EEE 221, ETE 221 – Signals and Systems

Experiment No: 05

System Modelling Using SIMULINK

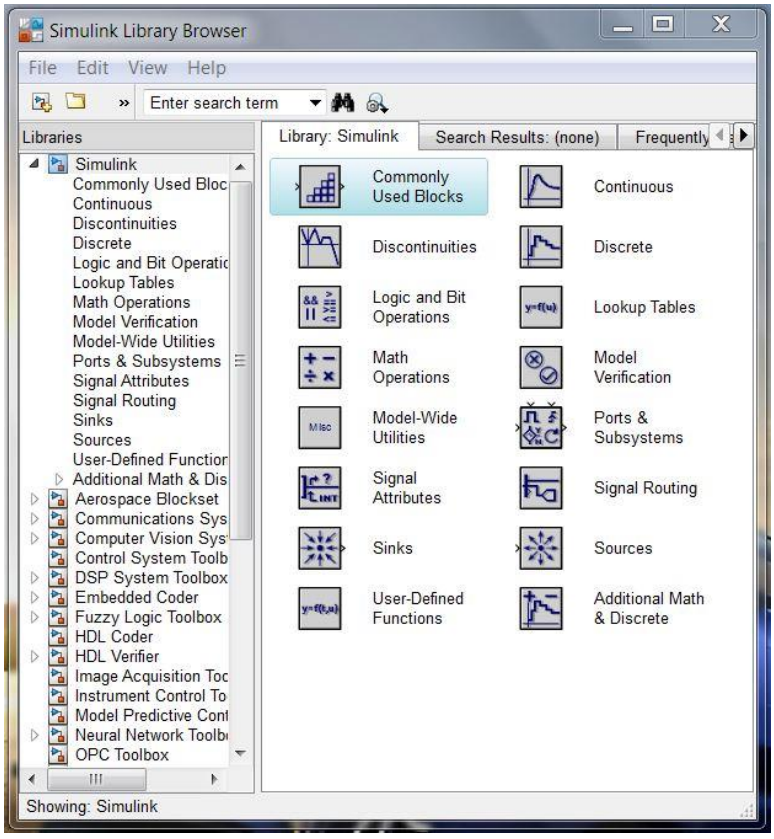
Prepared by: Mehrab Hossain Likhon

SIMULINK is a software package for modeling, simulation and analyzing dynamical systems. It is built on top of MATLAB, so you must have MATLAB to use SIMULINK, it is included in MATLAB software and is also available separately.

Starting SIMULINK:

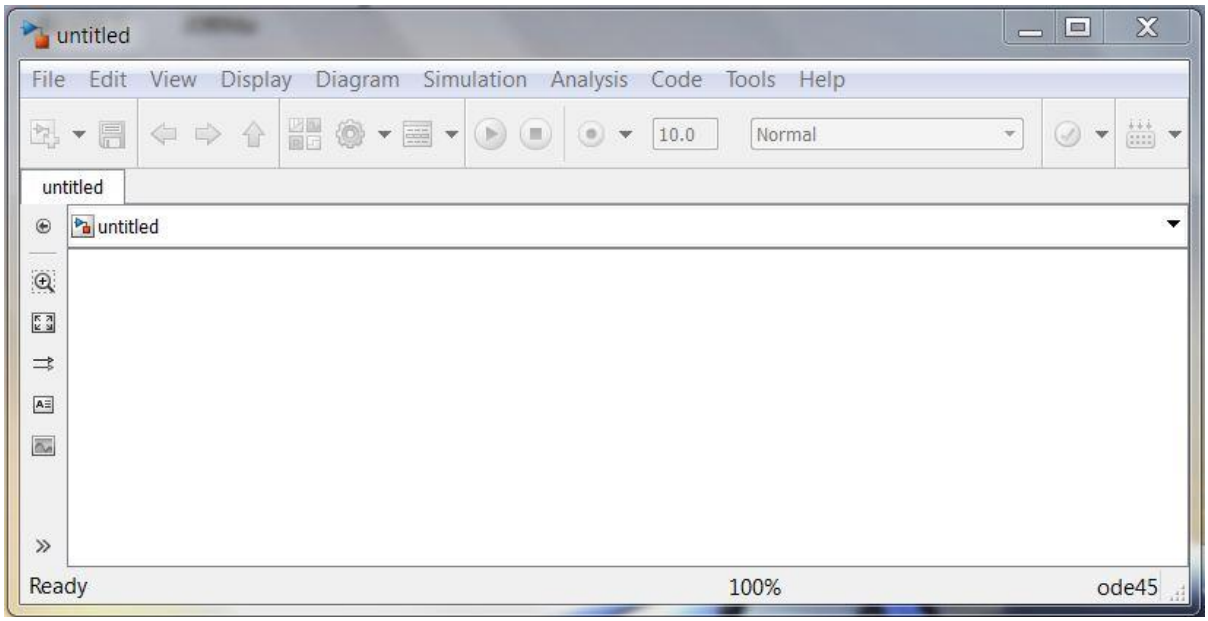
1. You can start Simulink by clicking on the Start button on the MATLAB desktop tool bar. Then click Simulink Library.
2. Type `simulink` in the command window.
3. From MATLAB toolstrip, go Home → Simulink Library.

