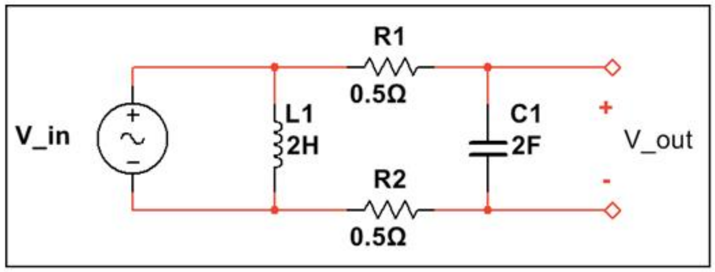
Silvia López de Diego TUid: 914476268
EE Science II
01/03/2013

Reworked Exam

1(a). Compute the Laplace transfer function for the network shown to the right.



In order to compute the transfer function of the network, the circuit was redrawn in the s domain. Figure 1 shows the diagram of the mentioned circuit.

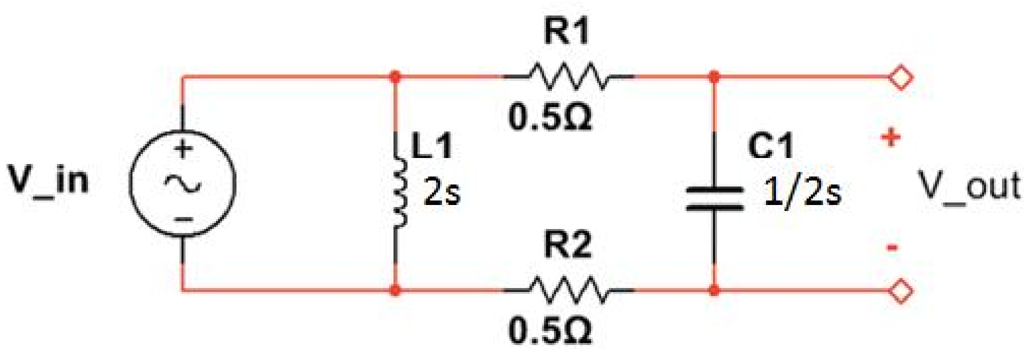


Figure 1

It can be seen in the figure that the elements of the circuit were written in their s domain representations: the capacitor was expressed as $\frac{1}{sC}$ and the inductor as sL.

If voltage division is applied across the Laplace circuit, the following expression is obtained:

$$V_{out} = V_{in}\left[\frac{\frac{1}{2s}}{1+\frac{1}{2s}}\right] \ldots\ldots (1)$$

This works because of the fact that the voltage across the inductor is the same as V_{in}, and the voltage across c_1 is the same as V_{out} (they are connected in parallel).

Since the transfer function of a network is by definition the relation between the input and the output of a linear time-invariant system, it can be calculated by solving for $\frac{V_{out}}{V_{in}}$, which according to equation (1), is given by:

$\frac{V_{out}}{V_{in}} = \frac{2s}{4s^2+2s}$ or $\frac{V_{out}}{V_{in}} = \frac{s}{2s^2+s}$ this transfer function can be simplified as:

$$\frac{V_{out}}{V_{in}} = \frac{1}{2s+1}$$

The analyzed circuit was implemented in Multisim and its frequency response was analyzed and compared to the frequency response of the transfer function plotted in MATLAB. Figure 2 shows the frequency response on a logarithmic scale of the circuit in Multisim. This was calculated through an AC analysis.

