EAST WEST UNIVERSITY

Course Name : Electronic Circuits

Course Code : CSE251

Section No :06

Group No :04

Experiment No : 01

Name of the Experiment : I-V Characteristics and Modeling of Forward Conduction of a Diode

Date of allocation :

Date of submission: 6-1-2020

Submitted To : Surajit Das Barman
Senior Lecturer

Student's Name : Apurba Roy
Student's ID :2018-3-60-063

Student's Name : Monjurul Alam
Student's ID :2018-3-60-035

1. To measure the I-V characteristics of forward conduction of a p-n junction diode.
2. To determine the models of forward conduction of a p-n junction diode.

Introduction:

Diode is one of the most basic non-linear electronic devices. An ideal diode acts like a switch for electric current, acting as a short circuit for current flow in one direction (forward bias connection) while behaving as an open circuit for current flow in the opposite direction (reverse bias connection). The characteristics of practical diodes are however somewhat different from those of ideal ones. The p-n junction diodes are one of the most popular types of diodes used in the industry. The forward bias current-voltage (I-V) characteristic of a p-n junction diode will be measured in this experiment.

Circuit Diagram:

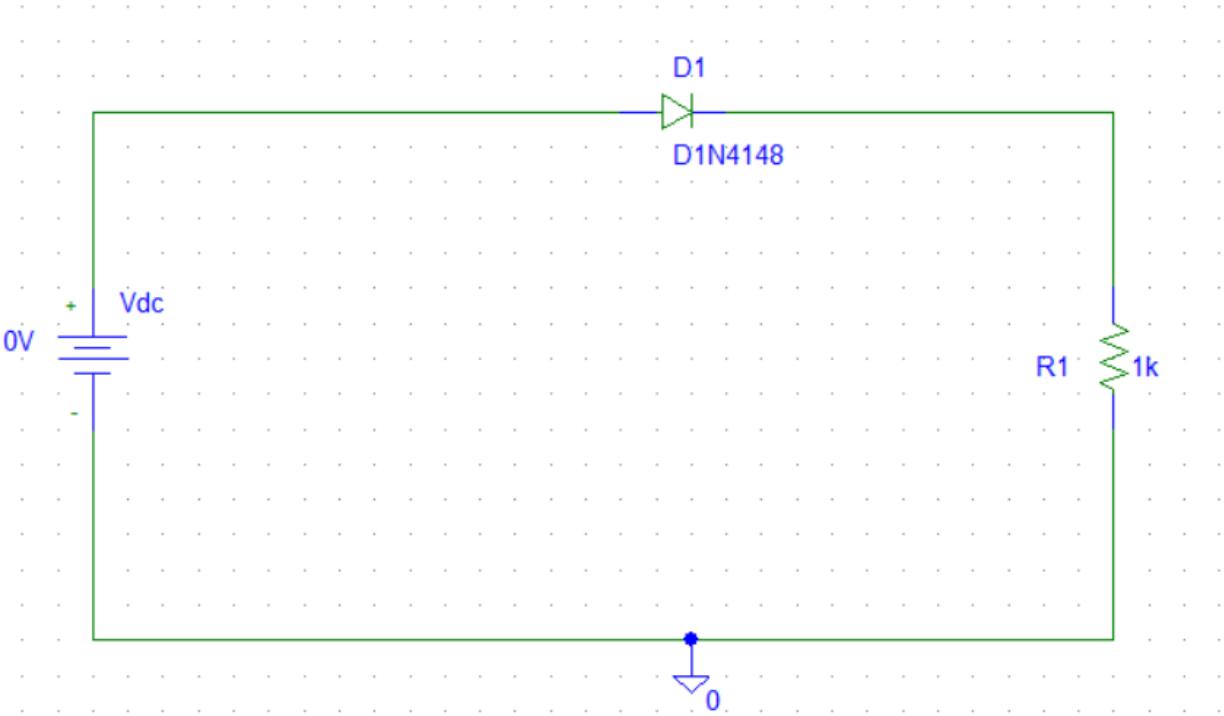


Figure 1. Circuit set up to measure forward bias I-V characteristics of a diode.

Apparatus:

- DC power supply
- Digital multimeter
- Diode (1 pc)
- Resistor 1KΩ
- Breadboard
- Connecting wires
- Matlab
- Pspice

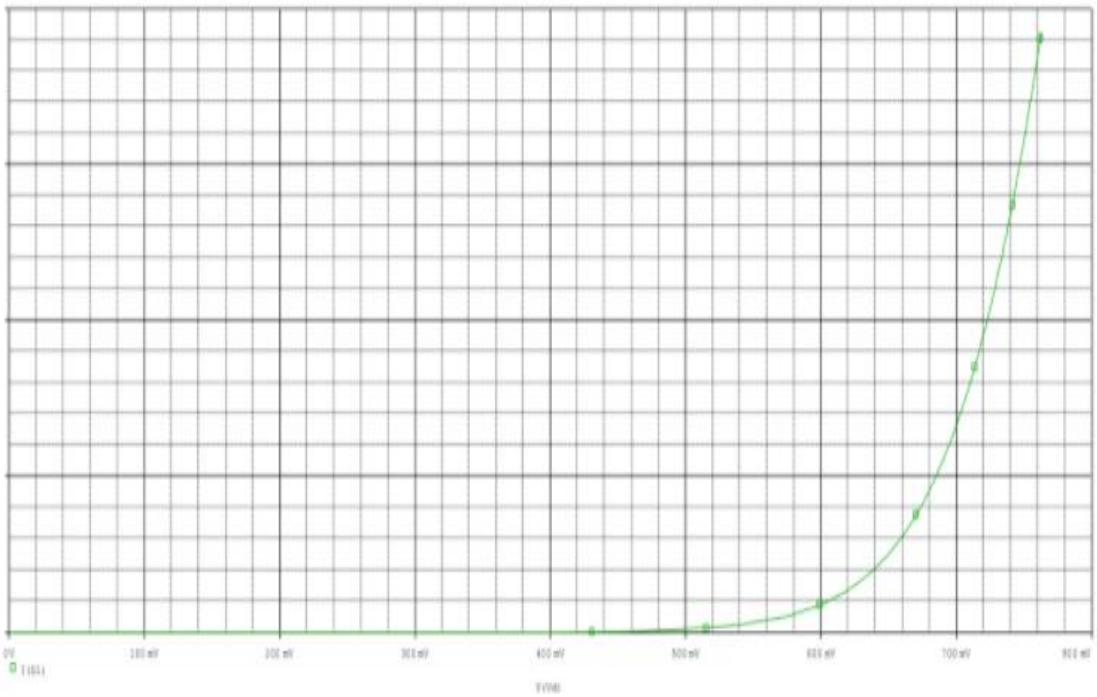
DATA SHEET:

V_s(V)	V_D(V)	V_R(V)	I_D(mA)
0	0	0	0
0.1	0.1	0	0
0.3	0.3	0	0
0.5	0.5	0	0
0.7	0.7	0	0
1	0.7	0.3	0.3
1.3	0.7	0.6	0.6
1.6	0.7	0.9	0.9
2	0.7	1.3	1.3
2.5	0..7	1.8	1.8
3	0.7	2.3	2.3
3.5	0.7	2.8	2.8
4	0.7	3.7	3.7
5	0.7	4.3	4.3

7	0.7	6.3	6.3
9	0.7	8.3	8.3
10	0.7	9.3	9.3
12	0.7	11.3	11.3
15	0.7	14.3	14.3

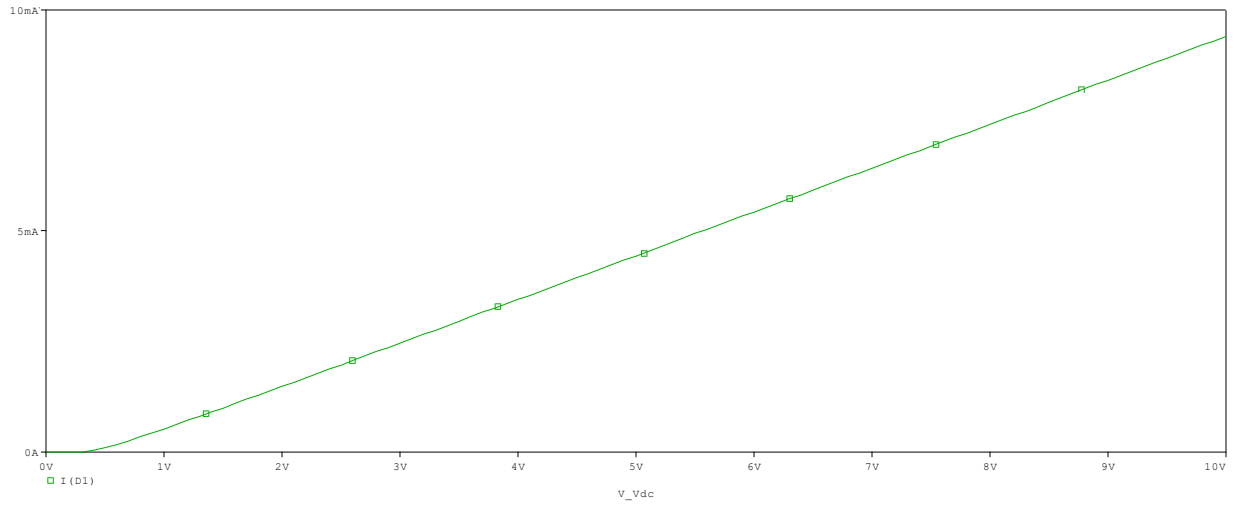
Ans to the post lab questions:

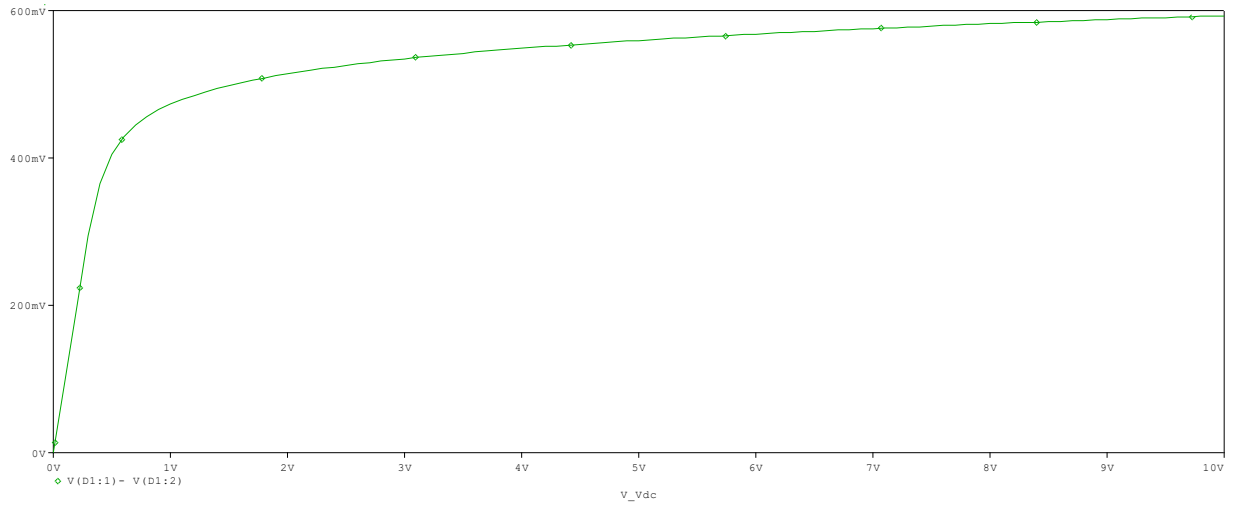
1. Using MATLAB



5. Simulate the circuit of the figure 1 for a DC bias (V_S) range of 0-5 volts using PSpice. Print the I_D vs. V_S and V_D vs. V_S plots generated by PSpice and attach them with your report. For simulation, use the DC SWEEP option of PSpice and the diode D1N4148. To modify the diode parameters, select the diode (it will turn red) and go to Edit → Model Edit Instance Model (Text). There, replace the values of I_S, N, V_j by your values calculated in steps 2 and 3 and click OK.

