Lab 02

spice

Contents

Contents 2
Part 1 (prelab) 3
PART2 5
1 In circuit1 (overdamping): 6
2 In circuit2 (critical): 8
3 In circuit3 (underdamping): 10
PART3 12
BOUNS 14

Part 1 (prelab)

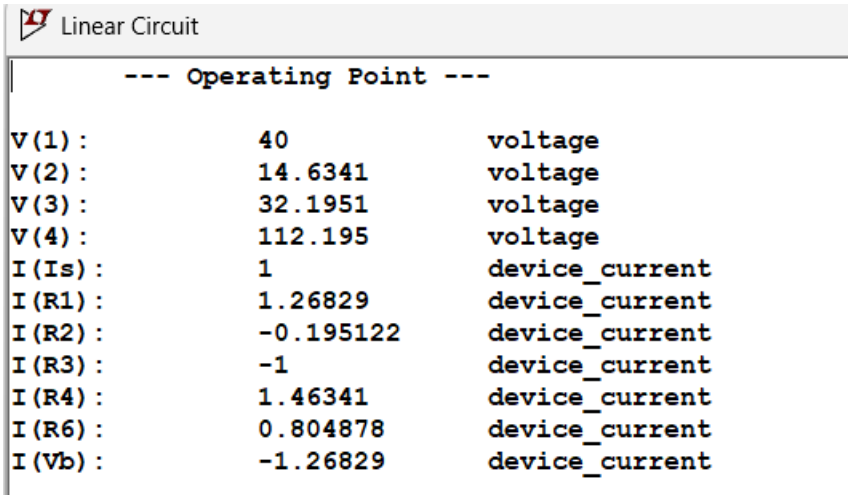
CODE IN PART1 FOLDER

```
% cleaning the workspace, and cmd window
clear all;
clc;

% running the first SPICE netlist
fprintf('the first netlist:\n');
[sum,num]=Solve_Circuit('circuit_1.cir');

clear all; % used to bypass an error only with Octave (not MATLAB)


fprintf('the second netlist:\n');
[sum,num]=Solve_Circuit('circuit_2.cir');
```

```
the first netlist:

V_1 = 30.000000
V_2 = 16.956522
I_Vb = -0.260870

the second netlist:

V_1 = 40.000000
V_2 = 14.634146
V_3 = 32.195122
V_4 = 112.195122
I_Vb = -1.268293
```

Linear Circuit

```
        --- Operating Point ---

V(1):            30              voltage
V(2):            16.9565         voltage
I(Is):           2               device_current
I(R1):           0.26087         device_current
I(R2):           0.565217        device_current
I(R3):           1.69565         device_current
I(Vb):           -0.26087        device_current
```

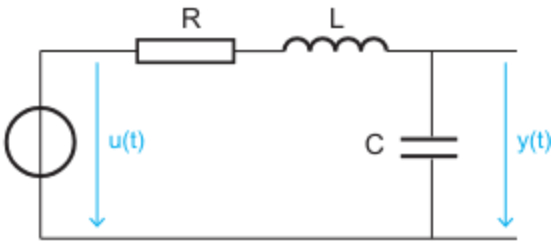


		LT SPICE	OUR CODE
CIRCUIT1	V(1)	30	30
	V(2)	16. 956522	16.9565
	I(VB)	-0.260870	-0.260870
CIRCUIT2	V(1)	40	40
	V(2)	14.6341	14.634146
	V(3)	32.1951	32.195122
	V(4)	112.195	112.195122
	I(VB)	-1.26829	-1.268293

Same results, but LT SPICE counts every current in the circuit, our code only calculate the current in the voltage sources.

PART2

CODE IN PART2 FOLDER



$$\frac{y(t)}{u(t)} = \frac{\frac{1}{sc}}{\frac{1}{SC} + R + LS} = \frac{1}{1 + RCS + LCS^2} = \frac{\frac{1}{LC}}{S^2 + \frac{R}{L}S + \frac{1}{LC}}$$