This opens the SIMULINK library window which is shown in following Figure



Lab Instructor: Mehrab Hossain Likhon
Email: mehrab.hossain@northsouth.edu

To begin to use SIMULINK, click **File→New→Model** from File menu. This opens a blank model window. You create a SIMULINK model by copying units, called blocks, from the various SIMULINK libraries into the model window.



Lab Instructor: Mehrab Hossain Likhon
Email: mehrab.hossain@northsouth.edu

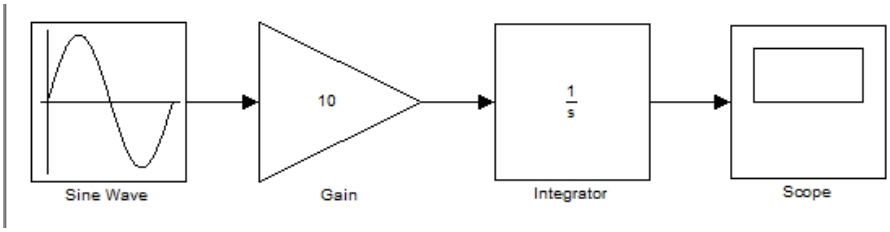
Practice 1:

We will use Simulink to solve the following problem for $0 \leq t \leq 13$

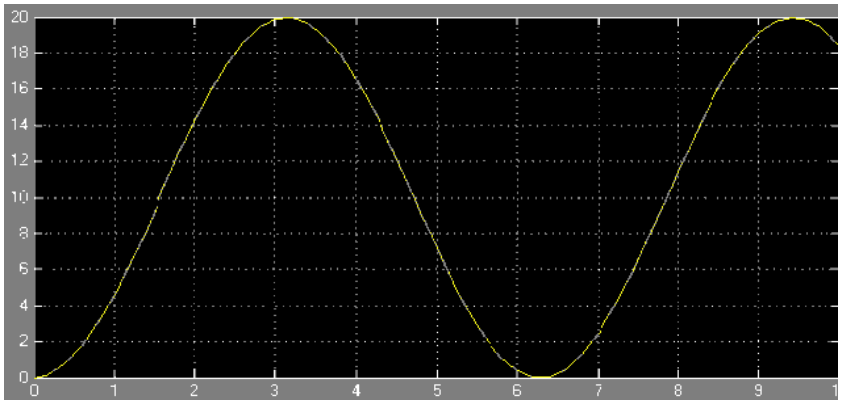
$$\frac{dy}{dt} = 10\sin(t), \quad y(0) = 0$$

The exact solution is $y(t) = 10(1 - cost)$.

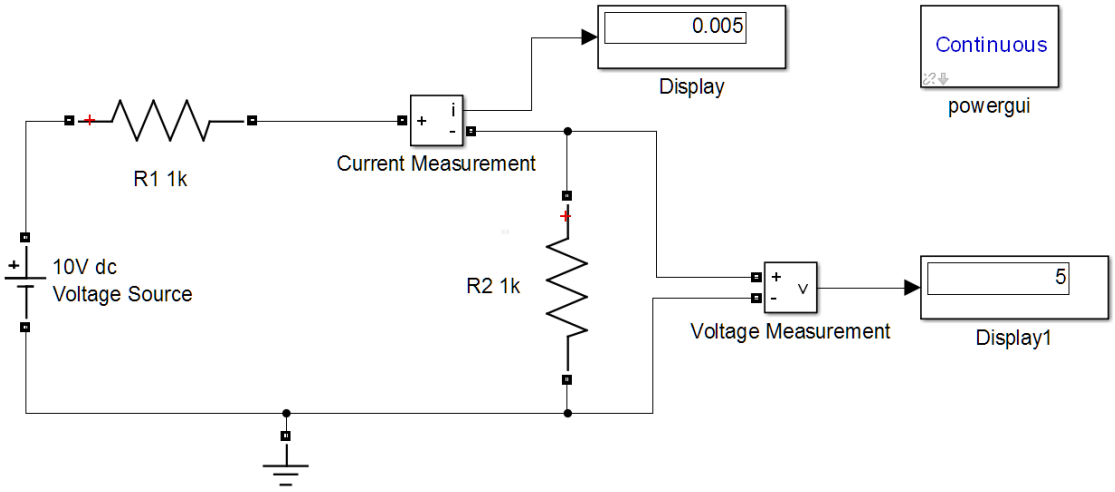
Solution:



1. In the command window, type **`simulink`** to start Simulink, then press enter.
2. Open a new model window.
3. Select and place (click and drag) the **Sine Wave block** from the **Sources library**, double click on it to open the block parameters window. Make sure the amplitude is set to 1, the frequency to 1, the phase to 0, and the sample time to 0, then click ok
4. Select and place the **Gain block** from the **Math operations library**, double click on it, and set the Gain value to 10 in the block parameters window, then click ok.
5. Select and place the **Integrator block** from the **Continuous library**. Double click on it to obtain the Block parameters window, and set the initial condition to 0 {y(0)=0} then click ok.
6. From the **Sinks library**, select the **Scope block**.
7. Once the blocks have been placed as shown in the following figure, connect the input port on each block to the output port on the preceding block. To do this, move the courser to an input port or an output port. The cursor will change to a cross. Hold the mouse button down and drag the cursor to a port on another block. When you release the mouse button, Simulink will connect them with an arrow pointing at the input port. The model should now look like the following figure.
8. Run the Simulation by clicking on the Simulation menu, and then clicking the start. Note: You can also start the simulation by clicking on the start icon on the toolbar.
9. You will hear a bell sound when the simulation is finished. Then double-click on the Scope block and then click on the binoculars icon in the scope display to enable auto scaling you should see an oscillating curve as shown below.



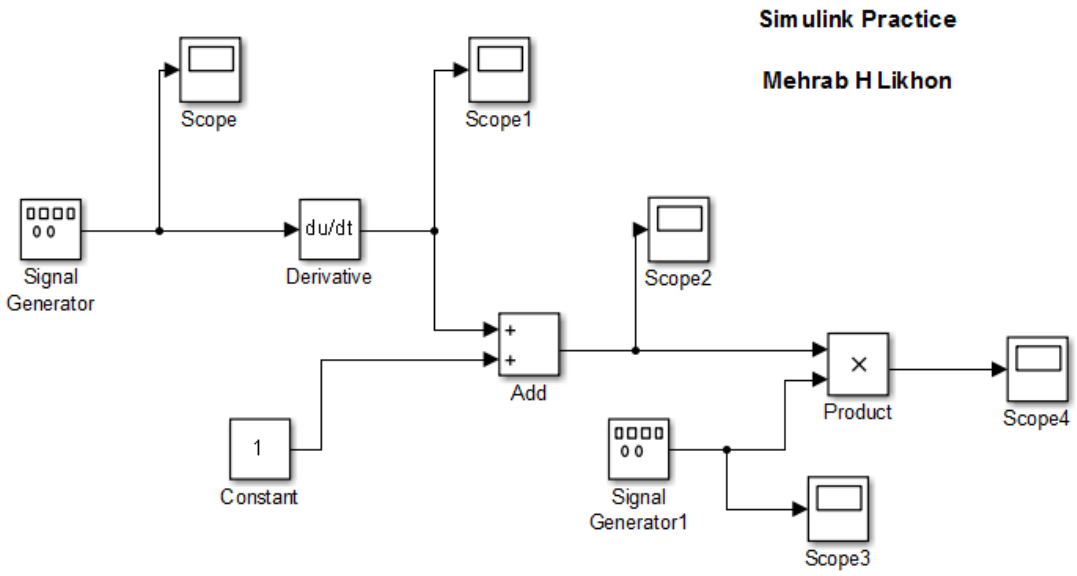
Practice 2: Simple Circuit Simulation



Steps:

1. In the command window, type **`simulink`** to start Simulink, then press enter.
2. Open a new model window.
3. Select and place (click and drag) the **DC Voltage Source** block from the **Simscape → SimPowerSystems → Specialized Technology → Electrical Sources → DC Voltage Source**, double click on it to open the block parameters window. Make sure the amplitude(V) is set to 10 V, then click ok
4. Select and place (click and drag) the **Series RLC Branch** block from the **Simscape → SimPowerSystems → Specialized Technology → Elements → Series RLC Branch**, double click on it to open the block parameters window. Make sure the **Branch type** to **R**, Resistance (Ohms) is set to corresponding 1000 Ω, then click ok
5. Select and place (click and drag) the **Current Measurement** block from the **Simscape → SimPowerSystems → Specialized Technology → Measurements → Current Measurement,** then click ok
6. Select and place (click and drag) the **Voltage Measurement** block from the **Simscape → SimPowerSystems → Specialized Technology → Measurements → Voltage Measurement,** then click ok
7. From the **Simulink → Sinks** library, select **Display** block and place (click and drag). From the **Simscape → SimPowerSystems → Specialized Technology** library, select the **powergui** block and place (click and drag).