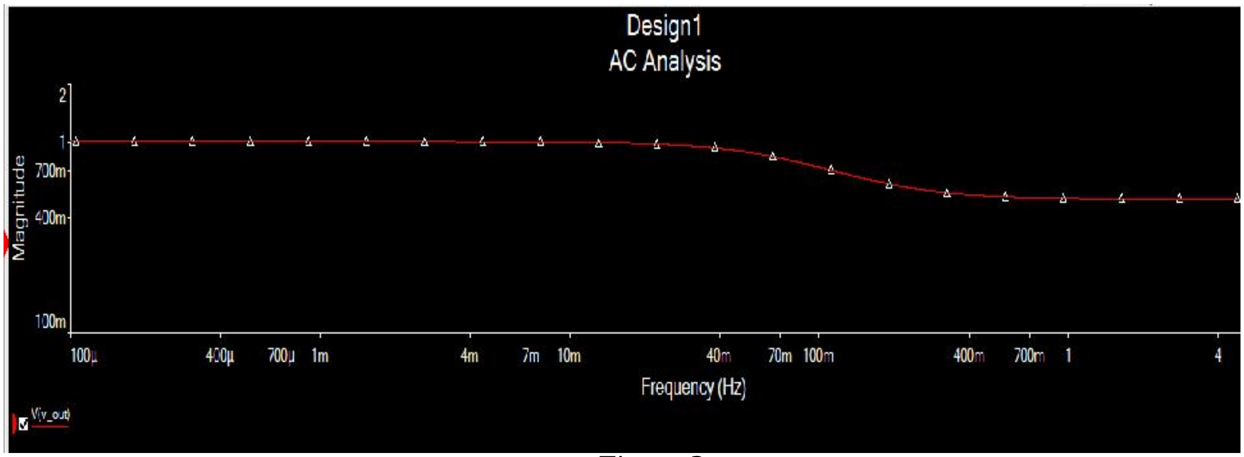


Figure 2

Similarly, the transfer function that was solved for the first part of the problem was plotted in MATLAB with the bode function (log scale). The absolute value of the magnitude was plotted as a function of the frequency of the circuit in Hz. Figure 3 shows the plot generated in MATLAB.

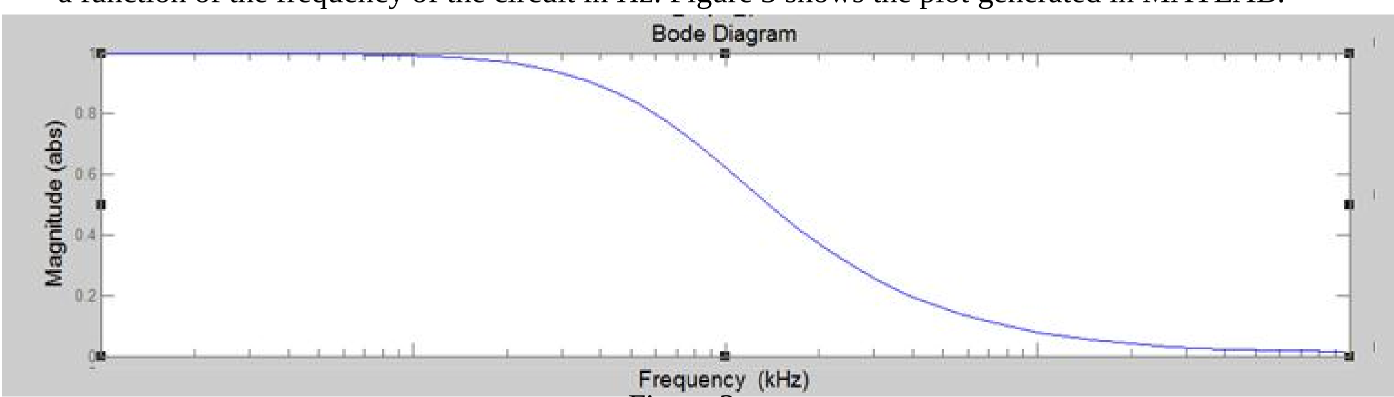


Figure 3

The commands used to graph the transfer function are shown in Figure 4. Similarly, the simulation

```
Command Window
>> tf([1],[2 1])

ans =

      1
  -------
  2 s + 1

Continuous-time transfer function.

>> bode(ans)
fx >>
```

Figure 4

1(b). Sketch the Bode plot (log magnitude as a function of log frequency). You can use MATLAB for this. However, either way, ~~explain~~ why this plot makes sense. An explanation is required to get maximum credit.

