

CET346 Final Project

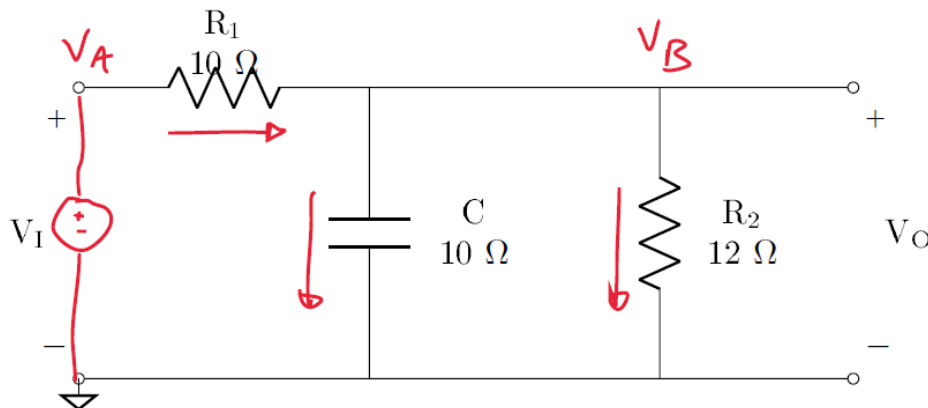
David J. Broderick
Spring 2020

1 PURPOSE

State what you think is the purpose of this project.

2 TRANSFER FUNCTION

I found the transfer function using nodal analysis. The circuit has two non-references node labeled V_A and V_B below.



In order to find the two unknown node voltages I wrote 1 KVL (since there is a single voltage supply) and 1 KCL to complete the system of equations.

$$\begin{aligned} &\underline{\text{KVL A}} \\ &V_I = V_A \\ &\underline{\text{KCL B}} \\ &\frac{V_A - V_B}{R_1} - \frac{V_B}{Z_C} - \frac{V_B}{R_2} = 0 \end{aligned}$$

The output of the circuit (labeled V_O) is simply V_B . To find the transfer function I divided the output by the input, V_{IN} . The following matlab code makes these calculations:

```

%% Finding a transfer function with Nodal Analysis
clear all
close all
clc
format short eng

R1=10;
R2=12;
C=1e-3;
syms s Vin Va Vb
Zc=1/(s*C);

e(1)=Va==Vin; %KVL A
e(2)=((Va-Vb)/(R1)) - ((Vb-0)/(Zc)) - ((Vb-0)/(R2)) == 0; %KVL B
sol=solve(e, Va, Vb);
Vout=sol.Vb-0;
H=Vout/Vin;
pretty(simplify(H))

```

The output of the code is the transfer function $H(s)$

$$H(s) = \frac{300}{3s + 550}$$

Given there are no zeros and a single pole I expect the frequency response of the circuit will be that of a low pass filter.

3 BODE PLOT

The Bode plot depicts the complex gain over a range of frequencies. I used the transfer function I found in the section above to plot the Bode plot in matlab and then validated the result using simulation in LTSpice.