$$\omega_0^2 = \frac{1}{LC}, \omega_0 = \frac{1}{\sqrt{LC}}$$

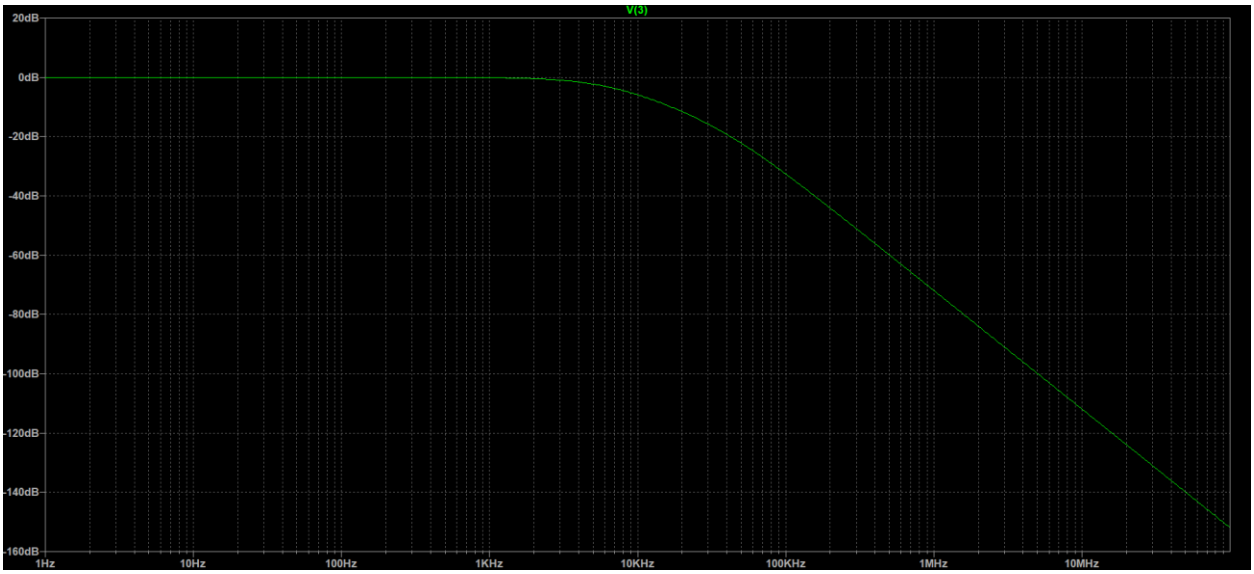
$$\frac{\omega_0}{Q} = \frac{R}{L}$$

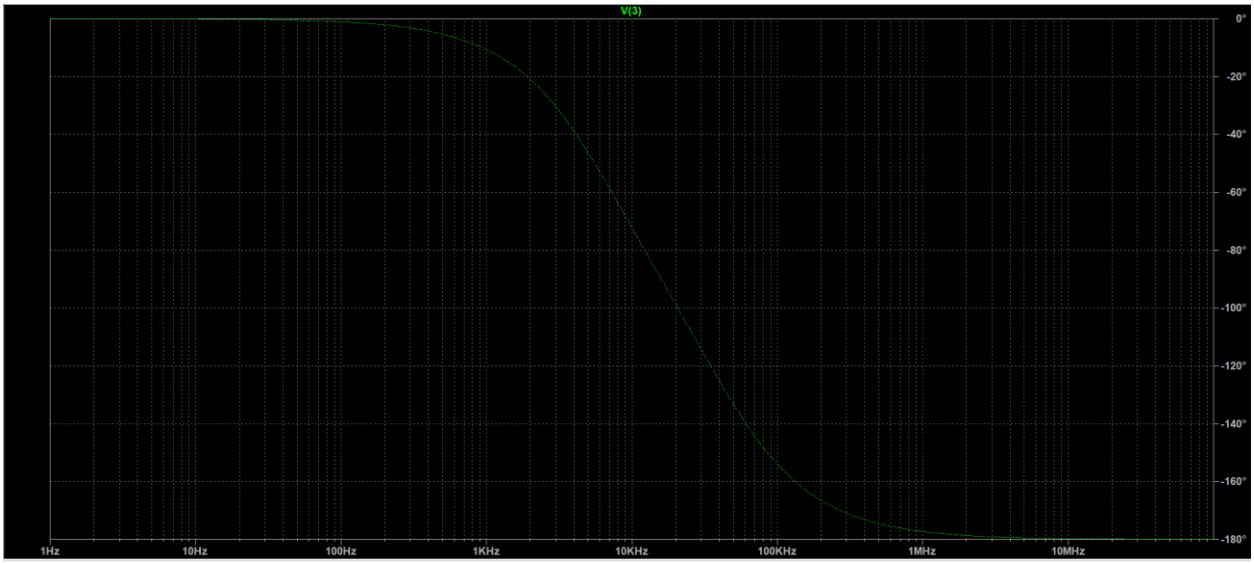
I added support for inductors, capacitors, and AC analysis, and changed the parsing of the netlist to read the user .ac command and also voltage source type.

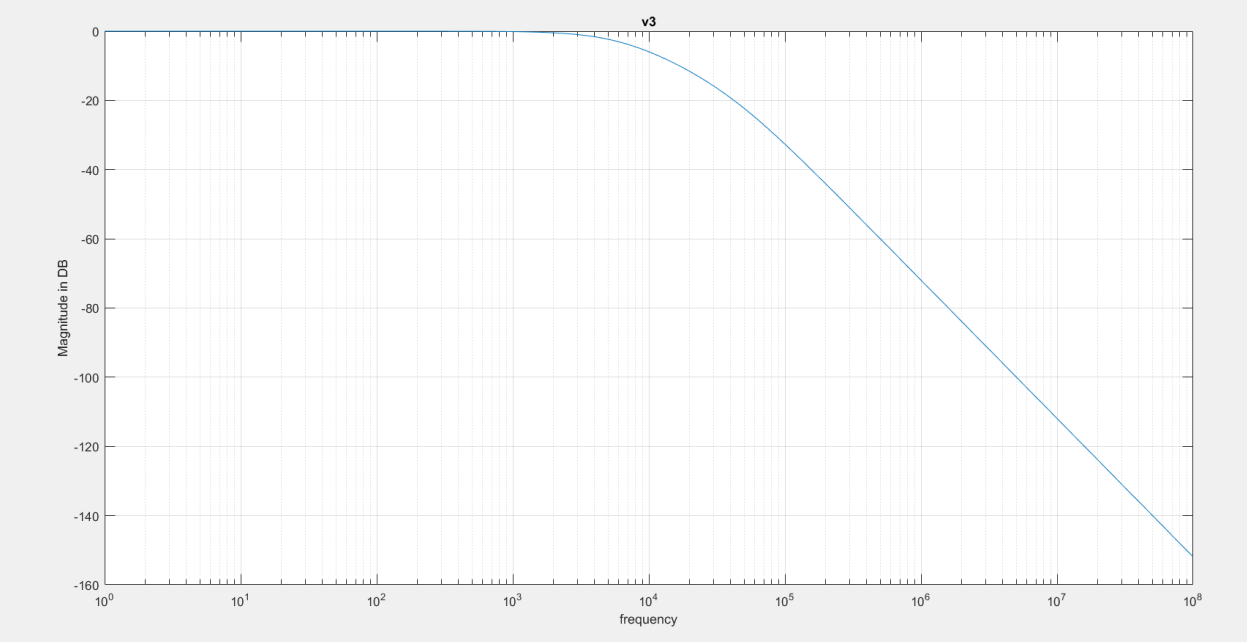
1 IN CIRCUIT1 (OVERDAMPING):

