Experiment -2-

Tutorial on Matlab Simulink blocks for control systems

Objectives:

To learn various Matlab Simulink blocks for control systems.

Introduction:

In this experiment, we introduce several Matlab simulink blocks that are considered essential for solving control systems, such as the transfer function, Gain, Sum, Step (source), Scope (sink or observation), and others.

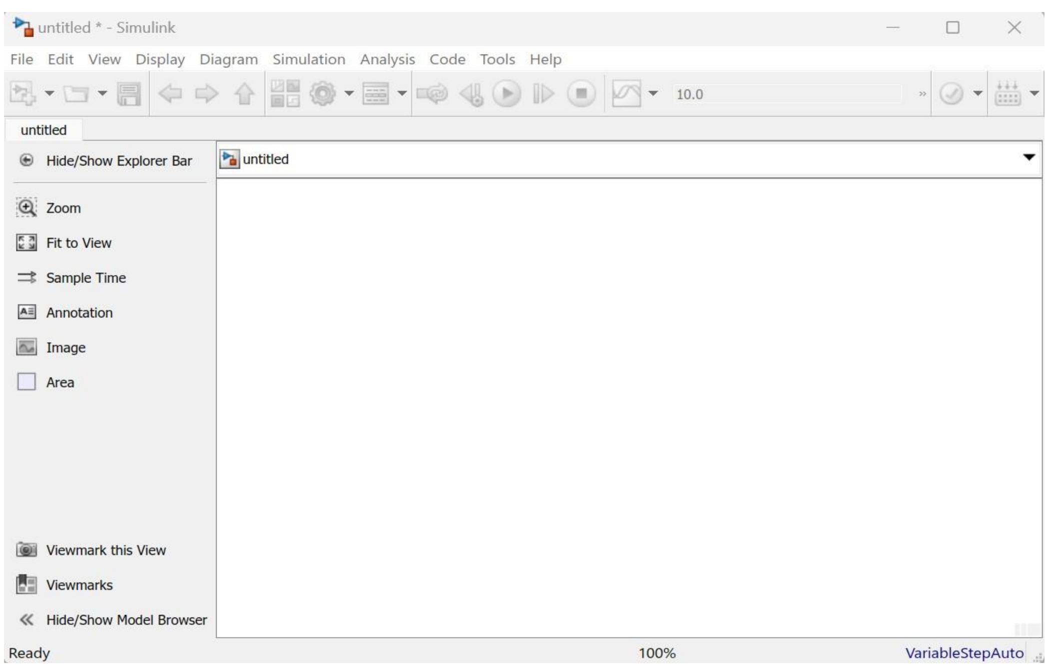
To learn how to create any system design, we will construct a similar control design to that used in the Experiment (1)→Example (1).

Firstly, you can open the Simulink either by typing the following command in the Matlab command window:

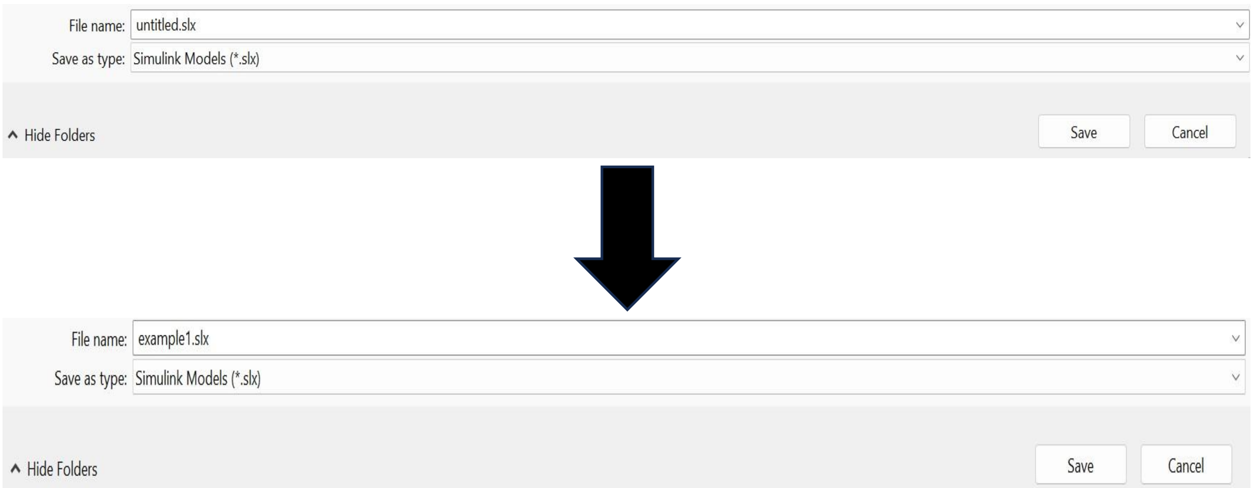
```
>> simulink
```

Or just go to Home→Simulink and select the new Blank Model.

