East West University
Department of Computer Science and Engineering

Course code: CSE251; Course name: Electronic Circuits
Expt No.: 1
Title: I-V Characteristics and Modeling of Forward Conduction of a Diode

Objectives:
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:

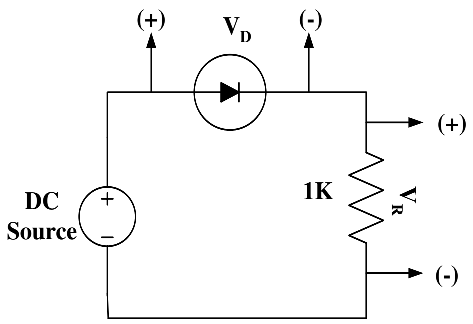


Figure 1. Circuit set up to measure forward bias I-V characteristics of a diode.

Equipments and Components Needed:
1. DC power supply
2. Digital multimeter
3. Diode (1 pc)
4. Resistor 1KΩ
5. Breadboard
6. Connecting wires

Lab Procedure:
1. Measure the resistance value of Figure 1 with the multimeter and write it down in KΩ in Table 1.
2. Connect the circuit as shown in Figure 1. Use the DC power supply unit as DC source.
3. Change the DC source and measure the values of V_D and V_R and write them in Table 1. **Continue measurement until V_D reaches to around 0.68 volts. Take around 25 to 30 readings by increasing the DC power supply with an increment of ≈ 0.5 V after V_D reaches to around 0.68 volts.**
4. Divide V_R by the measured value of resistance in KΩ. This is diode current I_D in mA.

Table 1. Experimental Datasheet.

V_D (V)	V_R (V)	I_D (mA) = V_R/R(KΩ)	Measured Value of R (KΩ)
1.			
35.			

5. Have the datasheet signed by your instructor.

Post-Lab Report Questions:

1. Using MATLAB, plot the I-V characteristics of the p-n junction diode in forward conduction. Label the axes appropriately and have it printed.
2. Use pencil to identify the points on your graph that are corresponding to I_D = 2 mA and I_D = 2.5 mA. Use these data points to calculate the diode parameters I_S and n from the equation $I_D = I_S \exp[V_D / nV_T]$. Use V_T = 0.0259V.
3. Determine the cut-in voltage from the printed graph by drawing extrapolated line with pencil.
4. If the diode resistance for the piecewise linear model is defined as $1/r_D = \partial I_D / \partial V_D = (I_{D2} - I_{D1})/(V_{D2} - V_{D1})$, calculate the value of r_D from the data points corresponding to I_D = 2 mA and I_D = 2.5 mA.
5. Simulate the circuit of 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.

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No.: 2
Title: Half-Wave Diode Rectifier Circuit

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.

Introduction:
A rectifier circuit converts an AC voltage with zero average into a unidirectional voltage with a non-zero average. The rectifier circuit can rectify both positive and negative half-cycles (full-wave rectifier) or only the positive half-cycle (half-wave rectifier) of a sine wave. A capacitor connected across the load resistor acts as a filter and reduces the ripple of the output voltage. The time constant of the RC network should be much larger than the period of the AC source voltage for effective filtering.

Circuit Diagram:

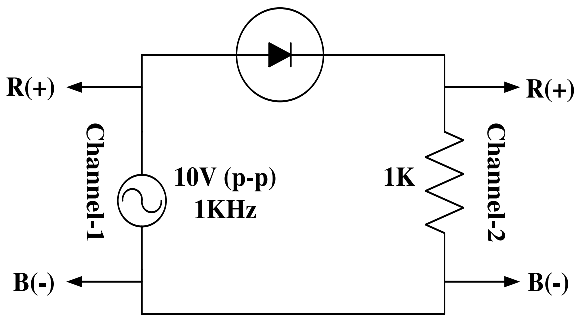


Figure 1. Set up for a half-wave diode rectifier circuit.

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

Pre-Lab Report Question:
1. Consider the half-wave rectifier circuit shown in Figure 1. It is fed by a 1.0 KHz sinusoid having a peak value of 5V. Design the circuit (determine the value of the parallel capacitor) so that the output will have a peak-to-peak ripple of $V_r = 0.5\text{V}$ using the formula $V_r = \frac{V_p}{fCR}$. Calculate the diode conduction time Δt and conduction angle $\omega\Delta t$ using the formula $\omega\Delta t = \cos^{-1}\left(\frac{V_p - V_r}{V_p}\right)$. Also calculate the average and peak values of

the diode currents using the formulas $i_{Davg} = \dfrac{V_p - \dfrac{V_r}{2}}{R}\left(1+\pi\sqrt{\dfrac{2V_p}{V_r}}\right)$ and $i_{D\max} = \dfrac{V_p - \dfrac{V_r}{2}}{R}\left(1+2\pi\sqrt{\dfrac{2V_p}{V_r}}\right)$.

Lab Procedure:

1. Measure the resistance and write it down.
2. Setup the circuit shown in Figure 1.
3. Setup a 10 volts peak-to-peak, 1 KHz sine wave signal from the signal generator and observe it in channel-1 of the oscilloscope.
4. Give input to the circuit and observe the output in channel-2 of the oscilloscope.
5. Observe both the input (in channel-1) and the output (in channel-2) signals by setting dual mode in the oscilloscope.
6. Measure the difference in peak values (ΔV_p) between the input and the output, and write it down.
7. Connect the capacitor from your design in the pre-lab report in parallel with the resistance and observe the output only.
8. Measure the time (Δt) 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.
9. Measure the peak-to-peak ripple voltage (V_r) from oscilloscope and write it down.
10. Measure the average value of output voltage (V_O) using the DC mode of the multimeter and write it down.
11. Have the datasheet signed by your instructor.

