ECE486 Lab 2: Digital Simulation

Scope and Objective

1. Study the dynamic response of 2nd order systems through digital simulation
   - Implement a transfer function with an all-integer block diagram in Simulink.
   - Implement a transfer function with State-Space function block in Simulink
2. Gaining insight to control system design through the characteristic responses of the systems
   - Investigate the effect of an extra zero
   - Investigate the effect of an extra pole

Introduction

Digital simulation packages are an essential part of control system design. Such packages are used routinely in control system analysis and design. This lab introduces Simulink, the digital simulation package sold with Matlab. Simulink will be used to study the dynamic response of a second order system,

$$F(s) = \frac{Y(s)}{U(s)} = \frac{b_1 s + b_o}{s^2 + a_1 s + a_o} \quad (1)$$

in both block-diagram:

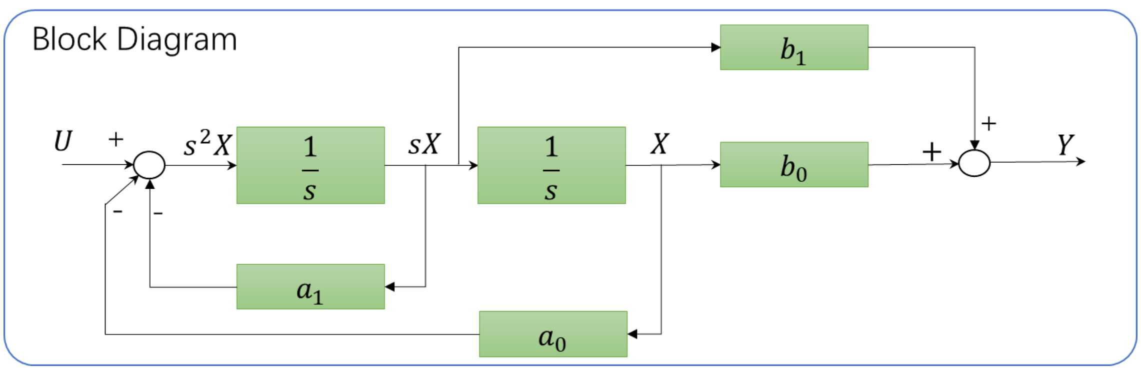


Figure 1: Block Diagram Representation of System with Transfer Function *F(s)*

and state-space forms:

$$\begin{pmatrix} \dot{x}_1 \\ \dot{x}_2 \end{pmatrix} = \begin{pmatrix} 0 & 1 \\ -a_0 & -a_1 \end{pmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} + \begin{pmatrix} 0 \\ 1 \end{pmatrix} u \quad (2); \qquad y = \begin{pmatrix} b_0 & b_1 \end{pmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} \quad (3)$$

together with the effects of an additional pole and an additional zero. The characteristic responses of these systems provide important insights into control system design.

Reading

- G. F. Franklin, *et al.*, Feedback Control of Dynamic Systems, 4th, 5th, 6th, 7th or 8th Ed., Sec. 3.5.

Modified from ECE486 Control Systems Lab Manual,
Department of Electrical and Computer Engineering, University of Illinois at Urbana-Champaign

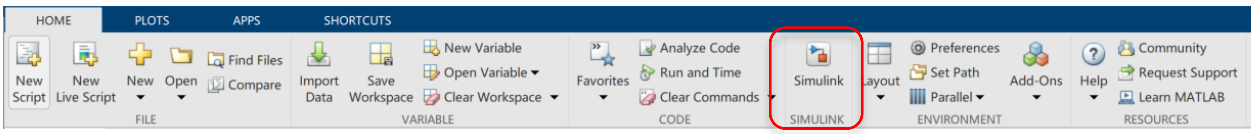
Prelab Exercise

$$H(s) = \frac{Y(s)}{U(s)} = \frac{25}{s^2 + 6s + 25}$$

1. Develop an all-integrator block diagram for the transfer function $H_1(s)$. What are the damping ratio and natural frequency? What the poles of the system?