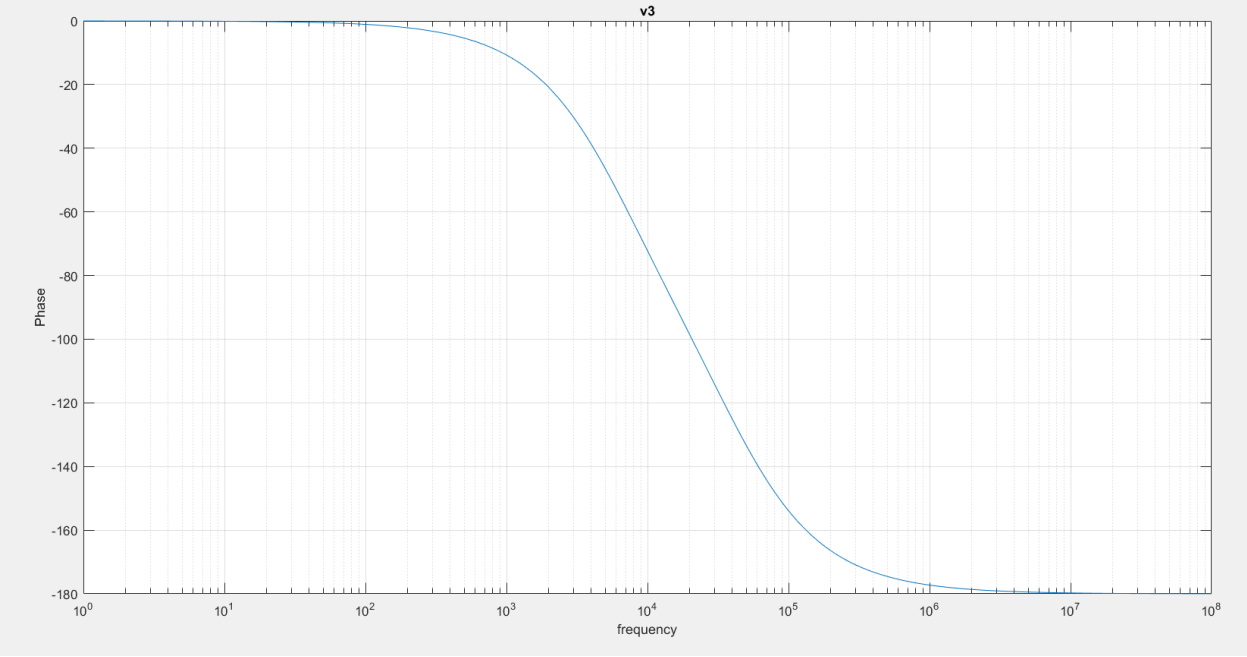
R=3000, C=10n, L=10m

$\omega_0 = 0.1MHZ$, Q=0.333





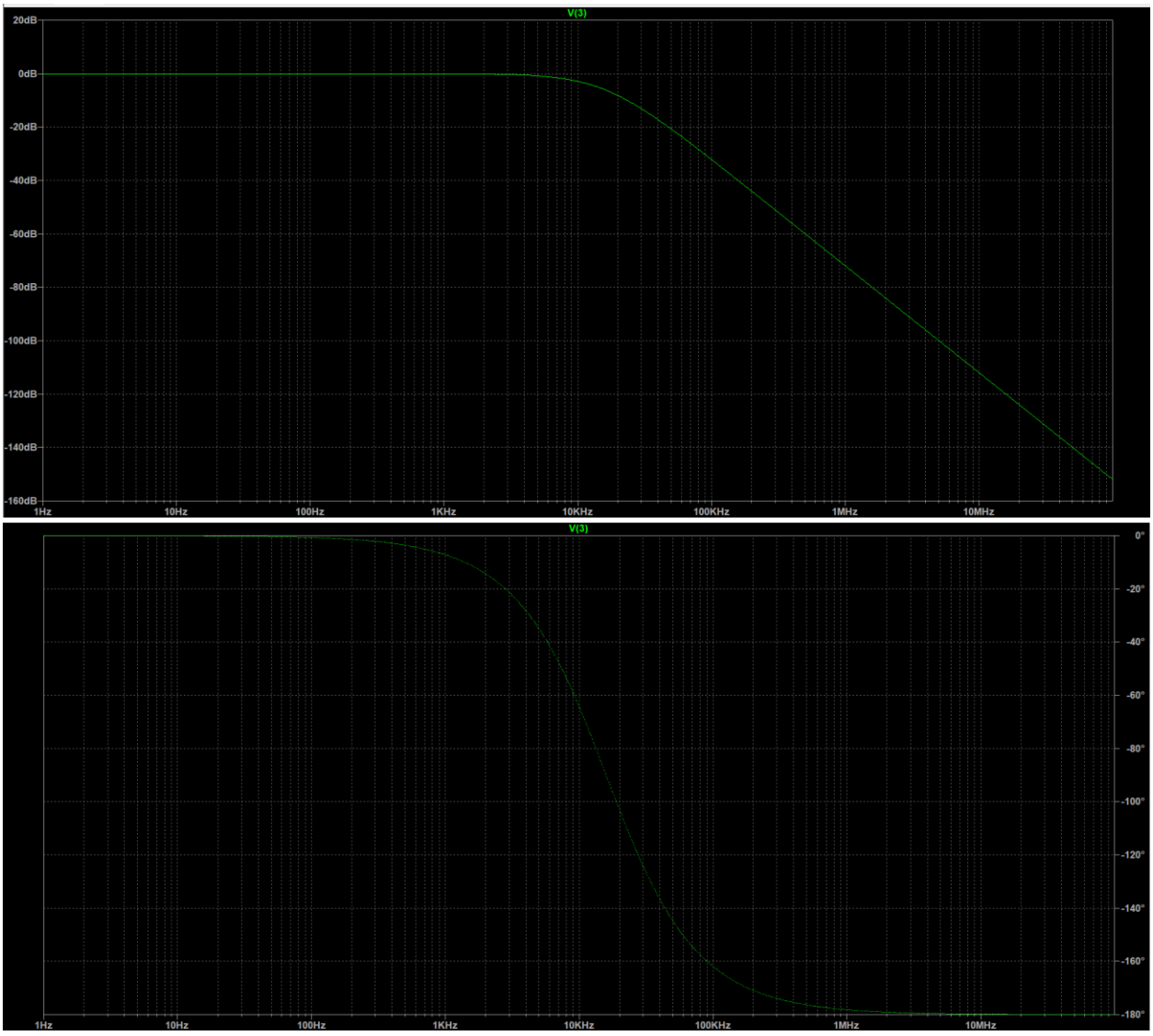

v3
Magnitude in DB
frequency
0
-20
-40
-60
-80
-100
-120
-140