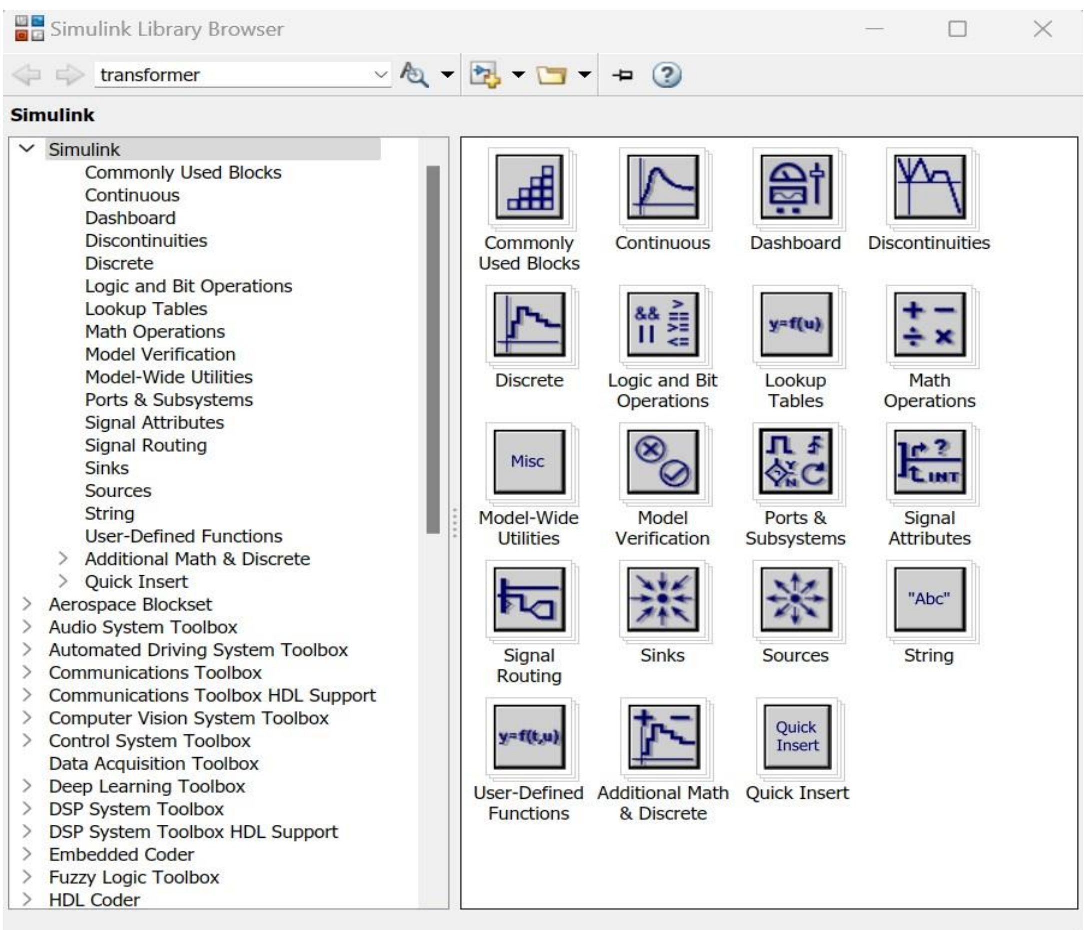
The following window is open:



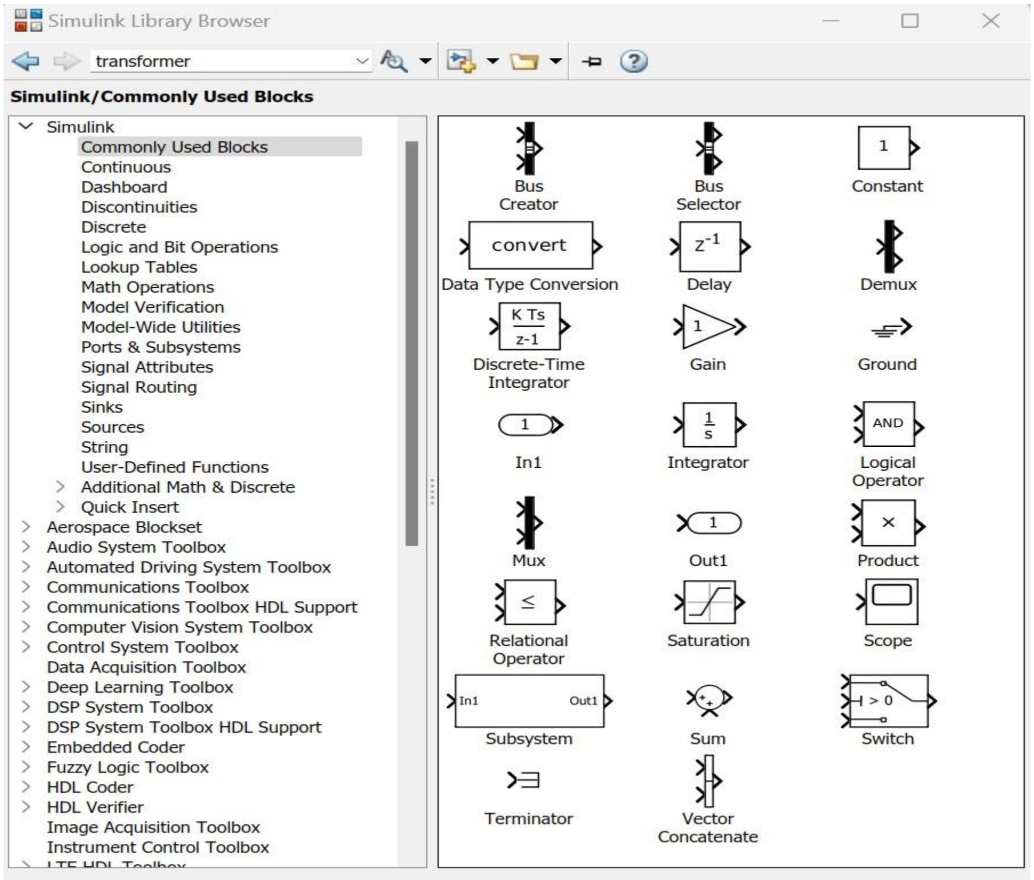
Next, go to File→Save→choose the directory location you want to save the file in→File name→change the file name from (untitled.slx) to (example1.slx)→Press Save.



