

EXPERIMENT NO: 8(a)

Author : Gaurav Supal, Vineeta Parmar
Date : March 17, 2016

Aim of the Experiment:

Analysis of Low Pass Filter using eSim.

Theory:

A low-pass filter is a filter that passes signals with a frequency lower than a certain cutoff frequency and attenuates signals with frequencies higher than the cutoff frequency. The amount of attenuation for each frequency depends on the filter design.

A simple passive RC Low Pass Filter or LPF, can be easily made by connecting together in series a single Resistor with a single Capacitor as shown below. In this type of filter arrangement the input signal (V_{in}) is applied to the series combination (both the Resistor and Capacitor together) but the output signal (V_{out}) is taken across the capacitor only.

Procedure:

1. Create the schematic of the Low Pass Filter as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for AC analysis from start frequency 1Hz to stop frequency 1MHz with 20 points in Decade mode.
6. Insert Source Details.
7. Convert KiCad netlist to Ngspice netlist.
8. Simulate the Ngspice netlist using Ngspice simulator.

Source Parameters:

For AC Voltage Source (V1):

1. Enter Amplitude Value - 10
2. Enter Phase Value - 0

Schematic Diagram:

The circuit schematic of Low pass filter register in eSim is as shown below:

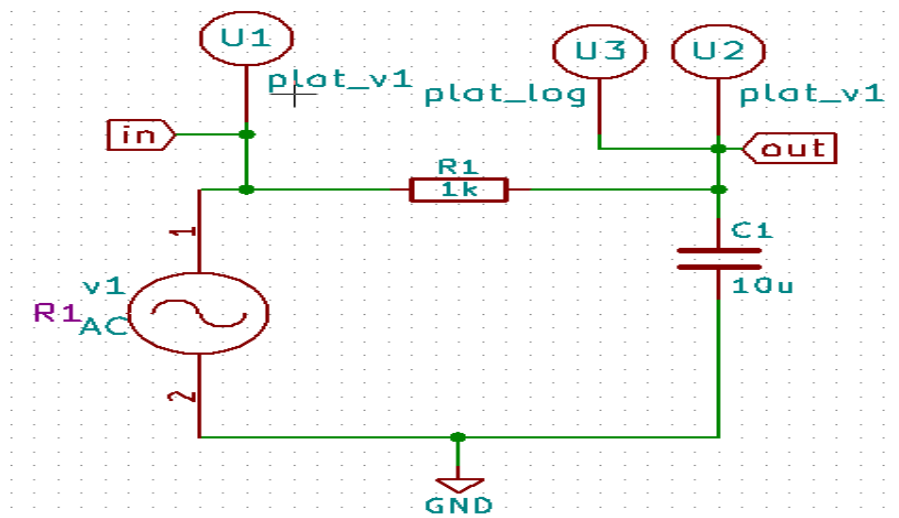


Figure 1: Low Pass Filter

Simulation Results:

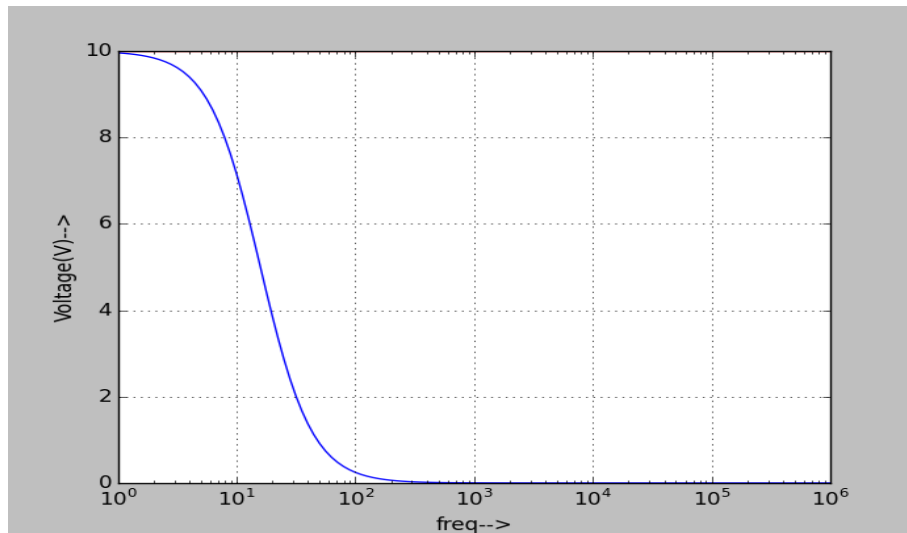


Figure 2: Python Plot

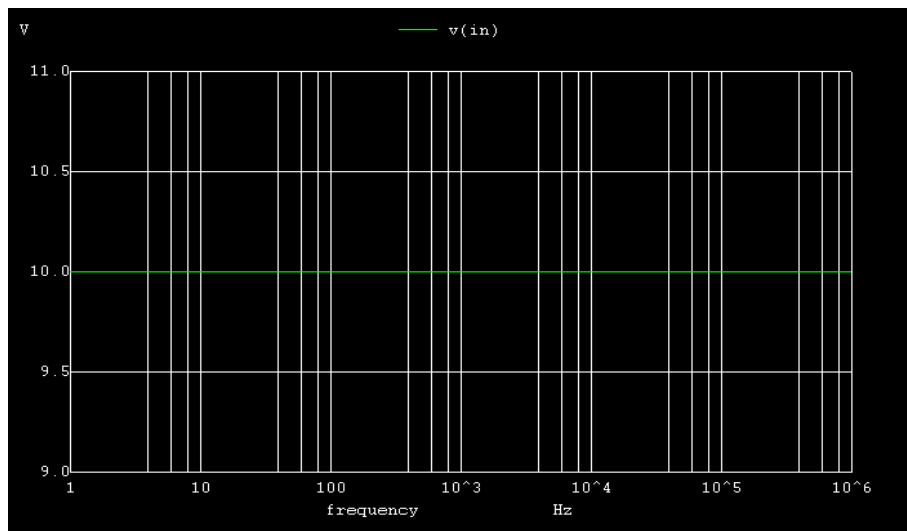


Figure 3: Ngspice Input Plot

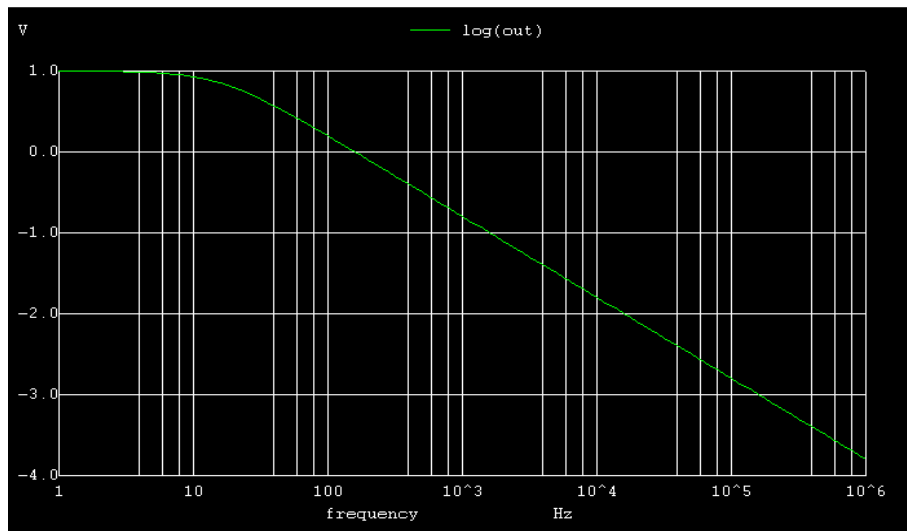


Figure 4: Ngspice Output Plot

Conclusion:

Thus, we have studied the low pass filter using eSim and we get the appropriate waveforms.