Practice 3: Signal Multiplication and Add Text



Create this system model on your model window. Select and drag the required blocks from Simulink Library Browser.

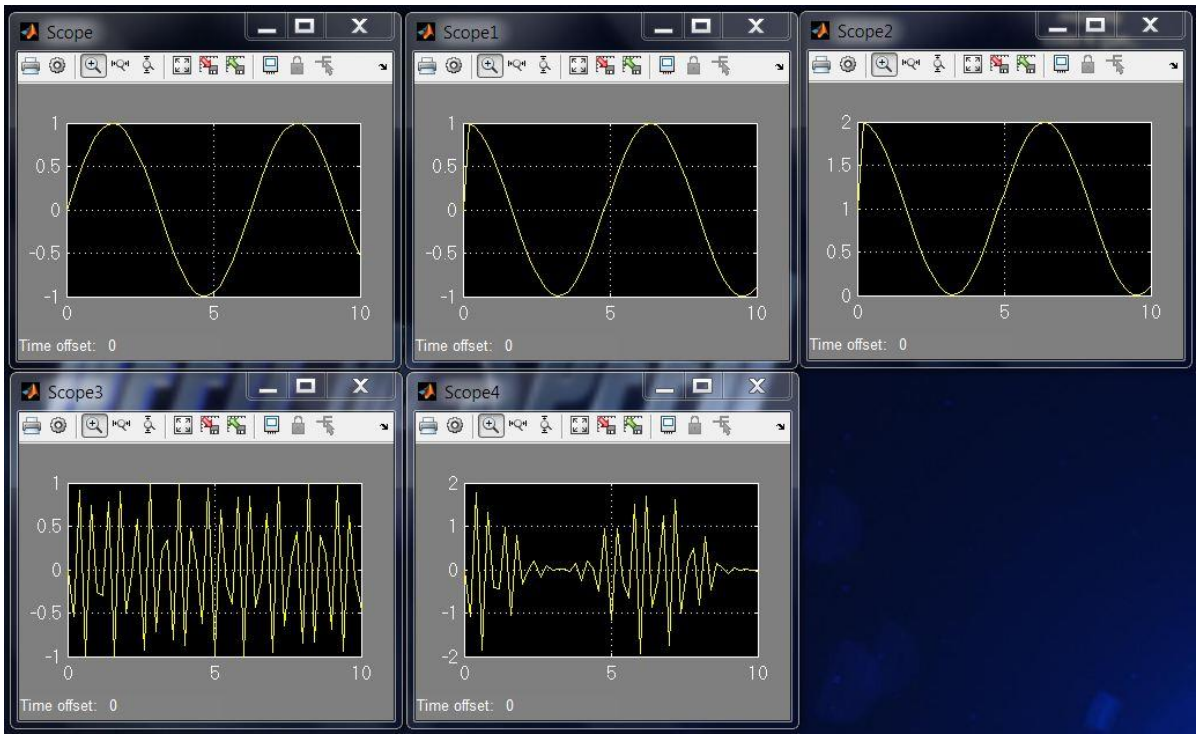
- Signal generator from **Sources**
- Derivative from **Continuous**
- Constant from **Sources**
- Adder from **Math Operation**
- Product from **Math operations**
- Scope from **Sinks**

