

**ES205 Analysis and Design of Engineering Systems:
An Introductory Tutorial: Getting Started with SIMULINK**

Lab 1:

What is SIMULINK? SIMULINK is a software package for modeling, simulating, and analyzing dynamic systems. It supports linear and non-linear system modeling in continuous time, sampled time, or a combination of the two. It provides a graphical interface for building models as block diagrams using the mouse by click-and-drag operations. It also provides a variety of simulated output devices that can be attached to the system to monitor the output or state variables of the system.

By defining the system graphically, the mathematical calculations required to simulate the differential equations that represent the models are hidden from your view. You only need to be able to supply the model structure and parameters, the simulation parameters, and the output variables and forms. You don't have to get bogged down in the math to determine the behavior of the system of interest.

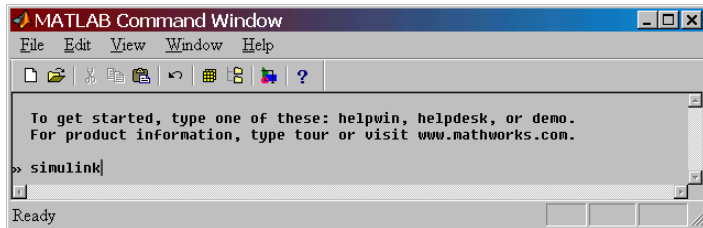
This tutorial will consist of 4 parts:

- A) Launching and becoming familiar with the SIMULINK environment
- B) Creating a simple 1st order model
- C) Creating a 2nd order model of a mass, spring, damper system
- D) Creating a model on your own of an electrical system.

Part A: Launching Simulink:

1) SIMULINK is run from the MATLAB prompt, `>>`, so the first step is to get MATLAB running.
-- Connect to the Network
-- Launch MATLAB. (if you have not already installed MATLAB you will need to do so at this time.)

2) At the prompt, `>>`, in the MATLAB Command Window, type `simulink` and press `Enter`.

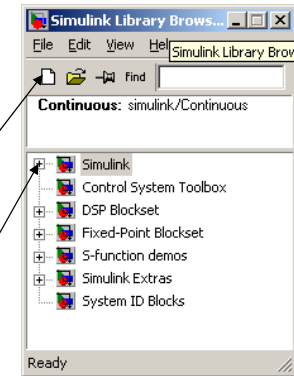


3) When SIMULINK opens you will see the box called the SIMULINK Library Browser. The Library consist of a number of different SIMULINK blocks with which a system model may be built.

To build a model, you first need to create a space to make the model.

Click the new model icon in the upper left corner to open a new SIMULINK file workspace.

Next, select the SIMULINK icon to expand the list of the available elements that are used to create a system model.

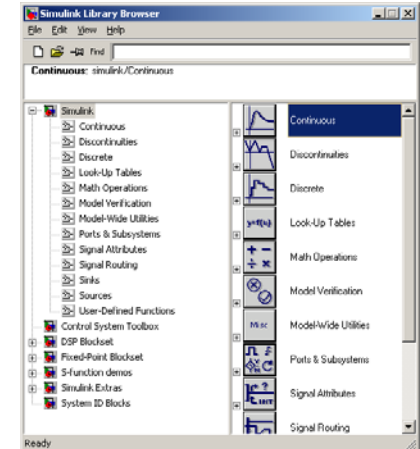


4) After making the selections in Step 3, the SIMULINK Library Browser will show folders that contain the most commonly used elements you will be using for model creation.

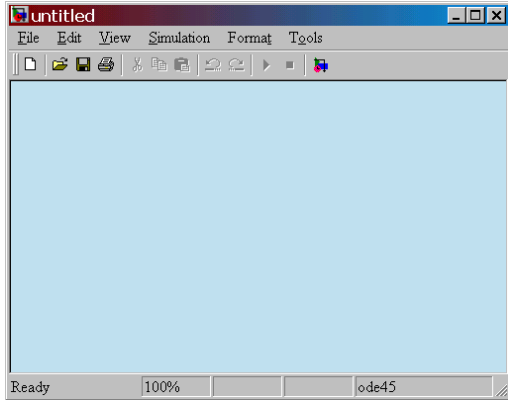
Click on each of the following folders and quickly note the elements that are contained in each.

