EXPERIMENT NO: 6(a)

Author : Gaurav Supal, Vineeta Parmar
Date : March 7, 2016

Aim of the Experiment:

Analysis of Inverting amplifier using eSim.

Theory:

In the Inverting Amplifier circuit the operational amplifier is connected with feedback to produce a closed loop operation. he junction of the input and feedback signal (X) is at the same potential as the positive (+) input which is at zero volts or ground then, the junction is a Virtual Earth. Because of this virtual earth node the input resistance of the amplifier is equal to the value of the input resistor, Rin and the closed loop gain of the inverting amplifier can be set by the ratio of the two external resistors.

An ideal voltage output for the op-amp inverting amplifier is given as:

$$\text{Vout} = -\left[\frac{Rf}{Rin}\right] \text{Vin}$$

Procedure:

1. Create the schematic of the Inverting Amplifier as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for transient analysis from 0 to 100 ms with a step time of 10 ms.
6. Insert Source Details.
7. Add ua741.sub Subcircuit file in Subcircuiut Tab for Op-Amp.
8. Convert KiCad netlist to Ngspice netlist.
9. Simulate the Ngspice netlist using Ngspice simulator.

Source Parameters:

Following are the input sine wave parameters:

1. Enter Offset Value- 0
2. Enter Amplitude - 2
3. Enter Frequency- 50
4. Enter Delay Time- 0
5. Enter Damping Factor- 0

Schematic Diagram:

The circuit schematic of Inverting amplifier in eSim is as shown below:

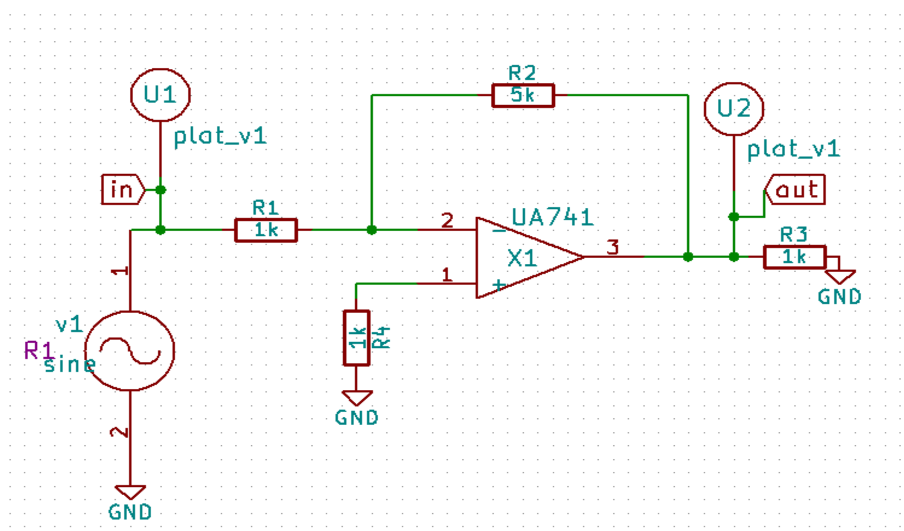

Figure 1: Inverting amplifier

Simulation Results:

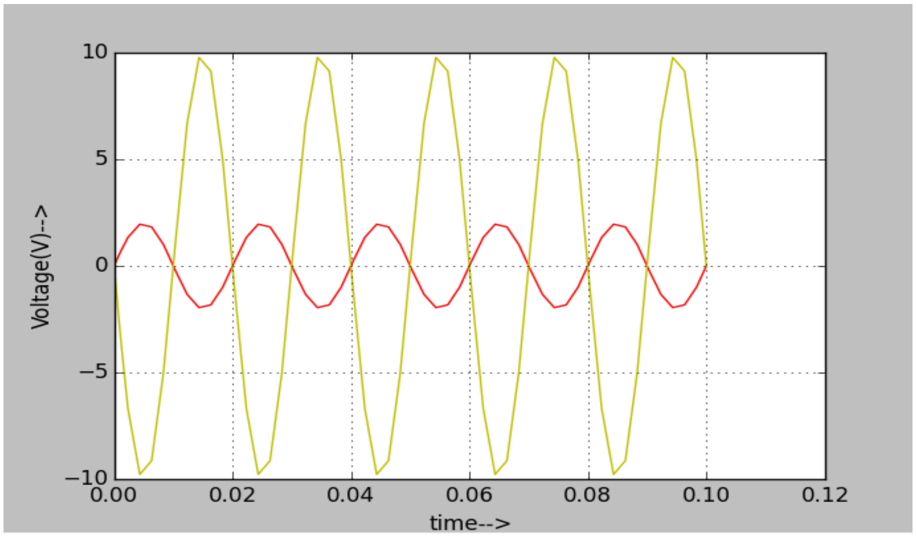

Figure 2: Python Plot Input and Output

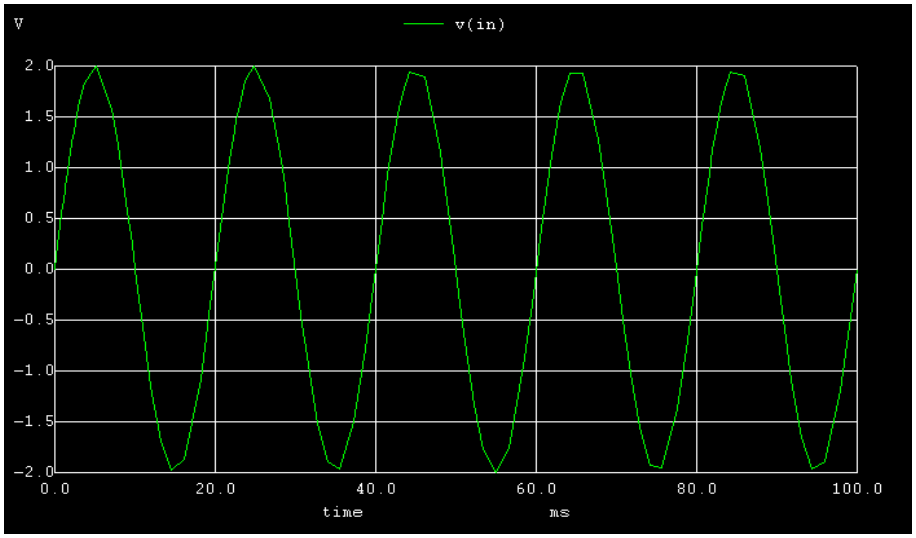

Figure 3: Ngspice Input Plot

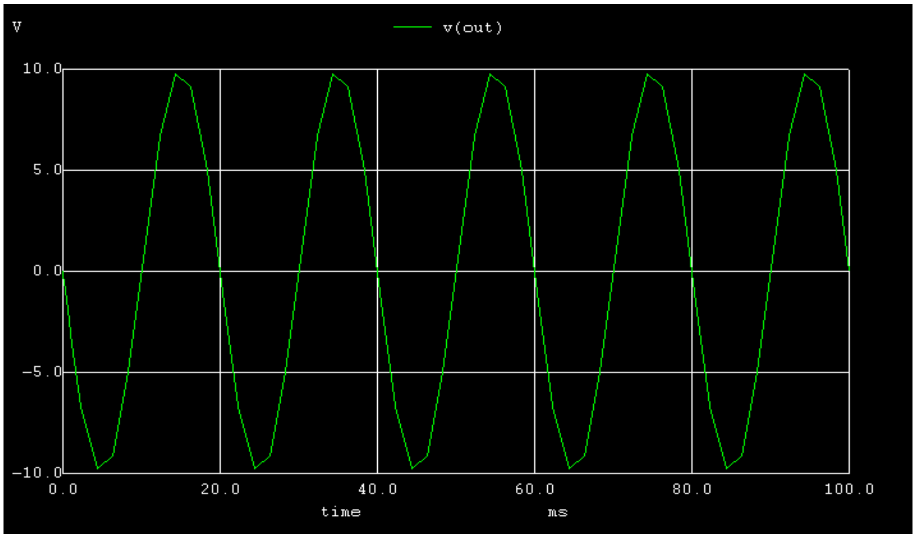

Figure 4: Ngspice Output Plot

Conclusion:

Thus, we have studied the Inverting amplifier using eSim and we get the appropriate waveforms.