\

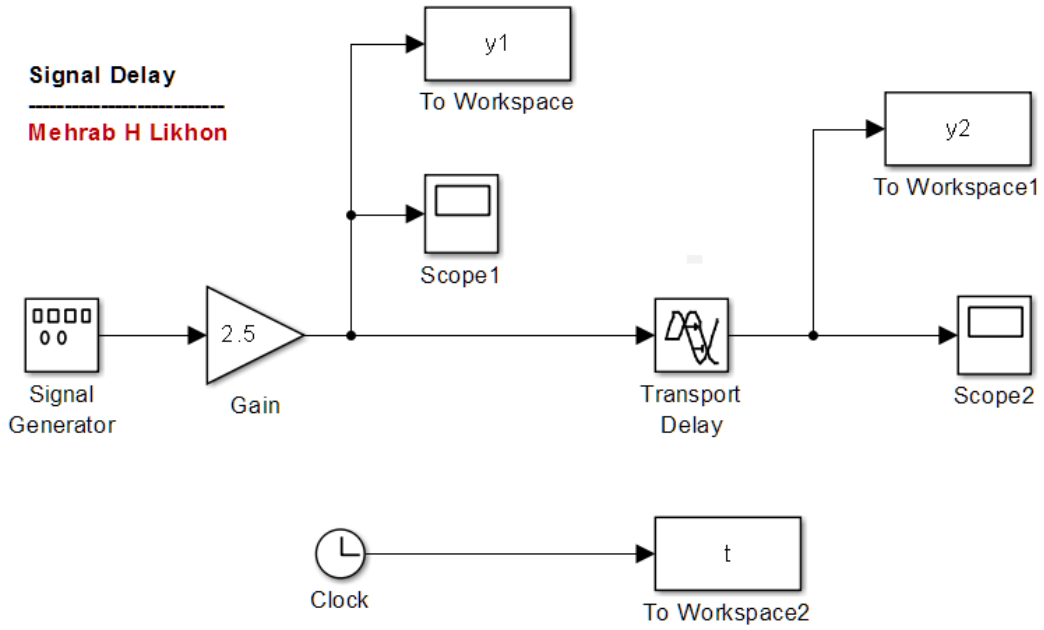
→ For signal generator: amplitude = 1, frequency = 1.

→ For signal generator 1: amplitude = 1, frequency = 20.

→ Run the model file and check all the outputs from scopes.



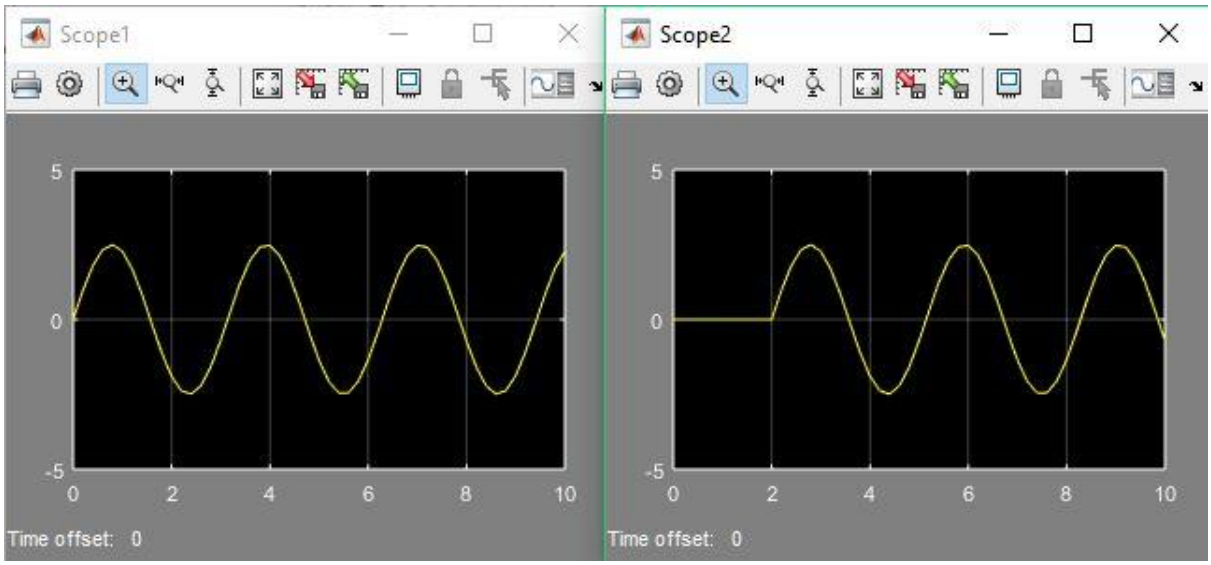
Practice 4: Signal Delay and Plot



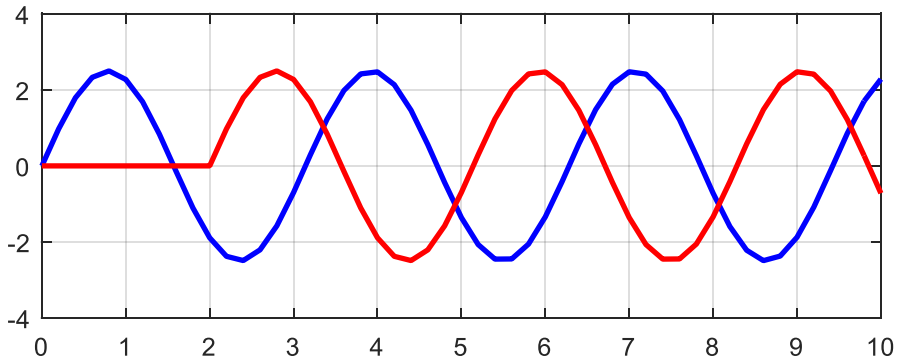
Create this system model on your model window. Select and drag the required blocks from Simulink Library Browser.