References:

<http://www.electronics-tutorials.ws/filter>

EXPERIMENT NO: 8(b)

Aim of the Experiment:

Analysis of High Pass Filter using eSim.

Theory:

A high-pass filter is a filter that passes signals with a frequency higher than a certain cutoff frequency and attenuates signals with frequencies lower than the cutoff frequency. The amount of attenuation for each frequency depends on the filter design.

A High Pass Filter or HPF, is the exact opposite to that of the previously seen Low Pass filter circuit, as now the two components have been interchanged with the output signal (V_{out}) being taken from across the resistor.

Procedure:

1. Create the schematic of the High Pass Filter as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for AC analysis from start frequency 1Hz to stop frequency 1MHz with 20 points in Decade mode.
6. Insert Source Details.
7. Convert KiCad netlist to Ngspice netlist.
8. Simulate the Ngspice netlist using Ngspice simulator.

Source Parameters:

For AC Voltage Source (V1):

1. Enter Amplitude Value - 10
2. Enter Phase Value - 0

Schematic Diagram:

The circuit schematic of high pass filter register in eSim is as shown below:

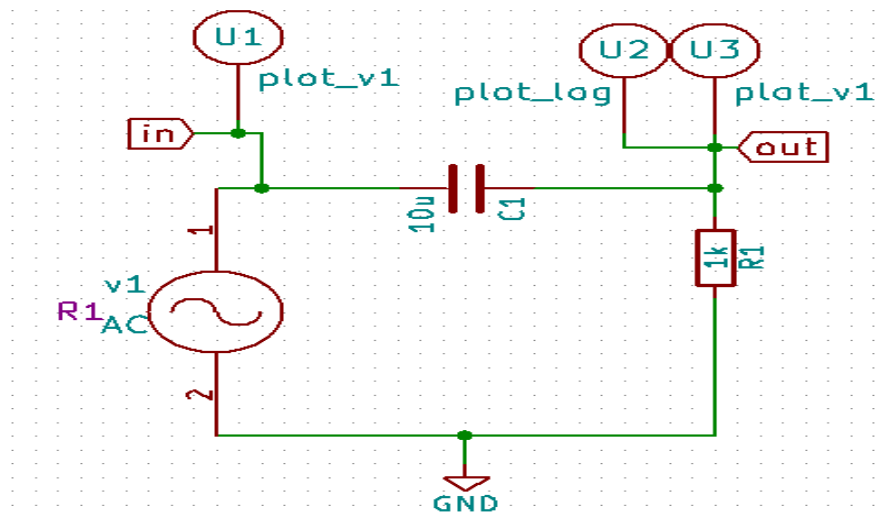


Figure 1: High Pass Filter

Simulation Results:

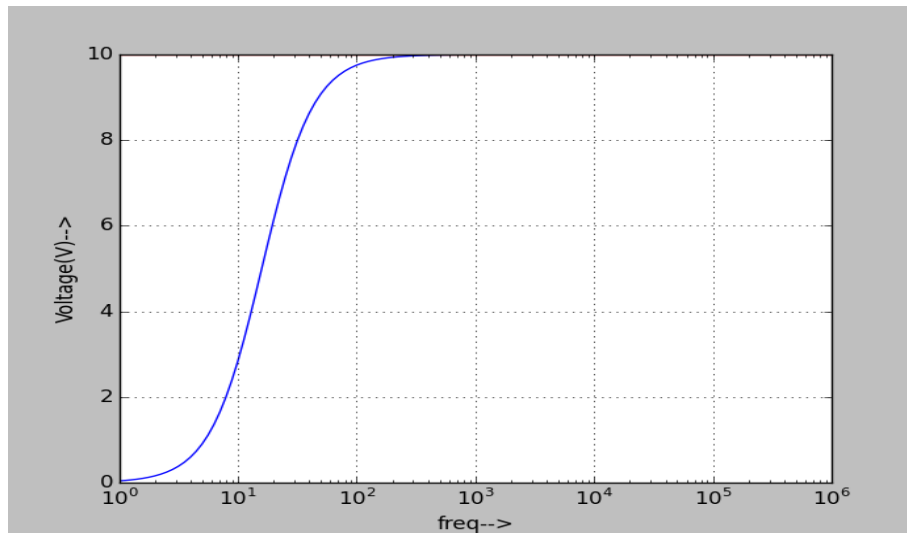


Figure 2: Python Plot

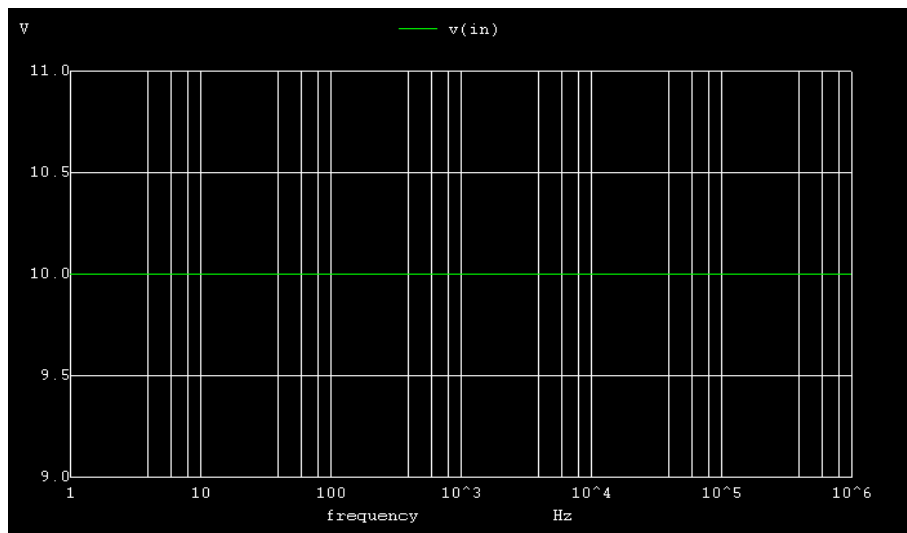


Figure 3: Ngspice Input Plot

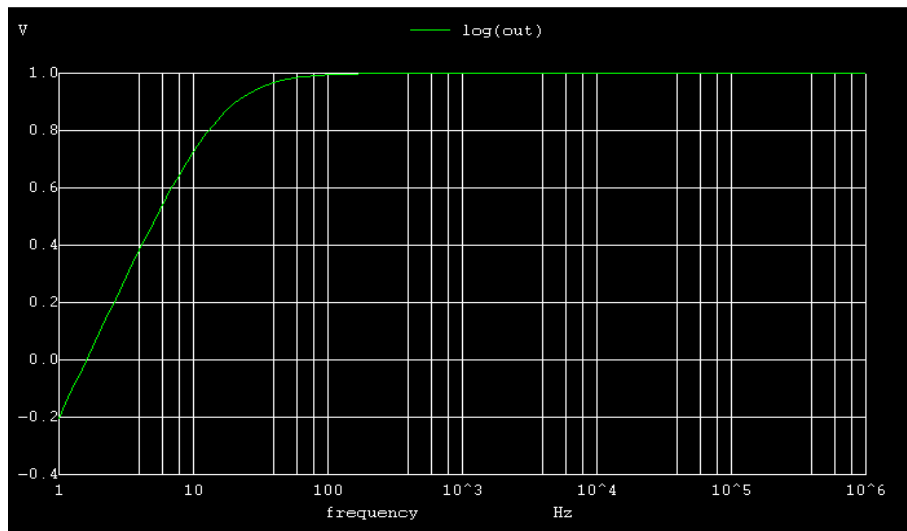


Figure 4: Ngspice Output Plot

Conclusion:

Thus, we have studied the high pass filter using eSim and we get the appropriate waveforms.

References:

<http://www.electronics-tutorials.ws/filter>