Lab 1: INTRODUCTION TO SIMULINK

Section 1 -- Background Information

This lab will introduce the use of Simulink, an extension to Matlab, for use in simulating control systems. In this lab you will build a model of a second-order system and observe the response to a step input.

1.1 What is Simulink?
- Simulink is an extension to Matlab.
- In Simulink, you build block diagram models of dynamic systems instead of text code.
- It is easy to model complex nonlinear systems.
- Simulink can model both continuous and discrete-time components.

Section 2 -- Procedure

There are three sections to this procedure. The first, Section 2.1 will show you how to start the Simulink utility and introduce you to the Simulink workspace. In section 2.2, you will determine the step response of a second-order control system, and in Section 2.3, you will modify your model to produce examples of overdamped and undamped system responses.

2.1 Simulink Introduction
2.1.1 Start Matlab. The computers in the controls lab will have the Matlab utility available from a Desktop folder named "SERVO LAB APPS." Double click the Matlab icon on the desktop. It will take a minute or two for Matlab to finish initializing. The Matlab window should appear as shown in Figure 1. Note: Setup may differ depending on MATLAB version you are using.

2.1.2 Start Simulink by clicking on the Simulink Library. A Simulink window will open as shown in Figure 2. This window shows the libraries of Simulink Blocks that are available for you to use in constructing Simulink models.
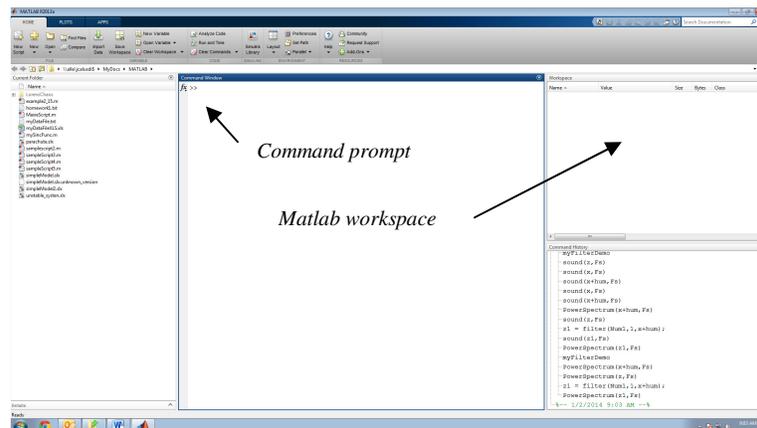
Figure 1. Matlab Main Window
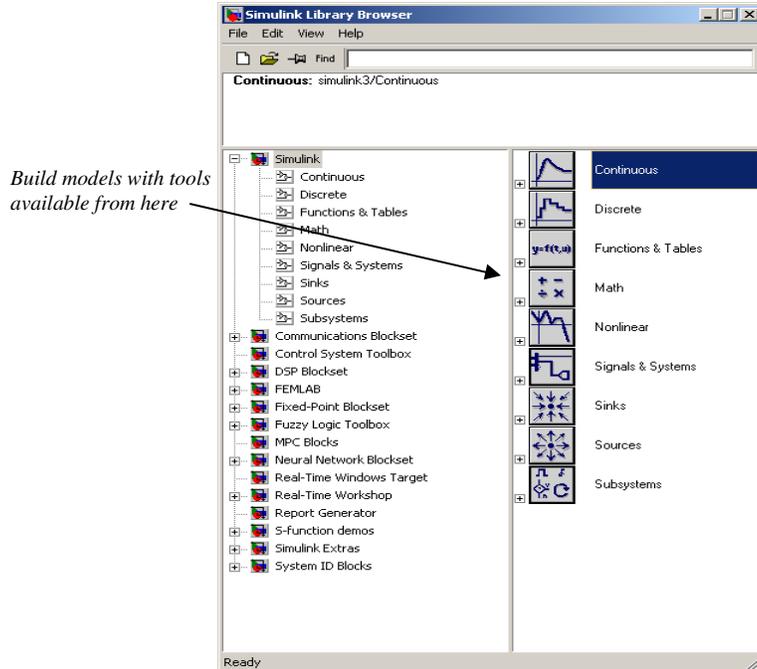
Figure 2. Simulink Library Browser Window

2.1.3 Open an existing model or create a new one. To create a new model, select File→New→Model, and new model window will appear as shown in Figure 3. An existing model will look like Figure 4.

