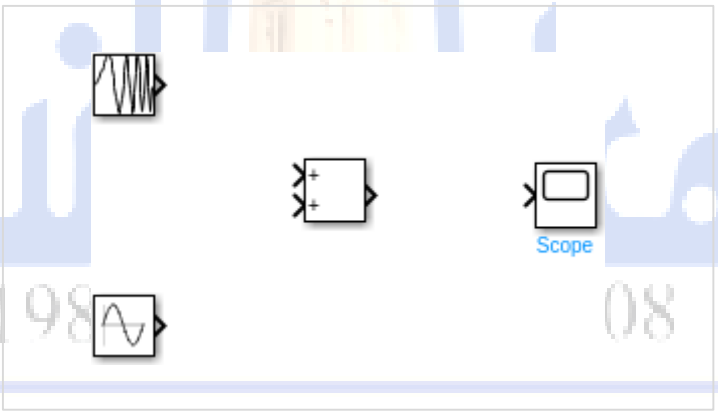
In this chapter, we will learn how to sum the two given signals and get the output.

Select the blank model and open Simulink library browser as shown below:

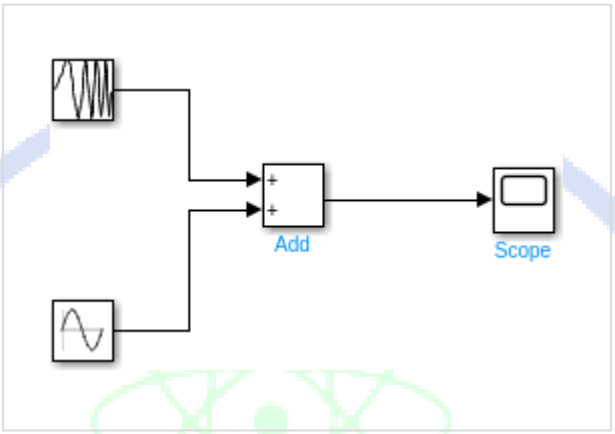


We are going to combine chirp signal and sine wave blocks by using add block from Math operation and see the final display.

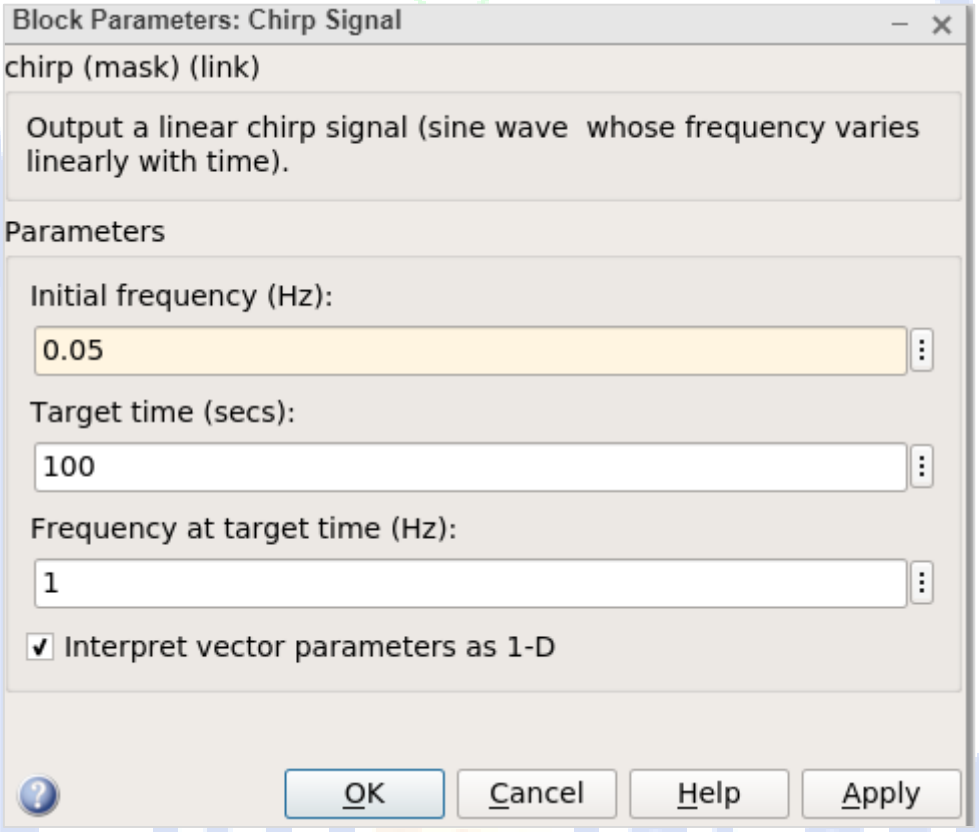
Let us pick the block we want. Select chirp signal and sine wave from sources library, add block from math operations, scope block from sinks library.



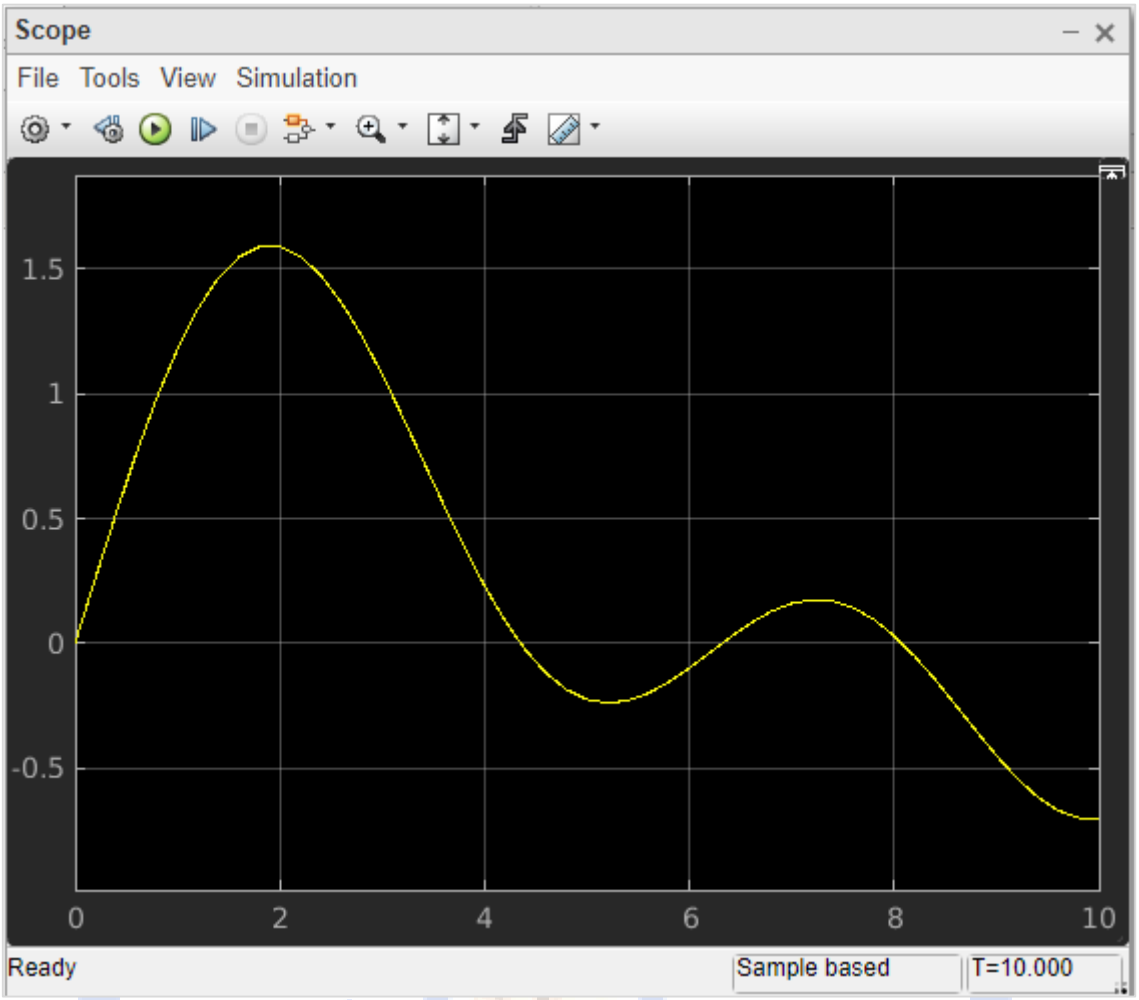
Join the lines to each block.



Double click chirp signal and change the initial frequency from 0.1 to 0.05 and click on Ok button.



The other blocks are kept as the default values. Now, click on run to see the output in scope as shown below.