References:

http://www.electronics-tutorials.ws/opamp

EXPERIMENT NO: 6(b)

Aim of the Experiment:

Analysis of Integrator using eSim.

Theory:

As its name implies, the Op-amp Integrator is an Operational Amplifier circuit that performs the mathematical operation of Integration, that is we can cause the output to respond to changes in the input voltage over time as the op-amp integrator produces an output voltage which is proportional to the integral of the input voltage.

In other words the magnitude of the output signal is determined by the length of time a voltage is present at its input as the current through the feedback loop charges or discharges the capacitor as the required negative feedback occurs through the capacitor.

An ideal voltage output for the op-amp integrator is given as:

$$\text{Vout} = - \left[\tfrac{1}{j\omega RC}\right] \text{Vin}$$

Procedure:

1. Create the schematic of the Integrator as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for transient analysis from 0 to 100 ms with a step time of 10 ms.
6. Insert Source Details.
7. Add ua741.sub Subcircuit file in Subcircuiut Tab for Op-Amp.
8. Convert KiCad netlist to Ngspice netlist.
9. Simulate the Ngspice netlist using Ngspice simulator.

Source Parameters:

Following are the Pwl parameters:

1. Enter Value (t1 v1 t2 v2 ..) = 0m 0 0.5m 5 25m 5 25.5m -5 50m -5 50.5m 5 75m 5 75.5m -5 100m -5

Schematic Diagram:

The circuit schematic of Integrator in eSim is as shown below:

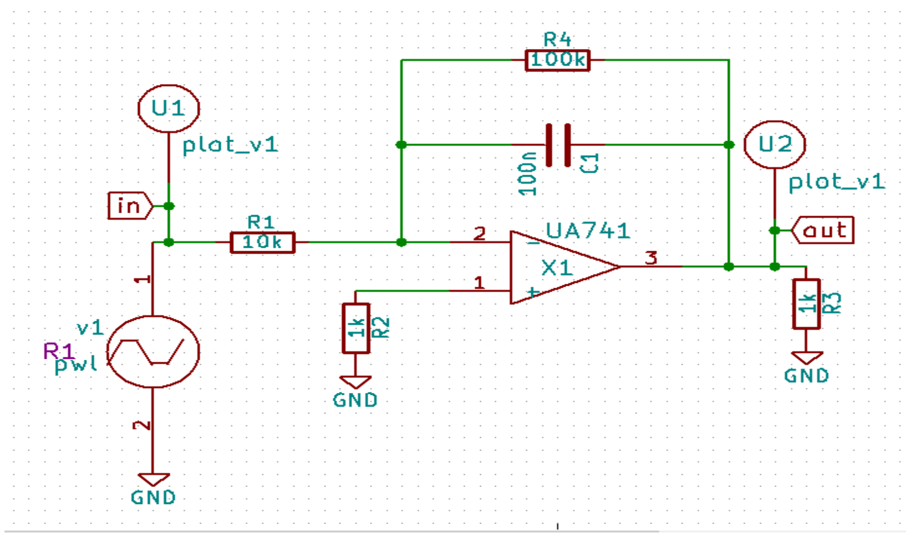

Figure 1: Integrator

Simulation Results:

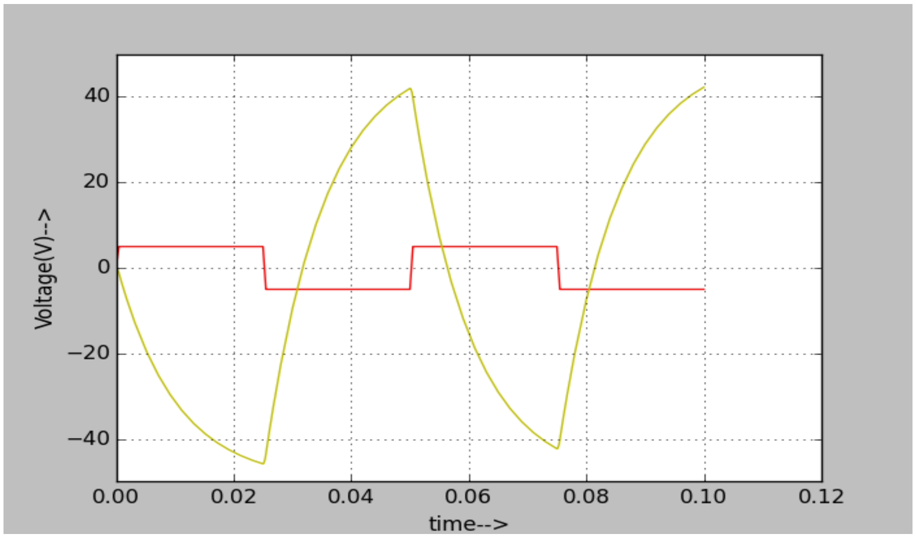

Figure 2: Python Plot

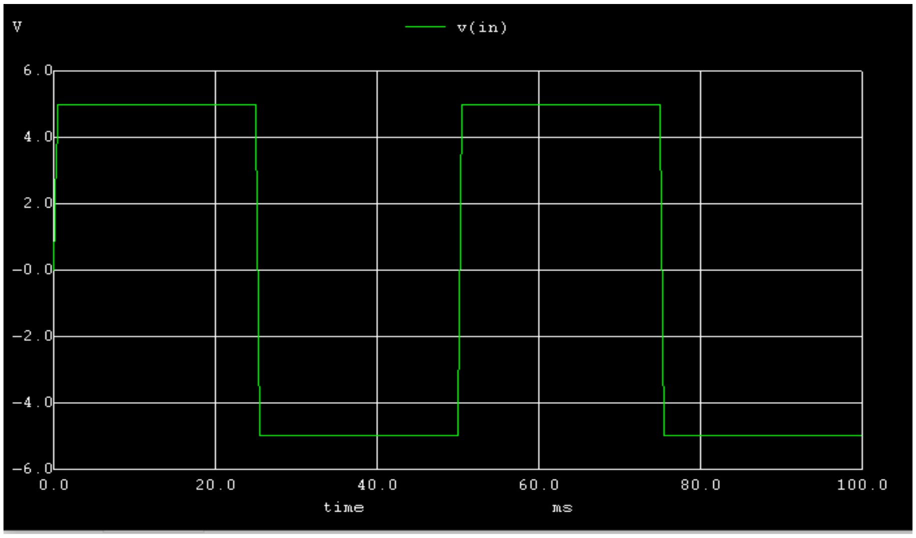

Figure 3: Ngspice Input Plot

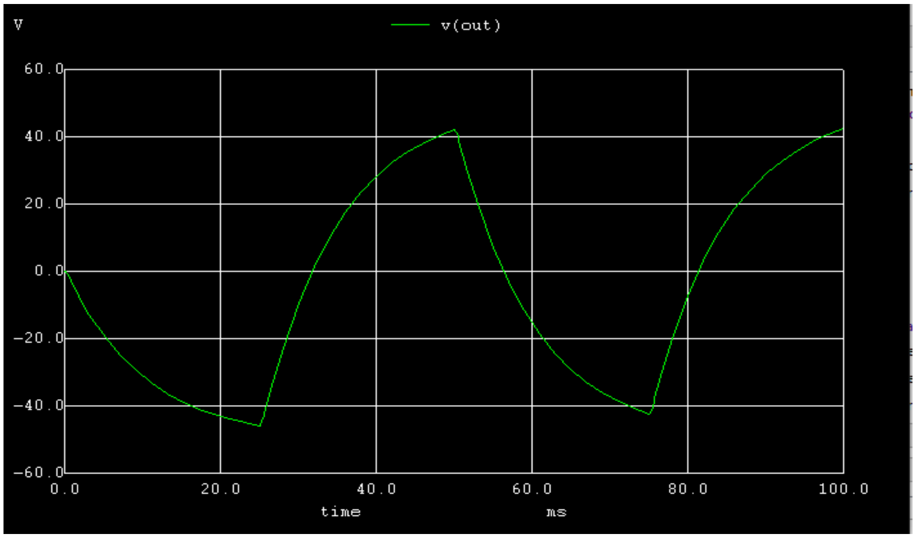

Figure 4: Ngspice Output Plot

Conclusion:

Thus, we have studied the Integrator using eSim and we get the appropriate waveforms.

