Ahmet Faruk Çolak
22102104
2024-2025 Fall

1-Introduction

This lab explores the frequency response of RC and RL circuits, focusing on first-order designs. The goal is to derive transfer functions, plot theoretical responses with MATLAB, and validate them using LTSpice. Component selection and corner frequency determination are key aspects of this work.

We also investigate adding RC stages to create higher-order filters and compare theoretical, simulated, and measured results. This hands-on approach helps reinforce key concepts such as gain and corner frequencies while adhering to design constraints.

2. Analysis

A- First Order RC Circuit

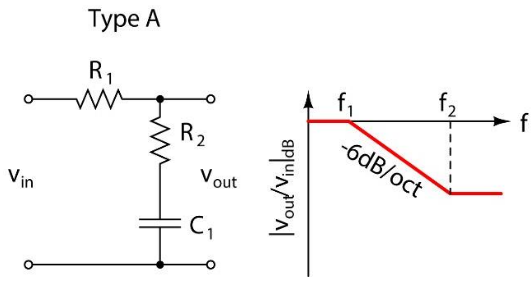

Figure.1: First Order RC Circuit

Given Values:

$ID: 22102104$

$f_1 = 5\ kHz, \quad f_2 = 120\ kHz$

Mathematical Derivations:

$$H(w) = \frac{1 + jwC_1}{1 + jwC_1(R_1 + R_2)}$$

To get 3dB Cut-off frequency before 120 kHz flatness,

$$\frac{1}{2} = \sqrt{\frac{1 + (wC_1)}{1 + wC_1(R_1 + R_2)}}$$

To get 6 dB/octave stop band, 3 dB cut-off frequency was selected as,

$$\frac{120000 \text{ kHz}}{\sqrt{2}} = 84.852 \text{ Khz}$$

From 84.852 kHz to 120 kHz, there is ½ octave and 3 dB which is 6 dB/octave. To find ½ octave, $\sqrt{2}$ is used.

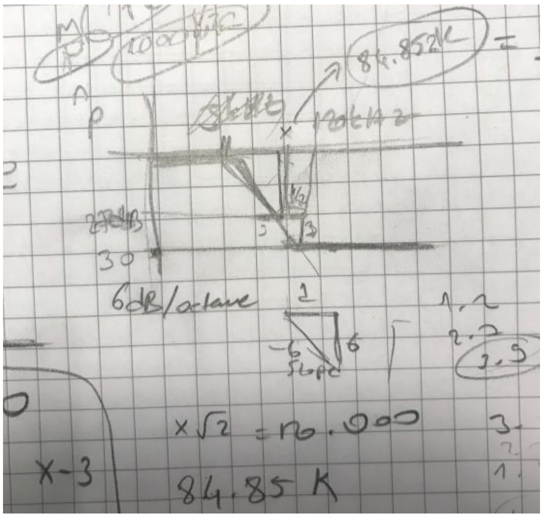

Figure.2: Cut-off Frequency Calculations to get 6db/octave

To select R_1, R_2 and C_2 simple python code was generated from mathematical derivations. The method while using python code is changing R and C values until finding the correct components.

Input:

```
import math as math
C1 = 330e-12
R1 = 2700
R2 = 33000
w = 120000 / (2**(1/2))

n=(w * C1)**2

d=(w * C1 * (R1 + R2))**2

nu = 1/2

equation = (1+n) / (1+d)

print(f"H(w) is: \n {equation} \n ")

print(f"H(w) should be around: \n {nu} \n ")
```