Post-Lab Report Questions:

1. Compare the measured value of ΔV_p with the built-in voltage of Expt 1.
2. Compare your measured Δt with your prelab value and make a comment.
3. Calculate the peak-to-peak ripple voltage from the formula $\omega\Delta t = \sqrt{\dfrac{2V_r}{V_p}}$ and compare it with your measured data and prelab data and make a comment.
4. Calculate the average output voltage from $V_O = V_p - \dfrac{V_r}{2}$ and compare it with the measurement.
5. With $I_L = \dfrac{V_O}{R}$, calculate the average and maximum diode currents using the formulas $i_{Davg} = I_L\left(1+\pi\sqrt{\dfrac{2V_p}{V_r}}\right)$ and $i_{D\max} = I_L\left(1+2\pi\sqrt{\dfrac{2V_p}{V_r}}\right)$ and measured value of V_r and compare with your pre-lab values and make a comment.
6. Simulate the half-wave rectifier circuit in PSPICE for C = 10 μF and submit the input and output plots (on same graph). Use transient analysis of PSPICE for 4 cycles of input (4 ms). Modify the diode parameters following the same procedure and the same parameters values used in Expt 1.

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No: 03
Title: Study of Zener Diode.

Objectives:
1. To measure the I-V characteristics of Zener diode.
2. To determine the voltage regulation for variable resistance and variable supply voltage.

Introduction:
The diodes we have studied before do not operate in the breakdown region because this may damage them. A *Zener diode* is different; it is a silicon diode that the manufacturer has optimized for operation in the breakdown region. It is used to build voltage regulator circuits that hold the load voltage almost constant despite large change in line voltage and load resistance. Figure shows the symbol of the Zener diode.

Fig.1: Symbol of Zener diode

The Zener diode may have a breakdown voltage from about 2 to 200 V. These diodes can operate in any of the three regions: forward, leakage and breakdown. Figure shows the 1-V graph of Zener diode.

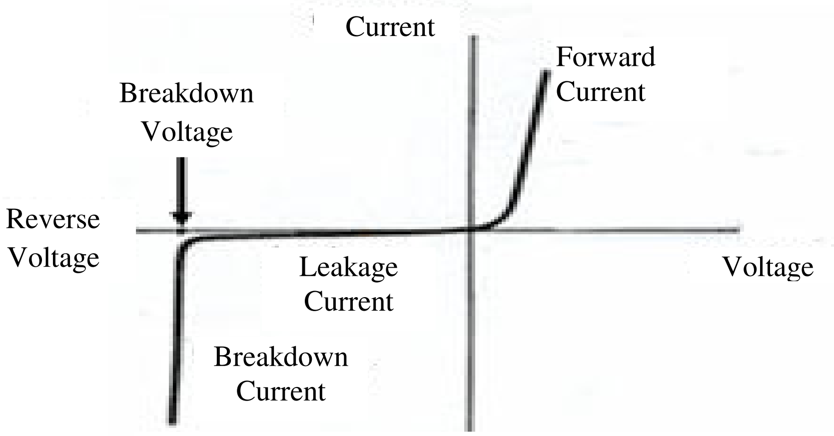
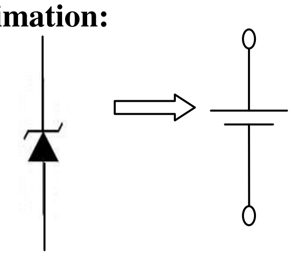


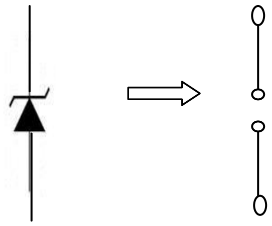
Fig. 2: I-V characteristic curve

- In the forward region it works as an ordinary diode.
- In the leakage region (between zero and breakdown) it has only a small reverse saturation current.
- In the breakdown it has a sharp knee, followed by an almost vertical increase in current without changing the voltage.
- The voltage is almost constant, approximately equal to V_Z over most of the breakdown region.

Approximation:



"**On**" $[V \geq V_z]$



"**Off**" $[V_z > V > 0]$

Pre-Lab Report:

1. Draw the I-V characteristic of Fig. 3 (circuit with no load)and
2. calculate V_L, V_R, I_Z for 10 V supply voltage.
3. Consider $R_L = 220\ \Omega$ and write down the condition for conduction.
4. Calculate R_{Lmin}, R_{Lmax}, I_{Lmin} for I_{Lmax} for I_{zmax}= 7.2mA and 10 V supply voltage in Fig. 4.
5. Calculate $V_{i\ min}$, and $V_{i\ max}$ for I_{zmax}= 7.2mA of Fig. 5.

Equipments:

1. Zener diode (6.2 volt)
2. Resistance (220 Ω, 470 Ω, 1 kΩ)
3. POT 10 kΩ
4. DC Power supply
5. Bread board
6. Multimeter.
7. Ammeter.

