Part I

Dynamics and motion control modeling exercises 2024

Introduction

This document gives part 1 of the exercises that should be performed during the course. They should be done in groups of two or three students.

General guidelines for the exercises

The result of each exercise should be one or more MATLAB scripts or Simulink models. Use the MATLAB command *sim* to start a Simulink simulation from within an *m*-script, see *help sim* in MATLAB for usage.

Observe that plots often must be scaled in a proper way such that the interesting part of the plot is highlighted.

The *subplot* and *hold* commands are useful when plots are complex and build upon each other. Use titles and labels and colours to increase the understanding of the results.

Physical Modeling using MATLAB Simscape

Physical modeling is a way of modeling and simulating systems that consist of real physical components. MATLAB Simscape is a toolbox of Simulink and provides fundamental building blocks from mechanical, electrical, hydraulic, and other physical domains. This exercise helps you learn basic skills of modeling simple mechatronic systems with Simscape.

A simple tutorial on building the Simscape model of a mass spring damper system is in the link.
http://se.mathworks.com/help/physmod/simscape/ug/creating-and-simulating-a-simple-model.html

Please go through the tutorial and then you should be able to solve Exercises 1 – 3 by Simscape.

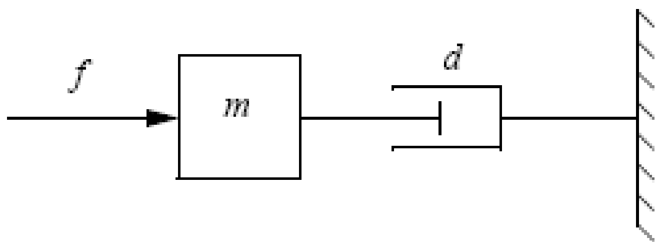
Another tutorial on building the Simscape models of RC and RLC circuits can be found in the link.

http://ece.wpi.edu/mathworks/ece2010_tutorial3.pdf

This will be helpful for solving Exercise 4.

EX. 1.

The mechanical system in the figure below consists of a load with mass $m = 0.1$ [kg], a damper $d = 0.5$ [Ns/m]. The damper is mounted between the load and a fixed point. An external force is applied directly on the load.



- Case 1: Input is *f*, output is the speed of the mass
- Case 2: Input is *f*, output is the position of the mass

Derive both a state space model and a transfer function model for each case in paper.

In MATLAB:

Build the above transfer function and state space models in MATLAB script.

Plot the frequency response (bode), the step response and the pole zero plot. What are the DC-gain and time constant of the system? Both read them from the plot and calculate them numerically.

In Simulink:

In one Simulink file, you should model both the systems with both a “Transfer Fcn” block and the block diagram of the state space model using 1/s blocks. Compare the results of both models to verify that they give the same results.

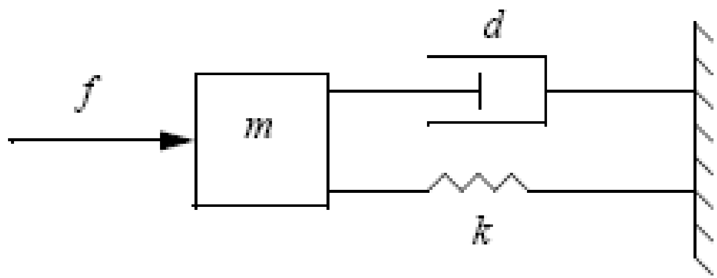
First simulate the model with the force as a step input, compare it with the Matlab step. Secondly, simulate the models with the applied force as a sine wave. Simulate the model with sine waves of frequency 1 rad/s and 10 rad/s. Plot both applied force and velocity in the same plot. Compare the results with the frequency Bode plot and draw conclusions. Is it correct?

In Simscape:

Build the Simscape model of the system and compare its output with the Simulink model. Apply all input signals in the previous question and verify if the Simscape model returns identical result.

EX. 2.

In addition to the system in EX. 1, a spring $k = 8$ [N/m] is connected in parallel with the damper.



- Case 1: Input is *f*, output is the speed of the mass
- Case 2: Input is *f*, output is the position of the mass

For both cases, derive both a state space model and a transfer function model in paper.

In MATLAB:

Let the output be the speed of the mass.

Plot the frequency response (bode), the step response and the pole zero plot, what are the natural resonance frequency, damped resonance frequency, and DC-gain?

Why is the resonance frequency that you can read in the pole zero map not exactly the same as the peak frequency that you can read at the peak gain in the bode plot?

Let the output be the position of the mass.

Make a step response of the system from external force to position. What are the natural resonance frequency, damped resonance frequency and DC-gain now?