Output:

```
H(w) is:
 0.5001745314931291

H(w) should be around:
 0.5
```

Figure.3: Python Code for components

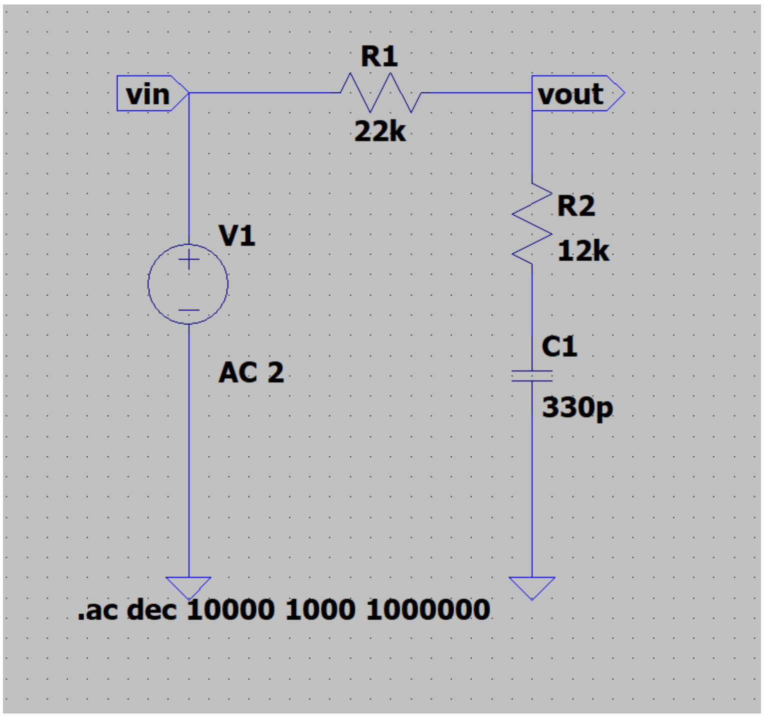

Figure.4: LTspice schematics for RC circuit.

At the end of the day, values found by the code were not used. However, the code gave strong insight to design a RC circuit.