The bode plot of the transfer function was also plotted with the Magnitude in decibels instead. Figure 5 shows the bode plot of the transfer function as Magnitude (decibels) as a function of frequency (KHz).

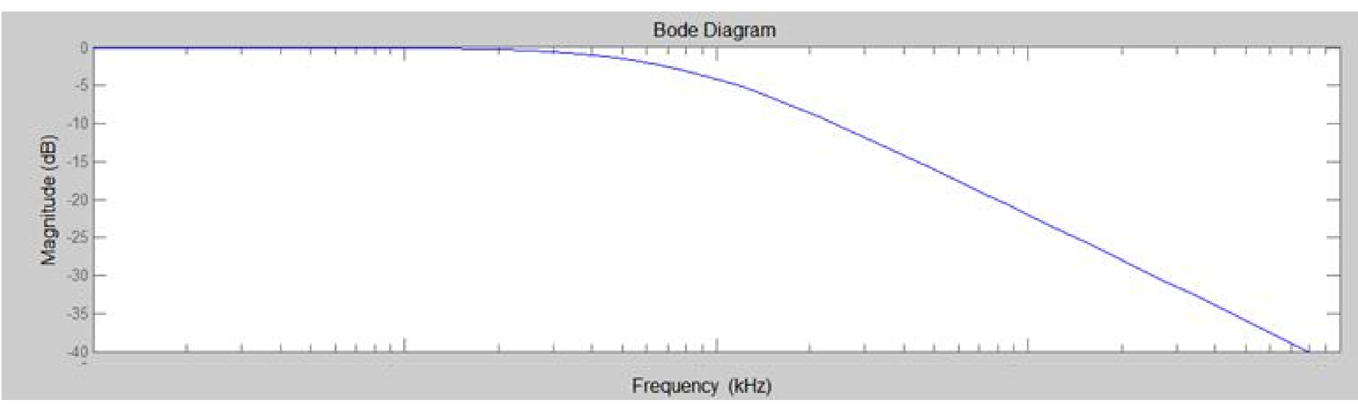


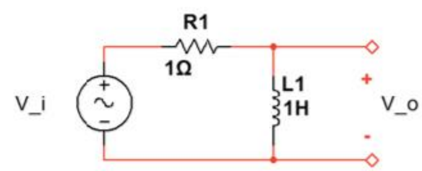
Figure 5

The plot shows that as the frequency increases, the magnitude, which is directly proportional to the voltage, decreases. The circuit works as a filter that attenuates the magnitude as the frequency increases. In this case, the plot suggests that the circuit is a first order low pass filter, because the transfer function is not affected by the inductor connected in parallel. The higher the

AC frequency in the input, the less reactance the capacitor has against the current, while lower AC frequencies result in more reactance. Basically, if the frequency is low, the capacitor tends to treat it as a DC signal, which increases the possibility that the signal will be blocked from ground and shunted to the output signal. Basically, the capacitor exhibits reactance, and blocks low frequency signals, causing them to go through the load instead. At higher frequencies, the reactance drops and the capacitor effectively functions as a short circuit.

With low frequencies, the capacitor has the time to charge up to the same voltage as the input signal. When the input signal wanes, the capacitor discharges its stored voltage to the output signal. However, with high frequencies, the capacitor does not have time to charge up to the same voltage as the input signal. What this means then, is that when the input signal wanes, the capacitor's discharged voltage is significantly decreased as compared to the original input signal.

2. Consider the circuit shown to the right. Compute the output voltage, $v_o(t)$ assuming $v_i(t) = u(t-1)-u(t-2)$. (Hint: sketch $v_i(t)$.)



As in the first problem, the first step to find the output voltage is drawing the circuit in the s domain. Figure 6 shows the diagram of the mentioned circuit.