Experimental Setup:

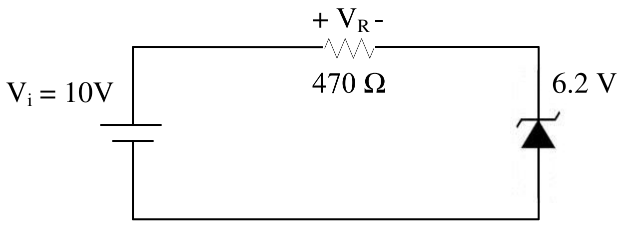


Fig. 3: Circuit with No Load

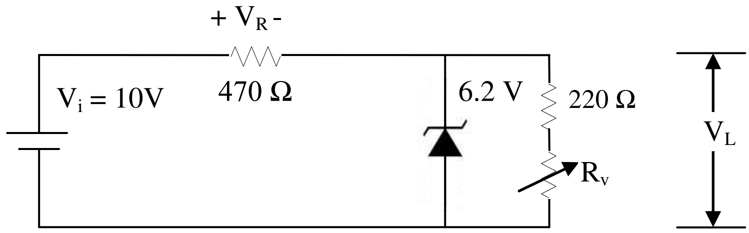


Fig. 4: Circuit with variable load

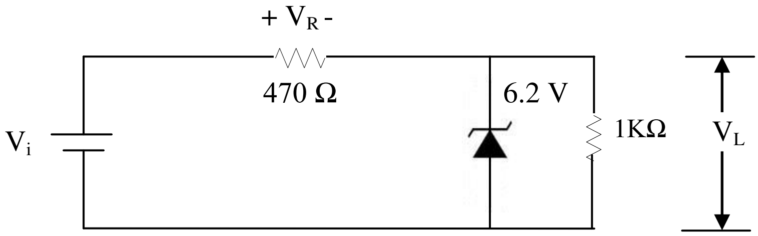


Fig. 5: Circuit with variable supply voltage.

Procedure:

1. Connect the circuit as shown in the Figure 3.
2. Vary the supply voltage from zero to 10 volts and complete Table 1.
3. Now, connect the circuit as shown in figure 4.
4. Keep the POT (variable resistance R_v) at its maximum value which you have calculated from pre lab.

5. Gradually decrease the POT resistance upto minimum R_v which you have calculated from pre lab , next calculate I_L by ohm's law.
6. complete the Table 2.
7. Replace load with 1 kΩ resistance, vary the supply voltage V_i and complete table 3.

V_i (Volt)	V_R (Volt)	V_z (Volt)	$I_z = V_R/R$ (mA)
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			

Table 1: Data for I-V characteristics

Vi (Volt) =10V	V_L(Volt)	I_L(Amp)

Table 2: Data for regulation due to load variation

V_i (Volt)	V_L(Volt)
1	
2	
3	
4	
5	
6	
7	
8	
9	
10	

Table 3: Data for regulation due to supply voltage variation

Post Lab Report Questions:

1. Plot the I-V characteristics of the Zener diode. Determine the Zener breakdown voltage from the plot.
2. Plot I_L Vs V_L for the data of table-2. Find the load regulation and compare it with the pre-lab data.
3. Plot V_L Vs V for the data of table 3. Find the line regulation compare it with the pre-lab data.

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No.: 4
Title: Adder and Amplifier Circuits Using 741 Op Amp

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).

The block diagram of the 741 Op Amp is shown in Figure 1. It has 8 pins. To identify the pins of the 741 Op Amp in the laboratory, place the Op-Amp on the trainer board in such a way that the **notch** is in your left side. Then the pin number should be identified as shown in Figure 1.

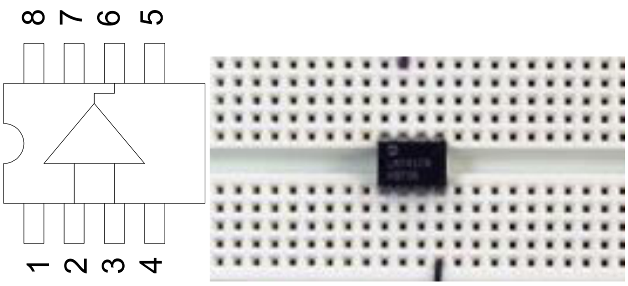


Figure 1. Block diagram of the 741 Op-Amp

The names of different pins are as follows:
1: Offset null (usually not used)
2: Inverting input terminal
3: Non-inverting input terminal
4: Negative DC power supply (usually 5-15V negative)
5: Offset null (usually not used)
6: Output terminal
7: Positive DC power supply (usually 5-15V positive)
8: Not connected (NC)

The Op Amp acts as an adder when the non-inverting input terminal is grounded and two or more signals are fed to the inverting input via resistors of appropriate values. The output becomes negative of the sum of the input signals.

A non inverting amplifier produces the amplified output equal to the input multiplied by the closed loop gain, which is determined by the appropriate resistances.

Circuit Diagram:

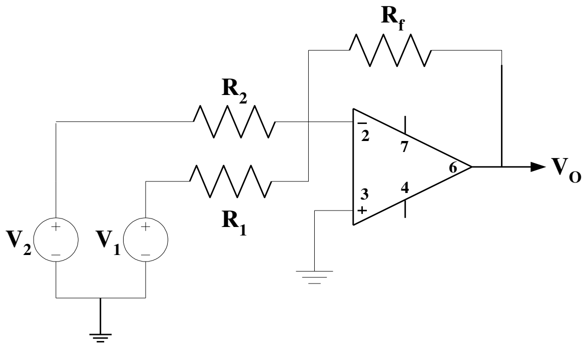


Figure 2. An adder circuit using 741 Op Amp.

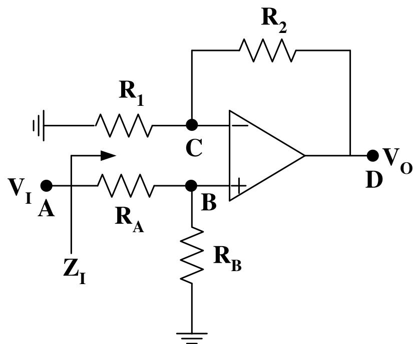


Figure 3. A non inverting amplifier circuit using 741 Op Amp.

