YILDIZ TECHNICAL UNIVERSITY

DEPARTMENT OF BIOMEDICAL ENGINEERING

BME2312 ANAOLOG ELECTRONICS LABORATORY

EXPERIMENT SHEETS

EXPERIMENT NO 1

Objectives

The objectives of Experiment 1 are

- to learn to read resistors by color bands
- to learn to use a multimeter
- to learn to use Oscilloscope
- to learn to build circuits using a breadboard
- to measure and calculate equivalent circuit resistance
- to verify Ohm's Law and Kirchhoff's Voltage Law.
- to build a low pass filter and analyze the output

A. BACKGROUND

Resistors

A component that is specifically designed to have a certain amount of resistance is called a resistor. The principal applications of resistors are to limit current in a circuit, to divide voltage, and, in certain cases, to generate heat. Although resistors come in many shapes and sizes, they can all be placed in one of two main categories: fixed or variable.

Fixed Resistors

Fixed resistors are available with a large selection of resistance values that are set during manufacturing and cannot be changed easily. They are constructed using various methods and materials. Figure 1 shows several common types.

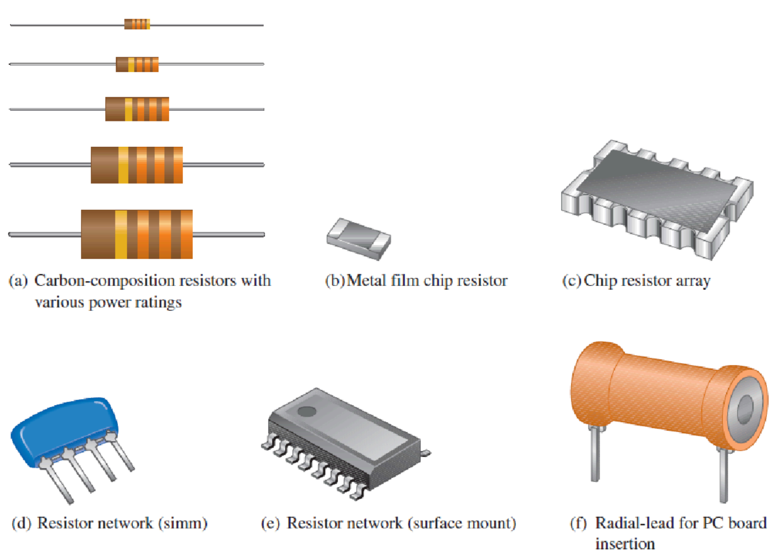


Figure 1. Typical Fixed Resistors

Resistor Color Codes

Fixed resistors with value tolerances of 5% or 10% are color coded with four bands to indicate the resistance value and the tolerance. This color-code band system is shown in Figure 2, and the color code is listed in Table 1. The bands are always closer to one end.

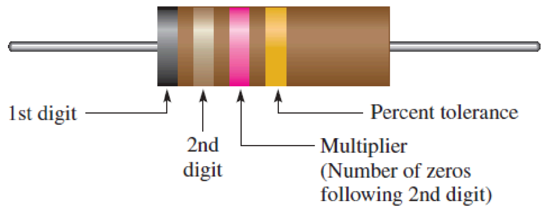


Figure 2. Color-code bands on a 4-band resistor.

The color code is read as follows:

1. Start with the band closest to one end of the resistor. The first band is the first digit of the resistance value. If it is not clear which is the banded end, start from the end that does not begin with a gold or silver band.
2. The second band is the second digit of the resistance value.
3. The third band is the number of zeros following the second digit, or the multiplier.
4. The fourth band indicates the percent tolerance and is usually gold or silver. If there is no band, it means 20% tolerance.
For example, a 5% tolerance means that the *actual* resistance value is within ±5% of the color-coded value. Thus, a resistor with a tolerance of ±5% can have an acceptable range of values from a minimum of 95 Ω to a maximum of 105 Ω.

Example: 470 kΩ □ Yellow, Violet, Yellow

Table 1 Resistor 4-band color code

	Digit	Color
Resistance value, first three bands: First band—1st digit Second band—2nd digit Third band—multiplier (number of zeros following the 2nd digit)	0	Black
	1	Brown
	2	Red
	3	Orange
	4	Yellow
	5	Green
	6	Blue
	7	Violet
	8	Gray
	9	White
Fourth band—tolerance	±5%	Gold
	±10%	Silver

Breadboard

A breadboard holds circuit components in place and connects them electrically. A breadboard is shown in Figure 3. The breadboard has many strips of metal that run underneath the plastic top. The metal strips are arranged as shown in the Figure 3. These strips connect to the holes on top of the board. This makes it easy to connect components together when building a circuit.