Answer:





Conclusion:

- The V-I characteristics or voltage-current characteristics of the p-n junction diode is shown in the figure. The horizontal line in the below figure represents the amount of voltage applied across the p-n junction diode whereas the vertical line represents the amount of current flows in the p-n junction diode

- An ideal diode, in forward conduction act like a short circuit but here in practical diode it consume some voltage to act like short circuit, but still it doesn’t act like short circuit fully because of leak voltage.

EAST WEST UNIVERSITY

Course Name : Electronic Circuit
Course Code : CSE251
Section No :06
Group No :04

Experiment No : 02
Name of the Experiment : Half-Wave Diode Rectifier Circuit

Date of allocation :
Date of submission: 6-1-2020

Submitted To: Surajit Das Barman
Senior Lecturer

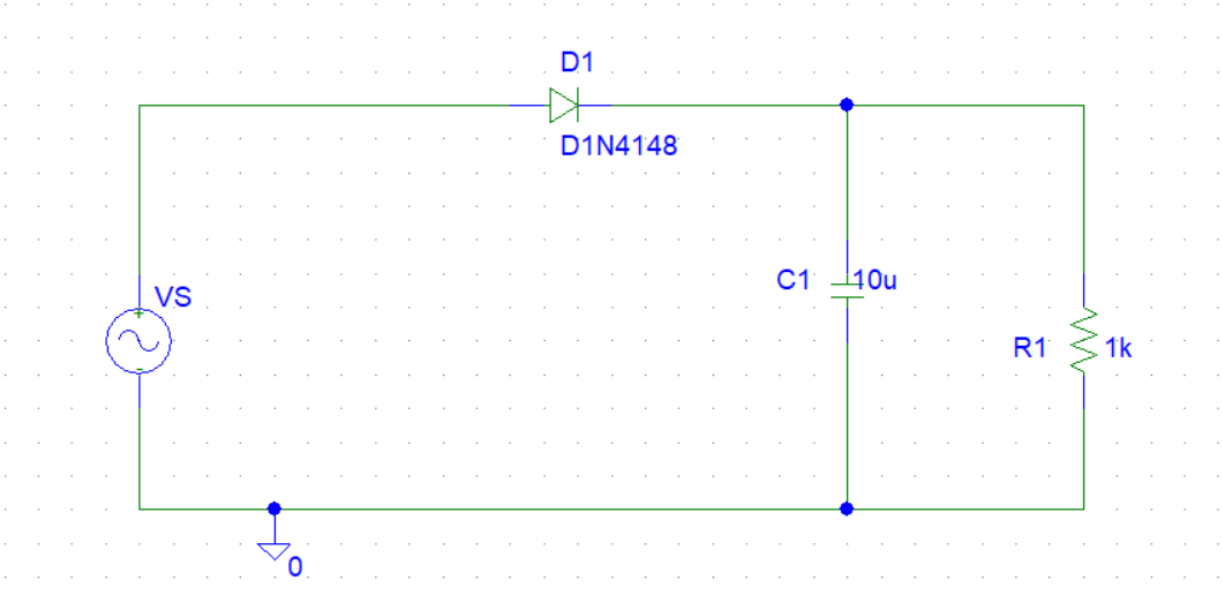
Student's Name : Apurba Roy
Student's ID :2018-3-60-063

Student's Name : Monjurul Alam
Student's ID :2018-3-60-035

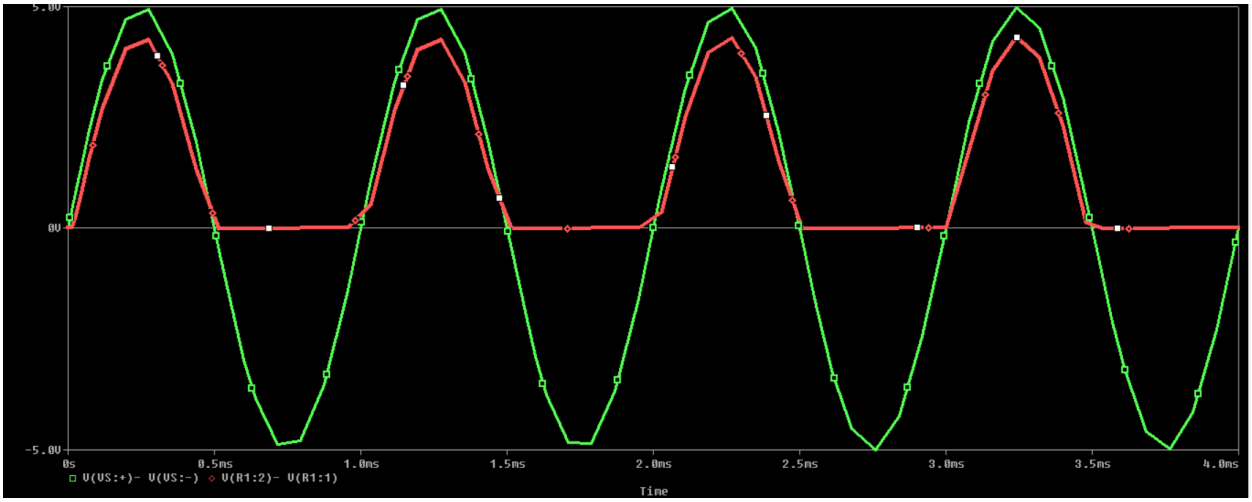
Objectives: 1. To study half-wave diode rectifier circuit. 2. To study the effect of a capacitor filter on the output of the rectifier circuit.