-160
10^0
10^1
10^2
10^3
10^4
10^5
10^6
10^7
10^8

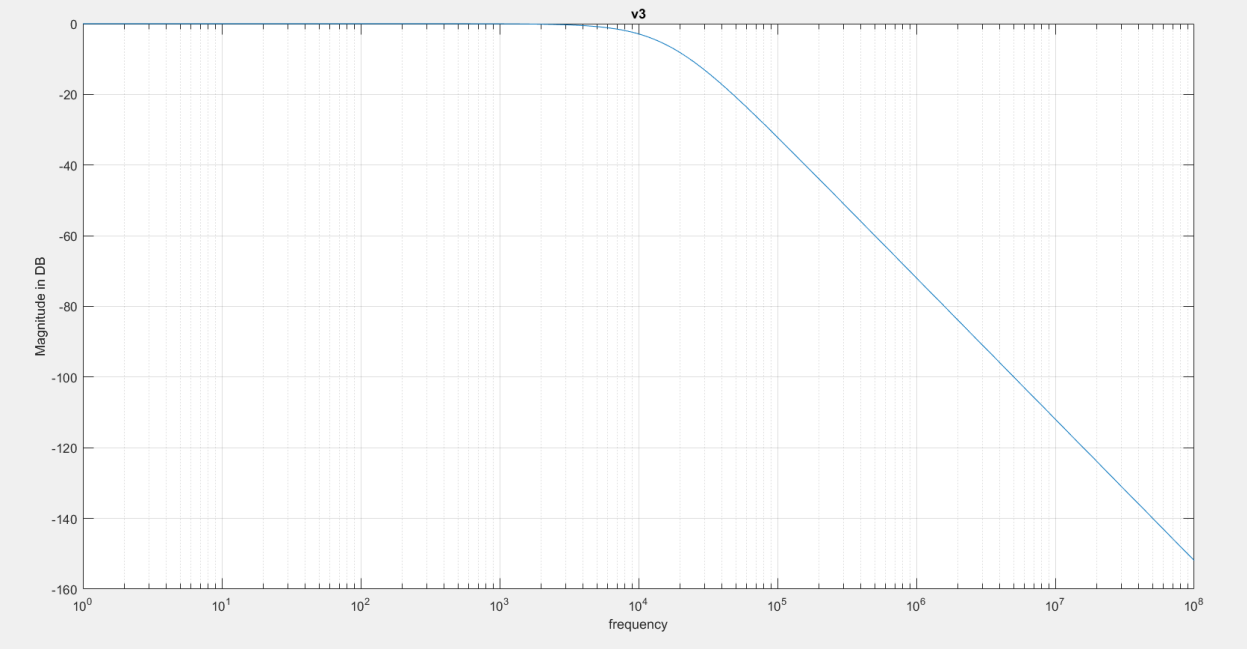

v3
Phase
frequency
0
-20
-40
-60
-80
-100
-120
-140
-160
-180
10^0
10^1
10^2
10^3
10^4
10^5
10^6
10^7
10^8