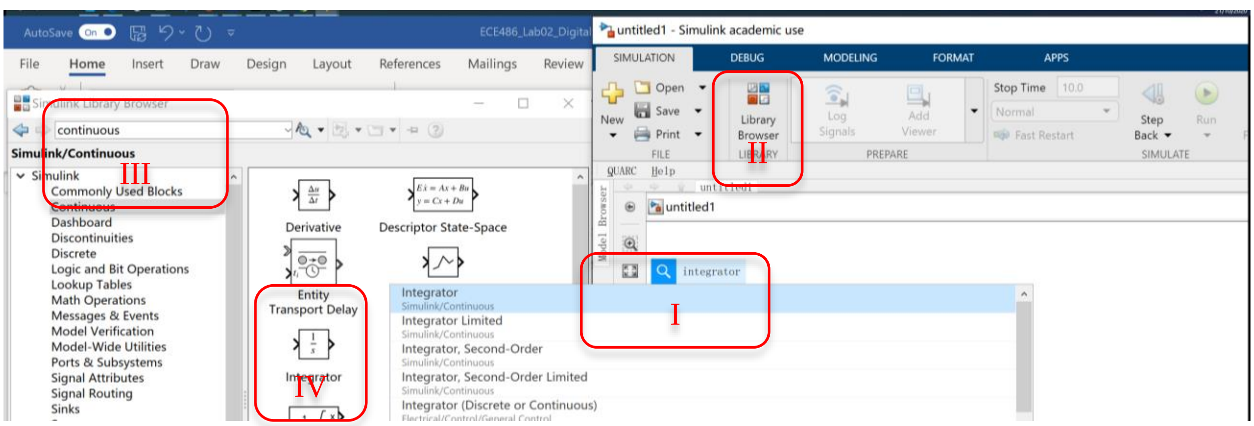
2. Simulate the all integrator block diagram using SIMULINK

a) Type *simulink* in MATLAB command line or click on the Simulink icon as shown



b) Click *Blank Model*. At the blank SIMULINK library file

c) Double click on the Model Window to open a field and type "integrator" as shown in (I) to select and place the Integrator block. Alternatively, click "Library Bowser". Search "continuous" to find the Integrator block and drag it to the model browser as illustrated by (II-IV).



d) Repeat the same for other block elements including Gain and Sum (from "Math Operation").

e) Copy and paste the necessary elements to construct your all-integrator block diagram in the model browser. You may can the direction of the block by right clicking and selecting *Rotate and Flip*.

e) Place a *Step* block from *Source* library and *To Workspace* block from *Sink* library

f) Find and add a *To Workspace* block to the output of the first integrator. (You may use the *Scope* block approach as you have learned in Lab 01 to log the data to the workspace too)

g) Label the output of the first integrator *y_dot* by double-clicking on the line representing the output

h) Double click on the Gain block and set the appropriate values

i) Double click on the Step clock to set the following parameters.

- Step time: 0;
- Initial Value: 0;
- Final Value: 1;
- Sample Time: 0

j) Double click on To Workspace block connect to the output and set the following parameters:

- Variable name: y
- Save format: Array

k) Double click To Workspace block connected to y_dot and set the following parameters:

- Variable name: y_dot;
- Save format: Array

l) Set the simulation parameters by clicking on the Simulation menu and choosing Configuration Parameters, to:

- Stop Time: 3 (chosen to capture the main response)
- Type: Variable-step
- Solver: ode45
- Max Step Size: 0.001(Smaller max step sizes give smoother responses, but take longer to compute)
- Click on *Data Import/Export* and uncheck the *Limit data points to last box*.

m) Run the simulation by clicking on the Simulation menu

3) In the MATLAB workspace, you will find three variables: *tout*, *y* and *y_dot*. SIMULINK always saves the time vector as tout. Plot the variable *y* and *y_dot* using tout as the time axis.

❧ LABORATORY EXERCISE ❧

PART I: STATE SPACE MODEL OF $H_1(s)$

1. Open a new SIMULINK model.
2. Drag in a *State-Space* block from the Continuous library.
3. In the MATLAB workspace window, set the following variables:

$$\mathbf{A} = \begin{bmatrix} -6 & -25 \\ 1 & 0 \end{bmatrix}, \mathbf{B} = \begin{bmatrix} 25 \\ 0 \end{bmatrix}, \mathbf{C} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}, \mathbf{D} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

4. Double-click on the *State-Space* block, and set the **A**, **B**, **C**, **D** parameters to "A", "B", "C", "D" respectively (without the quotes). This causes SIMULINK to look up the values in the MATLAB workspace when the model is run.
5. Open the Sources library and drag a *Step* block into the model window. Connect the *Step* block to the input of the *State-Space* block, and change the parameters to those used in the prelab.
6. Open the Signals Routing library and drag a *Demux* block into the model window. Connect the *State-Space* output to the *Demux* input.
7. Open the Sinks library to find the *To Workspace* block. Connect each output of the *Demux* to a separate *To Workspace* block. Set the variable name of the top *To Workspace* block to *y dot*, and the bottom one to *y*. Make sure both *To Workspace* blocks are set to an Array type save format (as in the prelab).
8. Set the Simulation Parameters as in the prelab, and run the simulation.
9. Make a plot containing subplots of both *y* and *y dot*. Label and print this for your report. Set `y1=y`, `y dot1=y dot`, and `tout1=tout` to save these responses for later.