In Simulink:

In the same Simulink file, you should model the system with both a "Transfer Fcn" block and the block diagram of the state space model using 1/s blocks. Compare the results of both models to verify that they give the same results

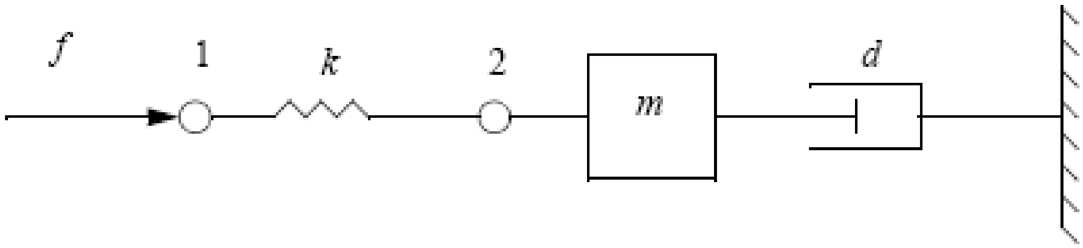
First simulate the model with the force as a step input, compare it with the MATLAB step. Secondly, simulate the model with the applied force as a sine wave. Simulate the model with three different sine waves: one with frequency below the natural resonance frequency, one with frequency at the natural resonance frequency and one at a frequency above the natural resonance frequency. Plot both applied force and velocity in the same plot. Compare the results with the frequency plot and draw conclusions. Is it correct?

In Simscape:

Build the Simscape model of the system with the position as the output and compare its result with the corresponding Simulink model. Apply all input signals in the previous question and verify if the Simscape model returns identical result.

EX. 3.

The spring is now mounted between the applied force and the load.



Derive the following three models based on Newton's laws of motion in paper. If you have limited time, you may only derive the transfer function models.

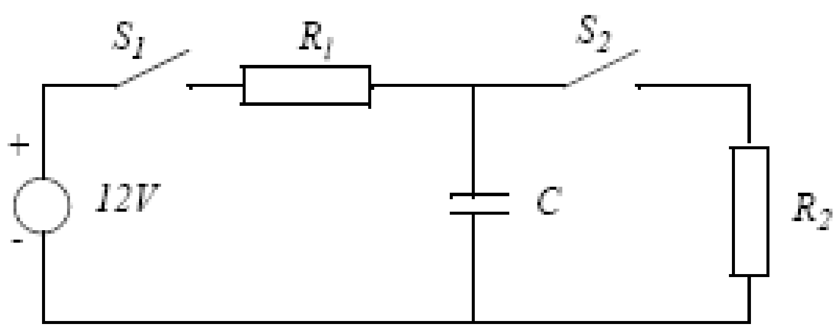
$$G_1(s) = \frac{v_2(s)}{v_1(s)}$$, from velocity at node 1 to velocity at node 2.

$$G_2(s) = \frac{v_2(s)}{f(s)}$$, from applied force to velocity at node 2.

$$G_3(s) = \frac{v_1(s)}{f(s)}$$, from applied force to velocity at node 1.

Do the same type of analysis in MATLAB, Simulink, and Simscape as above. Do you find any problem with the step response of $G_3(s)$, why?

EX. 4. An electric network with switches



At time zero the switch S_1 is closed, and it is kept closed rest of the simulation time, so the voltage source provides a step input of 12 V to the circuit. Switch S_2 is open at time zero. Derive the transfer function model for the voltage over the capacitor as the output and the voltage of the source as input. At time 0.2s the switch S_2 is closed, derive a second model for the voltage over the capacitor. Derive both transfer function models and state space models using Kirchhoff laws.

In MATLAB:

Plot the frequency response (bode), the step response and the pole zero plot of both the model with S_2 open and when S_2 is closed. What are the DC-gains and time constants of the models? Both read them from the plot and calculate them numerically.

In Simulink:

The same Simulink file compares direct transfer function blocks and a block diagram model using only **gain**, **sum** and **integrator** blocks. The model must have a time-dependent switch on function to show a continuous simulation time of 0.3 second. The **switch** block is found in the “Signal Routing” library of Simulink and time is outputted by a **clock** block in the “Sources” library.

Simulate the model with a s input voltage of 12V, $R_1 = 200\Omega$, $R_2 = 200\Omega$ and $C = 10\mu F$.

Note that the voltage source of 12 V is applied to the second transfer function model (after S_2 is on) at 0.2 second.

Do the block diagram model and the transfer function model have the same output? If not, which model is correct? Tip: The initial values in the transfer function block are always 0, but when S_2 is switched on, the initial voltage value of the capacitor is not 0.

In Simscape:

Repeat the circuit model in Simscape and verify the result is identical to the Simulink block diagram model.