- Continuous
- Discontinuous
- Discrete
- Look-up Tables
- Math Operations
- Sinks
- Sources

When needed, these elements may simply be clicked, dragged, and dropped into the model workspace.



During Step 3 you also opened a window called Untitled, which is the model workspace where the models will be created.

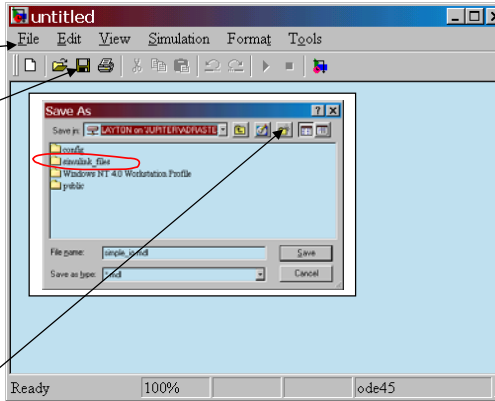


5) You have not yet created a model, but if you had, you would want to be able to save the model.

To save a model, pull down the **File** menu and select **Save_As** or Click on the File Save Button

While we won't save a file right now, we will have you create a new folder to save any models you will create.

Create a new folder called, **simulink_files** by clicking on the new folder button and supply the correct name.



Part B) Example 1: A Simple Model

1) The first model you are going to build will solve a simple 1st order differential equation given by

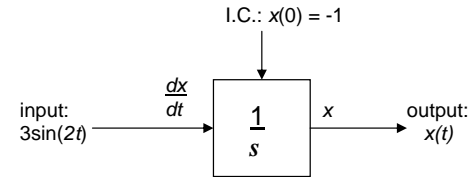
$$\frac{dx}{dt} = 3 \sin(2t)$$

with an initial condition $x(0) = -1$.

2) To create a model of this equation, we first sketch a **simulation diagram** of the mathematical model.

The simulation diagram will consist of:

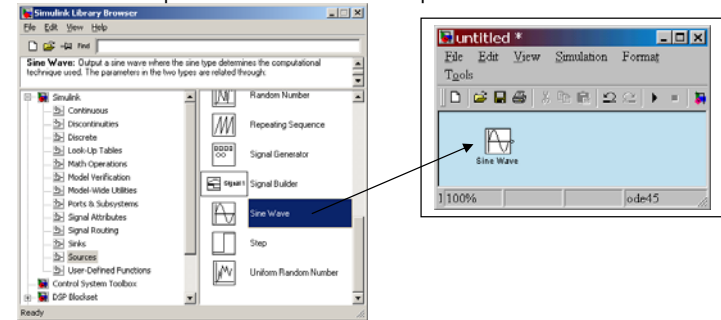
- an input --> the forcing function, $3 \sin(2t)$
- an output --> the solution, dx/dt
- a parameter --> the initial condition, $x(0) = -1$
- an operator --> integration block



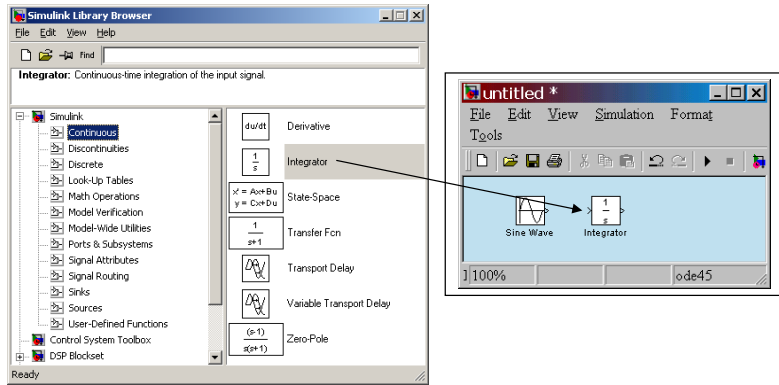
3) To construct this model in Simulink, will require three elements:

- a Sine Wave Block
- an Integrator Block
- a Scope Block

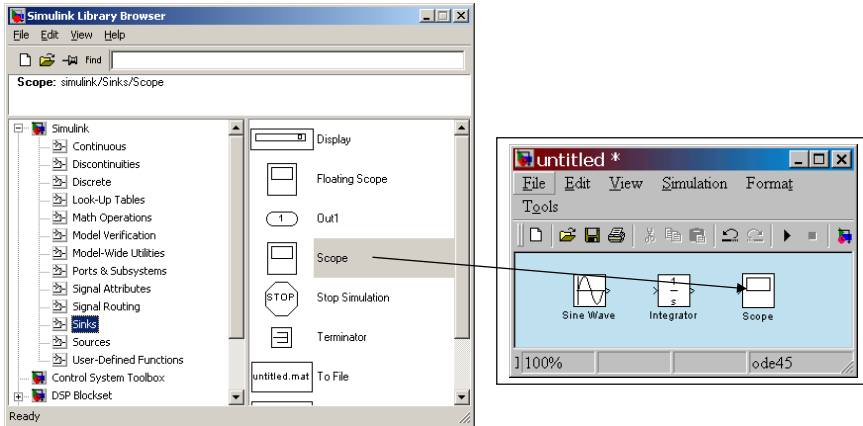
Locate the Sine Wave under the Sources folder and Click, Drag, and Drop it in the model workspace window to create the Input Block



4) Locate the Integrator under the Continuous folder and Click, Drag, and Drop it in the model workspace window to create the Operator Block.



5) Locate the Scope under the Sinks folder and Click, Drag, and Drop it in the model workspace window to create the Output Block

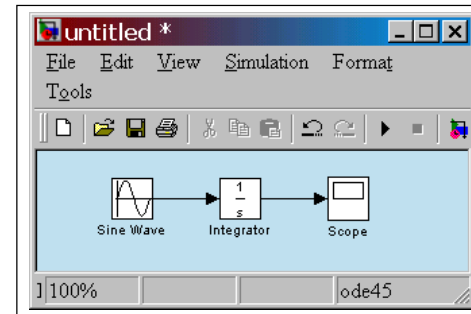


6) Connecting the blocks.

Blocks can be connected by dragging a line from the **output** of one block to the **input** of another block.

Make the following connections:

- Output of the Sine Wave --> Input of the Integrator
- Output of the Integrator --> Input of the Scope



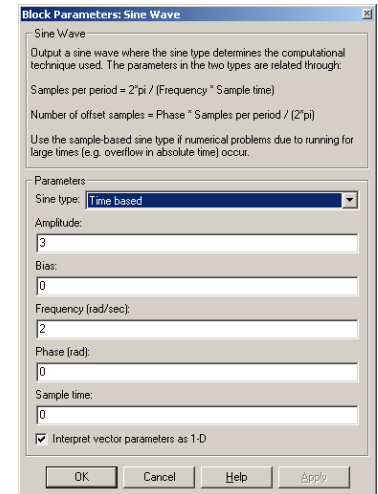
The arrows indicate the direction of the signal flow.

7) Selecting Simulation Parameters.

Most blocks have different parameters that are associated with them. To access the parameters, simply double-click on the block of interest. This will bring up a dialog box which allows the parameters to be changed.

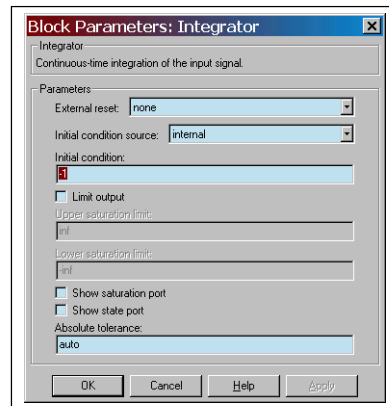
Double click on the Sine Wave and set the following parameters:

- Amplitude = 3
- Frequency = 2



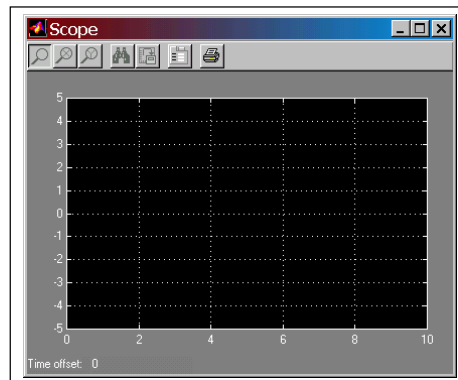
Double click on the Integrator Block and set the parameter for the initial condition.

$$x(0) = -1$$



Double Click on the Scope Block.

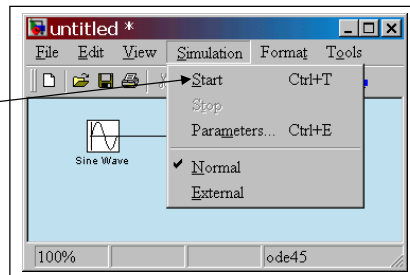
This does not have any parameters which need setting, but instead displays the scope display screen.



8) Now that the model has been defined and created, the model simulation is ready to run.

To Start a simulation, pull down the Simulation menu, and select Start.

View the output of the simulation on the Scope window.



9) Simulation Results

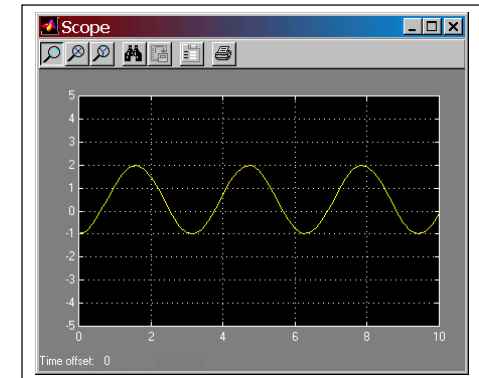
Your results should look like the figure.

Measure the magnitude of peaks and valleys and the period of the signal.

Peak = _____

Valley = _____

Period = _____



10) Simulation Verification:

To verify that this plot represents the solution to the problem, let's look at the analytical solution.

$$\frac{dx}{dt} = 3 \sin(2t) \quad \text{with } x(0) = -1$$

This equation can easily be solved by integration. The result is

$$x(t) = \frac{1}{2} - \frac{3}{2} \cos(2t)$$

This is the function that was plotted in the SIMULINK Scope window.

Save this SIMULINK model in the `simulink_files` folder as as `Example1.mdl`

Part C: Example 2. a mass, spring, damper system

1) The second model will use SIMULINK to create a model of a mass-spring-damper system which may be modeled with a 2nd order differential equation. Other parameters of the system include:

- initial conditions: $x(0) = 0$ and $dx/dt(0) = 0$
- the input $f(t)$ is a step function with magnitude 3 at $t=0$
- mass, $m = 0.25$
- damping coefficient, $c = 0.5$
- spring constant, $k = 1$

The model equation:

$$m \frac{d^2 x}{dt^2} + c \frac{dx}{dt} + kx = f(t)$$

Assume all units are consistent for the model.

You will work through the following steps:

- Draw the simulation diagram
- Create and save the SIMULINK Model
- Run the Simulation.

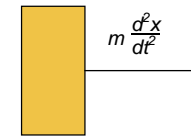
2) In the space below attempt to draw the **simulation diagram** for this model:

3) Let's walk through the steps.