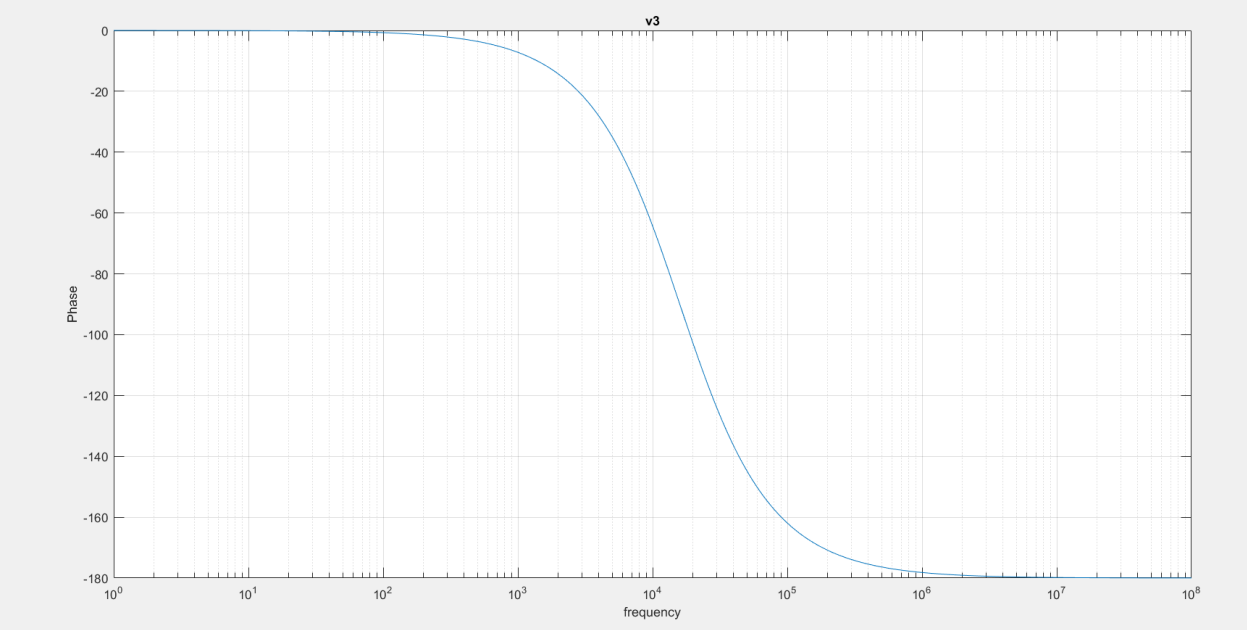
2 IN CIRCUIT2 (CRITICAL):