2.1.4 Help or Instructions. Pull down on the help menu in either the Matlab or the Simulink library window. Matlab and Simulink have extensive on-line help menus which include tutorials and demonstrations to help you. Also type **>>demo simulink** at the command prompt to see other help options.
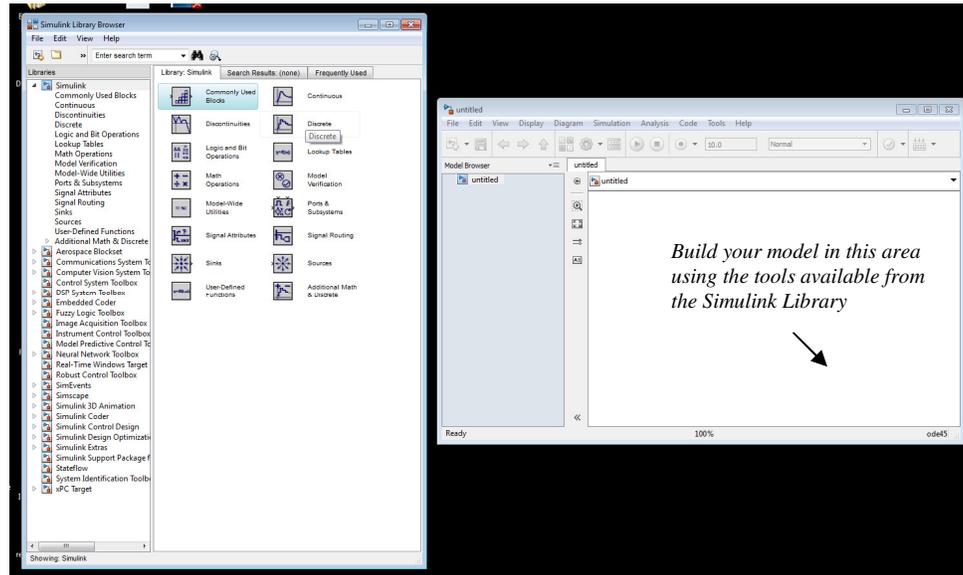
Figure 3. Simulink Model Window with Untitled Workspace for Building a New Model.
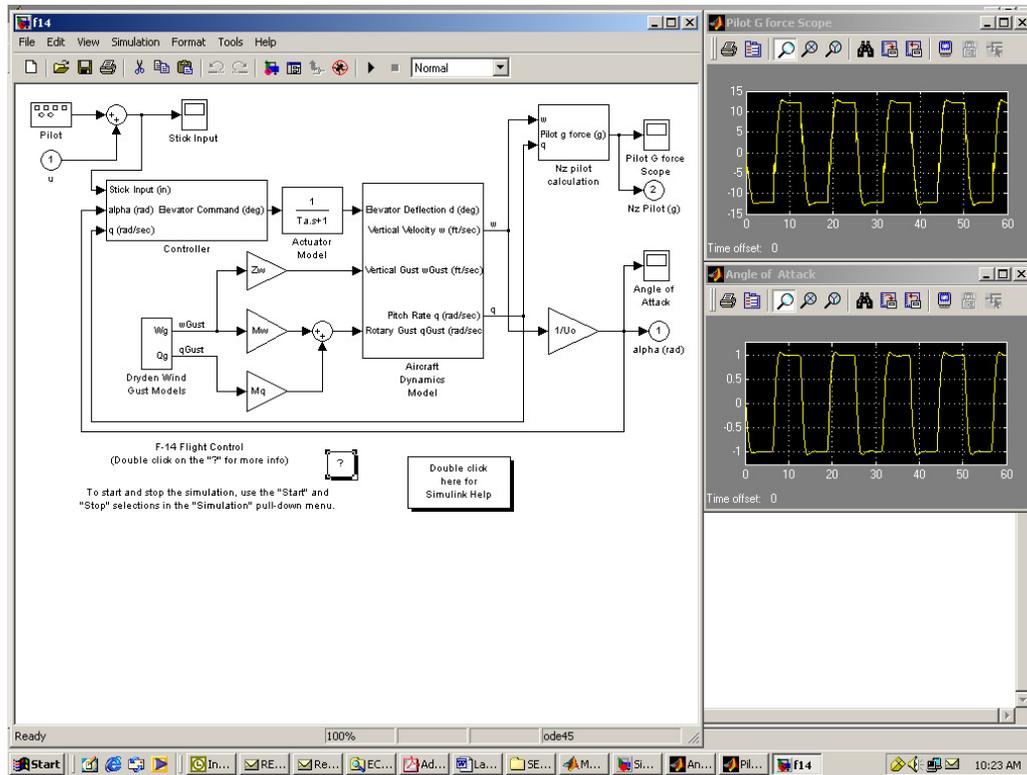
Figure 4. Existing Simulink Model, demo – f14.mdl