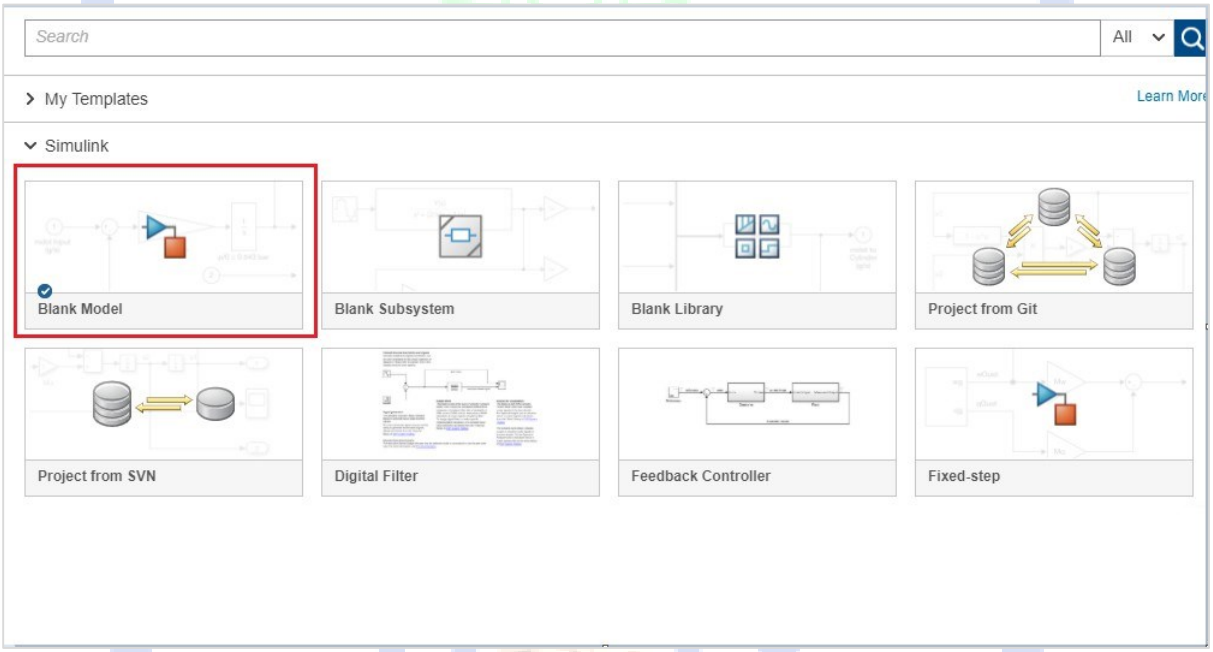


10. MATLAB Simulink — Build Model and Apply If-Else Logic

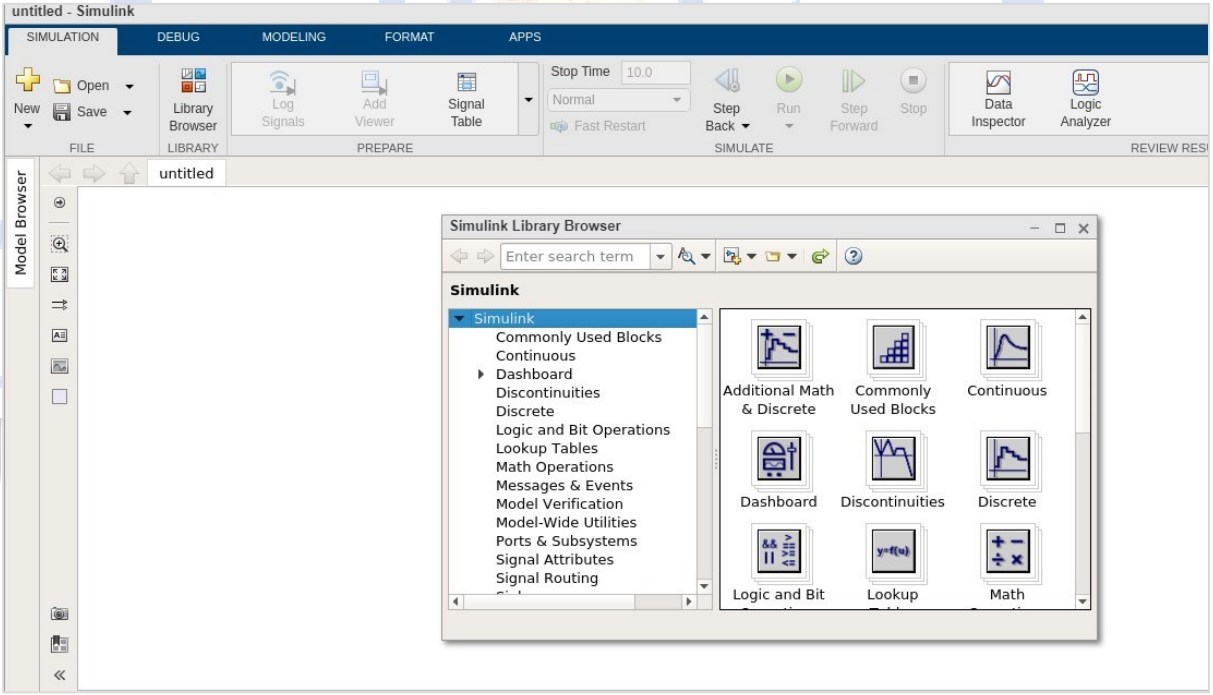
In this chapter, we will create a model and apply if-else logic to it.

Let us first collect blocks to create our model.

Now, open MATLAB Simulink (blank model) and the Simulink library browser as shown below:



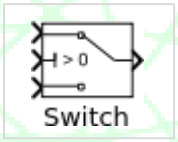
Click on the Blank Model and open Simulink library browser as shown below:



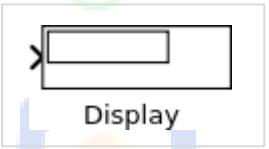
The blocks we require to build the model with if-else logic is as follows:

Constant block from Commonly used blocks

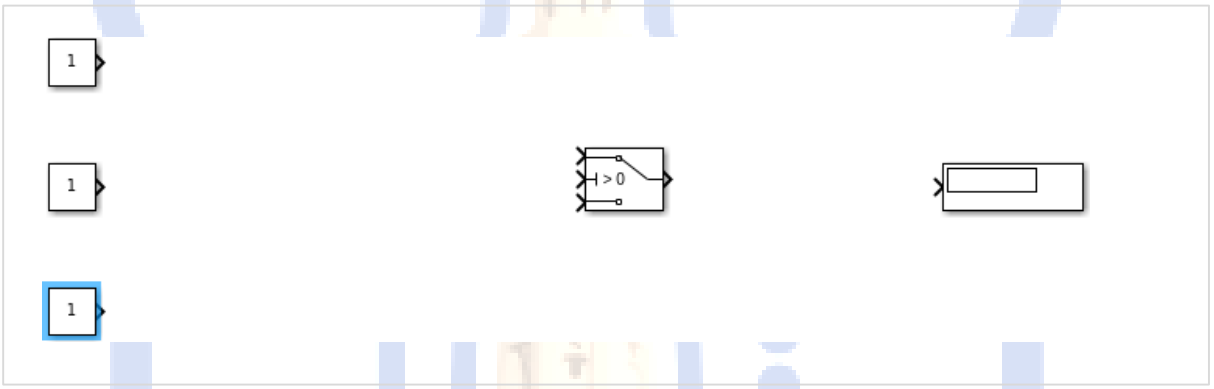
Switch block from Signal Routing



Display block from Sinks

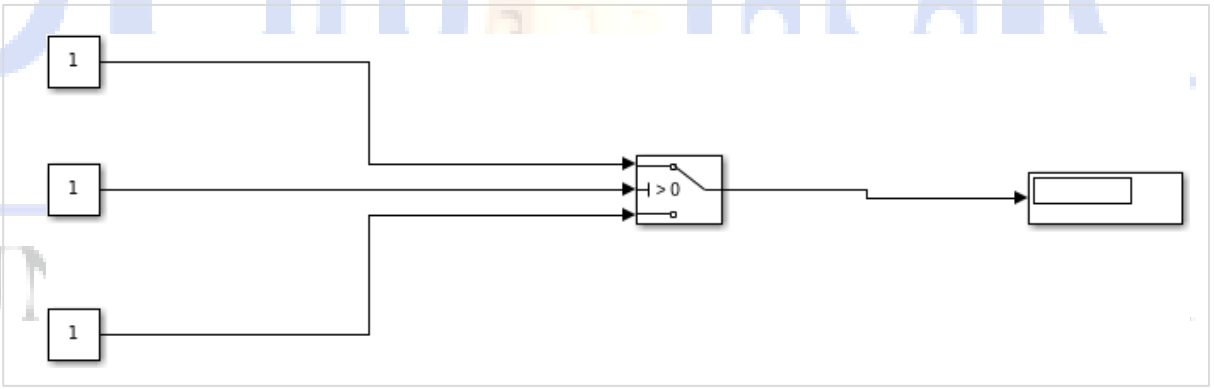


Let us now get all the blocks together to create a model as shown below:



Let us now connect the lines with each block. So you can see that the constant block has one output and the switch has three inputs and one output. We are going to connect them to the display block.