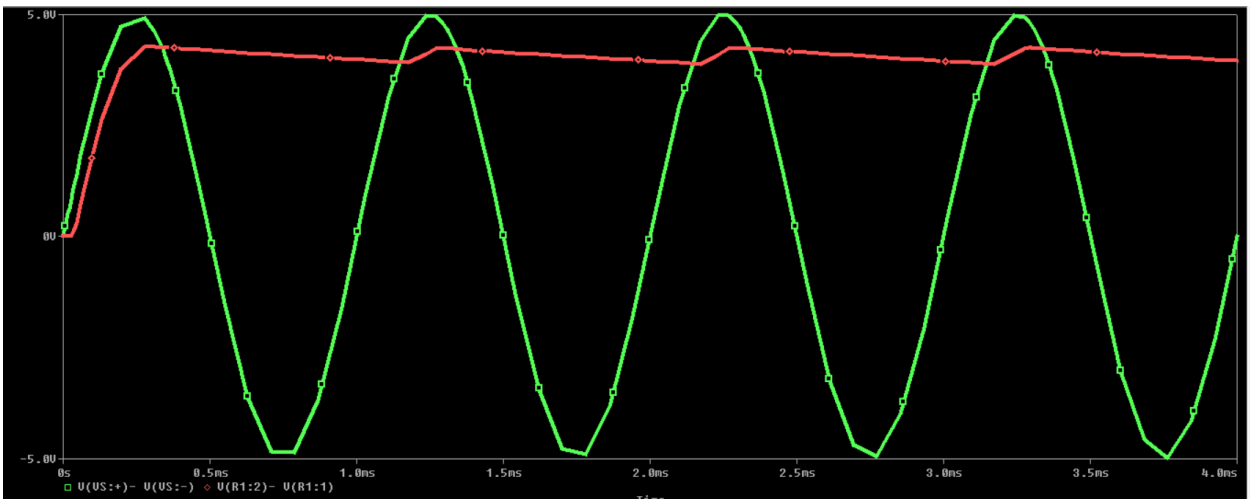
Equipments and Components Needed:
1. Signal Generator
2. Digital multimeter
3. Diode (1 pc)
4. Resistor 1K□ (1 pc)
5. Capacitor 10□F (1 pc)
6. Breadboard
7. Connecting wires

Circuit diagram:



Graph with filter and without filter:





Lab procedure question:

6.Measure the difference in peak values (□Vp) between the input and the output, and write it down.

Answer:

Output voltage-4.98

Input voltage-4.31

peak values (Vp**)**=(4.98-4.31)=.67

8. Measure the time (□t) during which the diode conducts (time between the lower peak to the upper peak of the ripple voltage, that is, the time of charging the capacitor) and write it down

Answer:
Upper peak time-1.257
Lower peak time-1.176
So,
Conduct time- (1.257-1.176)
=0.081
Upper voltrage-4.19
Lower voltage-3.92
So,
Ripple voltage-(4.19-3.92)
=0.27

10. Measure the average value of output voltage (VO) using the DC mode of the multimeter and write it down

Answer:
Given,
$V_p = 5$
$V_r = 4.19$

We know,

$$V_0 = V_p - 0.5 * V_r$$
$$= 5 - 0.5 * 4.19$$
$$= 2.905$$

Conclusion :

We connect this circuit using Pspice software. I think if we could do this same experiment in the lab, the measure value would change a little bit and most importantly we could learn how to connect the circuit for real life.

EAST WEST UNIVERSITY

Course Name : Electronic Circuits

Course Code : CSE251

Section No :06

Group No :04

Experiment No : 04

Name of the Experiment : Adder and Amplifier Circuits Using 741 Op Amp

Date of allocation :