2.2 Build a Second Order System Model and Simulate the Step Response
2.2.1 Develop Simulink Model. In the model window place the following items on the diagram and wire them together as shown in Figure 5. From the directory of tools select the following:

<u>From Directory</u>   <u>Select Items</u>
Sources →   Step, Clock
Continuous →   Transfer Function[1]
Sinks →   Scope, To Workspace (You will need two of these. Label one of them
                   "Output" and the other "Time")

To wire items together, click on a device's terminal then drag (hold left mouse key down) until you get to the target device's terminal. To split wires, hold CTRL key down, then click on the wire you want to split, drag to the desired terminal. Move your blocks or wires around until the items appear as shown in Figure 5.

2.2.2 Step Input. Double click on Step function block and a window will appear as in Figure 6 to set the parameters for the step function that will be applied to the input of the system under test. Set the following parameters: Step time= 0; Initial value=0; Final value=1; Sample time=0.

---

[1] A ***Transfer Function*** is a mathematical representation of a system and is simply the ratio of the system input to the system output. You will learn more about transfer functions later in the course.
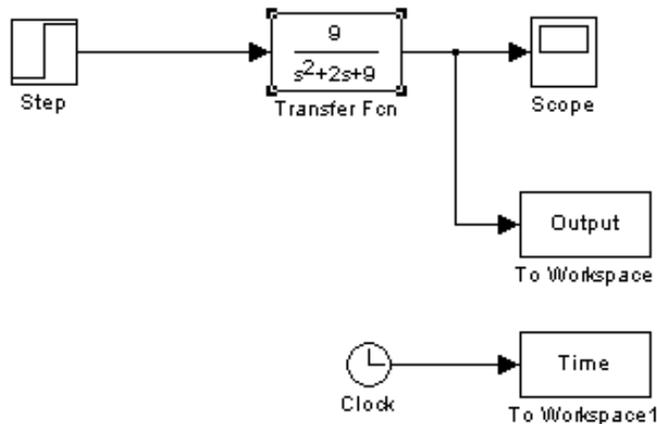
Figure 5. Simulink Diagram for Section 2.2

2.2.3 Setup Transfer Function. Double click on the transfer function block and the window will open as in Figure 7. Enter the coefficient values for the system transfer function given below. The coefficients for numerator and denominator polynomials are entered from highest order (s^2) to lowest (s^0). Delineators for transfer function coefficients can be spaces or commas. Values should be 9 for numerator and 1, 2, 9 for denominator.

$$G(s) = \frac{9}{s^2 + 2s + 9}$$
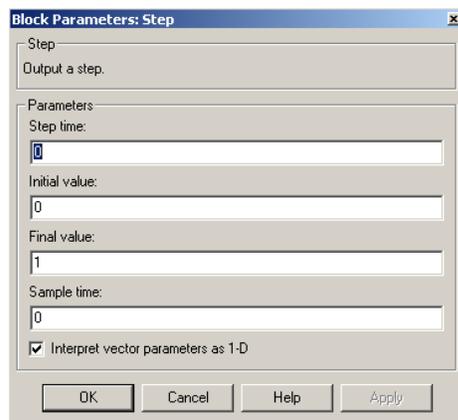
Figure 6. Step Function Setup
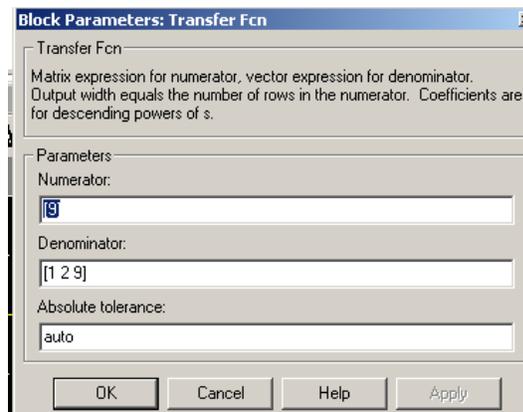
Figure 7. Transfer Function Setup