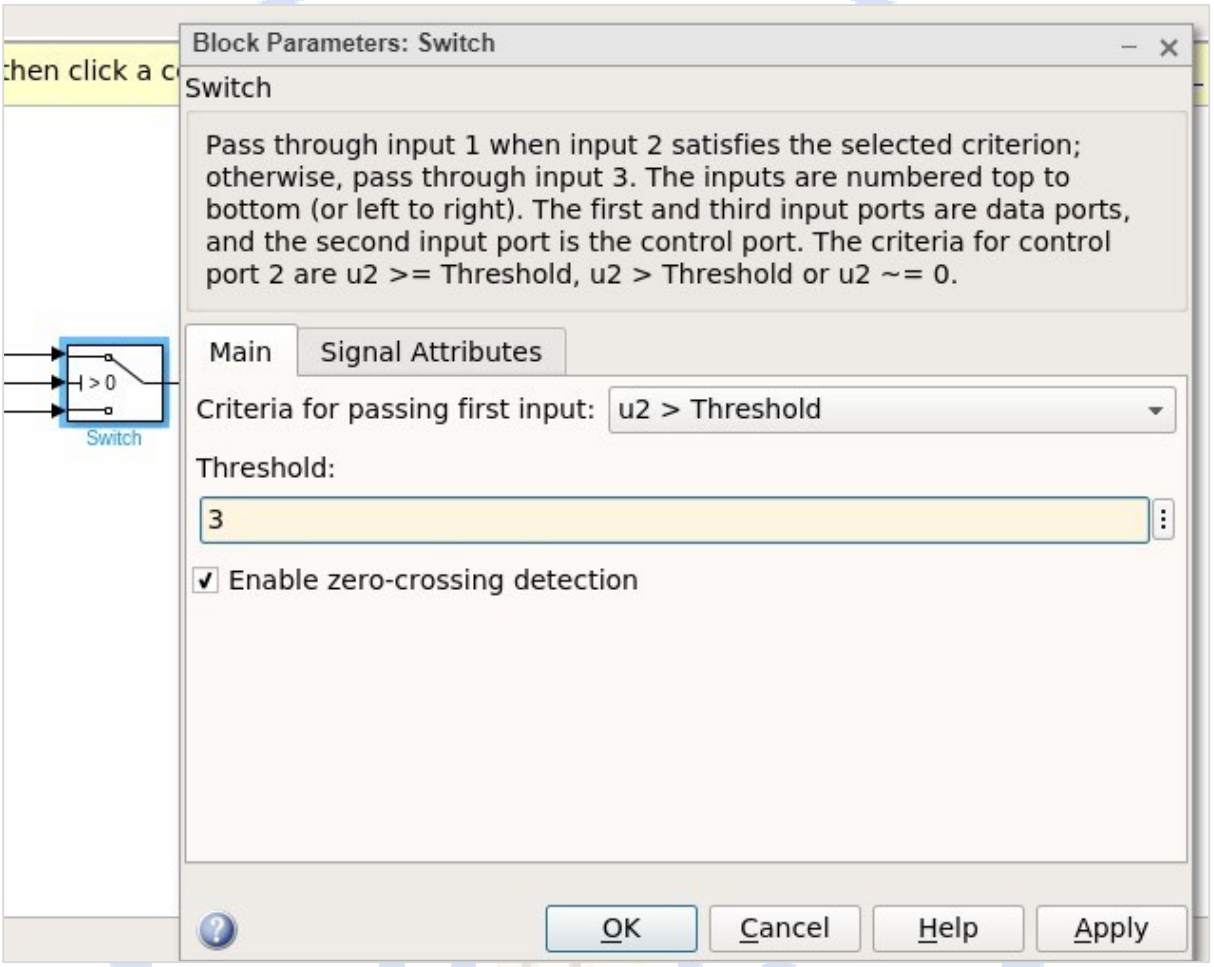
After connecting the lines, the model is as shown below:



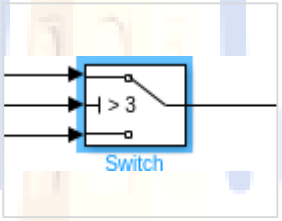
Now, double click the switch block and add a threshold.

The threshold value will be compared with the block in the center. Based on the constant value of the middle block, the first block value will be displayed or the last constant block value will be displayed.

Let us add a threshold value to the switch as shown below:

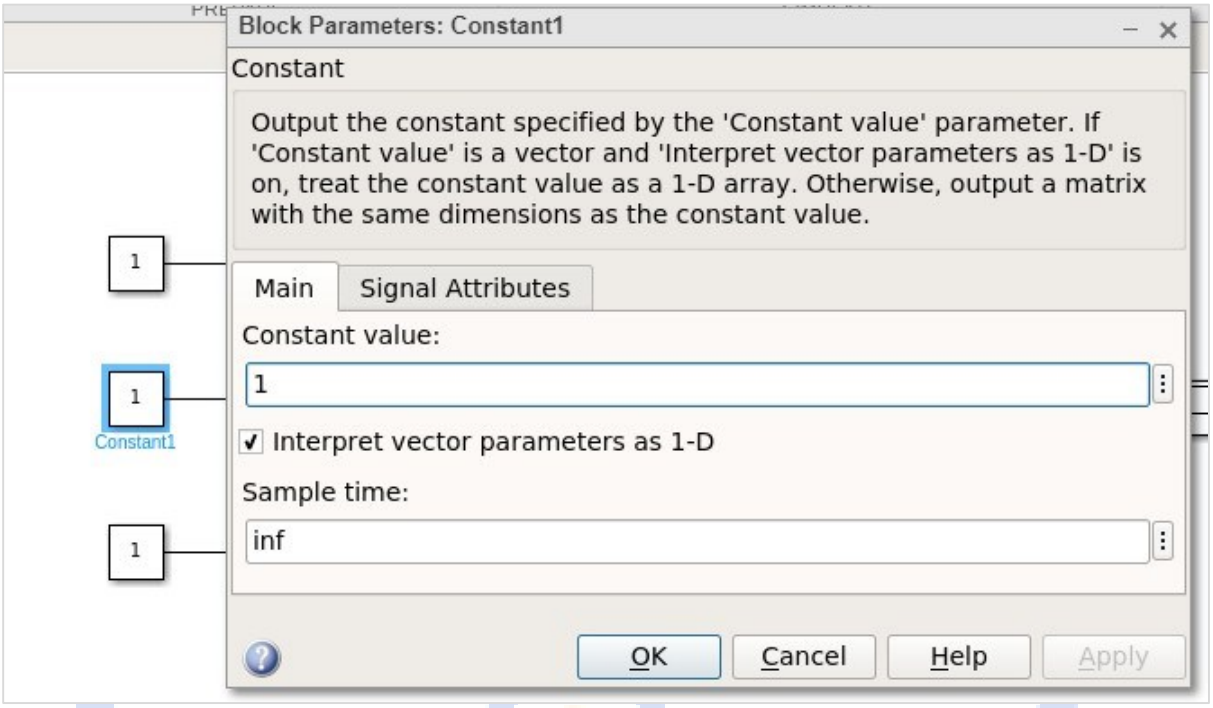


The threshold value given is 3. Click on OK to update the threshold. Now the threshold value is seen inside the switch block as shown below:

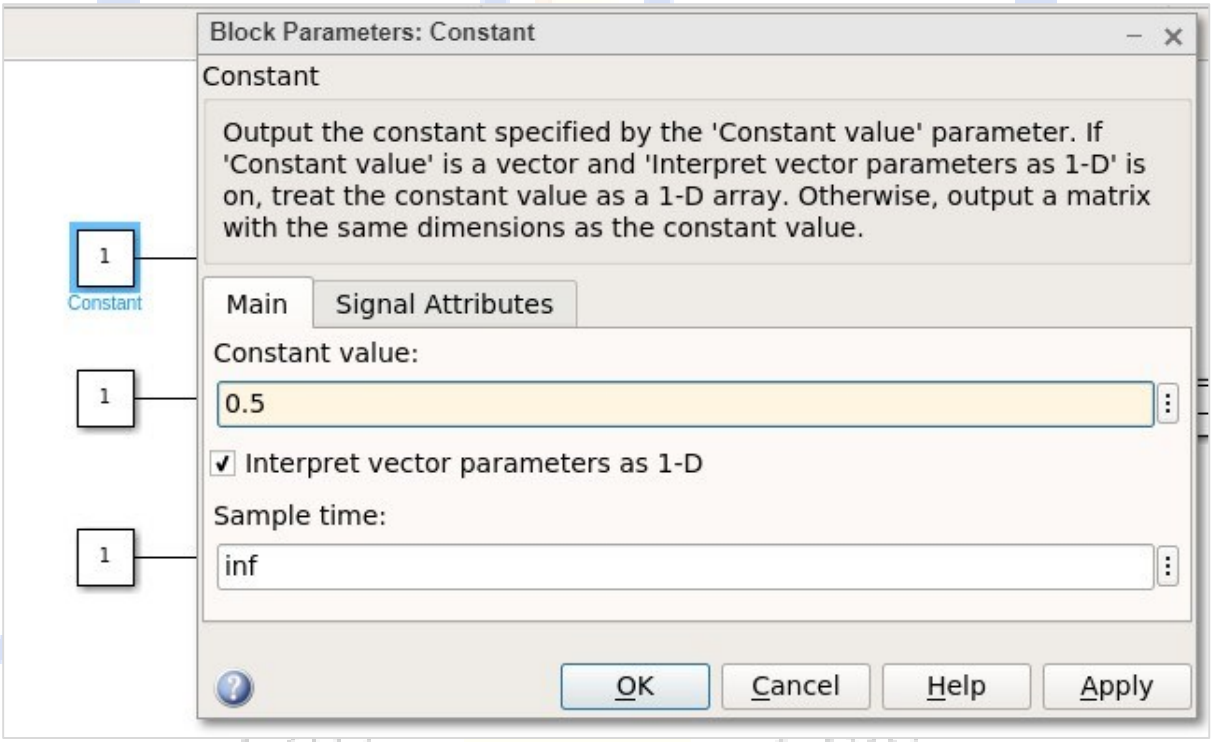


The middle constant block will be compared with the switch threshold and accordingly the display will be decided.

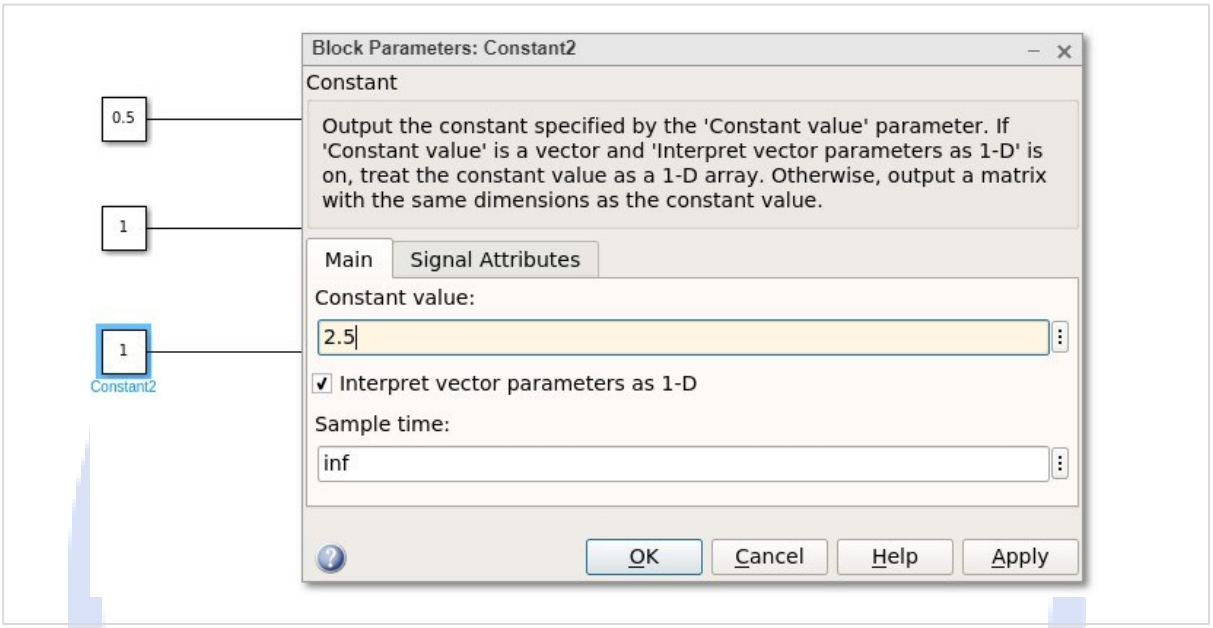
Let us now update the middle constant block with some value as shown below:



The value of the constant block is 1. Let us now change the first constant block and give it a value as 0.5 as shown below:

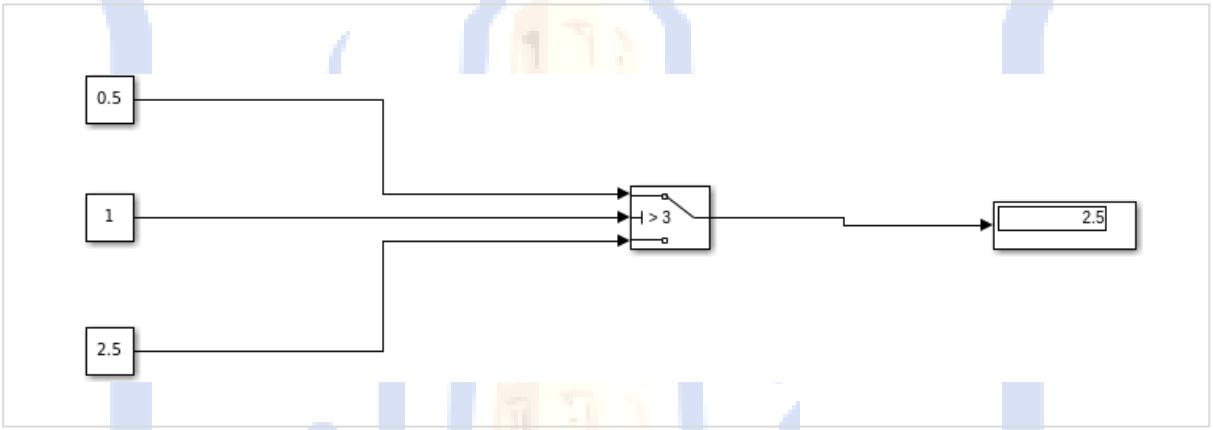


Let us now change the last constant with value as 2.5 as shown below:

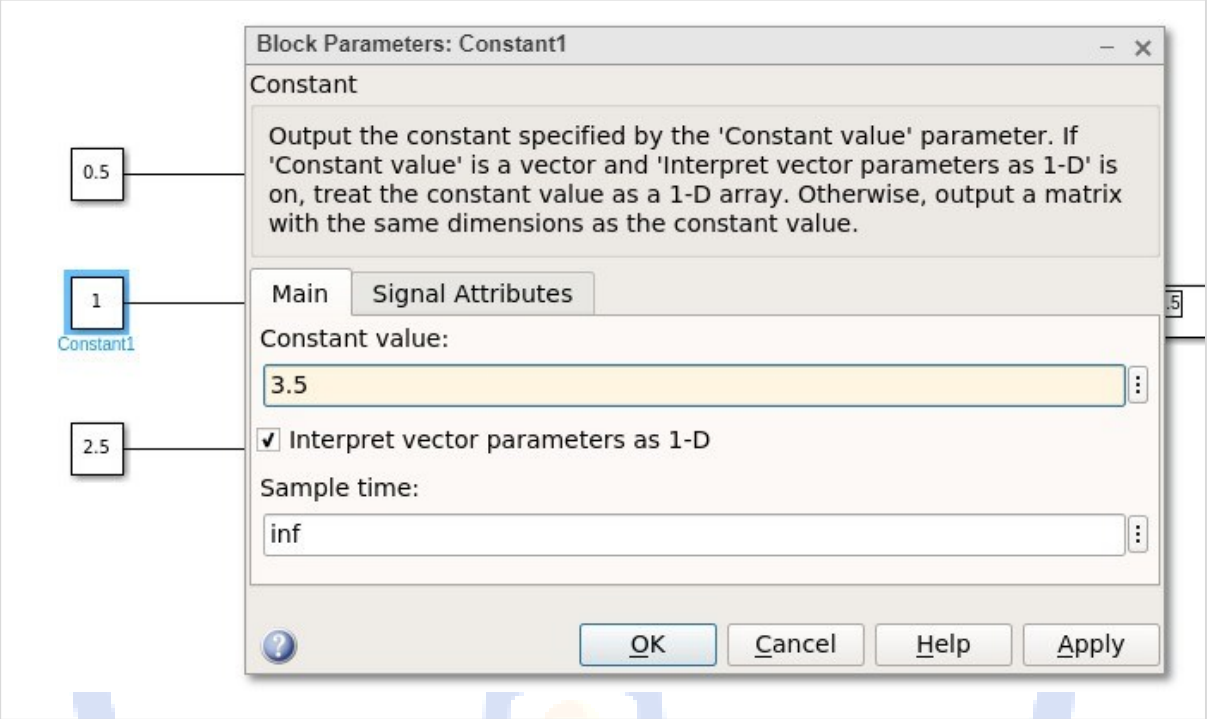


Hence, the first constant value is 0.5, the middle constant value is 1 and the last one is 2.5. The middle constant value 1 will be compared to switch threshold value i.e. 3 as (1 >3). It will print the value as 2.5 the last constant value.