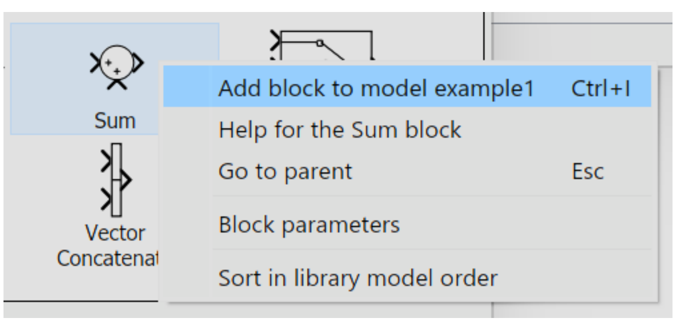
Now, from the new saved window, go and select Library Browser, where a new window is presented as shown in the following:



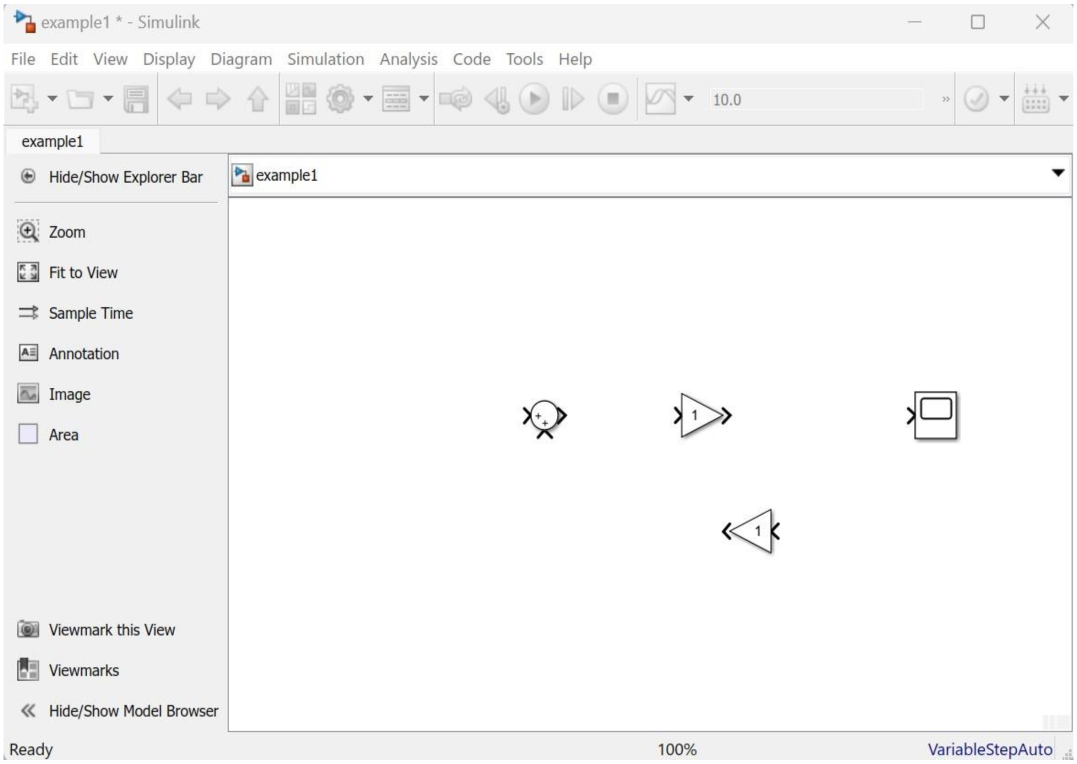
Move the pointer and select (Commonly Used Blocks)



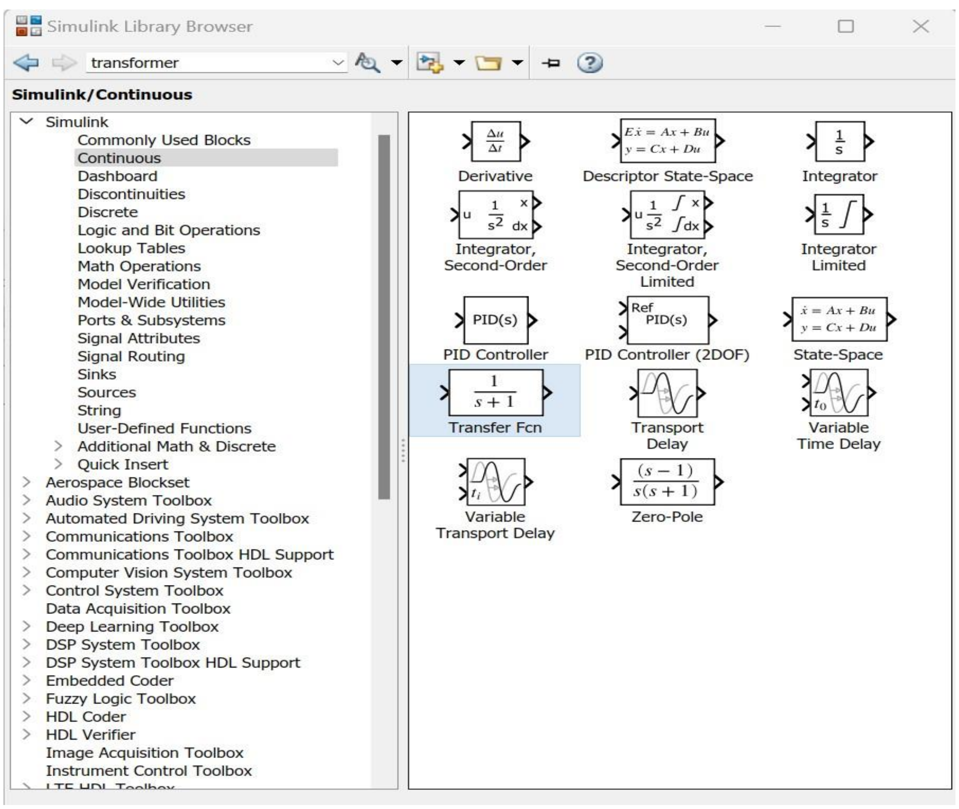
From Commonly Used Blocks, select the Sum block and move it to example1 sheet either by directly dragging the Sum block by the pointer to the sheet or simply pressing right click→ Sum block→Add block to model example1.