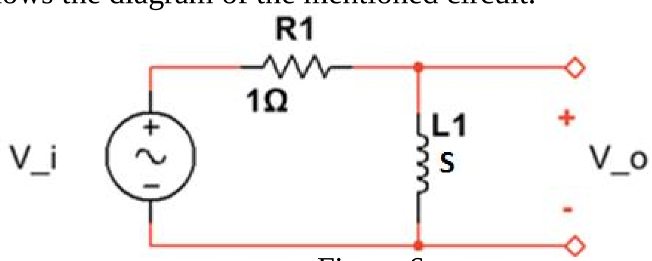


Figure 6

In this circuit the input is defined by a Heaviside function, so the input, given by

$V_i = u(t-1) - u(t-2)$ can be converted to the Laplace transform as follows:

$$L\{u(t-1) - u(t-2)\} = \frac{e^{-s} - e^{-2s}}{s}$$

The voltage of the circuit can be calculated through the voltage division rule. In this sense, V_o is given by the following expression:

$V_o = \frac{s}{s+1} . V_{in}$ or $V_o = \frac{s}{s+1} . \frac{e^{-s}-e^{-2s}}{s}$ …………………………………………………….(1)

the expression can be simplified as $V_o = \frac{e^{-s}-e^{-2s}}{s+1}$ or $V_o = \frac{e^{-s}}{s+1} - \frac{e^{-2s}}{s+1}$.

If the inverse Laplace transform is applied, the function can be written as follows:

$$L^{-1}\left\{\frac{e^{-s}}{s+1}-\frac{e^{-2s}}{s+1}\right\} = u(t-1)e^{1-t}-u(t-2)e^{2-t}=V_o$$

$u(t-1)e^{1-t}-u(t-2)e^{2-t}=V_o$……………………………………………..(2)

This expression was calculated in MATLAB in order to corroborate the inverse Laplace function. Figure 7 shows that the inverse Laplace function calculated matched the function calculated in MATLAB.

```
>> F

F =

(exp(-s) - exp(-2*s))/(s + 1)

>> ilaplace(F,s,t)

ans =

heaviside(t - 1)*exp(1 - t) - heaviside(t - 2)*exp(2 - t)
```

Figure 7

The function obtained was also plotted in MATLAB to compare the graph with a transient analysis that was run in Multisim. Figure 8 shows the resulting plot in MATLAB.