Equipments and Components Needed:

1. Digital trainer board
2. DC power supply
3. Signal generator
4. Oscilloscope
5. Digital multimeter
6. 741 Op Amp (1 pc)
7. Resistor (as required from pre-lab design)
8. Breadboard
9. Connecting wires

Pre-Lab Report Question:

1. DESIGN OF ADDER CIRCUIT: Design the adder circuit shown in Figure 2 (determine the values of the resistors) so that it implements the function $V_O = -(V_1 + 2V_2)$. The available set of resistances that you can use is 100Ω, 1KΩ, 2.2KΩ, 3.3KΩ, 5.6KΩ, and 10KΩ. For design safety, take R_f as high as possible. In your design, if exact value of resistance is not available, choose the nearest one.
2. DESIGN OF AMPLIFIER CIRCUIT: Design the non inverting amplifier circuit as shown in Figure 3 (determine the values of the resistors) so that the closed loop gain is $\frac{V_O}{V_I} = \frac{R_2}{R_1} = 5$. The available set of resistance that you can use is 100Ω, 1KΩ, 2.2KΩ, 3.3KΩ, 5.6KΩ, and 10KΩ. For design safety, take R_2 as high as possible. For proper operation of the amplifier circuit, assume that $R_A = R_1$ and $R_B = R_2$. In your design, if exact value of resistance is not available, choose the nearest one. With your design, set V_I

= 1V and calculate the voltages at nodes A, B, C, and D. Also, calculate the currents through the resistances R_1, R_2, R_A, and R_B and the input impedance $Z_I = \frac{V_I}{I_I}$.

Lab Procedure:

ADDER CIRCUIT:

1. Collect the resistances of your design from the lab assistant and measure them and write them down.
2. Connect the circuit as shown in Figure 2 with the resistance values from your prelab design. Use a +15V DC power supply to terminal 7 and -15V DC power supply to terminal 4 of the Op Amp from the digital trainer board.
3. Use 5V from digital trainer board as V_1 and 2V from DC power supply as V_2. Measure the output using multimeter and write it down.
4. Replace the V_1 by a 5V peak to peak 1 KHZ sine wave from the signal generator and observe the output in channel-2 in DC mode. Invert channel-2 and write the amplitude.

AMPLIFIER CIRCUIT:

5. Collect the resistances of your design from the lab assistant and measure them and write them down.
6. Set up the circuit as shown in Figure 3 using the resistances from your prelab design. Use a +15V DC power supply to terminal 7 and -15V DC power supply to terminal 4 of the Op Amp from the digital trainer board.
7. Set V_I = 1V from the DC power supply.
8. Measure the voltages at nodes A, B, C, and D using multimeter and write them down.
9. Measure the voltages across resistances R_1, R_2, R_A, and R_B and write them down.

10. Have the datasheet signed by your instructor.

Post-Lab Report Questions:

ADDER CIRCUIT:

1. From the measurement in step 3, verify your design.
2. Does the amplitude measured in step 4 verify your design? Explain.
3. Simulate the circuit shown in Figure 2 in Pspice. Use V_1 a 5V peak to peak, 1KHZ, 0^0 phase sine wave and V_2 a 5V peak to peak, 1KHZ, 90^0 phase sine wave. Perform simulation for 4 cycles (transient analysis for 4 ms) and attach the printed output with your report.
4. Write the expressions of V_1 and V_2 of Question 3 in phasor domain, add them, and write the result in time domain as a sine function. Compare it with the Pspice output in terms of amplitude, phase angle, and time period.

AMPLIFIER CIRCUIT:

5. Compare the measured voltages at nodes A, B, C, and D in step 8 with your prelab results.
6. From your measured voltages at nodes B and C in step 8, comment on the virtual ground of Op Amp.
7. From your measured voltages at nodes A and D in step 8, calculate the gain and verify with prelab result.
8. From the measured voltages across the resistances in step 9, calculate the currents through them and compare them with your prelab results.
9. From your measured voltages at node A in step 8 and across R_A in step 9, calculate Z_I and compare it with your prelab result

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No.: 5
Title: Signal Integration and Differentiation Using 741 Op-Amp

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. In Expt. No. 3, we use the Op-Amp together with the only resistors to study adder and amplifier circuits. In this experiment, we will study integrator and differentiator circuits by using a capacitor either in the feedback path or in the input path.

Circuit Diagram:

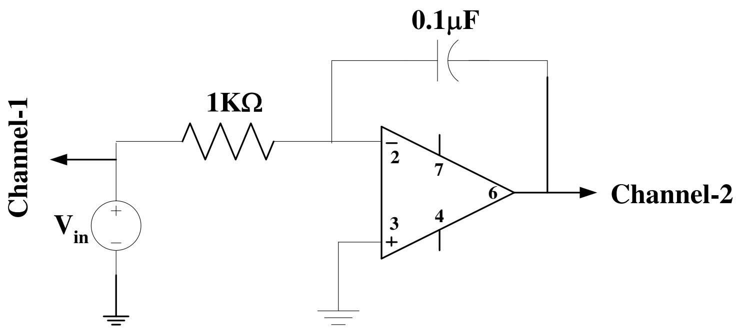


Figure 1. An Op-Amp integrator circuit.

