

EM 406 Vibrations

LAB 2: Experimental Determination of Frequency Response

Objectives

1. Take frequency response data (magnitude only, one frequency at a time)
2. Collect swept sin data at two different rates (70 seconds, 35 seconds)
3. Generate FRF from swept sin data using tfestimate (70 second data)

In this lab we will be experimentally determining the frequency response of a mass-spring-damper system and then constructing a frequency response plot to visually convey this information. We will be using the experimental setups in C-116. Figure 1 show one of the carts of the system you will be using in this lab. This cart is located between two springs and slides on bearings. The position of the cart is measured by an encoder which is located behind the cart.

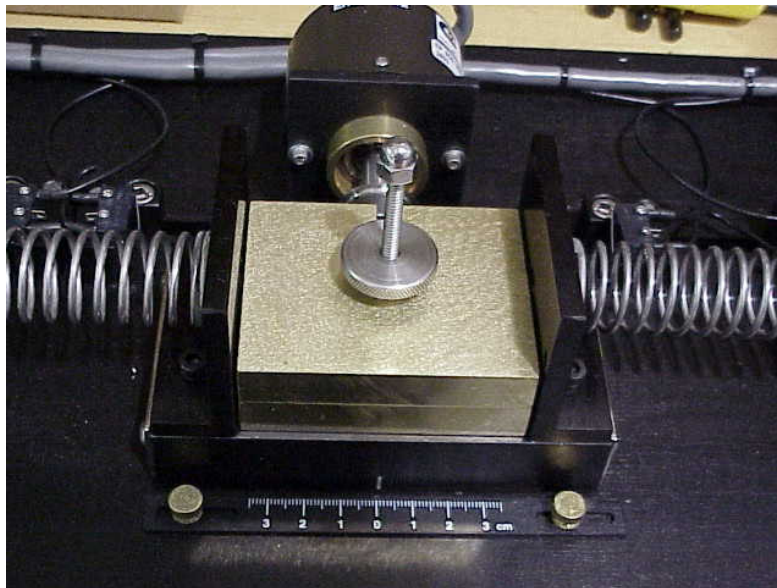


Figure 1. One of three carts in the mass/spring/damper system. Each cart is connected via springs, and the position of each carts is measured by an encoder .

To measure the frequency response we will have the motor generate an input sinusoid with a known frequency and known amplitude. This input signal will be applied to the first cart via a rack and pinion mechanism, as shown in Figure 2. The system will record the amplitude of this input signal as well as the positions of each of the carts in the system as a function of time. These cart positions will initially display transient responses, which will decay away. The cart positions will eventually oscillate with the same frequency as the input signal. However, it is likely that the positions of the carts will not have the same amplitude as the input sinusoid and there may be a phase shift between the input signal and the positions of the carts. It is this change in amplitude and phase between the input signal and output signals (the positions of the carts) as a function of frequency that the frequency-response plot conveys graphically. We will only use the first cart for today's lab. Next week we will use this data to be sure to save it.

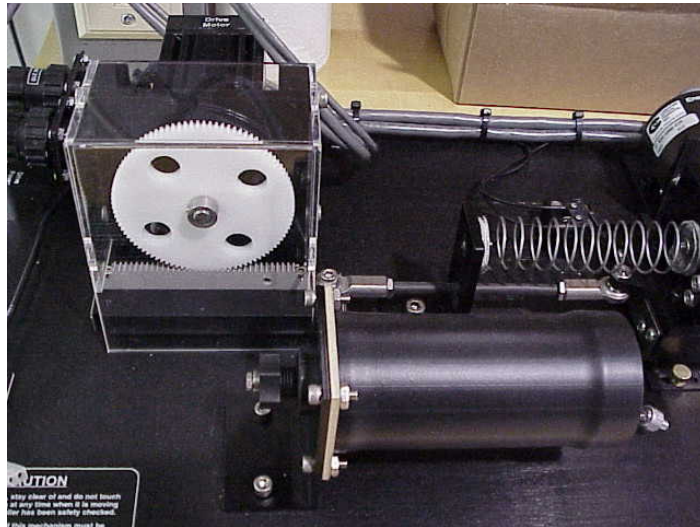


Figure 2. The rack and pinion (at the end of the spring on the far left) connecting the motor to the first cart

For you to do in lab:

Getting Started

1. Set up the environment.

Configure the hardware in 1 DOF (degrees of freedom) mode with one 500 g brass masses on the first carriage. Use two springs to connect the cart to ground. Disconnect the damper. Ask your instructor for help if you are not sure how to configure the station.