2.2.4 Output To Workspace Block Parameter Settings. The "To Workspace" blocks will each create an array in the Matlab workspace with the name that you specify in the appropriate parameter block. This allows you to plot using Matlab's extensive plotting capabilities (Simulink's plotting capabilities are somewhat limited) and to use Matlab routines to analyze and manipulate the data resulting from the simulation. Double click on one of the "To Workspace" blocks to open the parameter window as in Figure 8, and give it an appropriate name, such as Output or Time. The Save format needs to be set to Array. Use

the default parameters for the remaining items in the windows. Do this for each of the two blocks in your model.

2.2.5 Setup Simulation Parameters. Select from top menu, Simulation → Simulation parameters to get the window shown in Figure 9. Under the solver tab, set Start Time=0.0, Stop Time= 10.0. Solver Options: This sets the numerical integration or differential equation solution parameters. Set type= Variable Step; ode45 (Dormand-Prince); use default settings for all other parameters. *Note: You can learn more about Simulink's solvers from the help pages.*
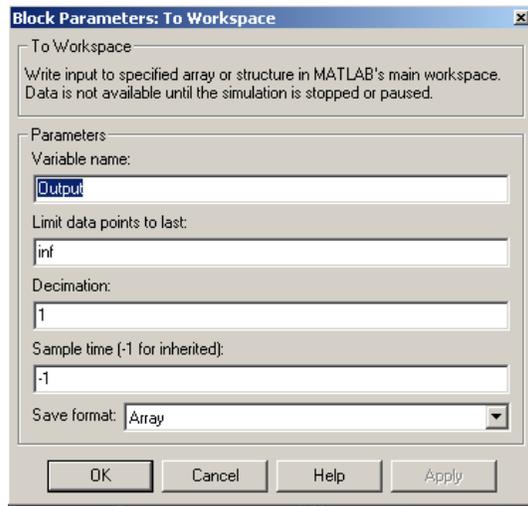
**Figure 8. Output to Workspace Block Setup**   **Figure 9. Simulation Solver Parameters**
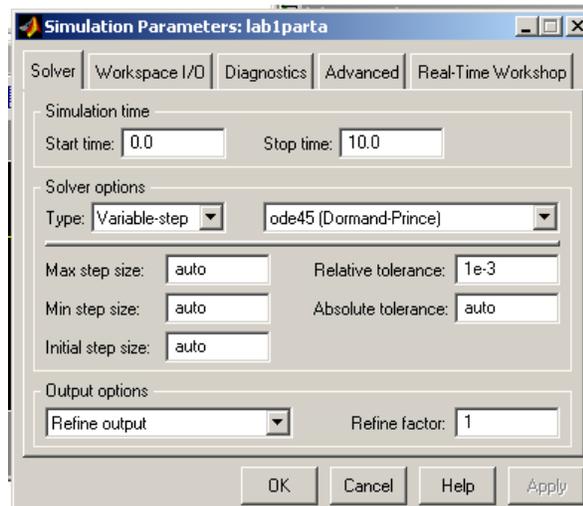
2.2.6 Run Simulation. First open the scope window by double-clicking on it. Then go to Simulation → Start. When the simulation has completed you will see the output on the scope. You should see something similar to Figure 10.
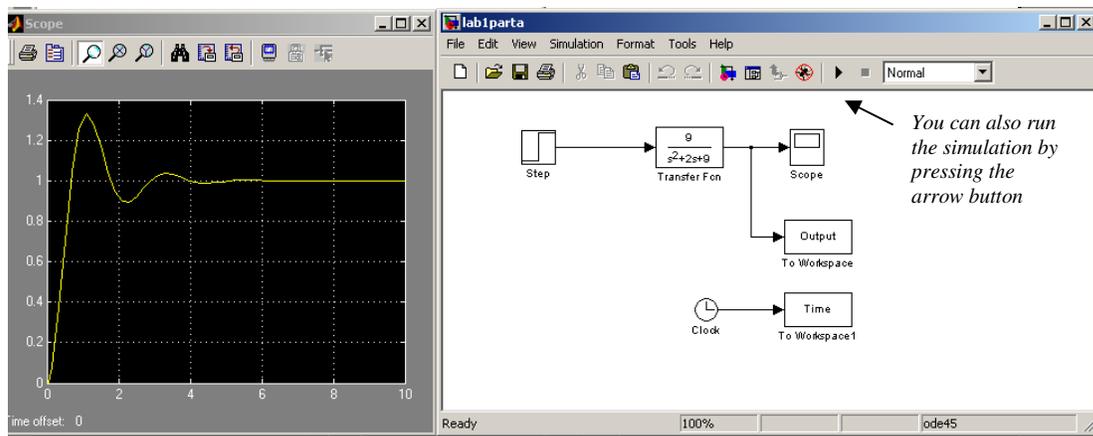