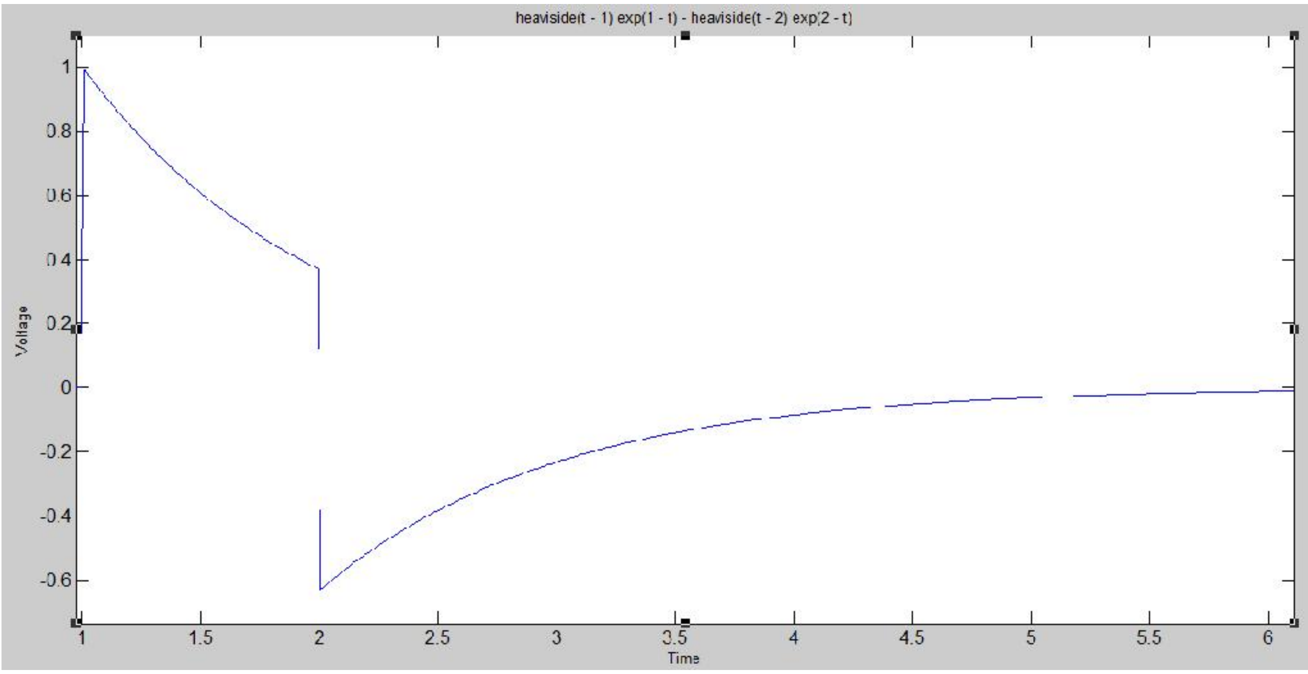


Figure 8

As it was said before, the circuit was implemented in Multisim with two step voltage sources that simulated the Heaviside functions that defined V_{in} of the network. Figure 9 shows the diagram of the circuit created in Multisim. On the other hand, Figure 10 shows the transient analysis that was run in order to analyze the output voltage of the circuit as a function of time.

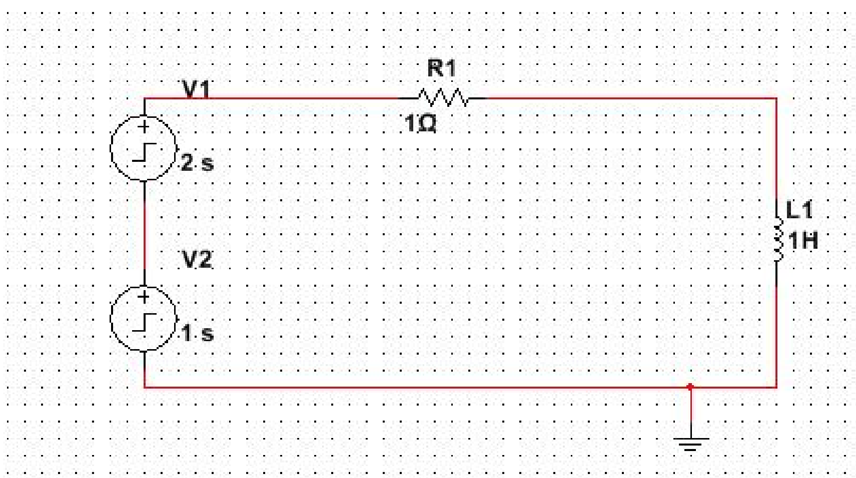


Figure 9

It can be seen that two step sources were used, because it was necessary to simulate a source that was driven by two different Heaviside functions with different arguments.