Part II: Effects of an extra Zero

We will now use the information from Appendix A section IV-c to introduce a zero into the all-integrator block diagram from the prelab. Sketch the block diagram by hand before implementing it in Simulink. This diagram will be similar to Figure 1.

$$H_2(s) = \frac{25\left(1 + \frac{s}{\alpha\zeta}\right)}{s^2 + 10\zeta s + 25}$$

1. Modify your all-integrator block diagram so that the zero in $H_2(s)$ is added. Remove the *y_dot* "*To Workspace*" block. In the block diagram, use variables like "a" or "zeta" for α and ζ; this way you can easily set their values in the Matlab workspace.

2. Notice that the zero is added at $s = -\alpha\zeta$, with ζ as found in the prelab. With this ζ, simulate the system with the real part of the zero in various regions. We will rename the variables after each simulation so they are not overwritten.

   ▷ ten times further left than the poles (save as $y2a$ and $tout2a$)

   ▷ on the real axis directly between the poles (save as $y2b$ and $tout2b$)

   ▷ halfway between the poles and the origin (save as $y2c$ and $tout2c$)

   ▷ in the right-half plane, with $\alpha = -3$ (save as $y2d$ and $tout2d$)

   ▷ to see the trend, also try $\alpha = -30$ (save as $y2e$ and $tout2e$)

Signal	zero location	α
$y2a$		
$y2b$		
$y2c$		
$y2d$		
$y2e$		

3. Overlay all these responses along with $y1$ on a single graph and label appropriately. Print a copy for the report. Note that in general $tout2a \neq tout2c$; since we are using variable-step integration, Matlab is free to choose which times it evaluates. Remember this; a common mistake is to use the wrong time data when plotting. | TA $\surd$: |

PART III: EFFECTS OF AN EXTRA POLE

$$H_3(s) = \frac{25}{\left(1 + \frac{s}{\alpha\zeta}\right)(s^2 + 10\zeta s + 25)}$$

1. Remove the zero from the model in Part II and add an extra pole instead, as in $H_3(s)$. Sketch the block diagram by hand before implementing it in SIMULINK.

2. The extra pole is added at $s = -\alpha\zeta$, with ζ as found in the prelab. With this ζ, simulate the system with the real part of the added pole:

   ▷ ten times further left than the poles (save as $y3a$ and $tout3a$)

   ▷ on the real axis directly between the poles (save as $y3b$ and $tout3b$)

   ▷ halfway between the poles and the origin (save as $y3c$ and $tout3c$)

   ▷ We do not want a pole in the right half plane because that is unstable. Try it and see what happens.

Signal	pole location	α
$y3a$		
$y3b$		
$y3c$		
$y3d$		

3. Overlay all the stable extra pole responses on a single graph and label appropriately. Be sure to include the response $y1$ from the first portion of the lab. Print.

   TA $\surd$:

✧ Report ✧

I: State Space Model of $\boldsymbol{H_1(s)}$

1. Compare the plots of *y_dot* and y as obtained in Part I of the lab to the plots made in the prelab. Do they appear to come from the same system? Attach the plots to your report, making sure they are clearly labeled.

II: Effects of an extra Zero

1. How are M_p, t_r, and t_s affected by the location of the zero? Answer this by measuring these values from the Part II plot and tabulating them. Include the values from Part I for comparison. Attach the plot to the end of your report.

2. A zero in the right half plane is called a non-minimum phase zero. What is interesting about the plot for this case?

3. Take $H_2(s)$, set ζ to the value found in the prelab, and separate the numerator terms to make a sum of two fractions. What does each term represent? As α changes, which term dominates? Be sure to consider the limits as α approaches 0 or ∞. What happens when α is negative?

III: Effects of an extra Pole

1. How are M_p, t_r, and t_s affected by the location of the additional pole? Answer this by measuring these values from the Part III plot and tabulating them. Include the values from Part I for comparison. Attach the plot to the end of your report.

2. Take $H_3(s)$, set ζ to the value found in the prelab, do a partial fraction expansion to separate the extra pole from the original system, and then separate the numerator as before. The expansion should look something like:

$$\begin{aligned} H_3(s) &= \frac{25}{\left(1+\frac{s}{\alpha\zeta}\right)(s^2+10\zeta s+25)} \\ &= \frac{k_1}{1+\frac{s}{\alpha\zeta}} + \frac{k_2 s + k_3}{s^2+10\zeta s+25} \\ &= \frac{k_1}{1+\frac{s}{\alpha\zeta}} + \frac{k_2 s}{s^2+10\zeta s+25} + \frac{k_3}{s^2+10\zeta s+25} \end{aligned}$$

Solve for k_1, k_2, and k_3 in terms of α. What does each of these three terms represent? How do k_1, k_2, and k_3 change as α changes? Be sure to consider the limits as α approaches 0 or ∞.