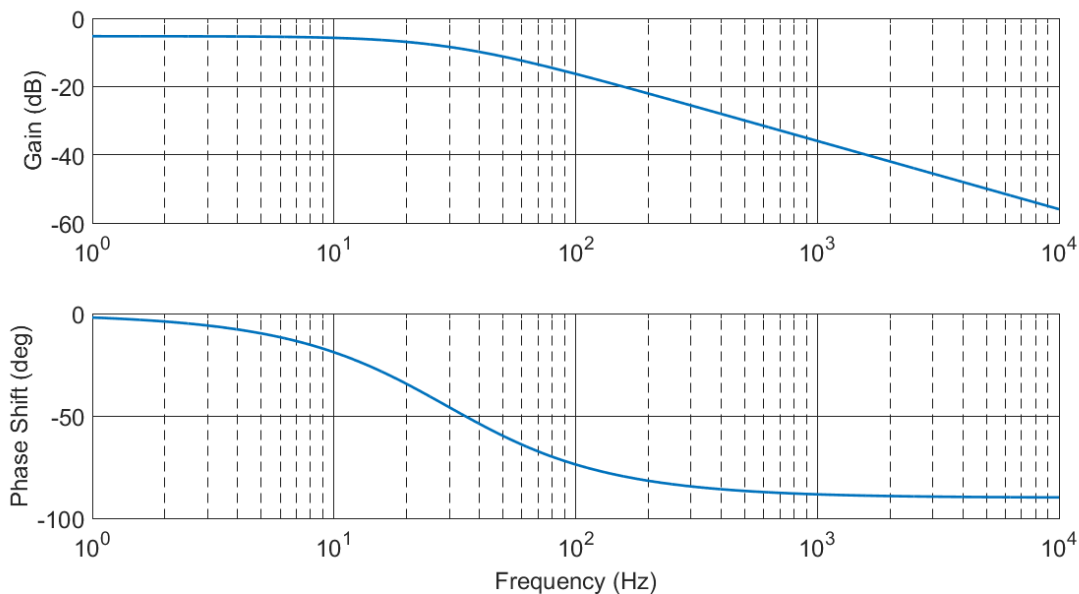
3.1 MATLAB ANALYSIS

The following code uses the transfer function in the variable H from the code above to generate the Bode plot:

```
%% Bode Plot
H=matlabFunction(H);
f=logspace(0,4,1000);
w=2*pi*f;
figure('Renderer','painters','Position',[391 289 1269 634])
subplot(2,1,1)
semilogx(f,20*log10(abs(H(j.*w))), 'LineWidth',2)
grid on
ylabel('Gain (dB)')
set(findall(gcf,'-property','FontSize'),'FontSize',18)
h=gca; h.MinorGridAlpha=1; h.MinorGridLineStyle='--'; h.GridAlpha=1;

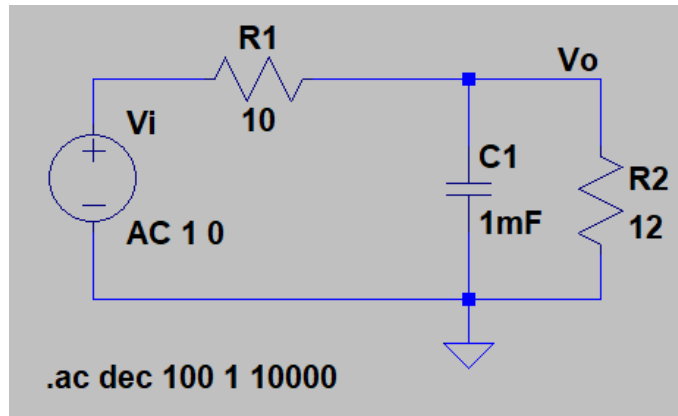
subplot(2,1,2)
semilogx(f,angle(H(j.*w))*(180/pi), 'LineWidth',2)
grid on
xlabel('Frequency (Hz)')
ylabel('Phase Shift (deg)')
set(findall(gcf,'-property','FontSize'),'FontSize',18)
h=gca; h.MinorGridAlpha=1; h.MinorGridLineStyle='--'; h.GridAlpha=1;
```

The following plot is the output of this code:

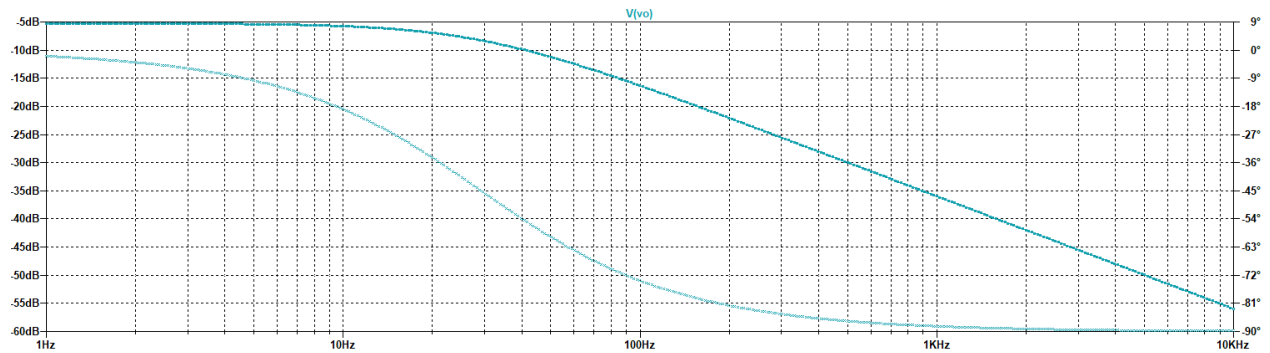


3.2 LTSPICE VALIDATION

I constructed the circuit in LTSpice and setup AC analysis to recreate the Bode plot. The schematic and spice directive are shown here:



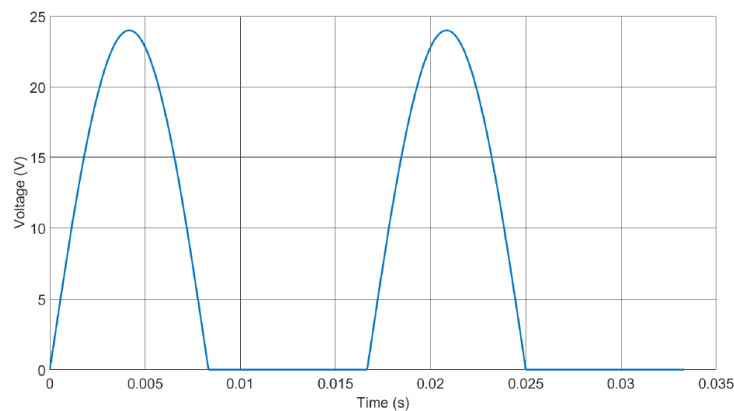
The plot below shows the output of the AC analysis:



Checking values between the Matlab plots and the LTSpice plot show that the results are consistent.

4 INPUT SIGNAL AND FOURIER SERIES

The plot below shows the assigned input signal:



I developed the following expression that represents the signal over a single period

$$v_{\pm}(t) = 24 \cdot \sin(\omega_0 t) \cdot u((T_0/2) - t)$$

where T_0 is 16.67 ms and ω_0 is 377 rad/s. I found the Fourier coefficients by integrating over the period from $t=0$ s to $t=T_0$. I found 201 Fourier coefficients each representing a multiple of ω_0 . The following code performs these integrals.

```

%% Find the Fourier coefficients
u=@(t) heaviside(t);
n=10;
w0=377;
T0=(2*pi)/w0;
x=@(t) 24.*sin(w0*t).*u((T0/2)-t);
t0=0;t1=T0;

for k=0:n;
    integrand=@(t) (x(t)).*cos(k.*w0.*t);
    a(k+1)=(2/T0)*integral(integrand,t0,t1);

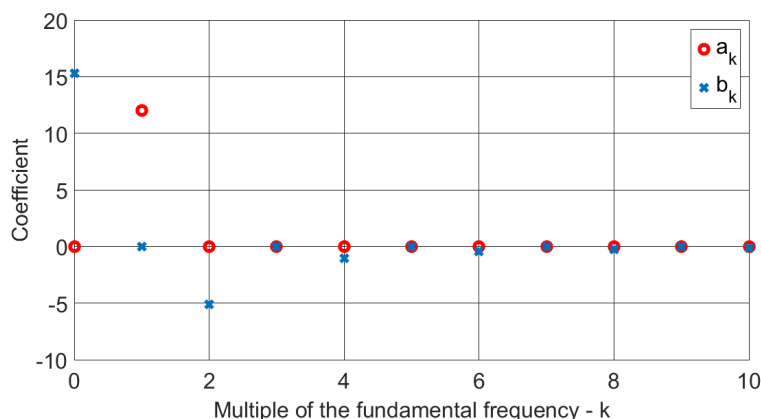
    integrand=@(t) (x(t)).*sin(k.*w0.*t);
    b(k+1)=(2/T0)*integral(integrand,t0,t1);
end

figure('Renderer','painters','Position',[391 289 1269 634])
hold on;grid on
plot(0:length(a)-1,b,'ro','LineWidth',4,'MarkerSize',10)
plot(0:length(a)-1,a,'x','LineWidth',4,'MarkerSize',10)

xlabel('Multiple of the fundamental frequency - k')
ylabel('Coefficient')
legend('a_k','b_k')
set(findall(gcf,'-property','FontSize'),'FontSize',24)
h=gca; h.MinorGridAlpha=1; h.MinorGridLineStyle='--'; h.GridAlpha=1;