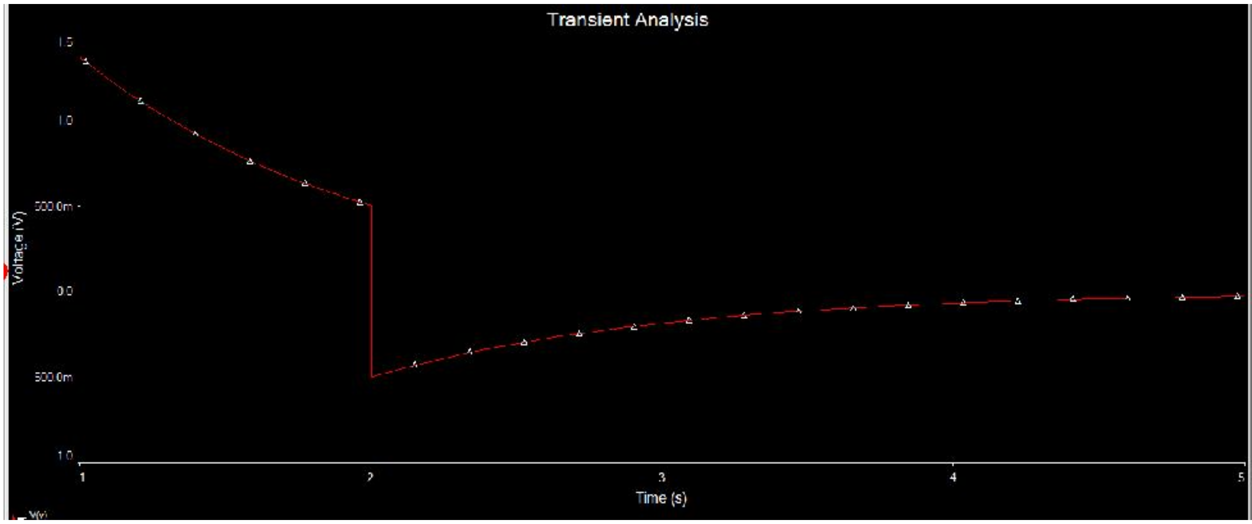
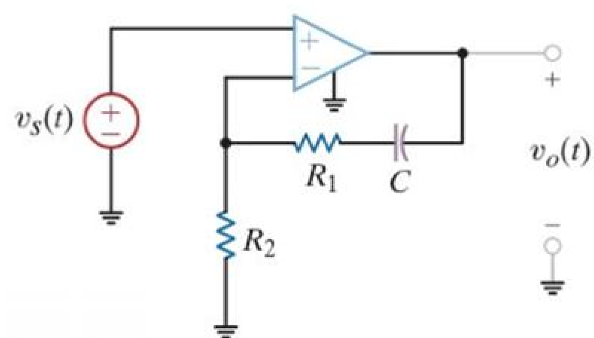


Figure 10

It can be seen that, at first the Heaviside function is high, charging the inductor. Within this charging process, there is a function that determines the output of the voltage (Equation (2)) that is a decaying exponential function. For this reason, the graph is decaying exponentially from 1 to 2. Then, the Heaviside function is low and the voltage, which goes negative in the first part of the cycle, starts approaching 0 that becomes a horizontal asymptote.

Figures 10 and 8 show that the transient response of the network calculated in Multisim matches the graph plotted in MATLAB and the analytical calculations.

3. Compute the Laplace transfer function of the circuit shown to the right. If you can't derive the expression, tell me as much as possible about the transfer function in a qualitative sense.



For this problem, it is important to recall that for an ideal operational amplifier, no current flows in or out of the device, and consequently $i_+ = i_- = 0$, and $v_+ = v_-$.

In this case, the voltage division rule can be used in order to determine V_{out}. To simplify the equation writing process, the impedance of the capacitor in the Laplace domain is determined by $\frac{1}{sC} = Z_C$. Figure 11 shows the diagram for the circuit in the Laplace domain.