- Signal generator from **Sources**
- Transport Delay from **Continuous**
- Gain from **Math Operations**
- Scope from **Sinks**
- To Workspace from **Sinks**
- Clock from **Sources**

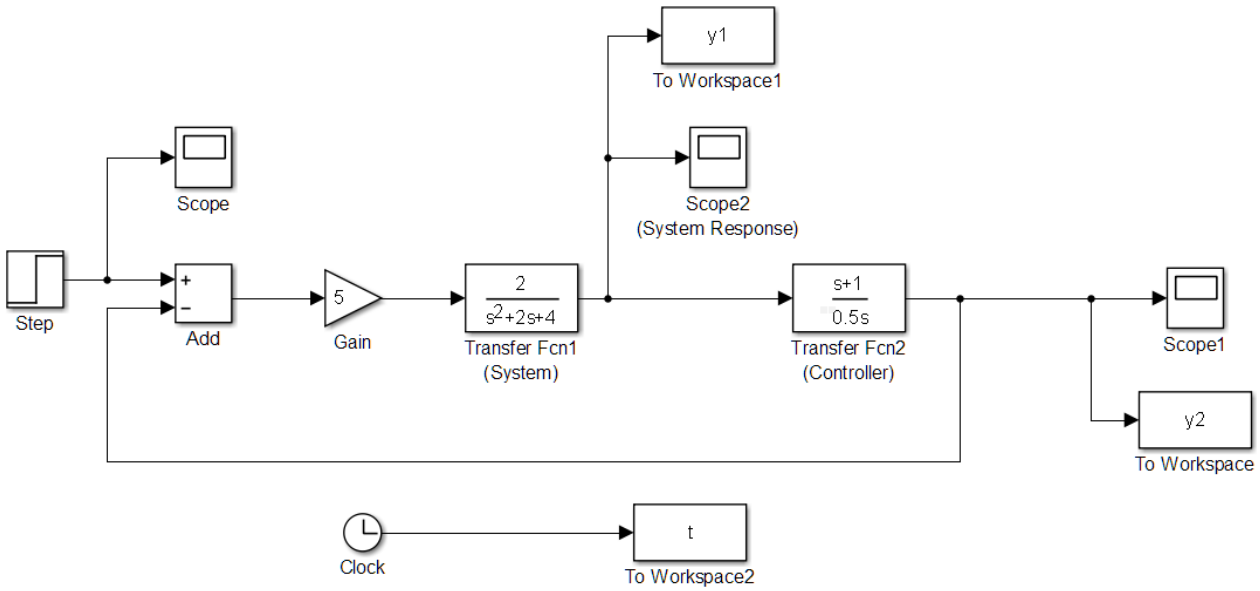
→ In Gain block, set gain to 2.5.
→ In Transport Delay block, set time delay to 2
→ In To Workspace block, change Variable Name (t, y1, y2 etc.) and set Save Format as Array
→ Run the model file and check all the outputs from scopes



→ Check workspace for your variables (t, y1, y2 etc.) and plot signals using these variables



Practice 5: Unit Step Response of a System and Design a Controller (with Negative Feedback)

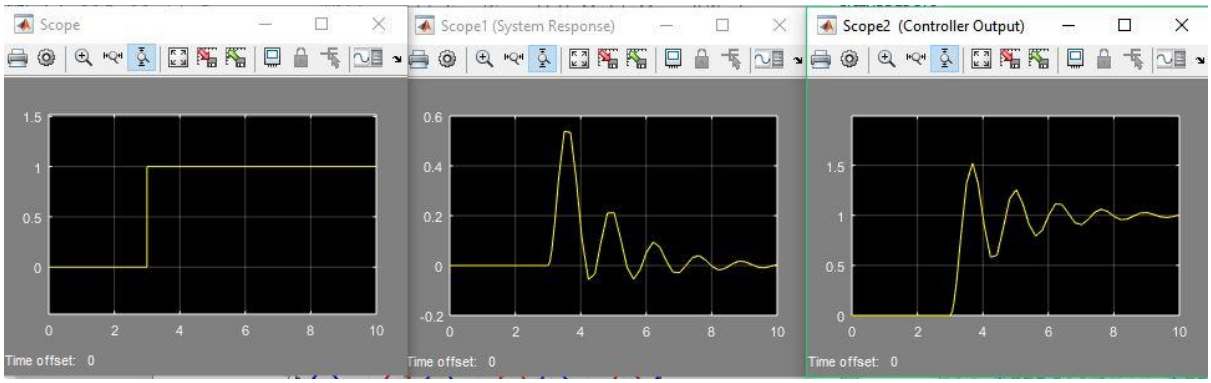


Create this system model on your model window. Select and drag the required blocks from Simulink Library Browser.