Now, the Sum block will move to example1 sheet. Repeat the above steps to insert the Scope block and two Gain blocks.

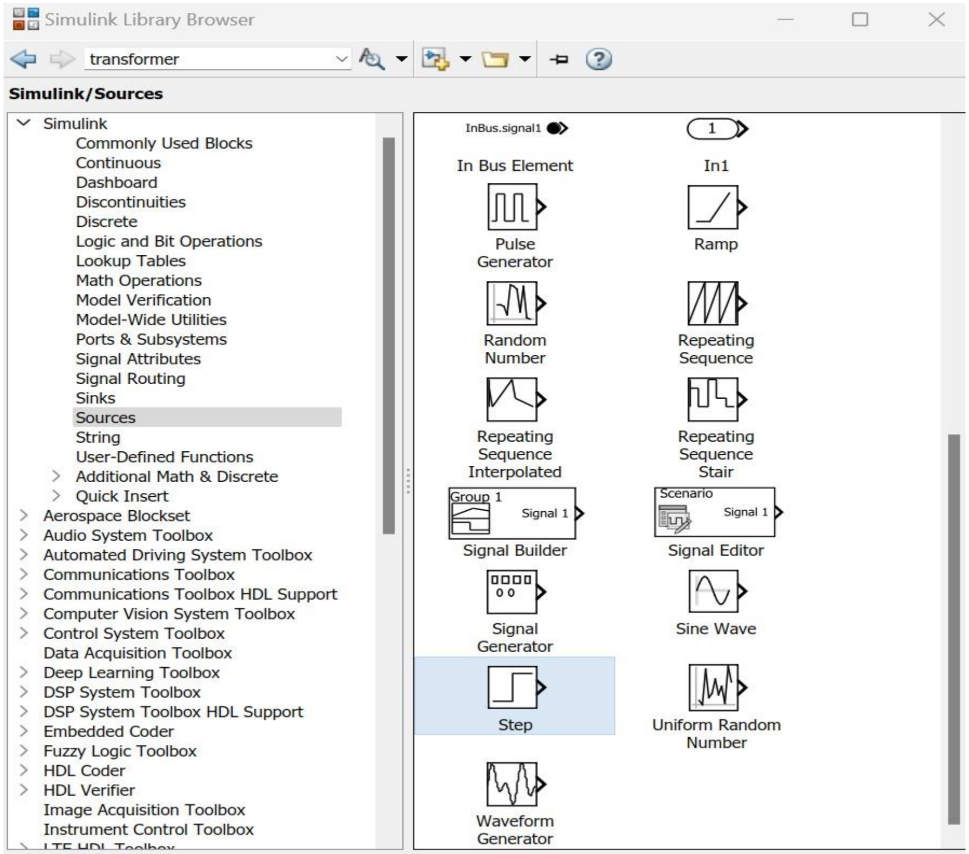


Now, go to the Simulink Library Browser and select Continous

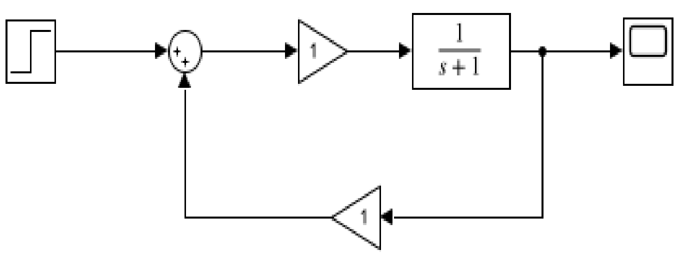


Insert the Transfer Fcn block into the example1 sheet model.

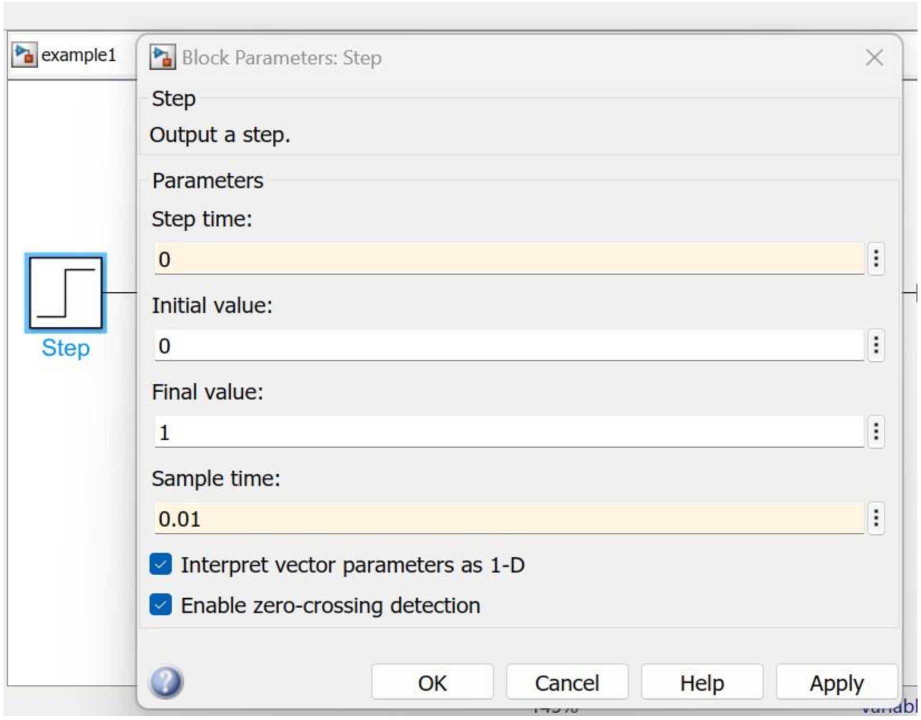
Select Sources from the Simulink Library Browser, select Step block, and move it to the example1 sheet model.