Date of submission: 6-1-2020

Submitted To : Surajit Das Barman
Senior Lecturer

Student's Name : Apurba Roy
Student's ID :2018-3-60-063

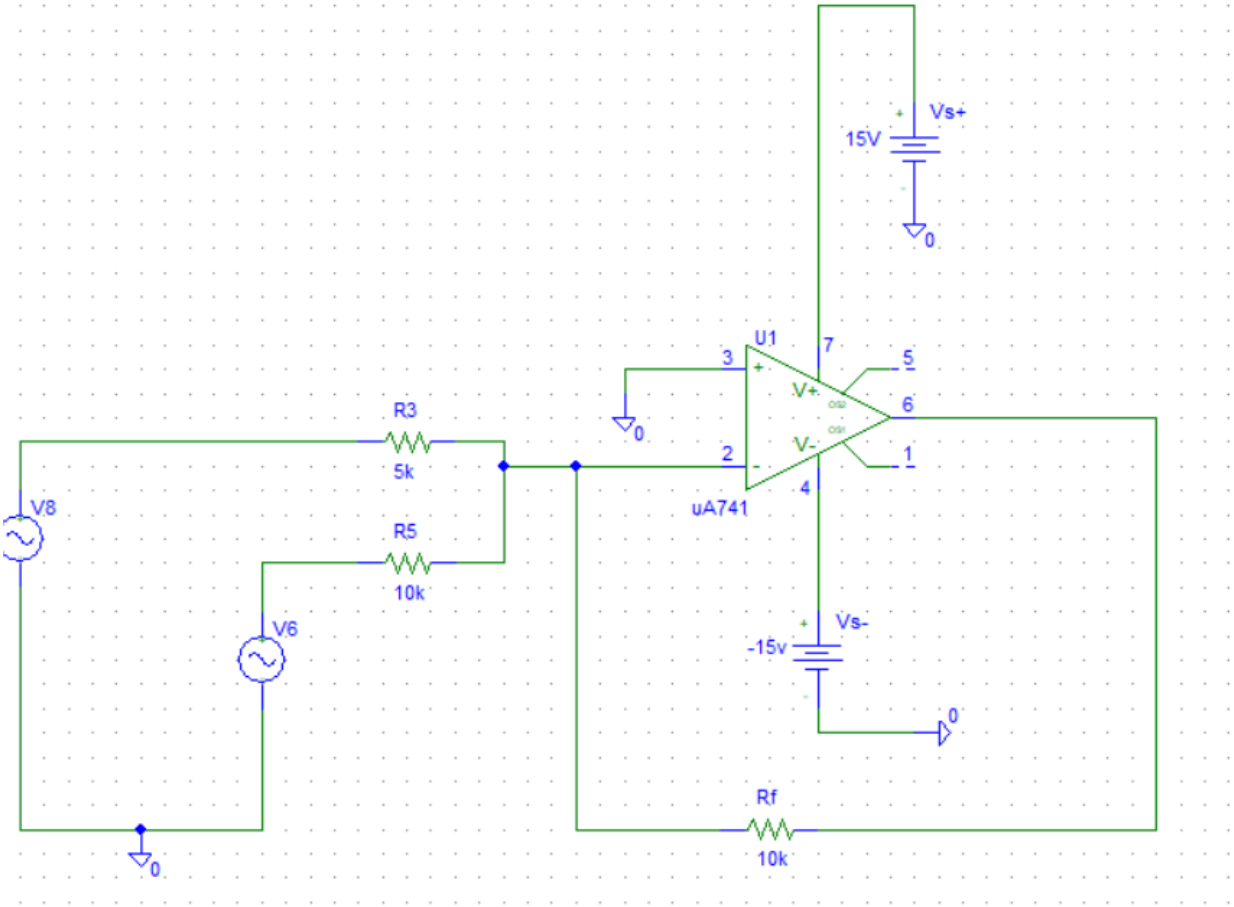
Student's Name : Monjurul Alam
Student's ID :2018-3-60-035

Objectives: 1. To familiarize with the 741 Op Amp Integrated Circuit (IC). 2. To design and construct an adder using 741 Op Amp. 3. To design and construct an amplifier using 741 Op Amp.

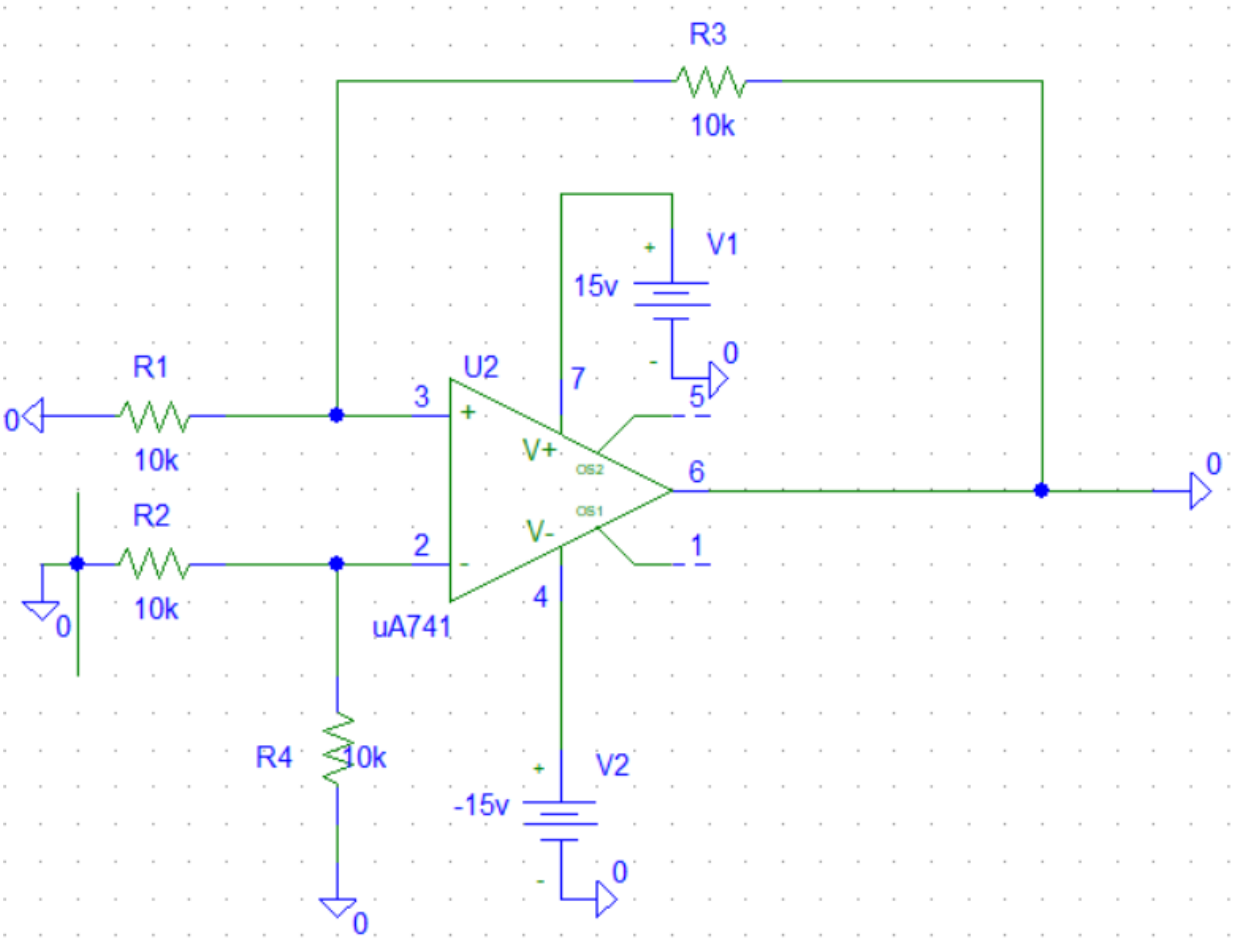
Introduction: Operational Amplifier (Op Amp) is a differential amplifier and can perform mathematical operations such as addition, subtraction, etc. This is an integrated circuit (IC).