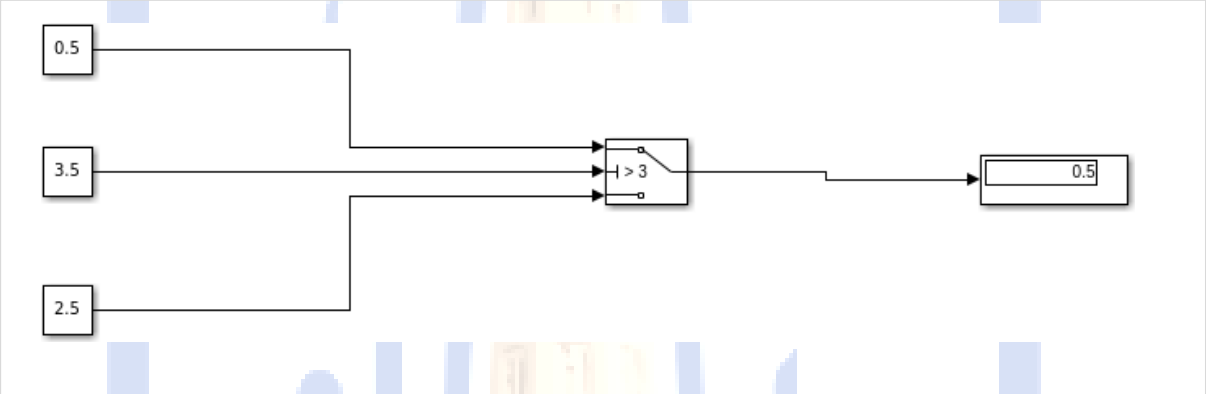
Click on the run button to get the output in the display block as shown below:



Let us now change the middle constant to a higher value than the threshold of switch and see the output:



The value is changed from 1 to 3.5. Click on OK and run the model to see the output in the display:

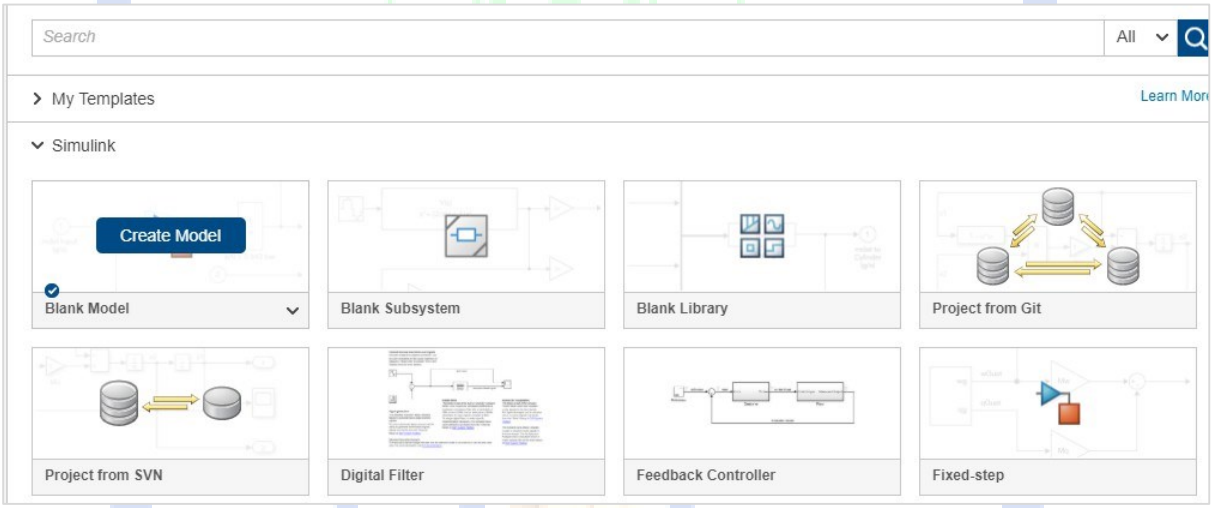


Now, since the value of the middle constant is greater, the value from the first constant is printed in display. If it is less, then the value from the last constant will be printed.

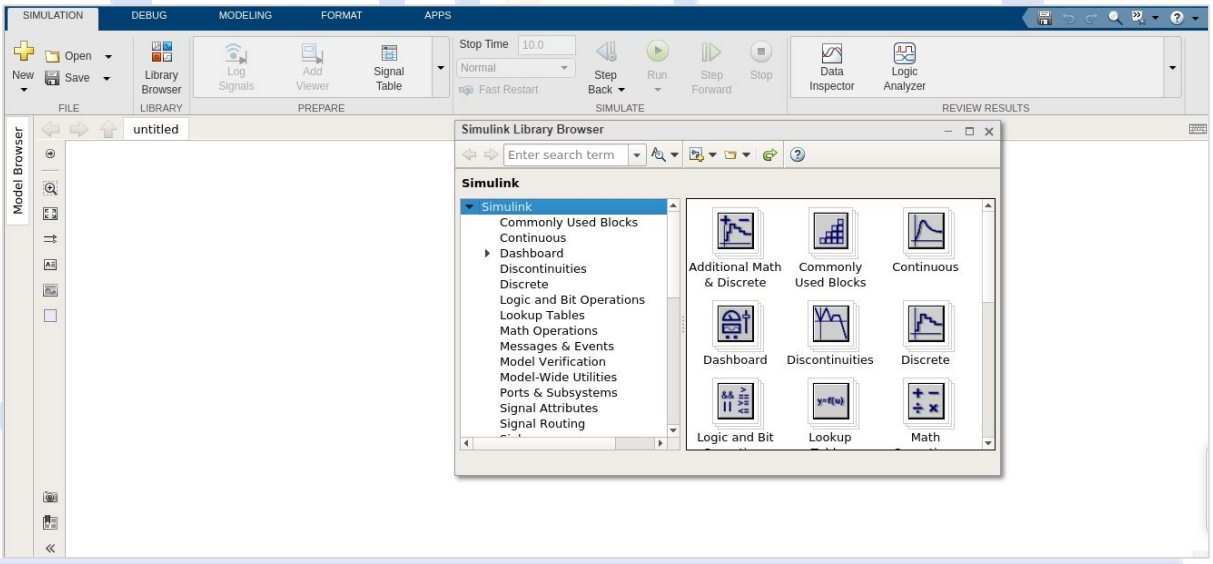
11. MATLAB Simulink — Logic Gates Model

In this chapter, let us understand how to build a model that demonstrates the logic gates. For example, gates like OR, AND, XOR etc.

Open the Simulink and open a blank model as shown below:

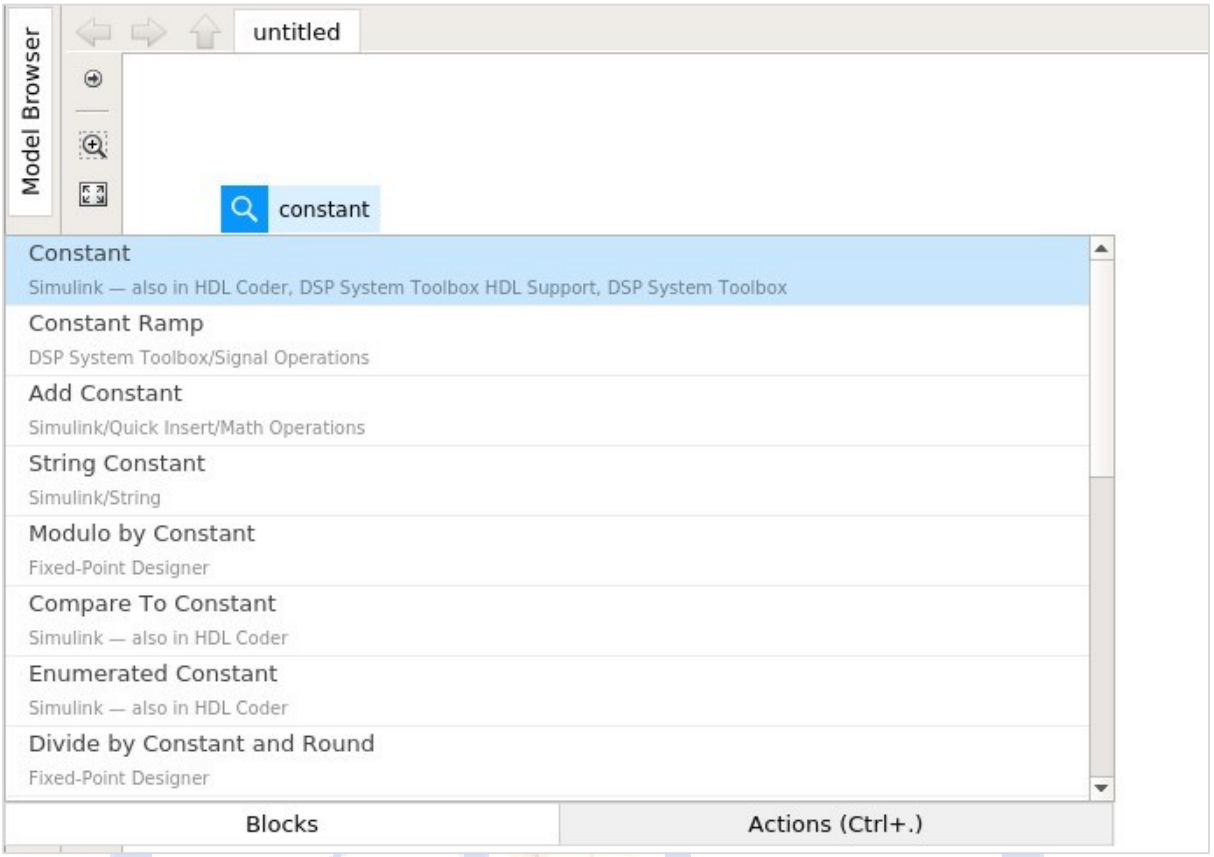


Click on blank model and select the Simulink library as shown below:

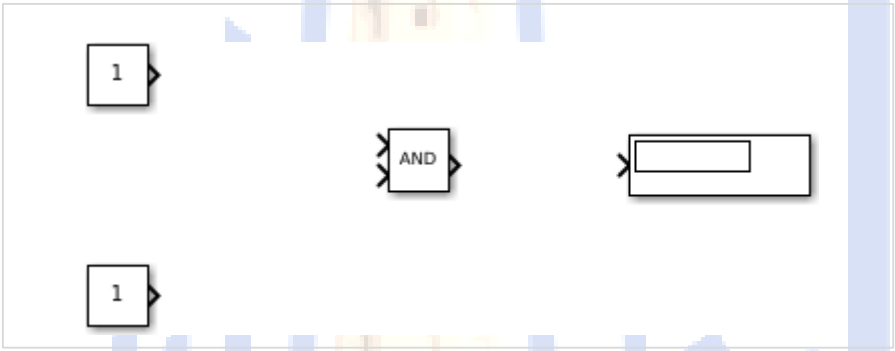


Let us select the block that we want to build a OR gate. We need two constant blocks to act as inputs, a logic operator block and a display block.

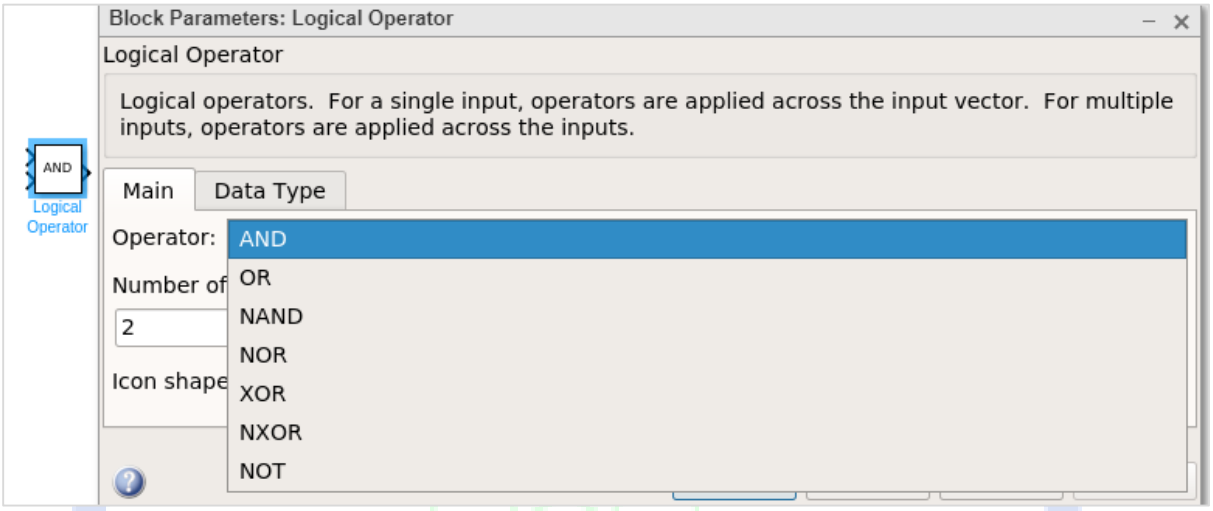
The constant and logic operator block will get from commonly used blocks library. Select the blocks and drag in your model or just type the name of the block in your model and select the block as shown below:



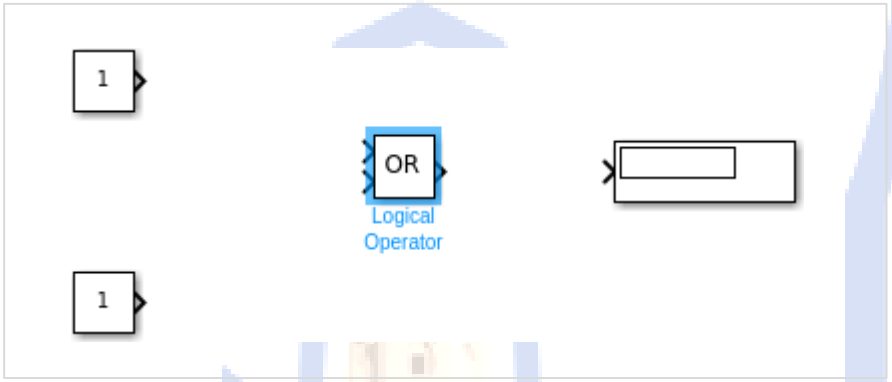
Select the constant block, we need two constant block, a logical operator and a constant. The blocks will look as shown below:



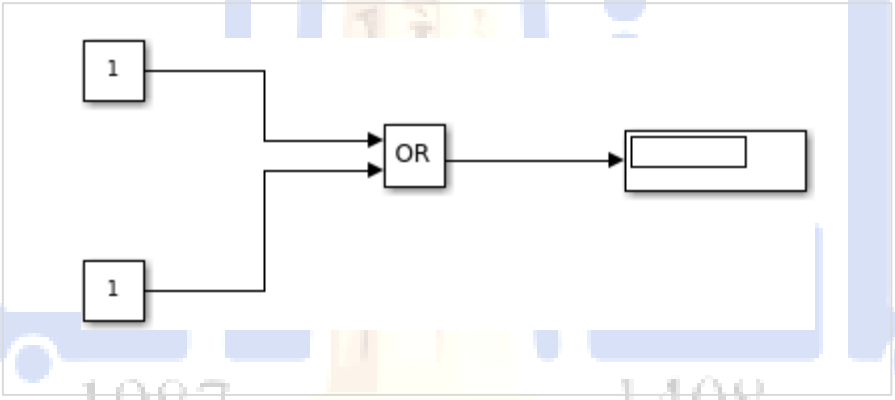
Right click on the logic operator block and it will display the block parameters as shown below:



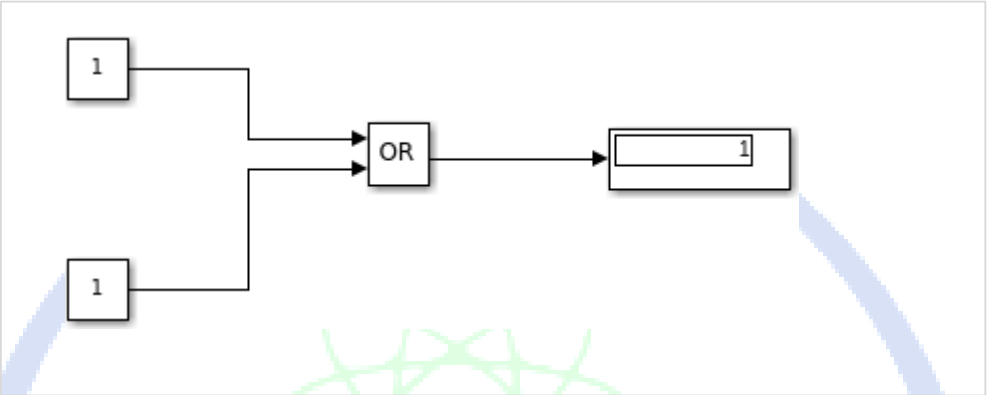
With logical operator you can use AND, OR, NAND, NOR, XOR, NXOR and NOT gates. Right now we are going to select the OR gate.