R=2000, C=10n, L=10m

$\omega_0 = 0.1MHZ$, Q=0.5



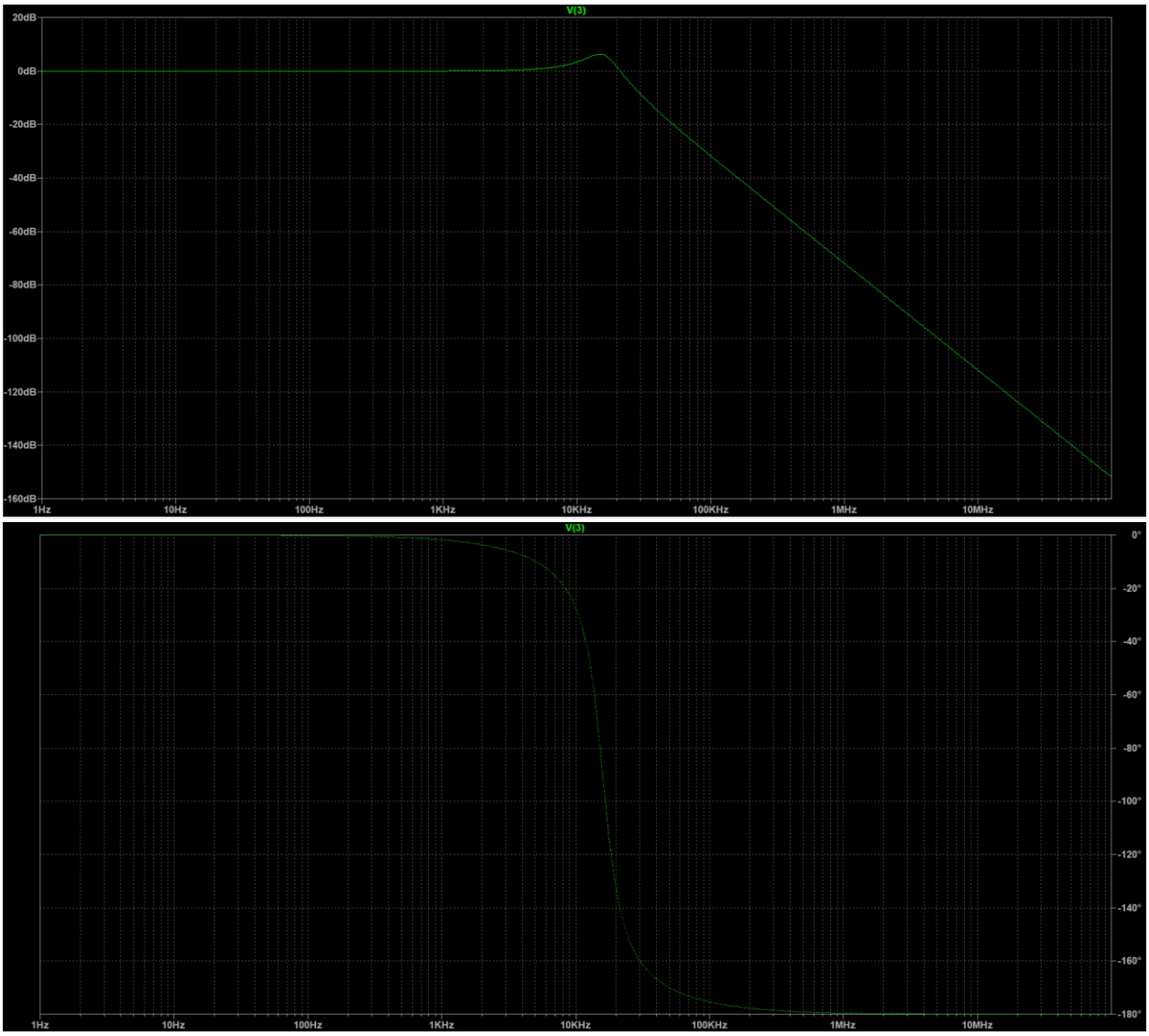

v3
Magnitude in DB
frequency
0
-20
-40
-60
-80
-100
-120
-140
-160
10^0
10^1
10^2
10^3
10^4
10^5
10^6
10^7
10^8

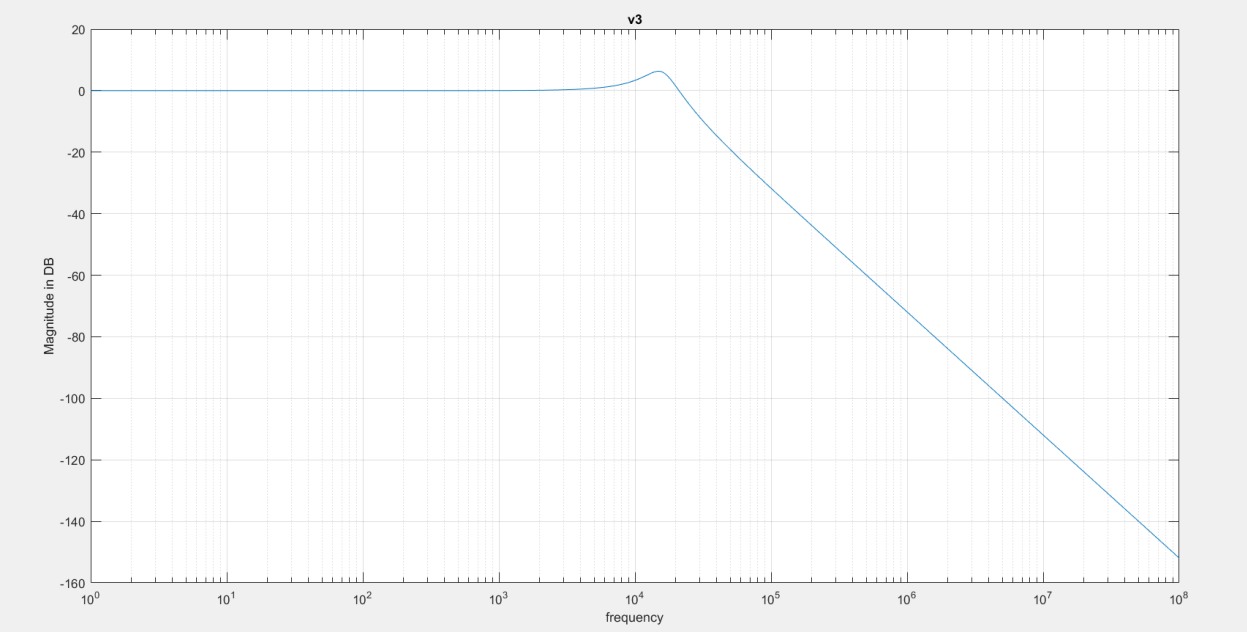

v3
Phase
frequency
0
-20
-40
-60
-80
-100
-120
-140
-160
-180
10^0
10^1
10^2
10^3
10^4
10^5
10^6
10^7
10^8