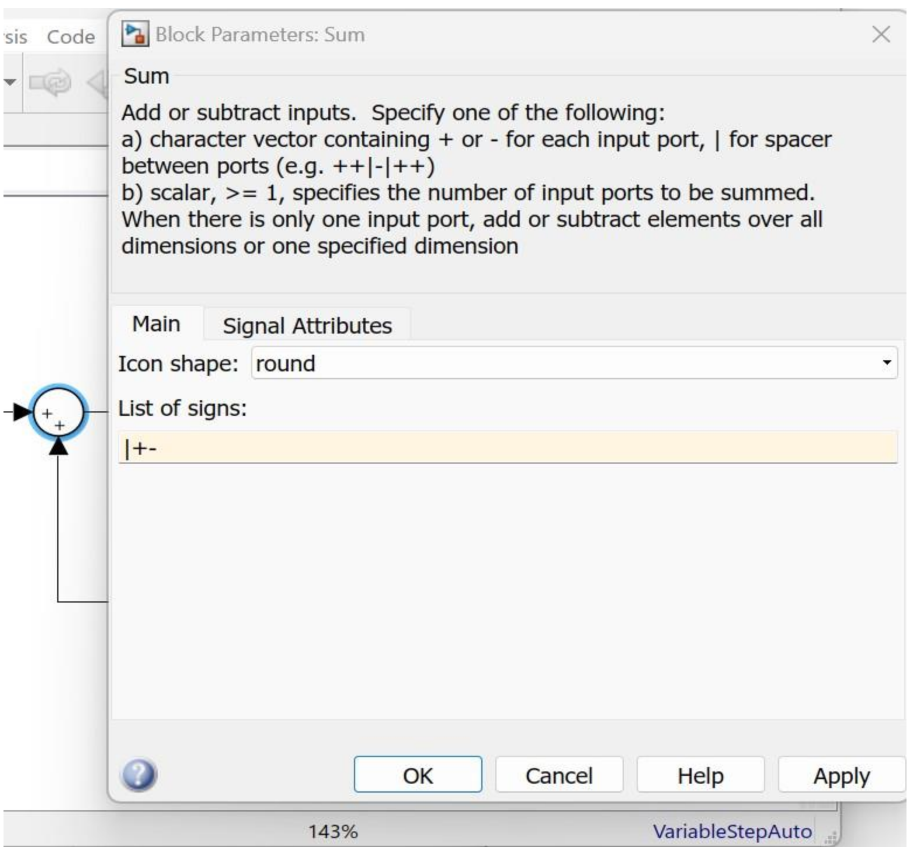
Now, connect all the blocks in the example1 sheet model



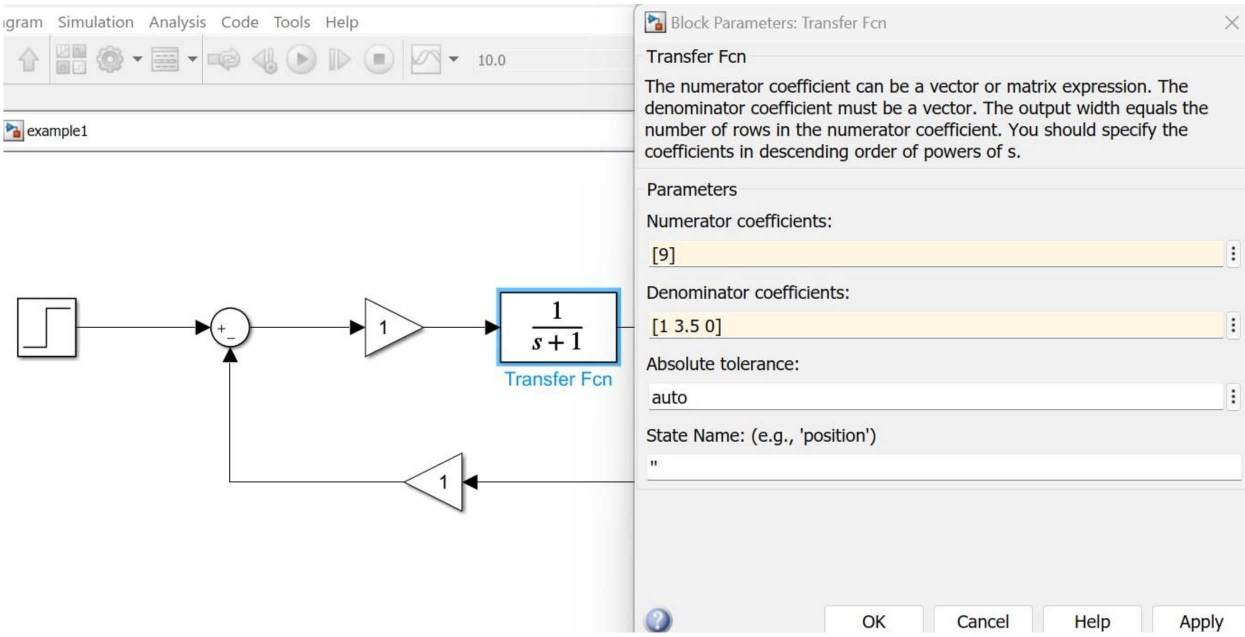
By pressing double clicks, modify the step response block, sum block, and transfer function block settings as shown below