```

The output of the code is a plot of the Fourier coefficients:



I have included the plot of the estimated function in the subsequent section along with the estimated output.

5 ESTIMATE OF OUTPUT

5.1 MATLAB ANALYSIS

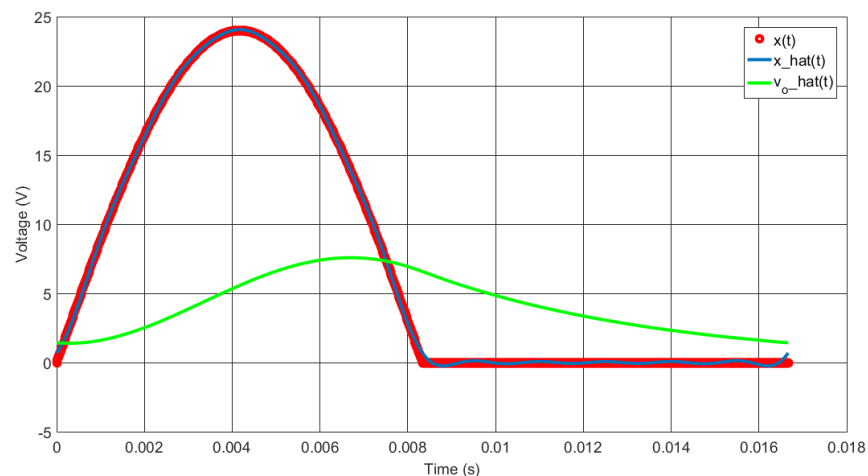
I used superposition to estimate the output by using the Fourier coefficients I found above as input phasors. I multiplied the input phasors by the gain at the corresponding frequency. I found the correct gain by evaluating the transfer function for each frequency. I moved each phasor representing the output back to the time-domain before summing the results. The following matlab code estimates the output of the circuit and adds the plot to that of the input signal.

```
%% Perform superpositon to estimate the output
vo_hat=(a(0+1)/2)*H(j*0); %DC input times DC gain

for k=1:n
    Vi=a(k+1)-j*b(k+1); %phasor for this frequency composed of a
    cosine and sine
    Vo=Vi*H(j*k*w0); %output=input times gain
    vo_hat=vo_hat+abs(Vo).*cos(k.*w0.*t+angle(Vo)); %move back to
    time-domain and add to previous result(s)
end

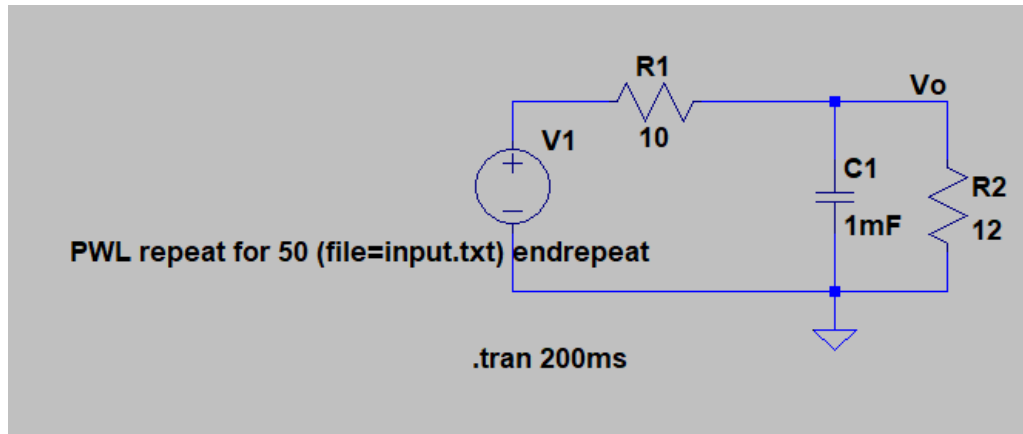
%% add the output estimate to the plot
plot(t,vo_hat,'g','LineWidth',3)
xlabel('Time (s)')
legend('x(t)','x_hat(t)','v_o_hat(t)')
set(findall(gcf,'-property','FontSize'),'FontSize',14)
h=gca; h.MinorGridAlpha=1; h.MinorGridLineStyle='--'; h.GridAlpha=1;
```