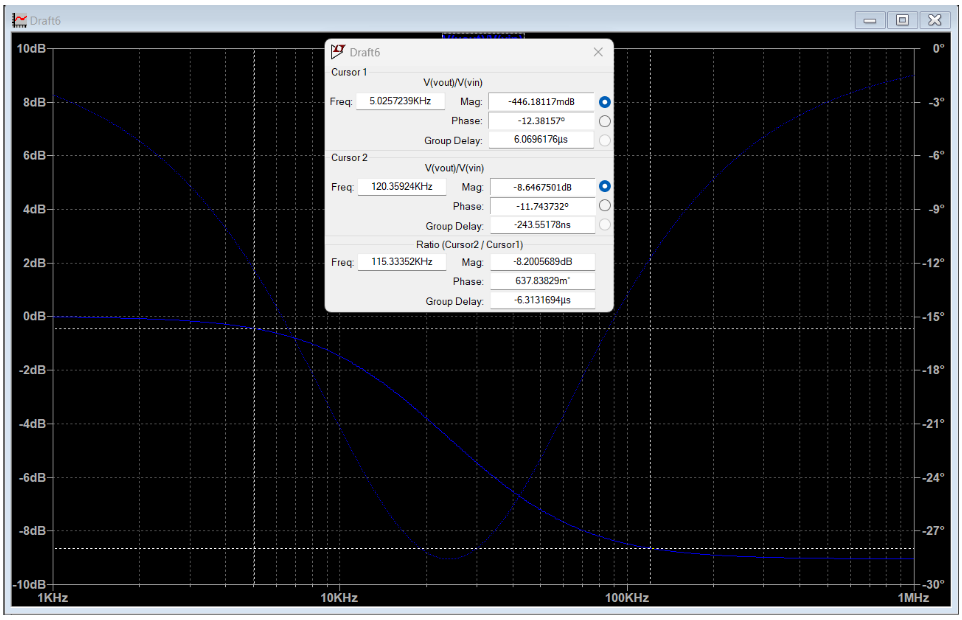

Figure.5: LTspice simulation for RC circuit

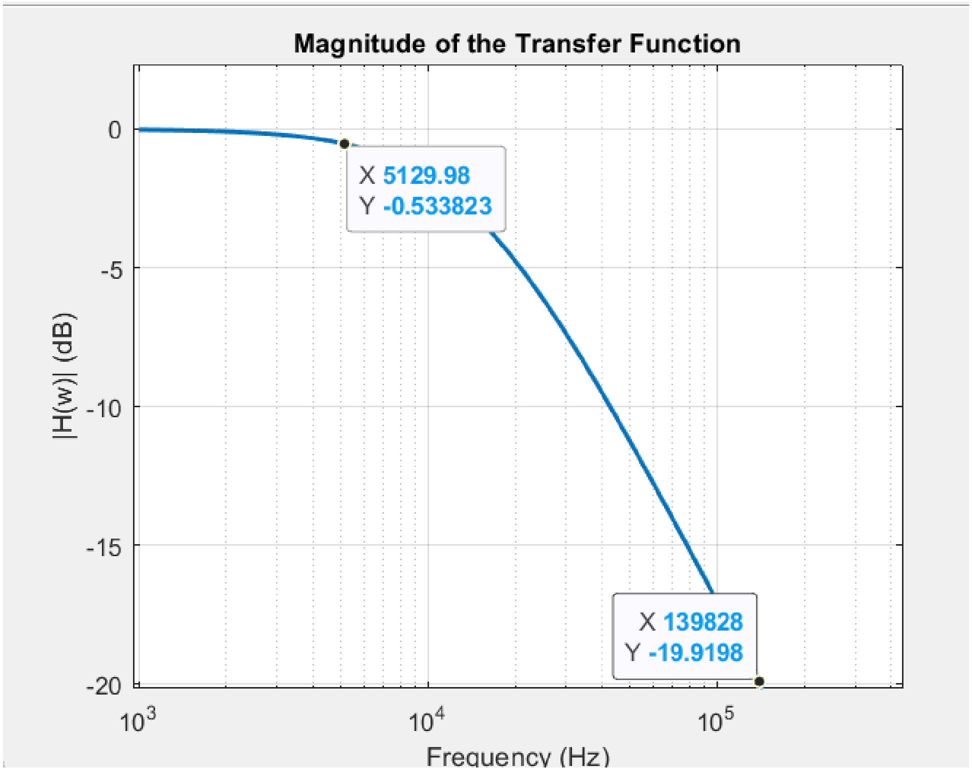

Figure.6: MATLAB simulation for RC circuit

```
% Define constants
C1 = 330e-12; % Farads
R1 = 12000;    % Ohms
R2 = 22000;   % Ohms

% Frequency range (logarithmic scale)
f = logspace(3, log10(150e3), 1000); % From 1000 Hz to 150 kHz
w = 2 * pi * f; % Angular frequency

% Transfer function calculation
numerator = sqrt(1 + (w .* C1).^2); % Numerator of the given formula
denominator = sqrt(1 + (w .* C1 .* (R1 + R2)).^2); % Denominator of the given formula
H = numerator ./ denominator; % Transfer function

% Magnitude of H(w)
H_abs = abs(H); % Calculate the magnitude

% Plotting
figure;
semilogx(f, 20 * log10(H_abs), 'LineWidth', 1.5);
grid on;
xlabel('Frequency (Hz)');
ylabel('|H(w)| (dB)');
title('Magnitude of the Transfer Function');
```