Log on the computer and start MATLAB 6.5.1 using the desktop icon. In the MATLAB Current Directory window, navigate to the C:\Documents and Settings\student\Desktop\EM406 directory. Open the Simulink *model* (not the directory) 'lab_two_sin.mdl'. Also open the model 'ECPDSPResetmdl.mdl'.

2. For each response you will need to do the following:

Make sure the interface box is energized (black button). In 'ECPDSPResetmdl.mdl' ensure the External/Normal window shows 'External'. Push the 'Connect to Target' button. The 'Play' button should become black. Push the black 'Play' button and wait about one second. The encoders are now reset. Now in 'lab_two_sin.mdl' ensure the External/Normal window shows 'External'. Push the 'Connect to Target' button. The 'Play' button should become black. Push the black 'Play' button and watch the response. If you get 'Internal Error' when pushing the 'Connect to Target' button, simply try again. Errors reading 'could not execute target data map file' indicate that you are working in the wrong directory. Change the working directory (at the top of the MATLAB command window to 'C:\Documents and Settings\student\Desktop\Em406 and try again.

If you get no response, or a very noisy response, try the following:

- a. Check that the ECP interface is on (you did press the black button, didn't you?).
- b. Are there cables connected to the system?

- c. Is the correct controller personality file loaded? Exit MATLAB and start the ECP executive, under
Start→Programs→ECP→ECP32
Utility→Download Controller Personality Files
and load the file C: Program files\ECP System\en\m210_rtw_3.pmc
- d. Close the ECP application, restart MATLAB, open the Simulink models, and try again.
- e. If the software still does not detect cart motion, then contact your instructor.

In most cases you should not have to use the ‘Build’ button unless Simulink gives you an error message declaring that the model needs to be rebuilt.

3. After each run, your data will be stored in the MATLAB workspace. You can use a command like the following to save your data to disk:

```
>> save run1hz
```

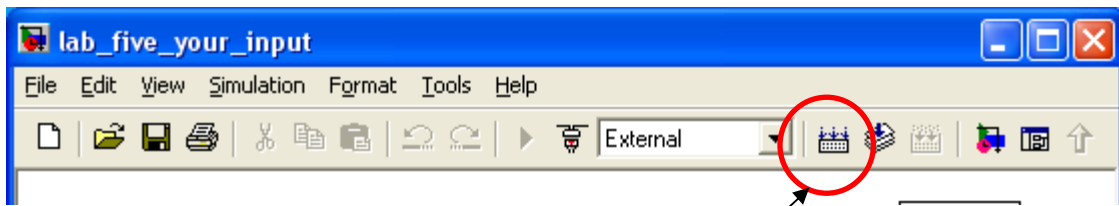
This saves the current workspace as ‘run1hz.mat’ in the current working directory. Be sure to save the workspace after each run. After you are done with the experiment, you will want to move your data to a different directory, flash drive, or your afs workspace. The third option can be done by using SecureFX.

Procedure for taking data

1. For frequencies 1.0 Hz, 1.5 Hz, 2.0 Hz, 2.5 Hz ... 7 Hz, run the system and collect the data. The simulation is set to run for 20 seconds. In most cases this time frame will allow the system to reach steady state. You can change the frequency by double-clicking the ‘Sine Wave’ block and setting the desired frequency. Note that the frequency is set as $x*(2*\pi)$, so that x is the input frequency in Hz. The amplitude is initially set at 0.01. Change this as needed to produce a response that is large enough to record, but not so large that the carts are pushed into the stops. *Be sure to record the input amplitude for each run on the lab worksheet.*

Clearly doing this one frequency at a time is a very time consuming process. In practice one would use an input such as band limited random noise or a swept sine (a chirp) and then use the fast Fourier transform (FFT) to determine the auto- and cross-spectra which can be used to determine the frequency response functions (FRFs). The frequency response plots we have been talking about in class are simply plots of the magnitude and phase of the FRFs.