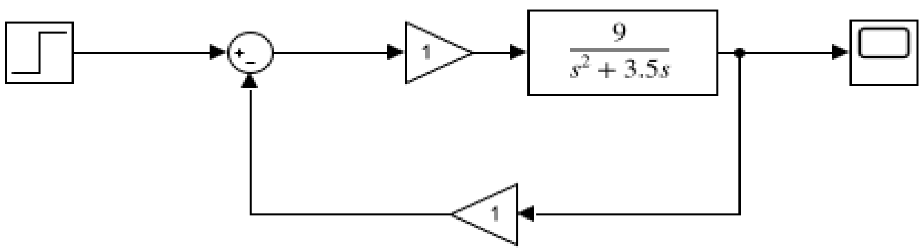
example1
Block Parameters: Step
Step
Output a step.
Parameters
Step time:
0
Initial value:
0
Final value:
1
Sample time:
0.01
Interpret vector parameters as 1-D
Enable zero-crossing detection
OK
Cancel
Help
Apply
Step



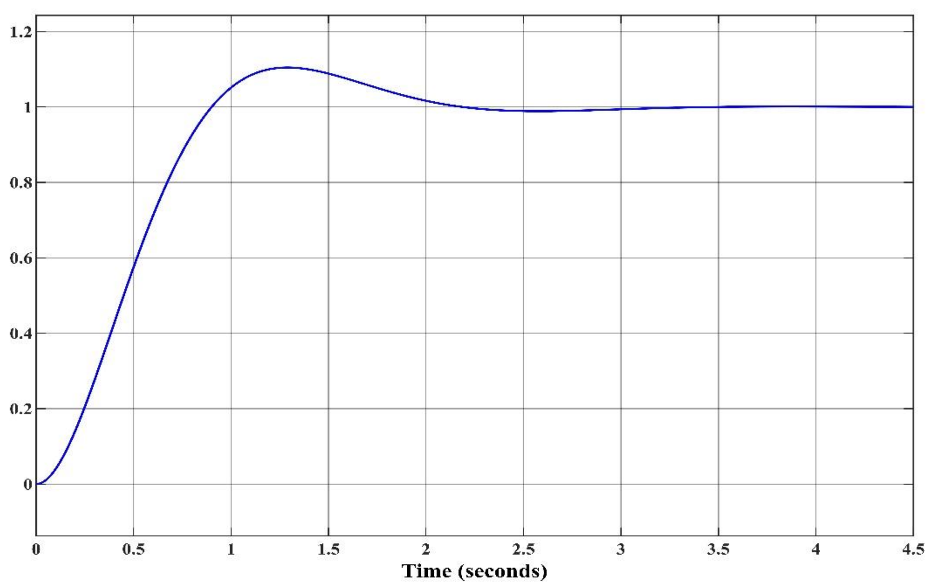
Code
Block Parameters: Sum
Sum
Add or subtract inputs. Specify one of the following:
a) character vector containing + or - for each input port, | for spacer between ports (e.g. ++|-|++)
b) scalar, >= 1, specifies the number of input ports to be summed.
When there is only one input port, add or subtract elements over all dimensions or one specified dimension
Main
Signal Attributes
Icon shape:
round
List of signs:
|+-
OK
Cancel
Help
Apply
143%
VariableStepAuto



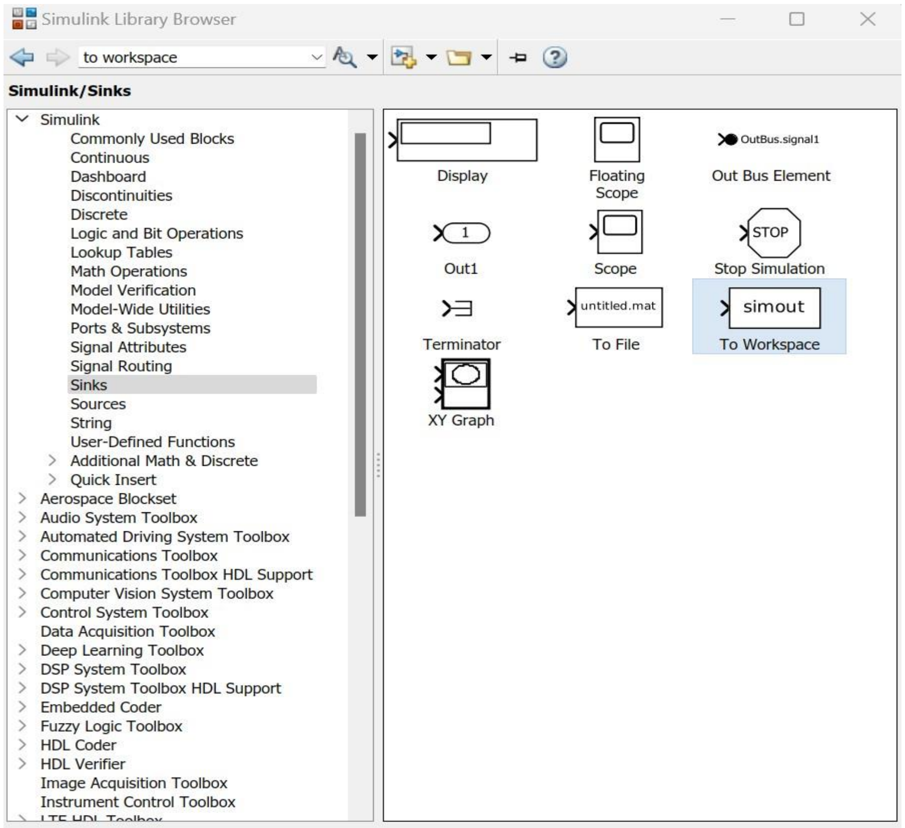
Now, the final design will be similar to the following