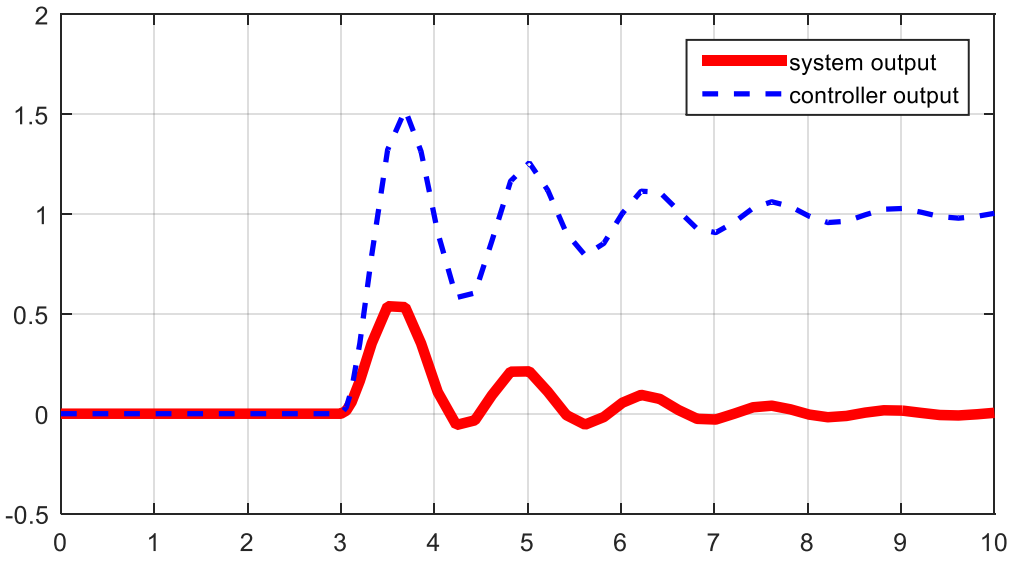
- Signal generator from **Sources**
- Add from **Math Operation**
- Gain from **Math Operations**
- Transfer Fcn from **Continuous**
- Scope from **Sinks**
- To Workspace from **Sinks**
- Clock from **Sources**

→ In Add block, change the list of input signs from (++) to (+−)
→ In Gain block, set gain to 5
→ In Transfer Fcn block, change the numerator and denominator
→ In To Workspace block, change Variable Name (t, y1, y2 etc.) and set Save Format as Array

→ Run the model file and check all the outputs from scopes

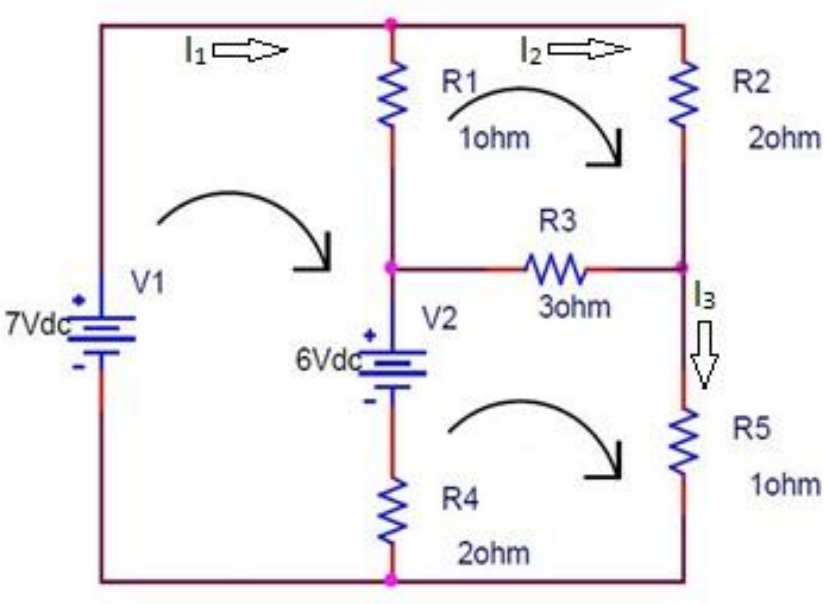


→ Check workspace for your variables (t, y1, y2 etc.) and plot signals using these variables