For comparison, start by solving the model equation for the highest order derivative term.

$$m \frac{d^2 x}{dt^2} = f(t) - c \frac{dx}{dt} - kx$$

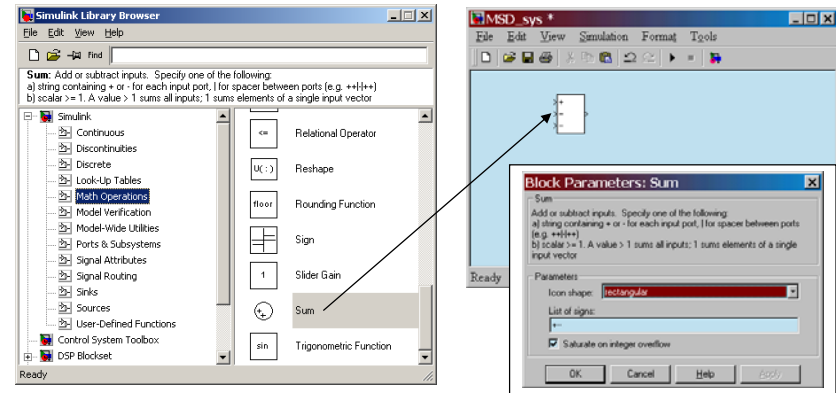
The first block you create should be a Sum block, where the output of the Sum block is the left-hand term of the equation above.



sum block

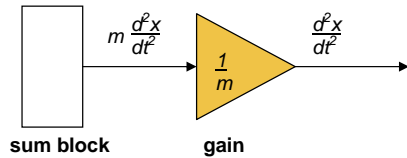
Now using SIMULINK, Create a new workspace..... **File New.....Model**

Locate the Sum Block in the _____ library and Click, Drag and Drop it into your new workspace.

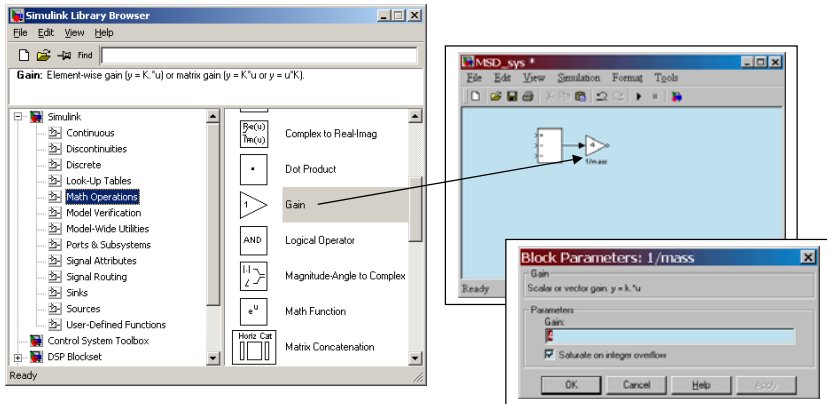


The Sum block is to have 3 inputs. Double click on the Block and set the parameters to **rectangular** and **+ - -**.
 4) Gain Block.

Add a gain (multiplier) block to normalize the coefficient, m, to modify the signal so it is equal to the highest order derivative term alone.



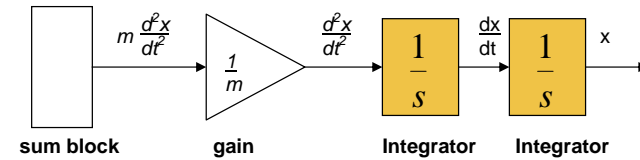
Click, Drag, and Drop a Gain Block from the _____ library into the SIMULINK workspace.



Double-click on the Gain block to change the block parameters.

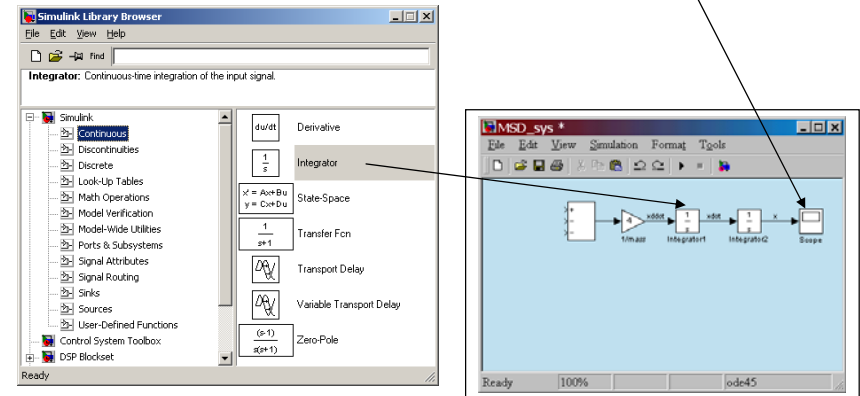
Use a Gain of _____

Add a title to the Gain Block.
 5) Add two integrators to your Simulation diagram.



