

**ECE 2111 Signals and Systems
Spring 2005, UMD**

Experiment 4: Analyzing Discrete Time Systems using Simulink

In this experiment you will use Simulink to generate the structure of discrete time systems and analyze them.

To start Simulink do the following:

1.- Run MATLAB and type simulink on the MATLAB command window

```
>> simulink
```

2.- In the simulink environment go to File and Open a New model. A blank window will appear. In this window you will bring blocks, from the different simulink libraries, and will connect them to have the desire flow of information.

3.- Go trough the simulink libraries and identify and bring to the blank window the following blocks. For each block double click on it to see the properties and parameters.

Sources:

- Sine Wave
- Signal Generator
- Constant

Sinks:

- Scope

Discrete:

- Unite Delay [1/z]

Linear:

- Gain
- Sum

Nonlinear:

- Abs
- Product

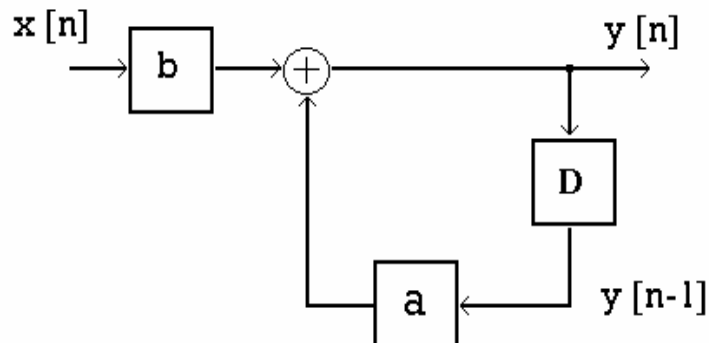
4.- First order system

Create in Simulink the following first order discrete time system, and simulate it. To do the simulation, first click on Simulation and choose: Parameters to set up the simulation time. For these examples choose Start time: 0.0 and Stop time: 100(200 or 300). To start the simulation click on Simulation, and choose: Start. To see the plots on the Scopes, double click on each of the Scope blocks. You will connect two scopes: one to the input signal $x[n]$, an other the output signal $y[n]$.

A first-order digital system can be expressed using the difference equation:

$$y[n] - a y[n-1] = b x[n]$$

A representation of this equation by a block diagram is:



4.1.- Assign to the constant gains **a** and **b** the values: **a = 0.9, b= 0.1**

For the simulation, choose the following inputs and simulation parameters

Input signal:

- a) Sine waveform, amplitude 1, frequency: 0.1(rad/sec), phase 0, sample time 0.5
In the simulation parameters, choose stop time = 300
- b) Sine waveform, amplitude 1, frequency: 0.2 (rad/sec)
In the simulation parameters, choose stop time = 100
- c) Sine waveform, amplitude 1, frequency: 0.4 (rad/sec)
In the simulation parameters, choose stop time = 100

For each input, compare the plots of the input signal $x[n]$, and the output signal $y[n]$

Is the system stable ? Explain.

4.2.- Assign to the constant gains **a** and **b** the values: **a = -1.5, b= 0.1**

Repeat steps a) to c) from section 4.1

For each input, compare the plots of the input signal $x[n]$, and the output signal $y[n]$

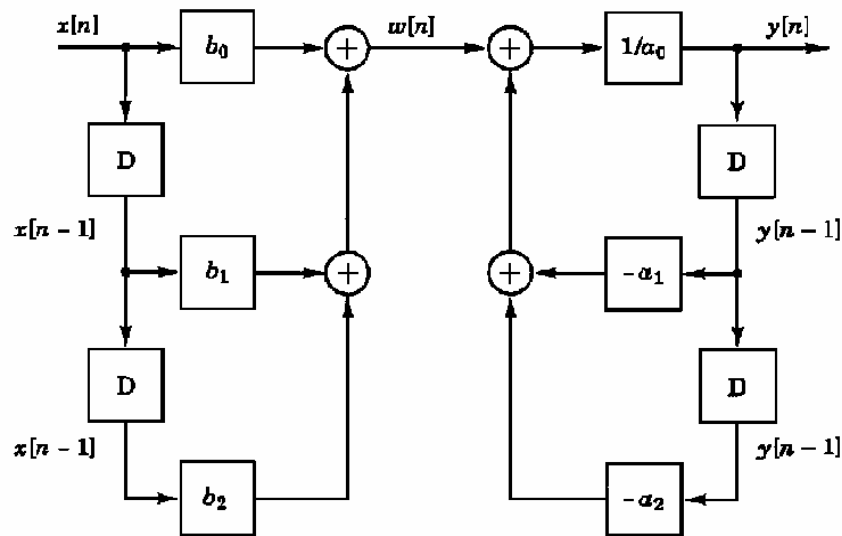
Is the system stable ? Explain.

5.- Second-order digital system.

A second-order digital system can be expressed using the difference equation:

$$a_0 y[n] + a_1 y [n-1] + a_2 y [n-2] = b_0 x[n] + b_1 x[n-1] + b_2 x [n-2]$$

A representation of this equation by a block diagram is:



5.1.- Assign to the constant gains **a_i** and **b_i** the values: **a₀ = 1, a₁ = -0.9, a₂ = 0, b₀= 5, b₁ = 0, and b₂ = -4.6**

For the simulation, choose the inputs and simulation parameters given in part 4.1

For each input, compare the plots of the input signal $x[n]$, and the output signal $y[n]$

Is the system stable ? Explain.

NOTE: Saving your files

You can save your **Simulink models** in postscript format using the MATLAB function: `print`. To do this, go to the MATLAB command window and type

```
>> print -s part5
```

This command will generate a Postscript file called `part5.ps` in the current directory. If you want to save it in a different directory give the complete path. (Note: to know which is the current directory type: `>> pwd` on the MATLAB command window).