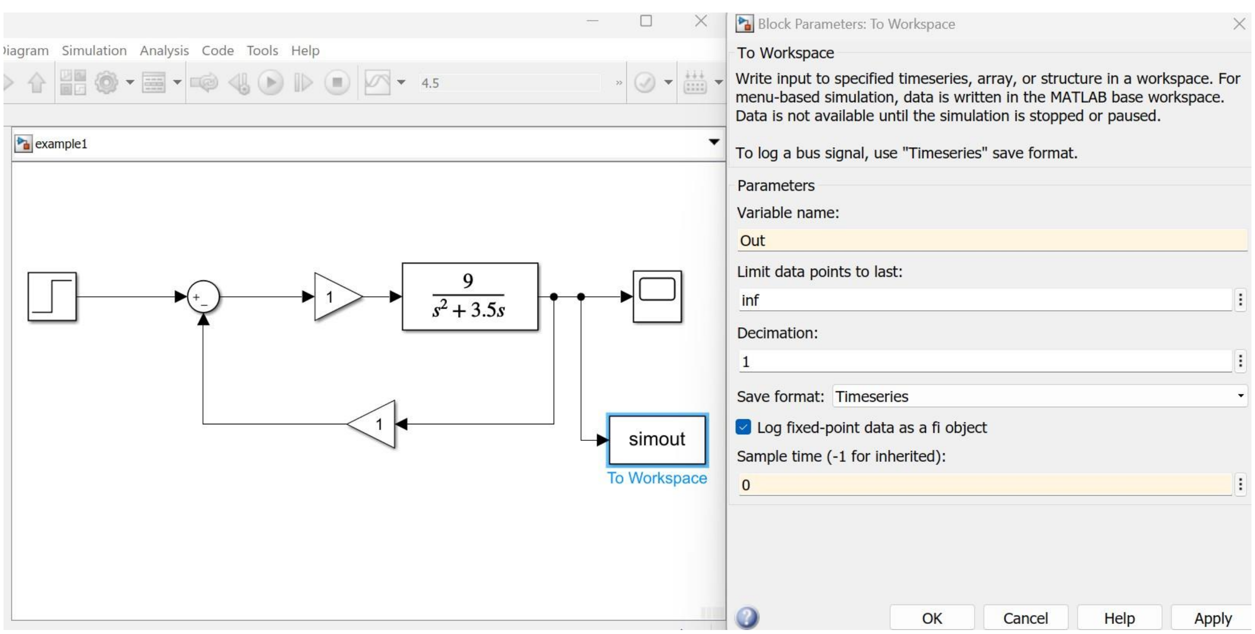
Set the Simulink time to (4.5) and press Run, and from Scope, display the obtained response



To get the step response characteristics, go to the Simulink Library Browser, select Sinks, and add To Workspace block to the example1 sheet model



Next, modify the To Workspace block parameters as shown below



Rerun the Simulink model and go to the command window and type the following code:

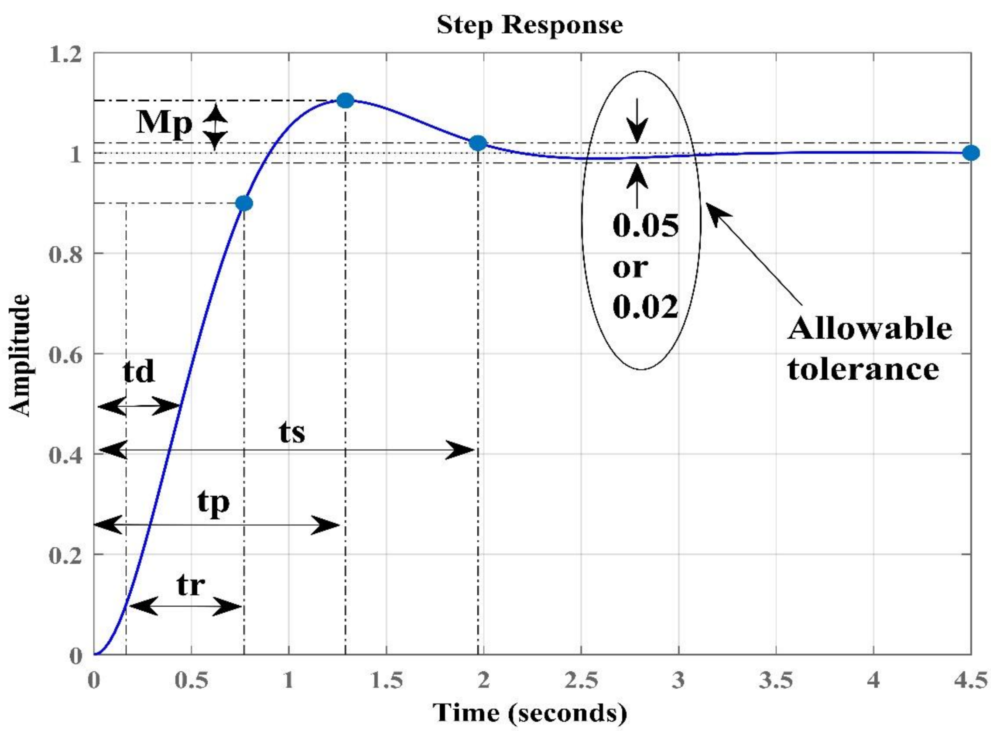
```
>> stepinfo(Out.Data, Out.Time)
```

You will get the following step response characteristics:

```
>>
ans =
  struct with fields:

        RiseTime: 0.6051
    SettlingTime: 1.9673
     SettlingMin: 0.9092
     SettlingMax: 1.1047
       Overshoot: 10.4429
      Undershoot: 0
            Peak: 1.1047
        PeakTime: 1.2900
```