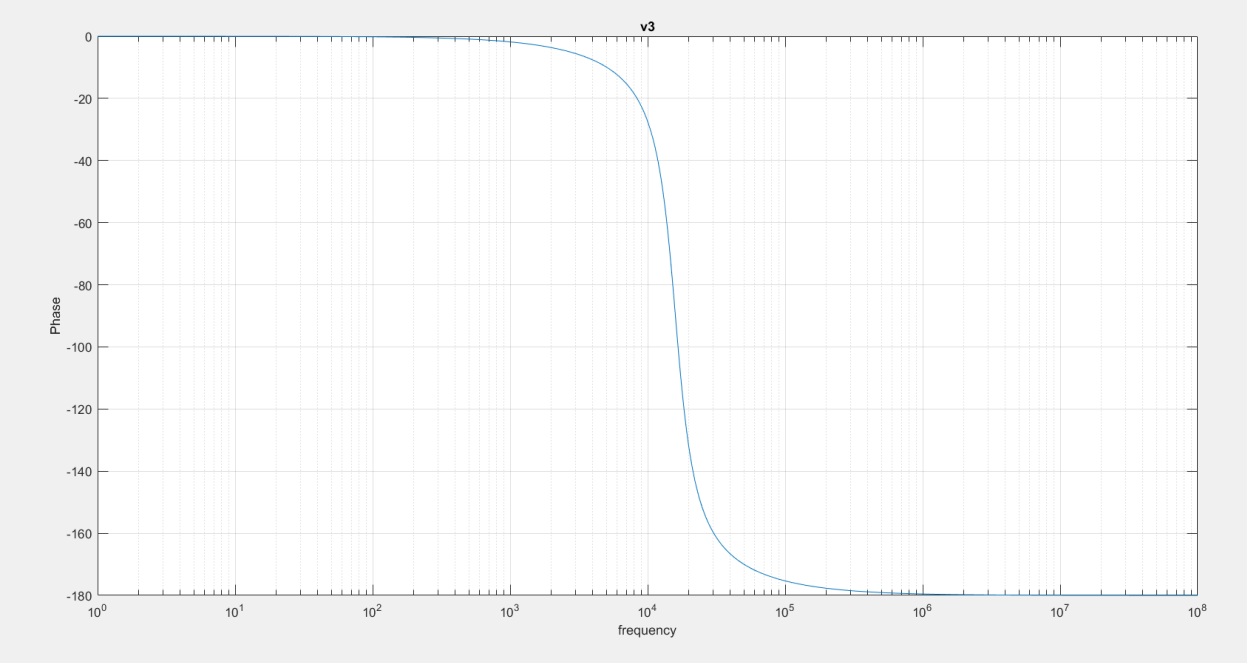
3 IN CIRCUIT3 (UNDERDAMPING):