Figure.7: MATLAB code for LPF

B- Adding another RC circuit

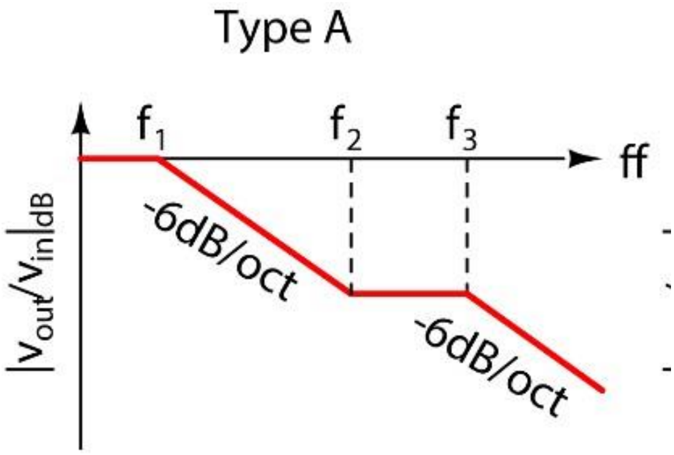

Figure.8: Final RC circuit

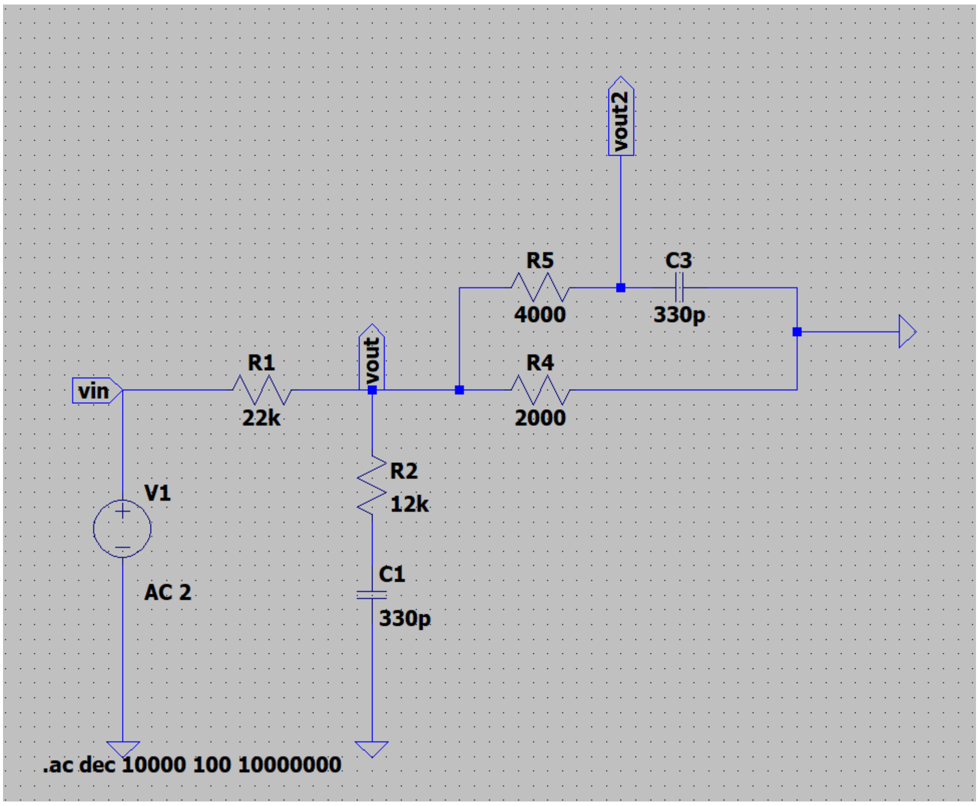

Figure.9: LTspice schematics of Final RC circuit

Transfer Function:

$$\frac{(1+\ jwC_1)}{(1+\ jwC_1(R_1+R_2))}\frac{1}{(1+jwC_2R_3)}$$

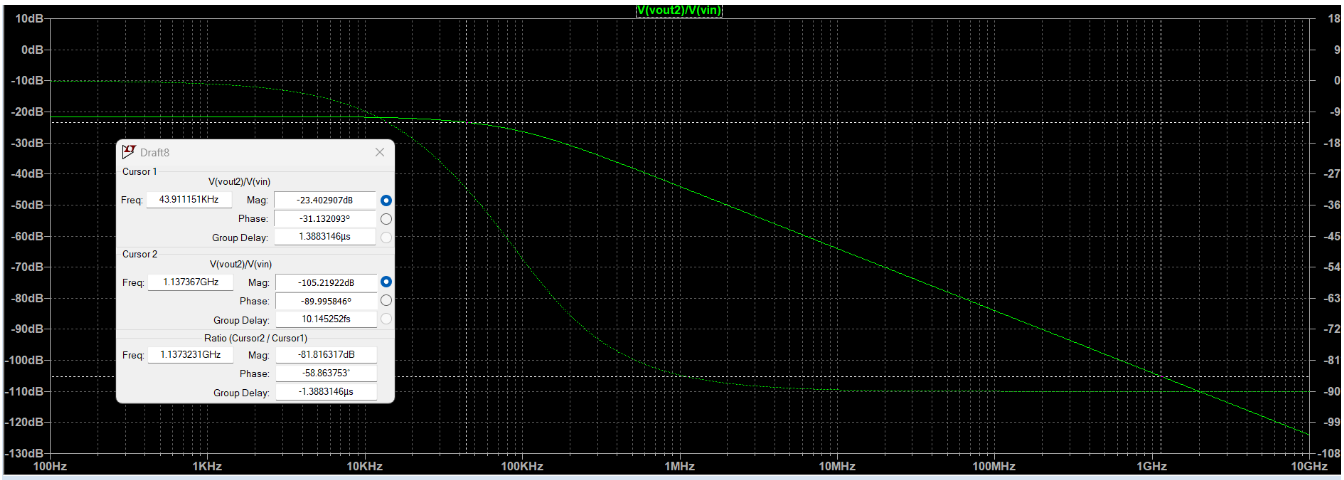

Figure.10: LTspice simulation of Final RC circuit, unsuccesfull

3. Conclusion

The first stage of the lab, focused on RC circuit design, was generally successful. We used Python code, as in the previous lab, to guide our component selection and gain insights into the circuit behavior. This understanding was further enhanced by deriving mathematical equations and using MATLAB for analysis. However, the second stage, where we added an additional RC stage, was challenging, and we faced difficulties that led to an unsuccessful outcome. Despite these challenges, the process provided valuable learning experiences in filter design and practical implementation.