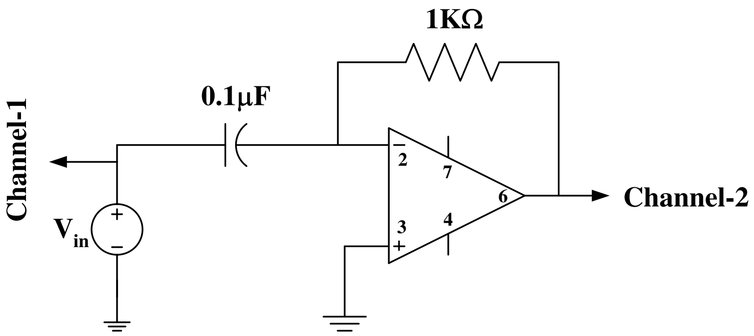


Figure 2. An Op-Amp differentiator circuit.

Equipments and Components Needed:

1. Digital trainer board
2. Signal generator
3. Oscilloscope
4. Digital multimeter
5. 741 Op Amp (1 pc)
6. Resistor (1KΩ 1 pc)
7. Capacitor (0.1μF, 1 pc)
8. Breadboard
9. Connecting wires

Pre-Lab Report Question:

1. INTEGRATOR: Consider the integrator shown in Figure 1 with $v_{in} = V_p \sin(\omega t)$. If the circuit is initially relaxed, find the expression of output and determine the frequency, f, at which the output amplitude is equal to $2V_p$. Also, determine the phase relation between input and output.
2. DIFFERENTIATOR: Consider the differentiator shown in Figure 2 with $v_{in} = V_p \sin(\omega t)$. If the circuit is initially relaxed, find the expression of output and determine the frequency, f, at which the output amplitude is equal to $V_p / 2$. Also, determine the phase relation between input and output.

Lab Procedure:

INTEGRATOR:

1. Measure the resistance and write it down. Connect the circuit as shown in Figure 1. Use a +5V DC power supply to terminal 7 and -5V DC power supply to terminal 4 of the Op-Amp. Use fixed 5V and -5V DC supply of digital trainer board or 0 to15V and 0 to -15V DC variable power supply from the digital trainer board.
2. Use a 2V peak to peak sine wave V_{in} from the signal generator. Set the sine wave frequency to the one that you have determined in your prelab design.
3. Observe the output in channel-2 and input in channel-1 using dual mode. Write the amplitudes of the input and output signals and the phase difference between them.
4. Change the input from sine wave to square wave (do not change the frequency and magnitude) and observe the output in dual mode. Draw both the input and output wave forms with voltage and time axes labels.

DIFFERENTIATOR:

5. Connect the circuit as shown in Figure 2. Use a +5V DC power supply to terminal 7 and -5V DC power supply to terminal 4 of the Op-Amp.
6. Use a 2V peak to peak sine wave V_{in} from the signal generator with frequency from you prelab.
7. Observe the output in channel-2 and input in channel-1 using dual mode. Write the amplitudes of the input and output signals and the phase difference between them.
5. Change the input from sine wave to triangular wave (do not change the frequency and magnitude) and observe the output in dual mode. Draw them with voltage and time axes labels.

6. Have the datasheet signed by your instructor.

Post-Lab Report Questions:

INTEGRATOR:

1. In the output expression from your prelab, put the measured value of R and the given values of C and V_p; calculate the amplitude of the output signal and compared it with the measured data.
2. Compare the phase relation between your prelab result and measured data.
3. Simulate the integrator of Figure 1 for 4 cycles using PSpice with the measured value of R and the square wave input that you have used in the lab and have the simulation result printed. Compare the simulation result with your measurement.

DIFFERENTIATOR:

4. In the output expression from your prelab, put the measured value of R and the given values of C and V_p; calculate the amplitude of output signal and compared it with your measured data.
5. Compare the phase relation between your prelab result and measured data.
6. Simulate the differentiator of Figure 2 for 4 cycles using PSpice with the measured value of R and the triangular wave input that you have used in the lab and have the simulation result printed. Compare the simulation result with your measurement.

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No.: 6
Title: Measurement of Parameters and I-V characteristics of an N-channel MOSFET

Objectives:
1. To measure the threshold voltage V_t and the process transconductance K_n of an N-channel enhancement type MOSFET.
2. To measure the I-V characteristics (I_D vs. V_{DS}) of an N-channel enhancement type MOSFET.

Circuit Diagram:

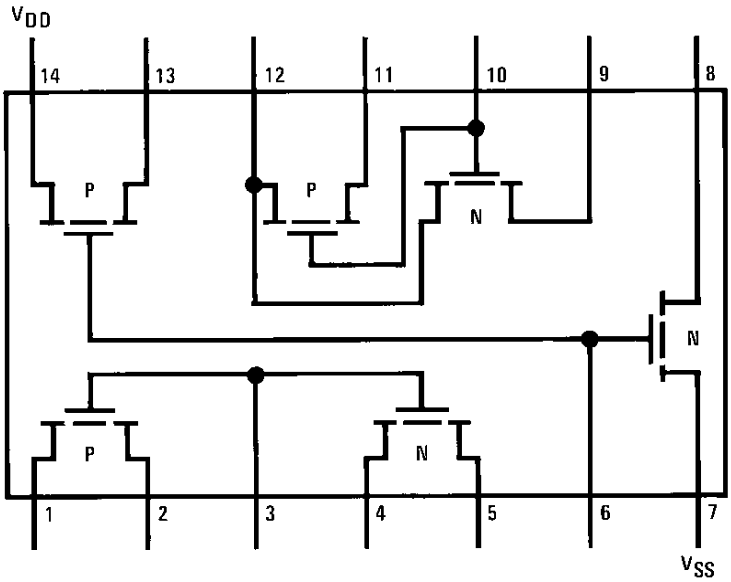


Figure 1. Pin diagram of CD4007C IC.