Circuit diagram:

Adder circuit :



Amplifier Circuits:

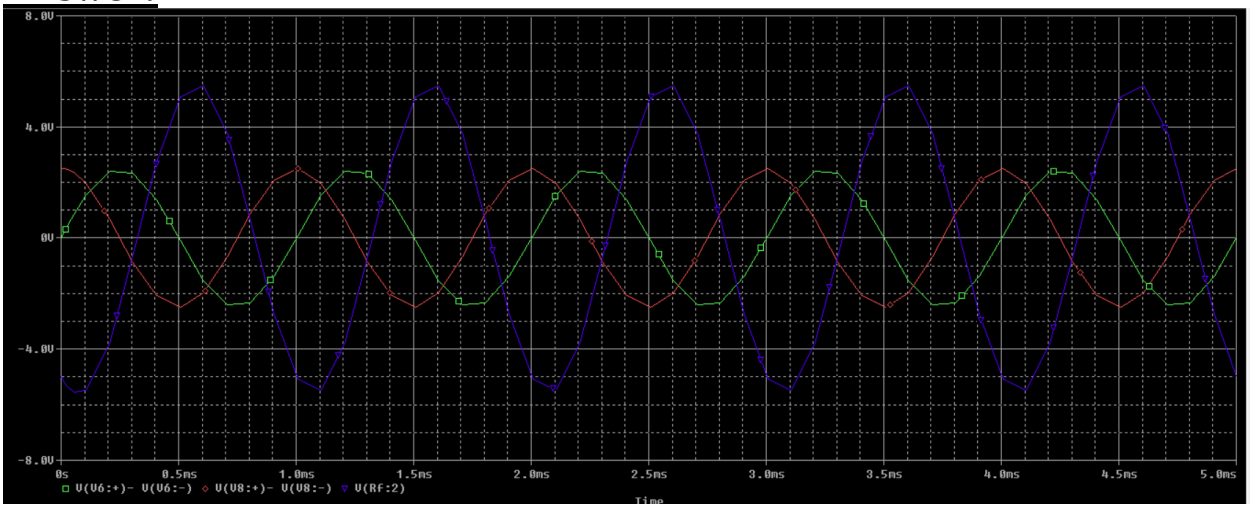


Post-Lab Report Questions:

ADDER CIRCUIT:

3. Simulate the circuit shown in Figure 2 in Pspice. Use V1 a 5V peak to peak, 1KHZ, 00 phase sine wave and V2 a 5V peak to peak, 1KHZ, 900 phase sine wave. Perform simulation for 4 cycles (transient analysis for 4 ms) and attach the printed output with your report.

Answer:



Conclusion :

We connect this circuit using Pspice software. I think if we could do this same experiment in the lab, the measure value would change a little bit and most importantly we could learn how to connect the circuit for real life.

EAST WEST UNIVERSITY

Course Name : Electronic Circuits

Course Code : CSE251

Section No :06

Group No :04

Experiment No : 05

Name of the Experiment : Signal Integration and Differentiation Using 741 Op-Amp

Date of allocation :

Date of submission: 6-1-2020

Submitted To : Surajit Das Barman
Senior Lecturer

Student's Name : Apurba Roy
Student's ID :2018-3-60-063

Student's Name : Monjurul Alam
Student's ID :2018-3-60-035

Objectives: 1. To study the responses of Op-Amp integrator to sinusoid and square waveforms. 2. To study the responses of Op-Amp differentiator to sinusoid and triangular waveforms.

Introduction: Operational Amplifier (Op-Amp) is a differential amplifier and can perform mathematical operations such as addition, subtraction, integration, differentiation, etc.

Circuit diagram:

integrator circuit:

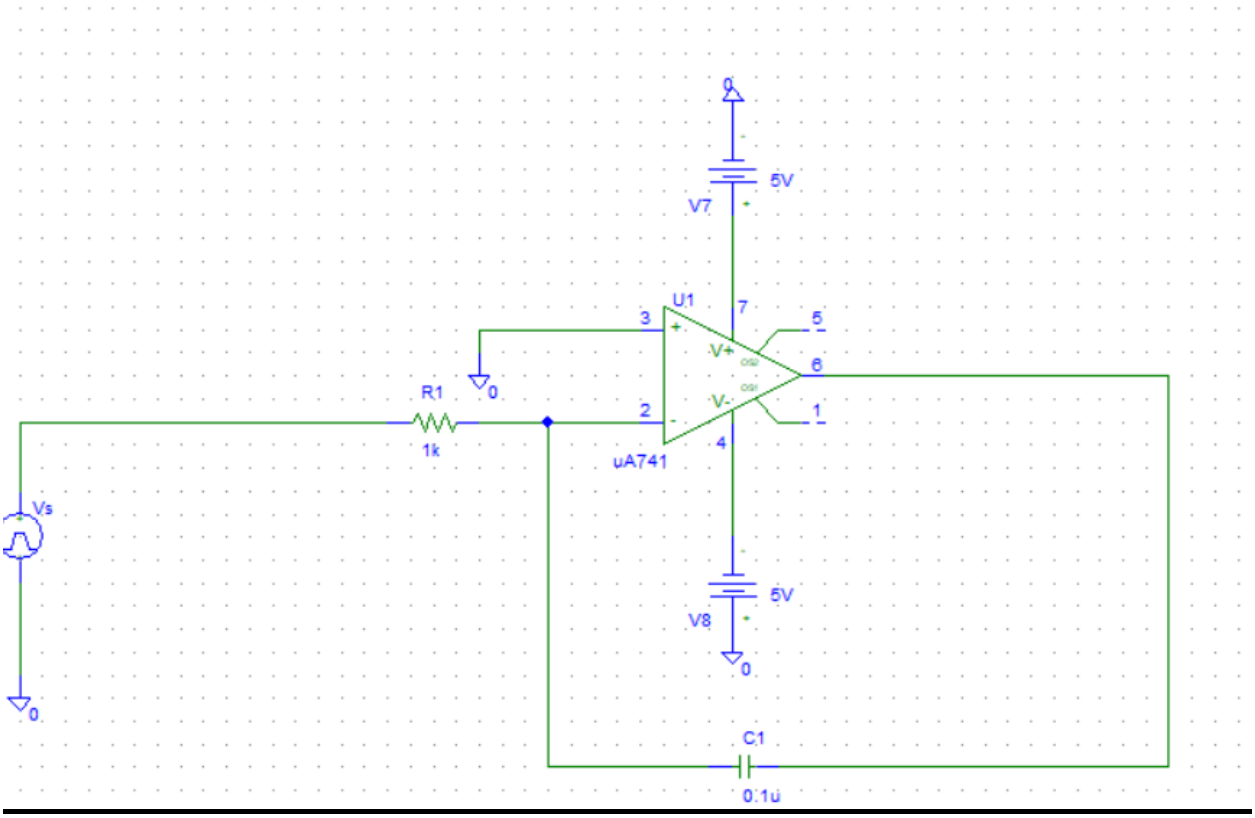


Figure 1. An Op-Amp integrator circuit

differentiator circuit:

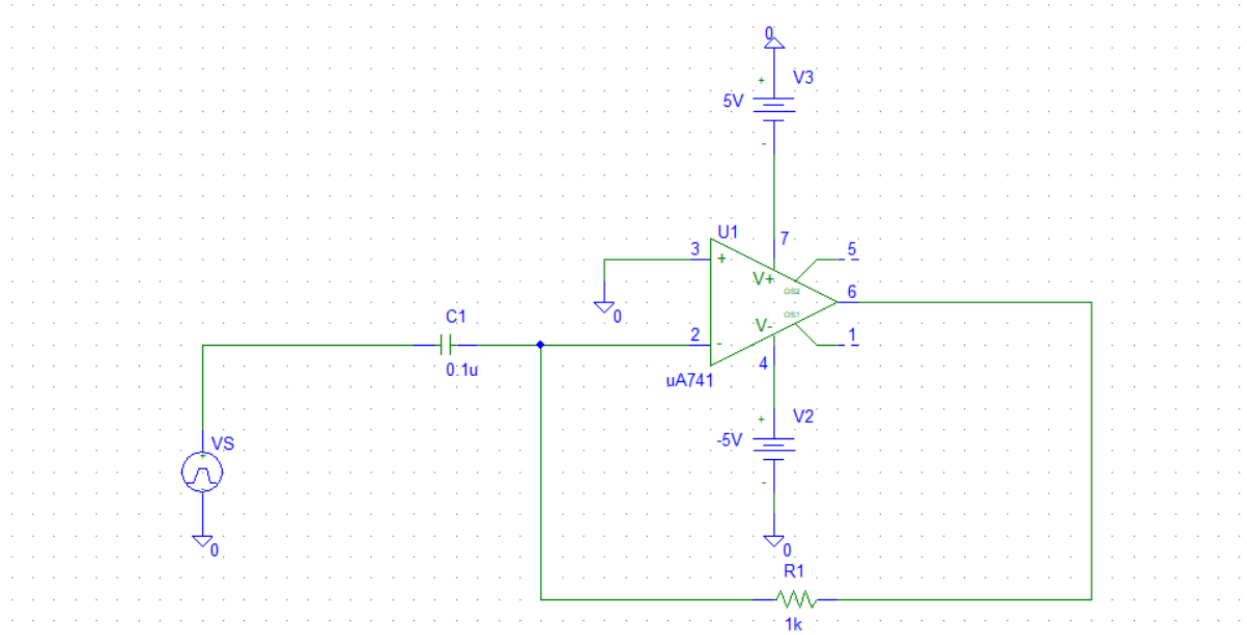
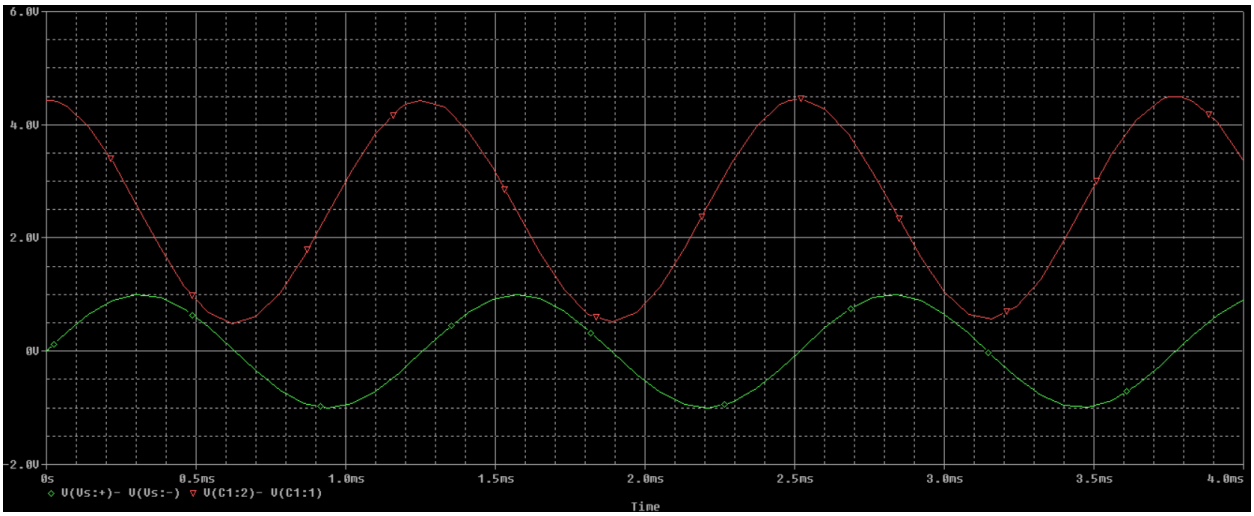