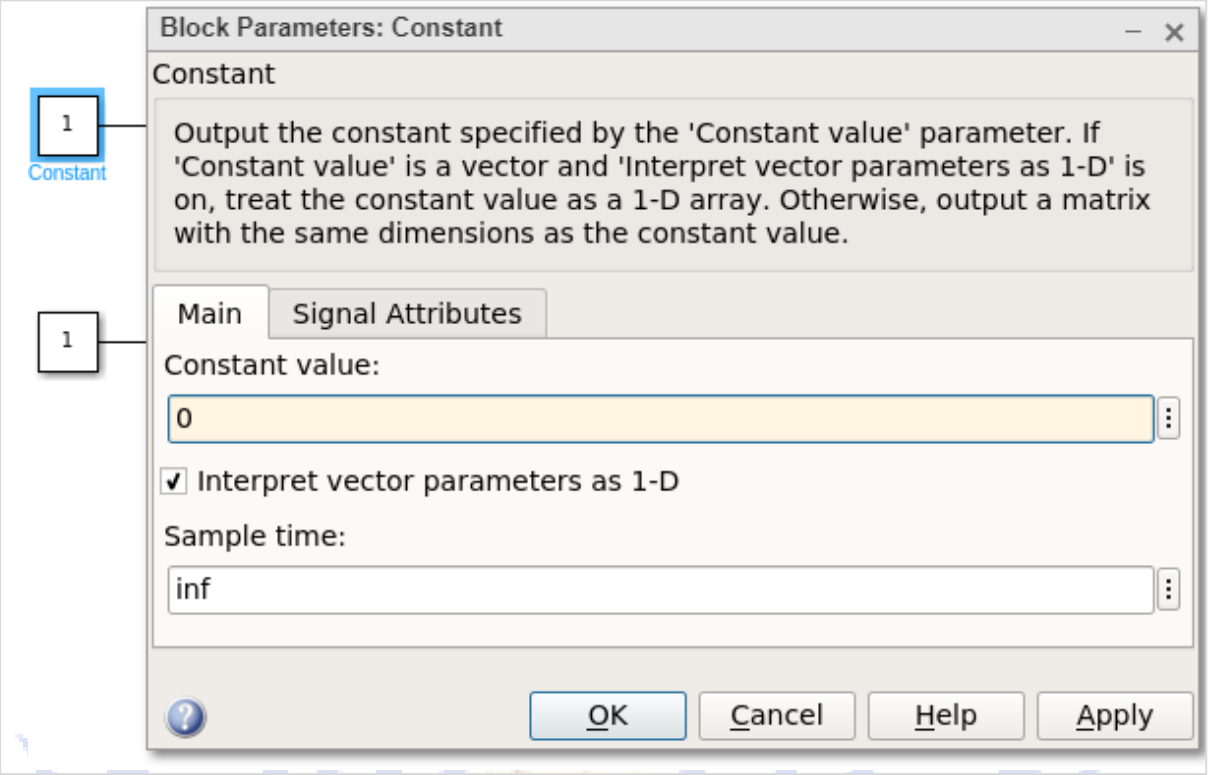
Now connect the lines and the model will be as shown below:



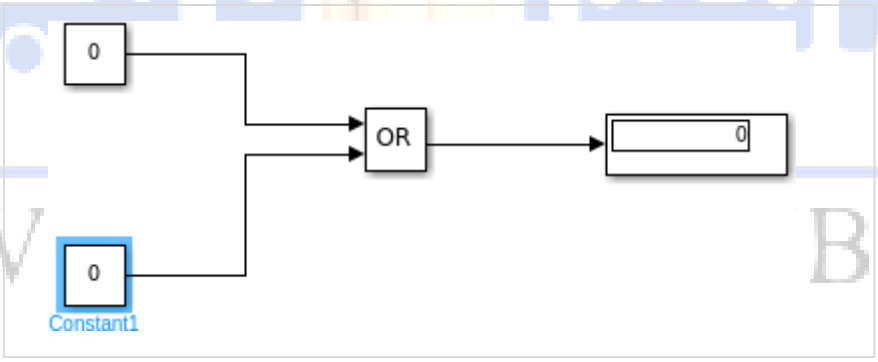
For an OR gate if the inputs are 1,1 the output will be 1. If the inputs are 0,0 the output will be 0. Right now, the constant has values 1,1. Let us run the model to see the output as shown below:



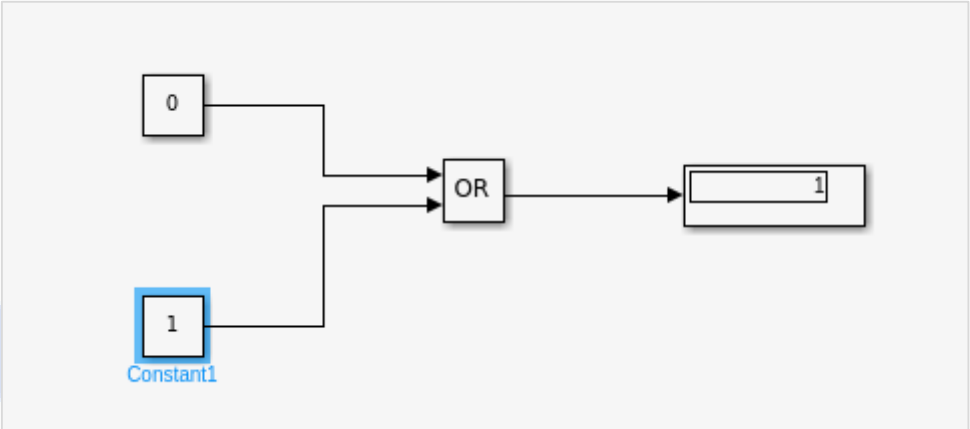
We can see in the display block the output shown is 1. Let us now change the constant value to 0. Right click on constant block and change the value as shown below:



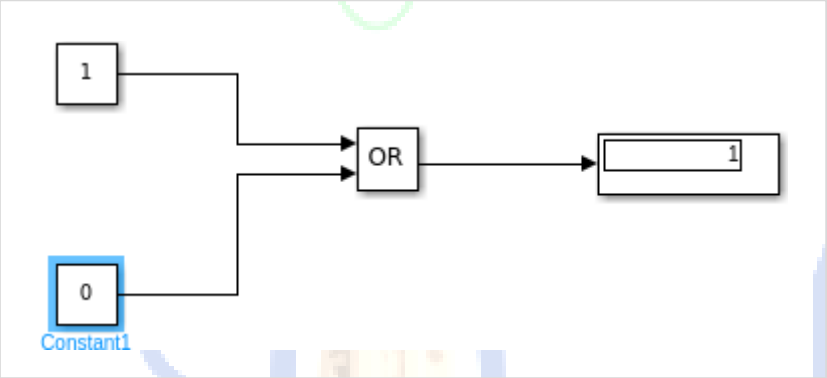
After changing the values of constant to 0, the output will become 0 when you run the model. The output is as shown below:



Let us now change the constant values to 0,1 and see the output:



With values as 1,0, the display will be as follows:

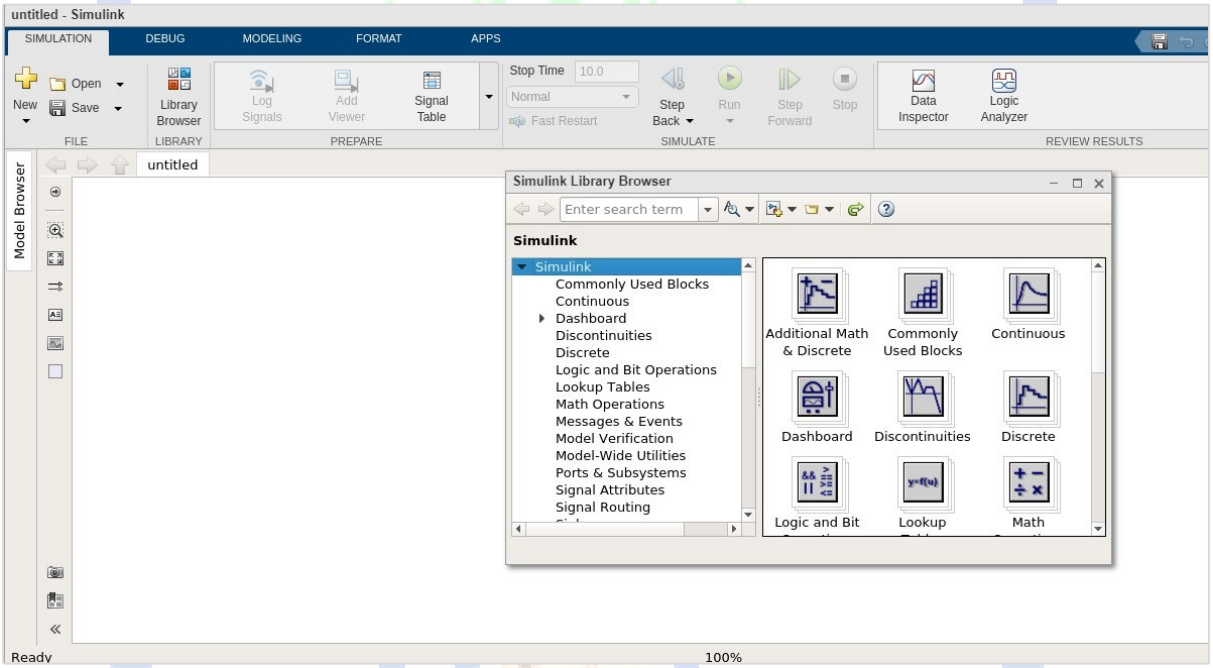


Similarly, you can design the AND and other gates.