Homework 5

1. Solve the following circuit using SIMULINK to find i1, i2, and i3. (Submit a printed copy of **model file**)



Answer: i1 = 3 amp, i2 = 2 amp, i3 = 3 amp

Steps:

i. Open **`simulink`** library, then open a new model window.

ii. Select and place (click and drag) the **DC Voltage Source** block from the **Simscape → SimPowerSystems → Specialized Technology → Electrical Sources → DC Voltage Source**, double click on it to open the block parameters window. Make sure the amplitude(V) is set to 7, then click ok

iii. Select and place (click and drag) the **Series RLC Branch** block from **Simscape → SimPowerSystems → Specialized Technology → Elements → Series RLC Branch**, double click on it to open the block parameters window. Make sure the **Branch type** to **R**, Resistance (Ohms) is set to corresponding 1 Ω, then click ok

iv. Select and place (click and drag) the **Current Measurement** block from **Simscape → SimPowerSystems → Specialized Technology → Measurements → Current Measurement,** then click ok

v. From the **Simulink → Sinks** library, select **Display** block and place (click and drag).

vi. From the **Simscape → SimPowerSystems → Specialized Technology** library, select the **powergui** block and place (click and drag).

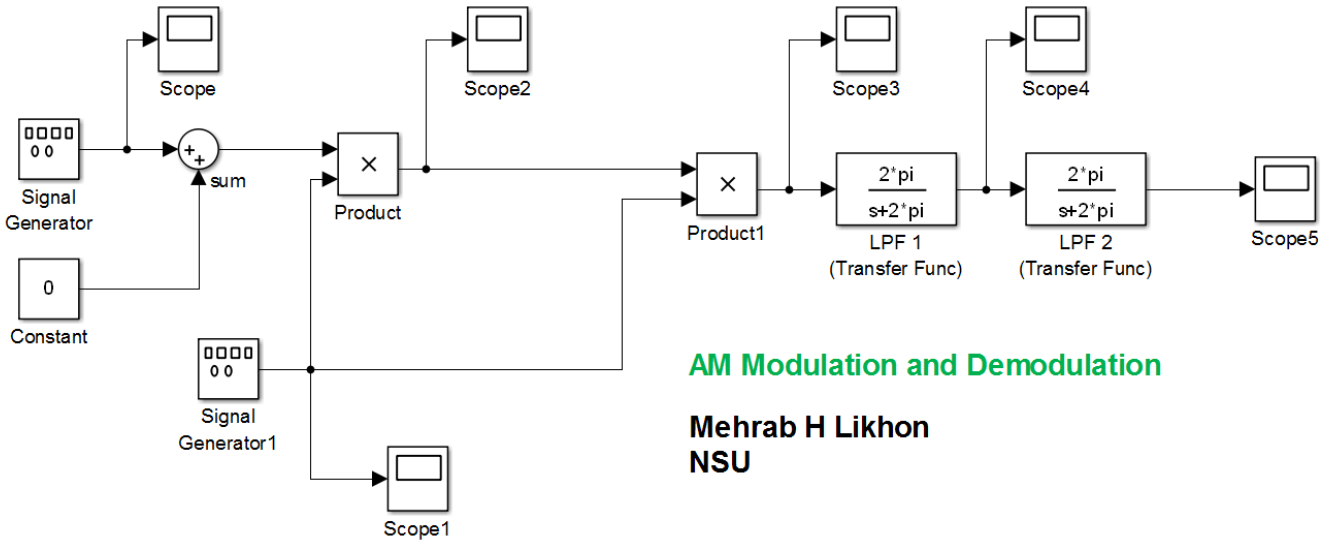
vii. Arrange the circuit according to the question and execute the answer as **i1=3, i2=2, i3=3.**

2. Create a SIMULINK model that will delay a discrete unit step input signal by 3 unit time. Also plot both input and delayed signal.

Lab Instructor: Mehrab Hossain Likhon
Email: mehrab.hossain@northsouth.com

3. Design the following model

Signal Multiplication and Noise Reduction from a Signal Using Filter



Build this model on SIMULINK model file. All the blocks in the corresponding library

- Signal Generator from Sources
- Constant from Sources
- Sum from Math Operations
- Product from Math Operations
- Transfer Func from Continuous
- Scope from Sink

→ For signal generator (message signal): amplitude = 1, frequency = 1.
→ For signal generator 1 (carrier signal): amplitude = 1, frequency = 20.
→ In Transfer Fcn block, change the numerator and denominator
→ Scopes should show output graphs like these-