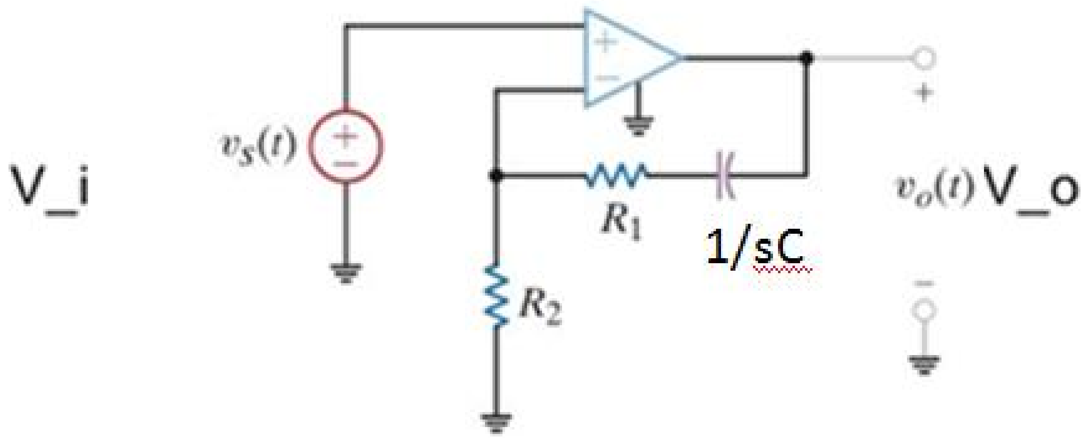


Figure 11

With the circuit in the Laplace domain, V_o can be calculated through voltage division, which gives the following expression:

$V_{in} = \frac{R_2}{R_2+R_1+Z_C} . V_o$ Solving for the transfer function of the circuit (V_o/V_{in}) the following equation is obtained:

$\frac{V_o}{V_{in}} = \frac{R_2+R_1+Z_C}{R_2}$ Separating the terms and simplifying the equation, it can be obtained that $\frac{V_o}{V_{in}} = 1 + \frac{R_1+\left(\frac{1}{sC}\right)}{R_2}$.

For this problem, values were assigned to the components and the transfer function of the circuit was calculated according to those particular values. The resulting function was then evaluated through a bode plot in MATLA and compared to an AC analysis simulated in Multisim. The values assigned to the components are the following:

$R_1 = 1\Omega, R_2 = 1\Omega, C = 1F$

In this sense, the expression for the transfer function found above becomes $\frac{V_o}{V_{in}} = 1 + \frac{1+\frac{1}{s}}{1}$ or $1 + \frac{s+1}{s}$. This expression can be simplified as $\frac{2s+1}{s}$.

Figure 12 shows the code used in order to generate this transfer function in MATLAB. Similarly, Figure 13 shows the bode plot generated in MATLAB in order to visualize better the transfer function of the circuit.

```
>> tf([2 1],[0 1])

ans =

  2 s + 1

Continuous-time transfer function.

>> tf([2 1],[1 0])

ans =

  2 s + 1
  -------
     s

Continuous-time transfer function.

>> bode(ans)
fx >>
```