2. Input a swept sine. Open the Simulink *model* (not the directory) ‘lab_two_swept.mdl’. This model is designed to input a swept sign with frequencies from 0.1 to 7.0 Hz over a period of 70s. As usual run ‘ECPDSPResetmdl.mdl’ prior to lab_two_swept.mdl. To find the approximate frequency at any instant in time simply divide the time by 10. **Watch the system closely during this test and record your observations on the worksheet.** Save your data for plotting.
3. Perform a second swept sin test by changing the period to 35 seconds rather than 70 seconds. Save your data for plotting. Just changing the chirp is not adequate. You will need to change the “simulation parameters” and make the final time 35 seconds. You will also need to rebuild the simulation. You can do so by clicking on the icon shown on the next page.



Click here to rebuilt the model

Analysis

1. Plot the results in Matlab and complete Table 1. The position of the first cart is called x_1 , and the time variable is called *time*. From each plot determine the amplitude (peak value) of the sinusoidal steady state response. Do this for each cart at each frequency, and record the values on the worksheet. Determine the steady-state amplification factor for each cart at each frequency. To convert to decibels take $20\log M$.
2. After you have saved your data for the swept sin test you can calculate the FRF by using the built-in Matlab function called “tfestimate” (Transfer function estimate). Next week we will discuss what this built-in function is doing. Sample Matlab code is shown below:

```
load swept_data70 % this assumes the data is called "swept_data70.mat"

dt = time(2)-time(1); % this finds the time step
N = size(x1,1); % this is the number of data points
T = N*dt; % this is the total length to time
fs=1/dt; % this is the sample frequency

f=force;

% the following command will calculate the transfer function at each
% frequency using a rectangular window, no overlap, and all the data
% points. Windowing is sometimes used to help reduce what is called
% "leakage" (you'll discuss this in future classes)

[Txy,F] = tfestimate(f,x,rectwin(N),0,N,fs);

% to plot the magnitude plot you need to take the magnitude
semilogy(F,abs(Txy))

figure

% to plot the phase (in degrees) plot you need to take the angle
plot(F,angle(Txy)*180/pi)
```

Construct the following plots:

1. Plot of the swept sin data for the two different sweep rates. Use subplot so the top plot is for a 70 s sweep rate and the lower plot is for a 35 s sweep rate.
2. Plot the magnitude frequency response plot using linear axes. Use “subplot” so the (1,1) plot is the magnitude plot from taking data one frequency at a time, the (1,2) plot is the magnitude of the FRF from the 70-s swept sin test, the (2,1) plot is for the other swept sin test and the (2,2) plot is all three magnitudes together. Make sure your units are consistent. Only plot the data for frequencies of 0 – 7.0 Hz.
3. Plot the magnitude frequency response plot for mass one using the following: Vertical axis for the magnitude is a linear scale, but the magnitude is in dB, and the horizontal axis for frequency

is a log scale. Only plot the data for frequencies of 1 – 7.0 Hz. Have the same subplots as discussed in step 2 above.

4. Plot the magnitude and phase of the transfer function for all the frequencies for the 70 s sweep test. Use subplot so the top plot is the magnitude and the bottom part is the phase in degrees.

Save all your data because next week we will use it to perform system identification. We will also use the chirp input and output data to determine FRF's and system parameters.

Lab 2 Worksheet

Return to CM: _____

EM406 Mechanical Vibrations
Rose-Hulman Institute of Technology

Names: _____

Table 1 – Frequency Response Data

Freq. (Hz)	File name	Input mag. (unknown units)	Cart 1 steady state mag. (cm)	H1 = Out1/In	H1 in dB
1.0					
1.5					
2					
2.5					
3					
3.5					
4					
4.5					
5					
5.5					
6					
6.5					
7					

Comment on your results and record observations made during the swept sin tests

Attach all figures to this worksheet.