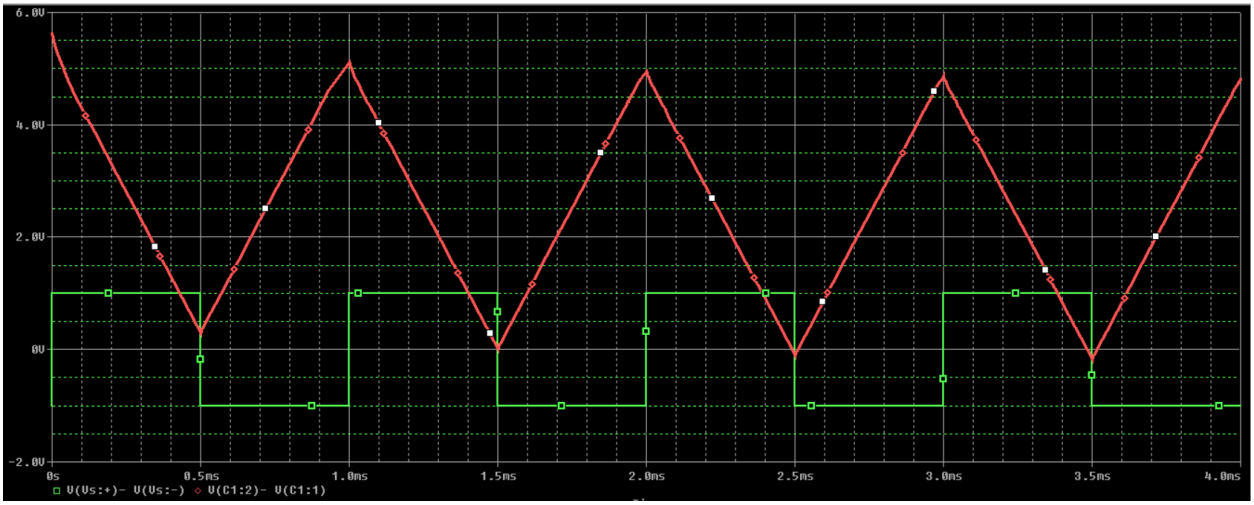
Figure 2. An Op-Amp differentiator circuit.

Post-Lab Report Questions:

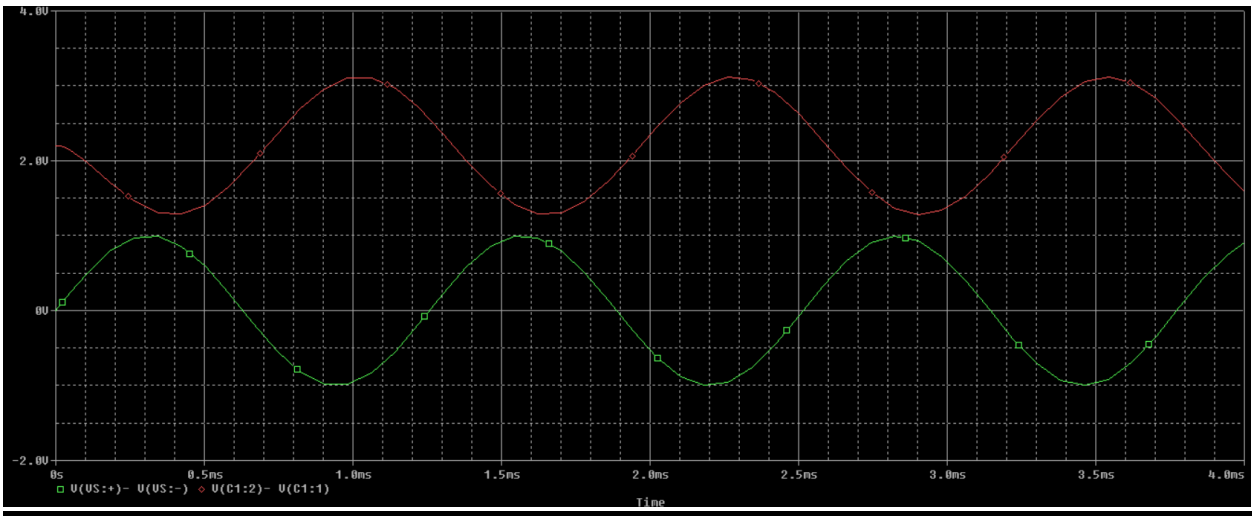
Answer:

Integrator:





Differentiator:



Conclusion:

We connect this circuit using Pspice software. I think if we could do this same

experiment in the lab, the measure value would change a little bit and most importantly we could learn how to connect the circuit for real life.