Figure 10. Simulation Results for Section 2.2

2.2.7 Make a Matlab Plot. Go to Matlab window and type **whos** at the command prompt. You will see the data arrays that Simulink generated during the simulation with the names you assigned during the setup. If you want to see this data, just type the name of the array at the command prompt. Make a plot of your data. It should appear as in Figure 11. Use the command: **>>plot(Time,Output)**. Add axes labels and a title to the plot.

2.2.8 Make Printouts of the System Diagram and Matlab Plot. Print the system diagram and the Matlab plot. Include these in your lab report. *hint: Copy figures to MS Word using Shift+PrintScrn on your keyboard to copy the screen. Then paste into a MS Word document and crop as necessary. You can even keep MS Word open while you are doing work in Matlab and Simulink..*
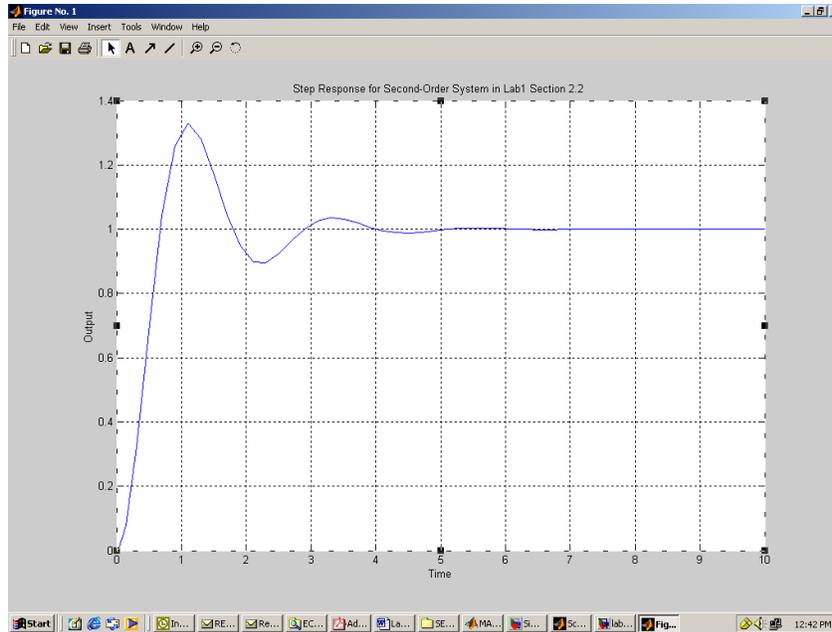

Figure 11. Matlab Plot Using Simulink-Generated Data

2.3 Explore Other Transfer Functions

In your Simulink model, edit the Transfer Function block so that it has one of the transfer functions shown in the table. Run the simulation for each transfer function and observe the system response. During this course you will learn more about these types of system responses. Include Matlab plots of these responses in your report.

Transfer Function, G(s)	
$\dfrac{9}{s^2+9s+9}$	Produces an example of an overdamped system response
$\dfrac{9}{s^2+9}$	Produces an example of an undamped system response

Section 3 -- Report

Submit a report after completing this lab. The report will have three main parts. In the first part, the Introduction, discuss the objective of the lab and briefly describe any special equipment, systems, software that was used to execute the lab procedure.

In the second part of the lab report, Results and Analysis, discuss the main parts of the procedure, results and observations. Include any plots that were generated from the data collected during the lab, and be sure to reference them in the text of this part of the report.

Finally, discuss the main points of the lab in the Conclusion of the report. Include any significant findings and recommendations you may have.