R=500, C=10n, L=10m

$\omega_0 = 0.1MHZ$, Q=2





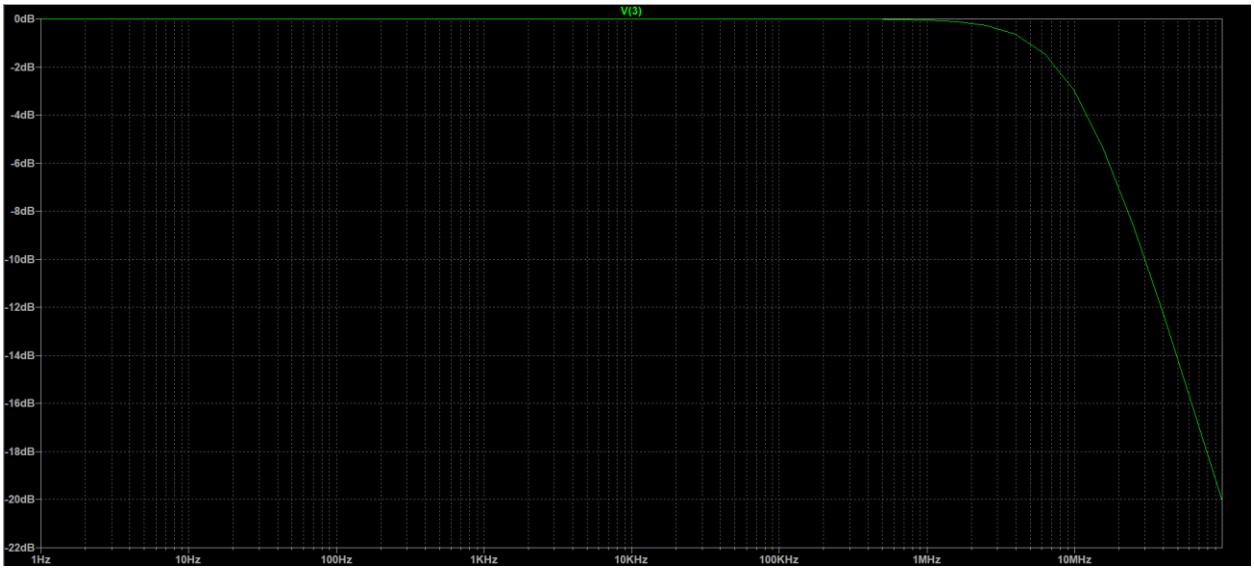


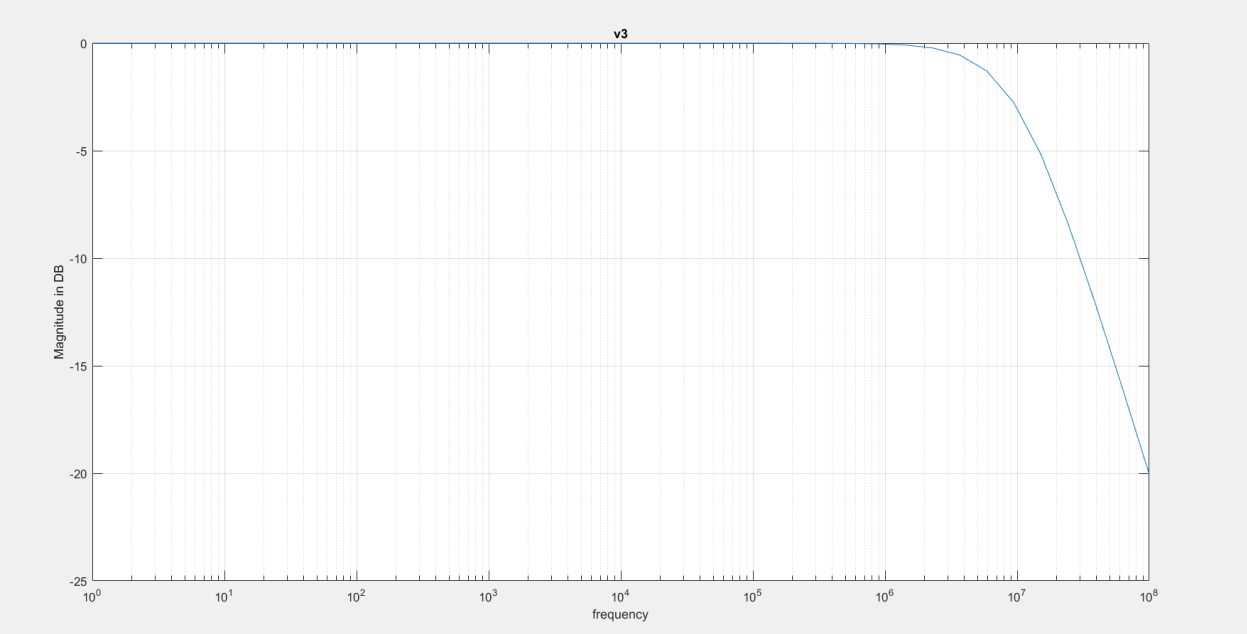
The results are pretty much the same.

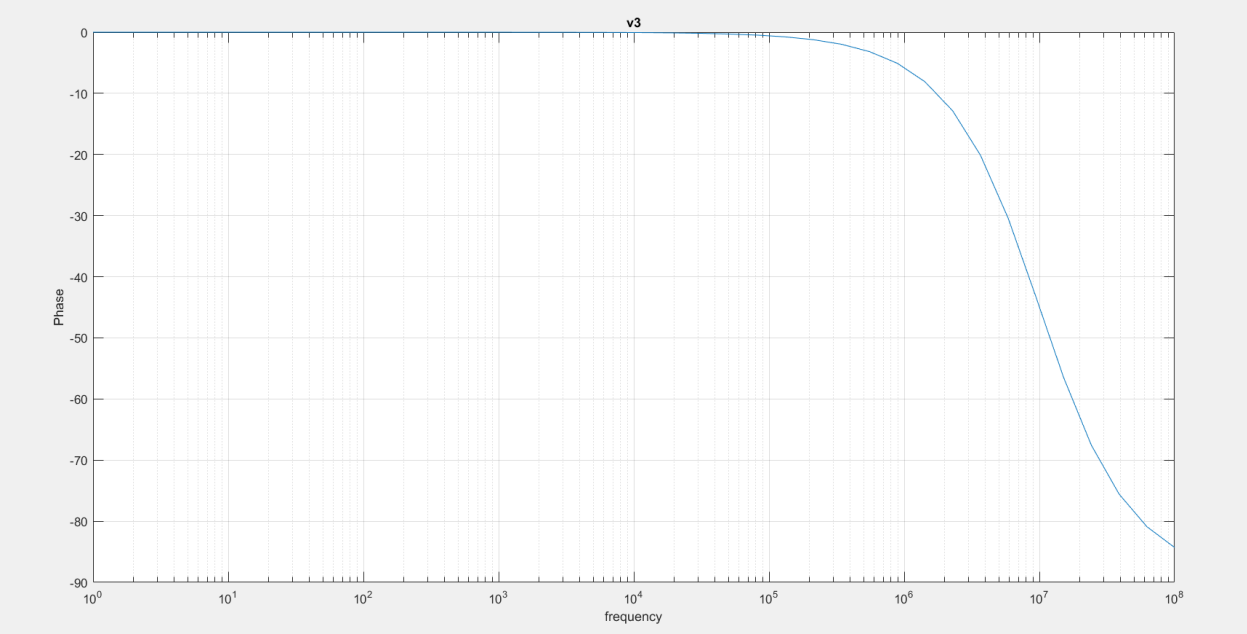
the code shows all bode plots for all unknown node's voltages, and currents in voltage sources, like we had in the lecture.

PART3

CODE IN PART3 FOLDER







The results are pretty much the same.

BOUNS

```
    if instances_key=='.'
       if strcmp(splitted_line(1),'.AC') || strcmp(splitted_line(1),'.ac')
     Node_1 = [splitted_line(2)];

        Node_2 = [splitted_line(3)]; %n
        Node_2=prefixes(Node_2);


        Values = [splitted_line(4)]; %fmin
        Values=prefixes(Values);

        Names = [splitted_line(5)];  %fmax
        Names=prefixes(Names);

   continue;
       end
       continue;
    end
```

```
[DEC n fmin fmax] = ParseNetlist(netlist, '.');

fmax=str2double(fmax)  ;
fmin=str2double (fmin) ;
n=str2double (n) ;

f=logspace(log10(fmin),log10(fmax),log10(fmax)*n) ;
for k=1:length(f)
            S=1i*2*pi*f(k) ;
      numeric_ans=subs(symbolic_ans);

      for ii = 1:1:numel(symbolic_ans)
 solv(ii,k)=double(numeric_ans(ii));
      end
end
```

The frequency isn't manually inserted but it changes with the netlist, also I made Prefixes a function in part3, as we use it a lot in this case (with fmax, fmin)