Another method is used to define the transient response of the system using the following graph:



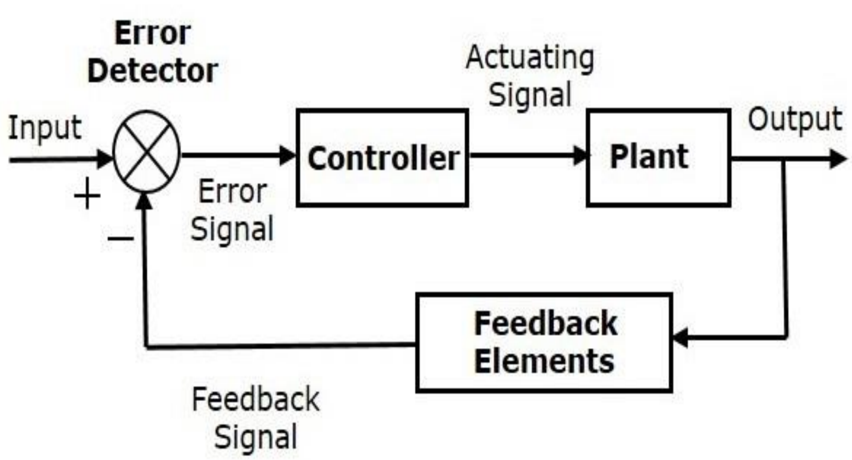
where **td** is defined as the Delay time, which is the time required for the response to reach half the final value at the first time, **tr** is defined as the Rise time, which is the time required for the response to rise from 10% to 90% of its final value, **tp** is defined as the Peak time which is the time required for the response to reach its first peak value, **ts** is defined as Settling time which is the time required for the response to reach its steady state value with an absolute percentage of the final value (2% or 5%), **Mp** is defined as the Maximum percentage overshoot which is the maximum peak value of the response measured from unity.

From the Scope, we can obtain all of these parameters, which are exactly the same values obtained in Experiment (1) and the values obtained from the previous method that employs the step info ().

From what you have learned in this experiment, obtain the output step response and the transient response characteristics for each case of the following exercises using the Matlab Simulink model.

Exercise 1:

Consider the following closed-loop block diagram

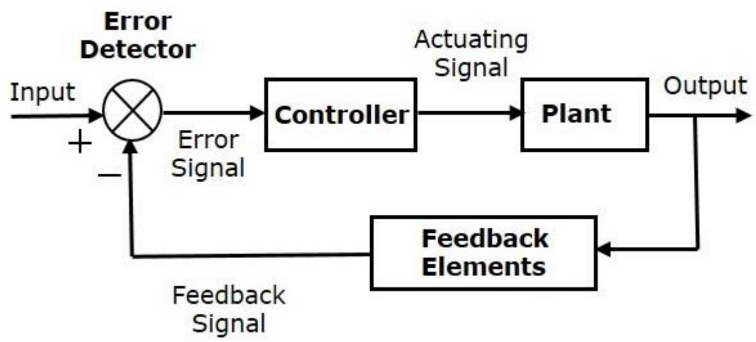


1- obtain the step response using the (step, feedback, tf) functions. where

$Input = unit\ step$		$Plant = \frac{\omega_n^2}{s^2 + 2\xi\omega_n s}$		
Case	**Controller**	ω_n	ξ	**Feedback**
Case-1	**1**	**4**	**0**	**0**
Case-2	**1**	**4**	**0.6**	**0**
Case-3	**s+4**	**4**	**0**	**1**
Case-4	**1**	**4**	**0**	**0.25*s**

Exercise 2:

Consider the following closed-loop block diagram

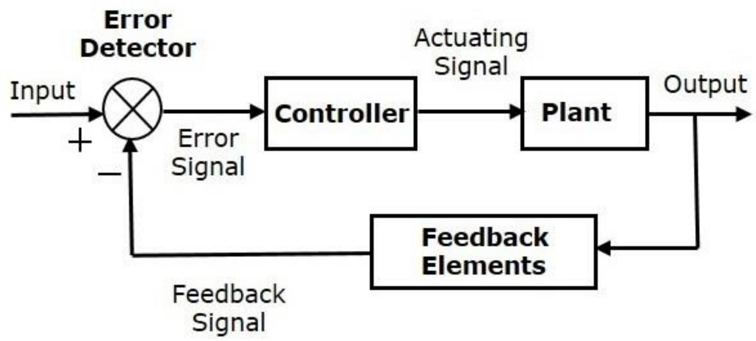


1- obtain the step response using the (step, feedback, tf) functions. where

$Input = unit\ step$		$Plant = \frac{\omega_n^2}{s^2 + 2\xi\omega_n s}$		
Case2	**Controller**	ω_n	ξ	**Feedback**
Case-1	**1**	**2**	**0**	**0**
Case-2	**1**	**2**	**0.3**	**0**
Case-3	**10(s+6)**	**2**	**0.3**	**1**
Case-4	**1**	**2**	**0.3**	**0.6*s**

Exercise 3:

Consider the following closed-loop block diagram



1- obtain the step response using the (step, feedback, tf) functions. where

$Input = unit\ step$			$Plant = \frac{\omega_n^2}{s^2 + 2\xi\omega_n s}$	
Case	**Controller**	ω_n	ξ	**Feedback**
Case-1	**1**	**1**	**0**	**0**
Case-2	**1**	**1**	**0.6**	**0**
Case-3	**10(s+4)**	**1**	**0.6**	**1**
Case-4	**1**	**1**	**0.6**	**0.8*s**