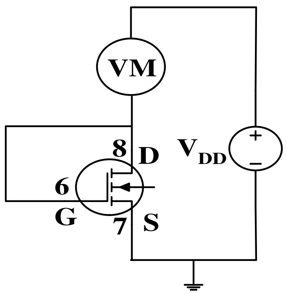


Figure 2. Circuit for measurement of V_t and K_n of an NMOSFET.

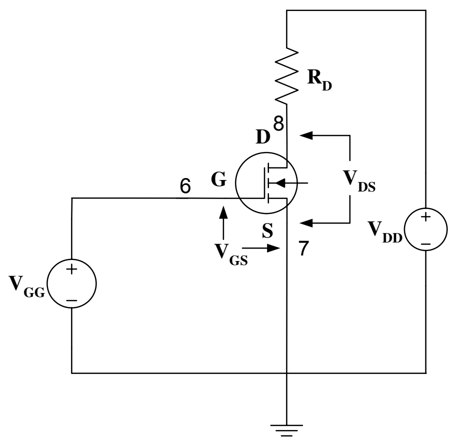


Figure 3. Circuit for measurement of I-V characteristics of an NMOSFET.

Equipments and Components Needed:

1. Digital trainer board
2. DC power supply
3. Digital multimeter
4. DC Voltmeter
5. CD4007C IC (1 pc)
6. Resistor (1KΩ 1 pc)
7. Breadboard
8. Connecting wires

Lab Procedure:

MEASUREMENT OF V_t AND K_n:

1. Measure the resistance and connect the circuit as shown in Figure 2. Note that a voltmeter (VM) is in series with the drain and V_{DD}; and **G** and **D** are shorted. **Use the pin numbers as shown in Figure 2.**
2. Set V_{DD} to 10V from the DC power supply unit and measure the reading of the voltmeter. The threshold voltage $V_t = V_{DD} –$ VM reading.
3. Now replace the voltmeter by 1KΩ resistance and measure the voltage drop across the resistance. Divide it by the resistance to get I_D. Measure V_{GS} and calculate the process transconductance from $K_n = 2I_D/(V_{GS}-V_t)^2$.

MEASUREMENT OF I-V CHARACTERISTICS:

4. Connect the circuit as shown in Figure 3 and set $V_{GG} = V_t+1V$ from the trainer board variable power supply.
5. Use the DC power supply unit as V_{DD}. Now change V_{DD} from 0 and measure V_{DS} and V_{RD} (voltage across R_D resistance). Calculate I_D from $I_D=V_{RD}/R_D$. Take around 15 data up to V_{DS} = 7V. **Be careful so that V_{DD} does not exceed 15V**.
6. Set V_{GG} to V_t+2V and V_t+3V and repeat step 5.

7. Have the datasheet signed by your instructor.

Post-Lab Report Questions:

1. You have V_t and K_n. Note that here K_n is equivalent to $K'_n(W/L)$ of the text. For three V_{GG} ($V_{GG} = V_{GS}$) values of Figure 3, use the linear (triode) and saturation current expressions to tabulate the I_D for each V_{DS} and plot the I_D-V_{DS} curves using your calculated and experimental data on the same graph. Use MATLAB for plotting.

$$I_D = K_n\left[(V_{GS} - V_t)V_{DS} - V_{DS}^2/2\right]; linear$$

$$I_D = (K_n/2)\left[(V_{GS} - V_t)^2\right]; saturation$$

2. Write your observation and comments on the calculated and experimental graphs, especially in the saturation regions.
3. For $V_{GG} = V_t + 3V$, take two experimental data points in saturation and calculate the slope. From the slope, obtain output resistance r_o.
4. Simulate the circuit shown in Figure 3 using PSPICE. For simulation use MbreakN3 MOSFET and DC sweep analysis with nested loop for the three different values of V_{GG}. To set the parameters, double click on MbreakN3 and set W and L to 1E-6 (1um). Now select MbreakN3 (it will turn red) and go to Edit→Model→Edit Instance Model (Text). Delete everything in the appeared window and write the followings (put your values of V_t and K_n) and click OK.

```
.MODEL MbreakN3 NMOS
LEVEL  = 3
VTO  = 1.8
KP  = 100E-6
```

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No.: 7
Title: Biasing of a Common-Source Voltage Amplifier

Objectives:
1. Identify an appropriate DC operation point for a NMOS transistor.

Theory: The common-source amplifier with a NMOS transistor is shown here.
1. The biasing is done by fixing the gate voltage with a voltage divider and also by using a source resistor R_S. The source resistor gives negative feedback and stabilizes the bias current as a function of temperature variations and transistor characteristics. This is a popular biasing scheme for discrete transistor circuits.
2. Select source resistor such that voltage Vs at source terminal is about 1/3rd to 1/5th of V_{DD}. The resistance R_D is chosen such that drain voltage V_D is about in the middle of V_{DD} and V_S. This is done so that the signal at the drain has a relatively large and symmetrical output swing.

$$i_D = \frac{1}{2} k_n' \frac{W}{L} (v_{GS} - V_t)^2 \qquad V_{GS} = V_t + \sqrt{\frac{2I_D}{k_n' \frac{W}{L}}}. \quad V_G = V_S + V_{GS} = V_S + V_t + \sqrt{\frac{2I_D}{k_n' \frac{W}{L}}}$$