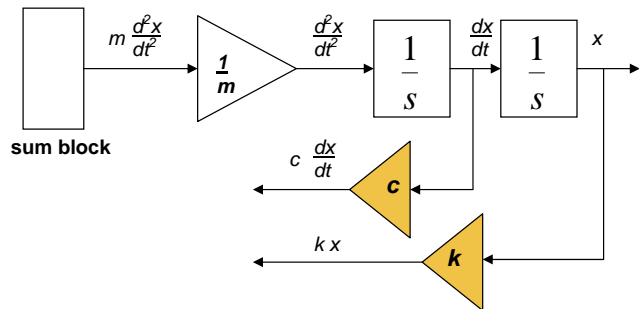
Click, Drop, and Drag two Integrator Blocks into your SIMULINK workspace from the _____ library.

Also:
 Set the initial condition parameters to zero for both integrators.
 Add a scope block to the output of the second integrator
 Connect the output ports to the input port as shown
 Label each of the signals by double clicking on the leader line.



6) Return Branch Gain Blocks

In the Simulation Diagram, connect the integrated signals with gain blocks to create the terms on the right-hand side of the model equation.



In SIMULINK, Drag in two additional Gain blocks from the Math library to the workspace.

--Flip these blocks by selecting each block and using..... **Format.....Flip Block**

--Double click on each of the blocks and set the appropriate parameters

c = 0.5

k = 1.0

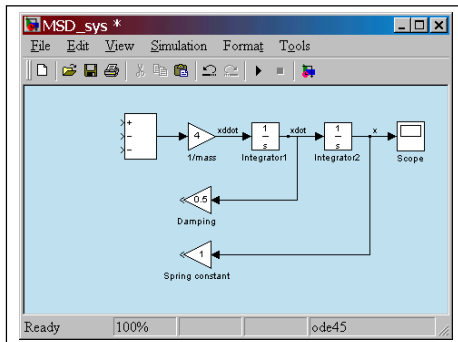
--Connect the signal lines:

Either Click the gain block input and drag to each of the branch points.

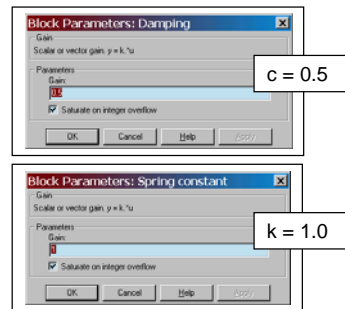
or

Ctrl-Click to select the branch point first and drag to the gain inputs.

--Add appropriate titles to the gain blocks



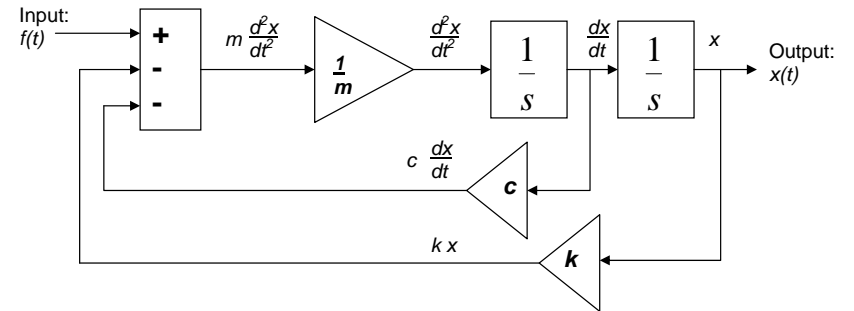
ES205 Getting started with Simulink



Page 13 of 16

7) Connect all Input Signals.