12. MATLAB Simulink — Sine wave

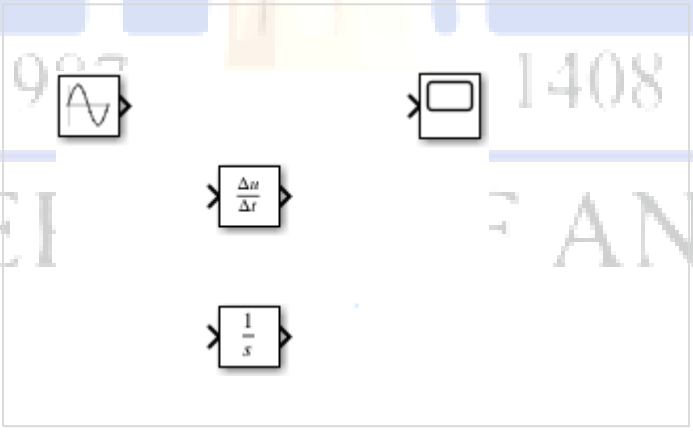
In this chapter we will integrate and differentiate sine wave by using the derivative and integrator blocks.

Open blank model and Simulink library as shown below:



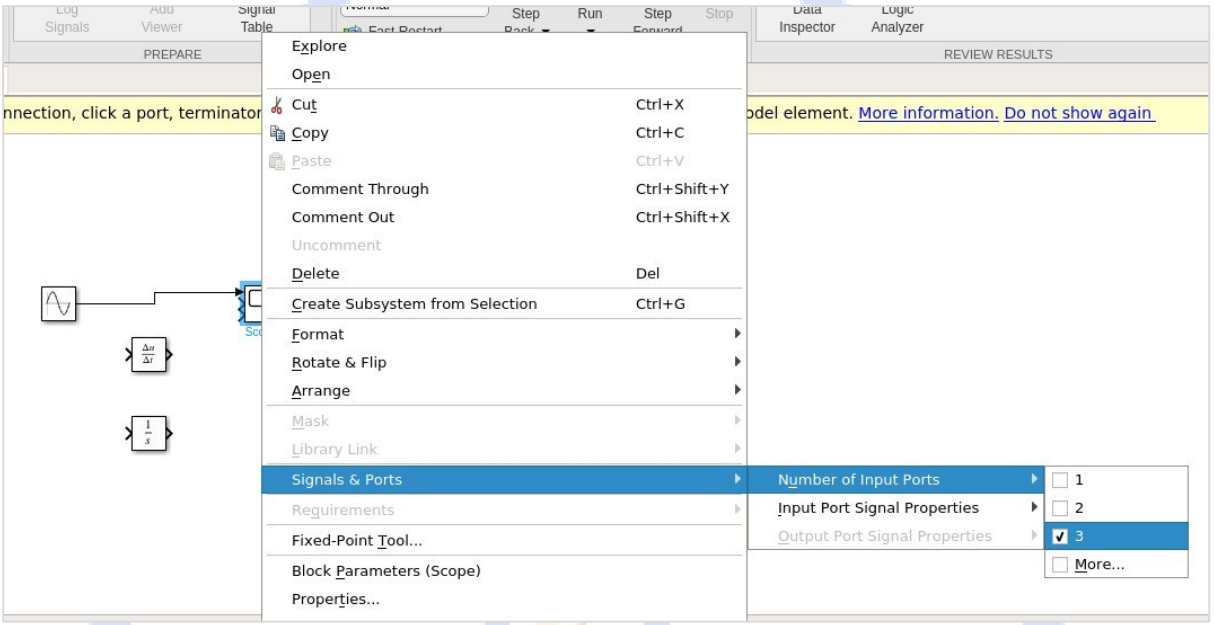
Let us pick the sine wave from sources library and scope block from sinks library.

We would like to add the derivative and integrator block from continuous library as shown below:

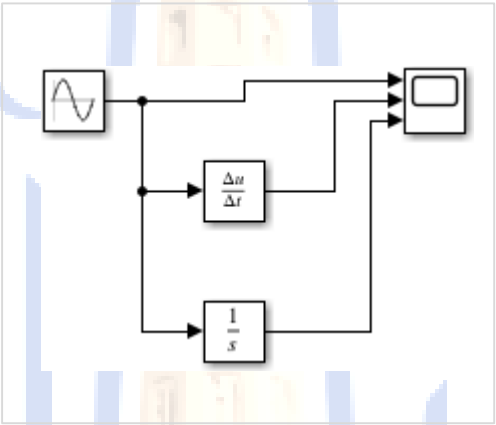


We would need 3 input ports for scope block as the sine wave, derivative and integrator block will be connected to it.

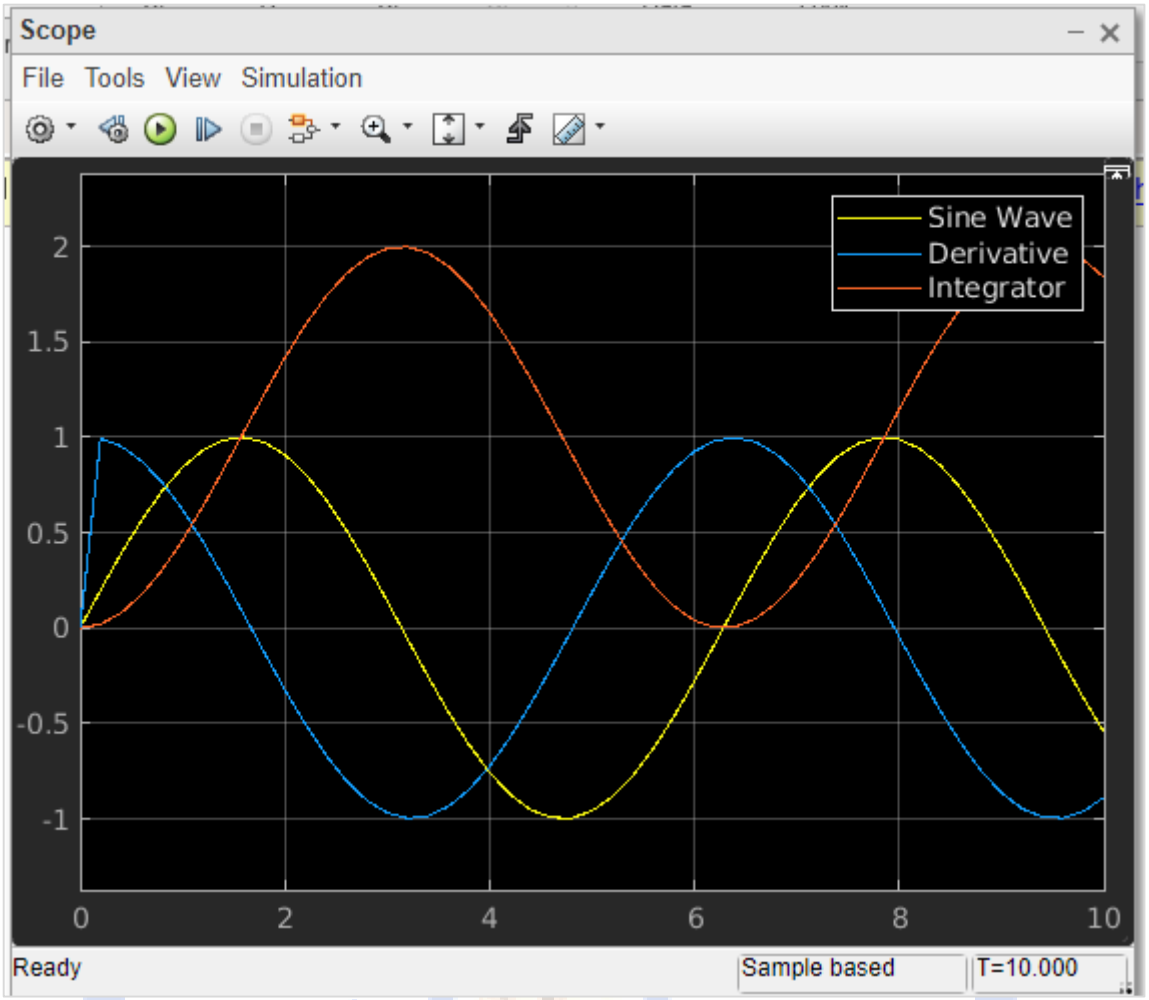
Right click on the scope block and change the inputs from 1 to 3 as shown below:



Connect the lines as shown below:



Now, run the model to see the display.



So, we have three signals sine wave, derivative and integrator.