3. We choose the resistors such that the parallel resistor is relatively large to ensure a large input resistance of the amplifier and prevent loading of the signal source $R_{in} = R_{G1} || R_{G2}$.
4. An important characteristics of a transistor is its *transconductance* g_m. It is a measure of the rate of change of output current with respect to input voltage v_{gs}. The g_m can be written as follows,

$$g_m = \frac{\partial i_D}{\partial v_{GS}} |_Q = k_n' \frac{W}{L} (V_{GS} - V_t) = \sqrt{2 I_D k_n' \frac{W}{L}} \; , \quad g_m = \frac{2I_D}{V_{GS} - V_t}$$

Circuit Diagram:

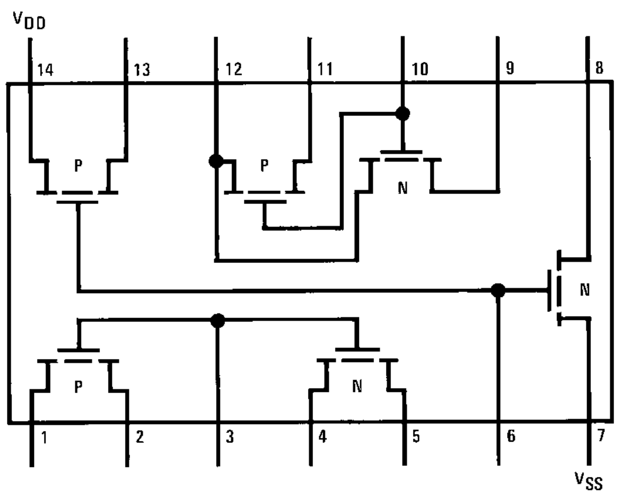


Figure 1. Pin diagram of CD4007C IC.

Equipments and Components Needed:

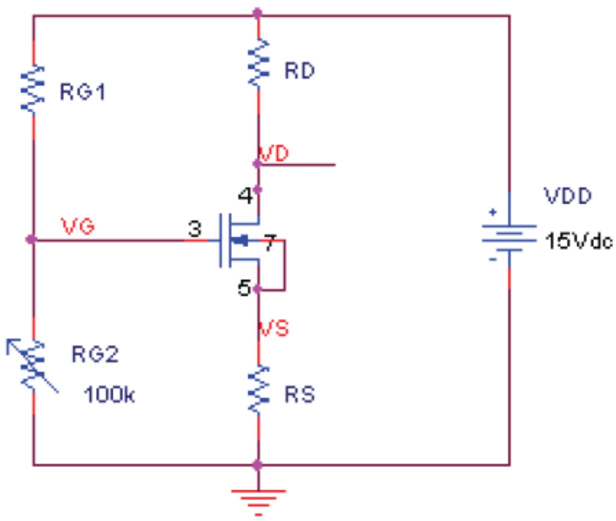
1. Digital trainer board
2. DC power supply
3. Digital multimeter
4. DC Voltmeter
5. CD4007C IC (1 pc)
6. Resistor (1KΩ 1 pc)
7. Breadboard
8. Connecting wires

Lab Procedure:
Pre-Lab:

a. Read: Section 4.5.2 (example 4.9) Sedra-Smith, 5th ed.
b. You have to bias the transistor with a bias current $I_D = 0.6mA$. Let the source voltage V_S be 4V.
c. Choose drain voltage V_D such that it is in the middle of V_{Sn} and $V_{DD.}$
d. The input resistance R_G should be larger than 15 kOhm.
e. The NMOS transistor (CD4007CN array) characteristics are: V_t= 1.2V , $k_n{}' W/L$=0.7mA/V^2,
a. Following example 4.9 procedure and information above determine the value of V_D, R_S, and R_D, R_{G1} and R_{G2}
b. What is the total DC power dissipation in the amplifier? (hint: power dissipation is $V_{DD}I_{total}$).

PROCEDURE:

In this part you will bias the transistor (Figure to the right) using pre-lab data.



a. Build the circuit to the right. Use the transistor between the pins 3, 4 and 5. Connect the pin 7 source (pin 5) of the NMOS transistor; drain-to-source short is done to eliminate the back-gate (body) effect on the threshold voltage. (If this transistor does not work try any of the other two NMOSes). For the biasing resistor R_{G2}, use a 100 kOhm pot. For R_{G1}, R_D and R_S use the values from pre-lab.
b. Set the pot R_{G2} such that the drain voltage V_D (pin 4 of MOSFET) is between 9 and 10 V. Note the drain voltage V_D=___V. Now measure the gate and source voltages. V_G=___V. V_S=___V. Calculate the drain current I_D = ___A?

East West University
Department of Computer Science and Engineering

Course: CSE251 Electronic Circuits
Expt No.: 8
Title: Introduction to Transistor

Objectives:

1. Identify base, emitter and collector terminals and connections of NPN and PNP transistors.
2. Demonstrate and measure the effects on base current of forward and reverse bias in the emitter-base circuit.

Theory:

It is convenient to represent the current voltage characteristics of transistor graphically. In a BJT common-emitter (CE) configuration, the emitter serves as the common terminal between input and output. The input is applied at the base terminal and the output is taken from the collector terminal. The typical CE output describes i_C as a function of V_{CE} with i_B as a parameter. In the active mode, i_C of practical BJTs shows some dependence on V_{CE} due to early effect. As a result, i_C characteristics are not horizontal straight lines. This dependency of i_c on V_{CE} is included in the equivalent circuit via an output resistance r_O.