References:

http://www.electronics-tutorials.ws/opamp

EXPERIMENT NO: 6(c)

Aim of the Experiment:

Analysis of Differentiator using eSim.

Theory:

This Operational Amplifier circuit performs the mathematical operation of Differentiation, that is it produces a voltage output which is directly proportional to the input voltages rate-of-change with respect to time. In other words the faster or larger the change to the input voltage signal, the greater the input current, the greater will be the output voltage change in response, becoming more of a spike in shape.

An ideal voltage output for the op-amp differentiator is given as:

$$\text{Vout} = \text{Rf*C}\ [\frac{dVin}{dt}]$$

Procedure:

1. Create the schematic of the Differentiator as shown in Figure-1.
2. Annotate the schematic.
3. Test Electric rules.
4. Generate the netlist.
5. Insert analysis for transient analysis from 0 to 100 ms with a step time of 10 ms.
6. Insert Source Details.
7. Add ua741.sub Subcircuit file in Subcircuiut Tab for Op-Amp.
8. Convert KiCad netlist to Ngspice netlist.
9. Simulate the Ngspice netlist using Ngspice simulator.

Source Parameters:

Following are the Pwl parameters:

1. Enter Value (t1 v1 t2 v2 ..) = 0m 0 0.5m 5 25m 5 25.5m -5 50m -5 50.5m 5 75m 5 75.5m -5 100m -5

Schematic Diagram:

The circuit schematic of differentiator in eSim is as shown below:

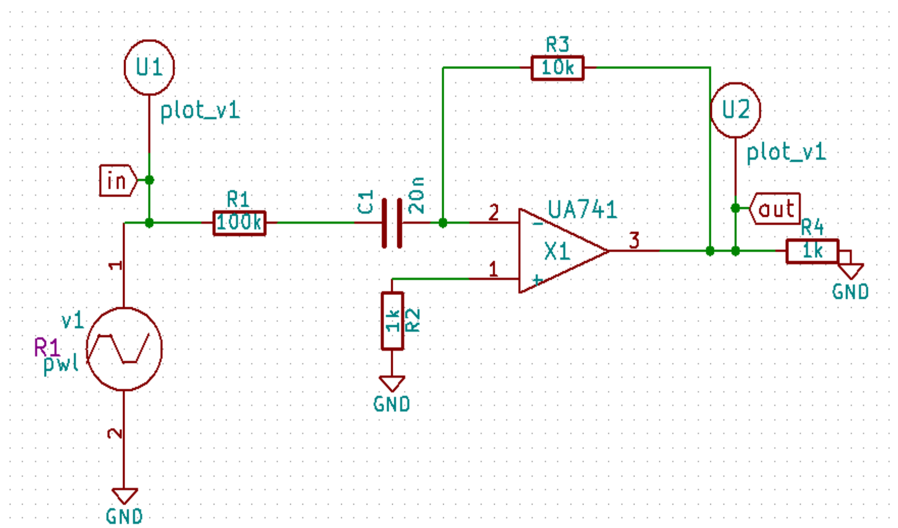

Figure 1: Differentiator

Simulation Results:

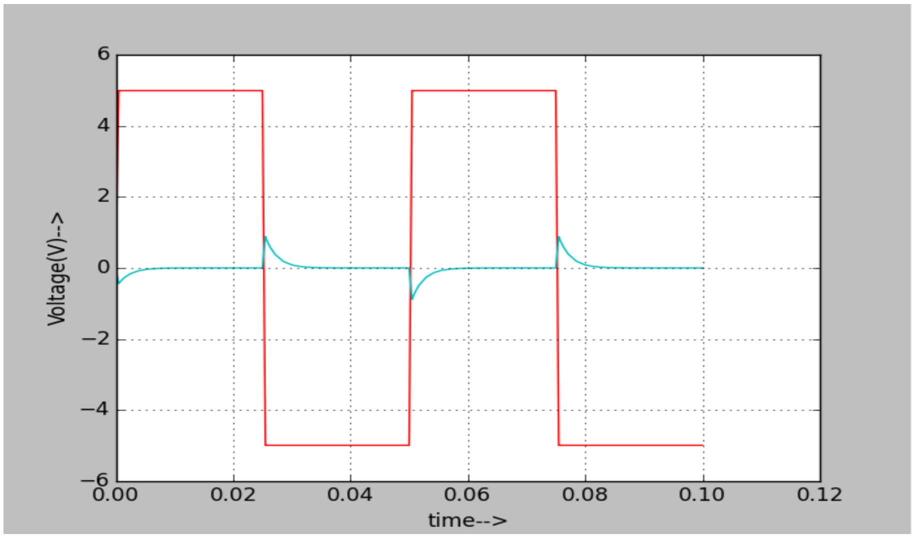

Figure 2: Python Plot

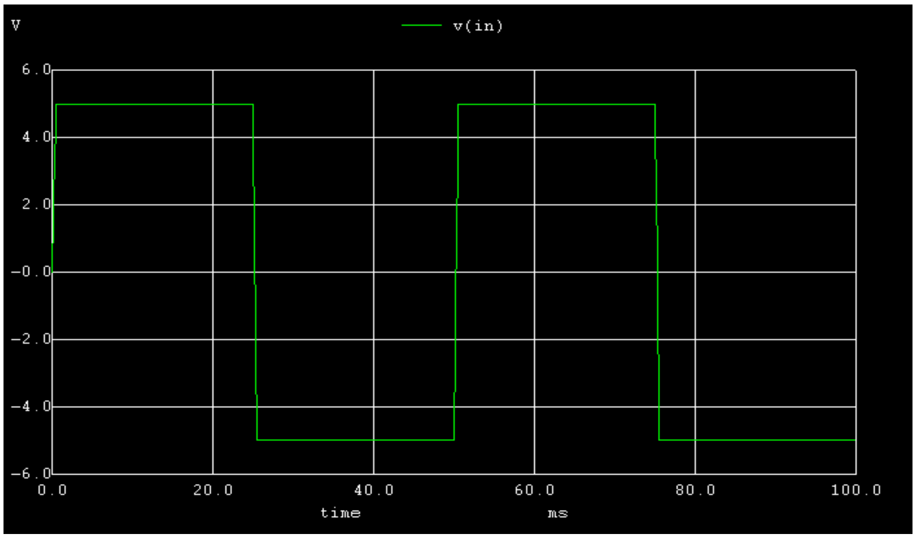

Figure 3: Ngspice Input Plot

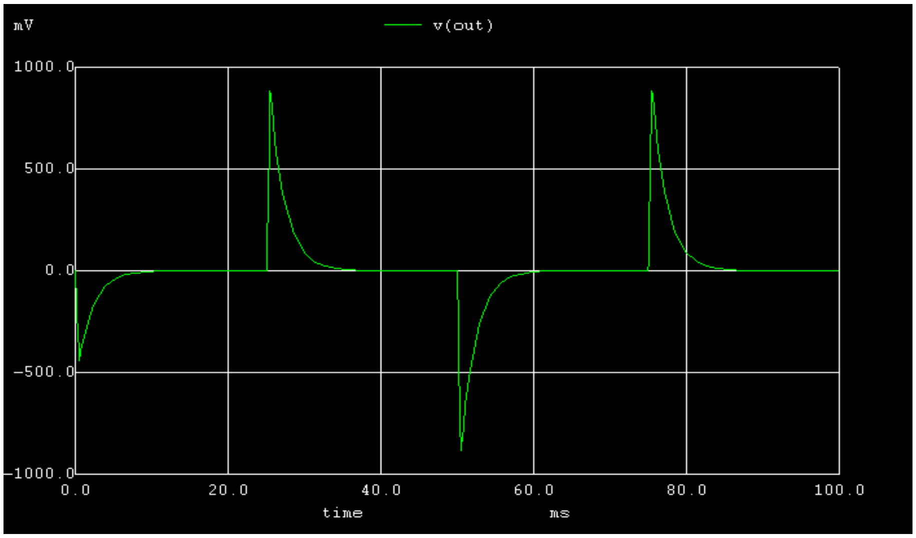

Figure 4: Ngspice Output Plot

Conclusion:

Thus, we have studied the Differentiator using eSim and we get the appropriate waveforms.

References:

http://www.electronics-tutorials.ws/opamp