Figure 12

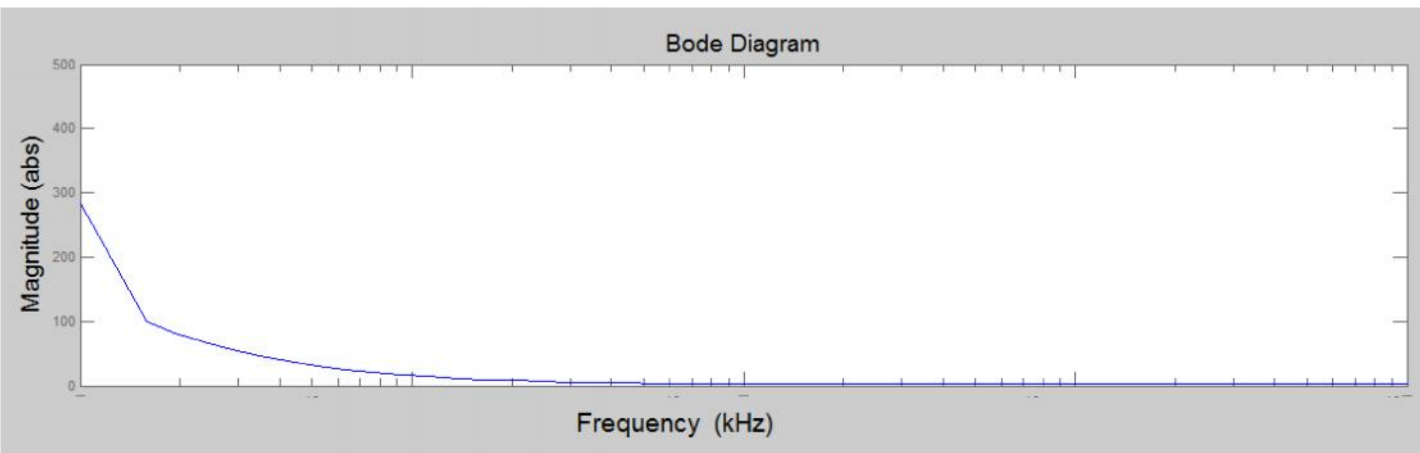


Figure 13

The circuit was also analyzed in Multisim. Figure 14 shows the diagram of the circuit that was simulated in Multisim, and Figure 15, shows the AC analysis that was run to plot the transfer function.

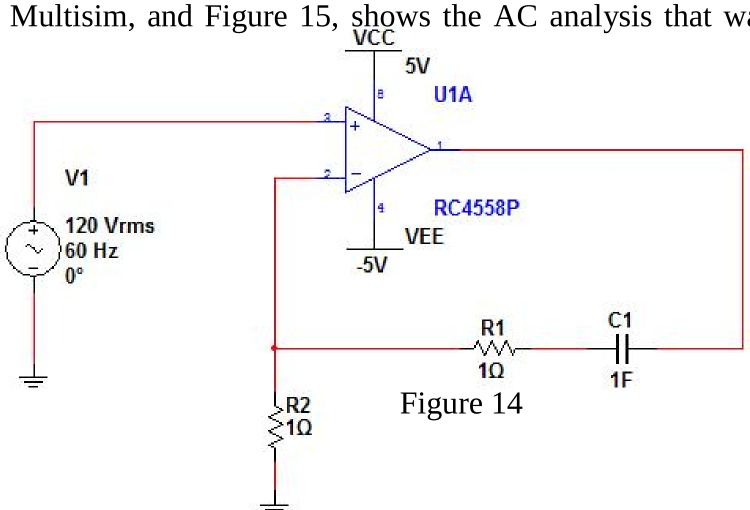


Figure 14

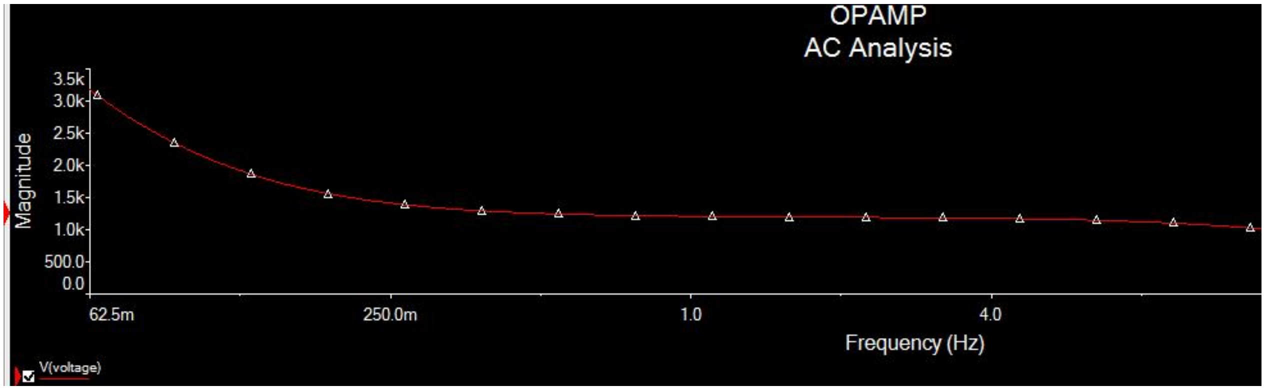


Figure 15

The picture show that the function is decaying as the frequency increases. If the circuit is treated as a DC network, the capacitor acts as an open circuit and therefore, the current does not pass through the branch of the circuit in which the capacitor is located, and therefore, the negative feedback is eliminated. Basically, when the frequency is low the gain of the opamp increases and tends to infinity, and when the frequency is high, the magnitude decreases and tends to zero.