Equipments and Components Needed:

1. Power supply
2. Multimeter
3. Resistor (100KΩ and 1.8KΩ)
4. Transistor (NPN and PNP)
5. Voltmeter
6. DC milli-ammeter and DC micro-ammeter

PROCEDURE:

1. Measure β (h_{FE}) of the transistor with the ammeter and record the value.
2. Construct the circuit as shown in the figure 1. Use DC power supply for V_{BB} and 0-15 V variable DC voltage from trainer board for V_{CC}. Connect the micro-ammeter in the base circuit and milli-ammeter in the collector circuit. Make sure to connect the ammeters with the correct polarity.
3. Adjust V_{BB} so that I_B ia about 5uA. Vary V_{CE} in steps of 1 V form 10 V to 1 V by changing V_{CC}. Keep I_B constant during this measurement. V_{CE} may be measured with digital multimeter. Next vary V_{CE} in steps of 0.1 V from 1 V to 0V. Measure I_C and V_{CE} in each step.

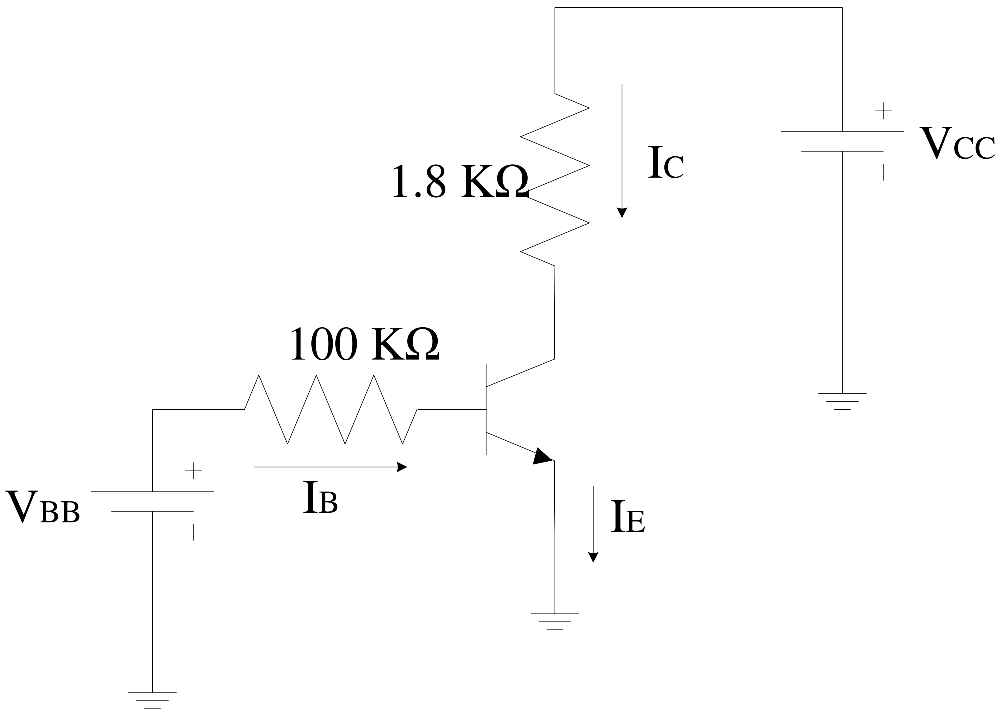


Figure 1 Circuit diagram for measuring I-V characteristics of Transistor

Post-Lab Report Questions:

Plot I_C-V_{CE} characteristics curves from the measured data using PSPICE and MATLAB

East West University
Department of Computer Science and Engineering

CSE251 Electronic Circuits
Lab Rules

Conduct:
1. Students must come to class on time. **No student will be allowed to enter into the lab class after 10 minutes of the start of the lab**.
2. Students should not use mobile phone in the lab class.
3. The students should collect the Lab-sheets a few days before the date of the experiment.
4. No lab makeup will be allowed except for unavoidable reasons.

Pre-Lab Report:
1. Students must bring completed pre-lab reports in the lab. **No student will be allowed to do the lab without pre-lab report.**
2. The pre-lab reports **should be hand-written** and should contain the following.
    (i) Top sheet containing Name of the University, Name of the Department, Semester, Course Number (Section), Course Title, Experiment Number, Experiment Title, Student ID, Student Name, Group Number, Group IDs, and Date of Performance.
    (ii) Objective of the Experiment.
    (iii) Circuit Diagram(s).
    (iv) Summary of the Experimental Procedure.
    (v) Answer(s) to the Pre-Lab Question(s), of any.
3. Have the Pre-Lab Report signed by your instructor.
4. **Students should not copy pre-lab report from others. In this case both the source and the copy will not be accepted and both the students will not be allowed to do the lab.**

Lab-Work:
1. Perform the lab-work as instructed in the lab-sheet of individual lab.
2. Have the Experimental Datasheet signed by your instructor.

Post-Lab Report:

1. **The post-lab report is due on the next lab class**. **The post-lab report should be computer-composed using MS Word. Graphs should be plotted using MATLAB**. **Circuit diagrams should be drawn using MS VISIO or PSPICE**. **Mathematical calculations should be composed using MS Equation Editor**.
2. The post-lab report should contain the following.
    - (i) Top sheet containing Name of the University, Name of the Department, Semester, Course Number, Course Title, Experiment Number, Experiment Title, Student ID, Student Name, Group Number, Group IDs, Date of Performance, and Date of Report Submission.
    - (ii) Objective(s) of the Experiment.
    - (iii) Circuit Diagram(s).
    - (iv) Experimental Datasheet.
    - (v) Answer(s) to the Post-Lab Report Question(s).
    - (vi) Conclusion.
    - (vii) Signed Pre-Lab Report.
3. **Experimental Datasheet has to be prepared as per the format given in the Lab Sheet. All comparisons needed for the Post-Lab Report Questions are to be made in a tabular form**.
4. **Students should not copy post-lab report from others. In this case both the source and the copy will not be graded**.