In the Simulation Diagram, connect all the input signals to the appropriate inputs of the Sum Block.



In the SIMULINK workspace:

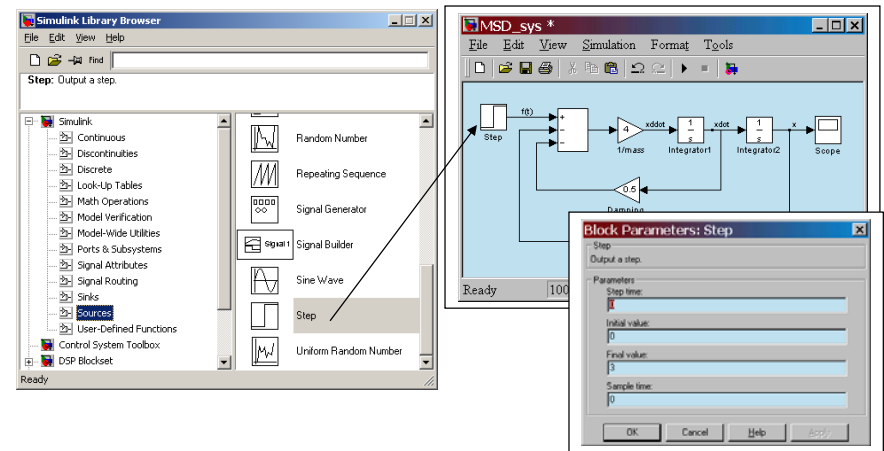
-- Add a Step Block from the Source library and set its parameters.

Step time = 0

Initial value = 0

Final value = 3

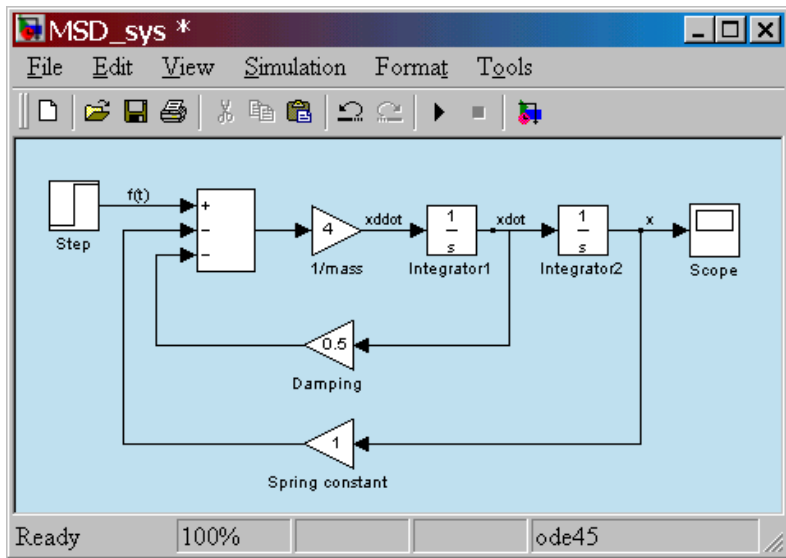
-- Connect the signal lines to the sum Block, paying attention to the signs of the inputs.



ES205 Getting started with Simulink

Page 14 of 16

8) Your completed SIMULINK model should look like the following. Complete any missing titles or labels.



Save your model as Example2.mdl

9) Running your simulation

Double click on the Scope to make its output visible.

Run your simulation.

What is its maximum value of x reached? _____

What is its final value of x reached? _____

Is this behavior underdamped, overdamped, or critically damped?

This concludes the SIMULINK Tutorial Module.

You are now to complete Part D: Lab 1 Worksheet. To complete the worksheet you are expected to understand and use the terms below.

Terms used on the worksheet:

Steady State Value is the final value of the system settles at after transient behavior has dissipated.

Overshoot is characterized as the maximum response swing past the steady state value.

Rise time is time required for the system to rise from ten to ninety percent of the steady state value.

Settling time is the amount of time the system takes to value settle close to the steady state condition (to within approximately 2% of the step size).