The plot below shows the output of this code



5.2 LTSPICE VALIDATION

I used the previously constructed circuit in LTSpice and setup a transient analysis to simulate the response of the circuit to the assigned input voltage. The schematic and spice directive are shown here:



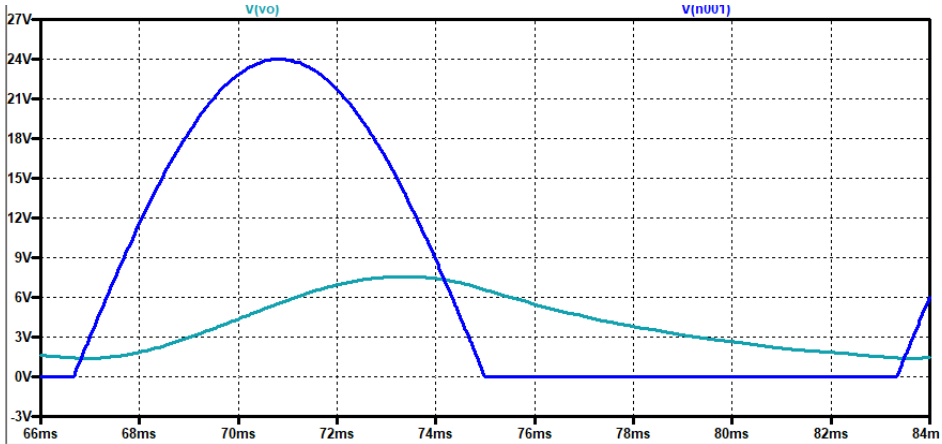
Matlab generated the file 'input.txt' using the function defined as $x(t)$ that was used as input to the Fourier analysis. The following code generated that file:

```
%% Write input waveform to CSV file for simulation  
csvwrite('input.txt', [t', x(t) '])
```

The content of that file is a list of value pairs indicating a time and a voltage. A sample of the file contents is shown here:

```
input.txt - Notepad  
File Edit Format View Help  
0,0  
1.6666e-05,0.1508  
3.3333e-05,0.30158  
4.9999e-05,0.45236  
6.6665e-05,0.60312  
8.3331e-05,0.75386  
9.9998e-05,0.90456  
0.00011666,1.0552  
0.00013333,1.2059  
0.00015,1.3564  
0.00016666,1.507
```

LTSpice simulated the output of the circuit as shown here:



Examination of the simulated results as compared to the output of the Fourier analysis show similar waveform shape and key values such as positive and negative peak voltages.