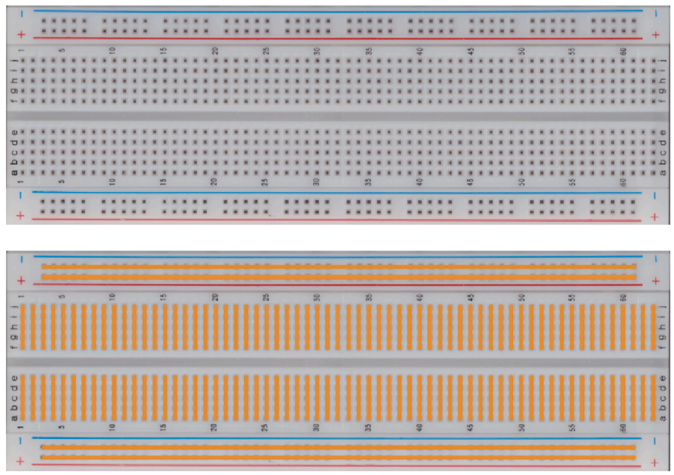


Figure 3. Breadboard and breadboard connection pattern

In Figure 4, a schematic of a circuit is given, and it is shown how to build it on the breadboard.

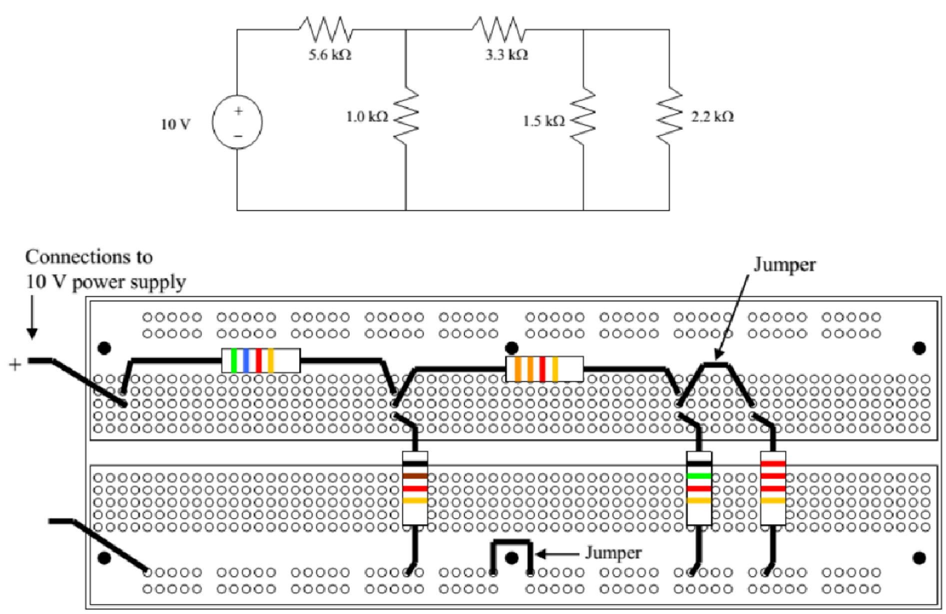


Figure 4.

Multimeter:

The multimeter, also called a volt-ohm meter (or VOM), is the basic tool for anyone working in electronics. You can see a fairly typical modern multimeter in Figure 5. You use a multimeter to take a variety of electrical measurements — hence the term "multi."

Figure 5. A simple multimeter

With this one tool, you can:

Measure AC voltages
Measure DC voltages
Measure resistance
Measure current going through a circuit
Measure continuity (whether a circuit is broken or not)

Voltmeter:

A voltmeter measures electrical potential between its terminals. Voltmeters are always placed in parallel with the circuit or circuit element where the voltage measurement is desired. Since the voltage across two or more parallel elements is the same, the voltage measured by the meter will be the same as the element to which the meter is connected. When using a non-auto-ranging meter, select the highest possible range and reduce the range as necessary until the desired level of accuracy is reached. Always start with a range higher than the expected value to prevent damage to the meter.

Ammeter:

An ammeter measures the current that flows between its terminals. An ammeter is always placed in series with the circuit or circuit element where the current flow is of interest. Since the current in each element of a series circuit is the same, the current flow through the meter will be the same as the current flow to the element of interest. Never connect an ammeter in parallel unless you intend to measure the short circuit current of a circuit or circuit element and you have made sure that destructive current levels won't be reached. When using a non-auto-ranging meter, select the highest possible range and reduce the range as necessary until the desired level of accuracy is reached. Always start with a range higher than the expected value to prevent damage to the meter.

Ohmmeter:

An ohmmeter measures the electrical resistance between its terminals. An ohmmeter is connected to the circuit or circuit element of interest after the element of interest has been isolated from the rest of the circuit. The element of interest has to be isolated from the rest of the circuit so that its resistance value isn't obscured by the resistance values of the other circuit components connected to the element of interest. Never connect an ohmmeter to an energized circuit or the meter could be destroyed.

Measurement Errors:

- ***Absolute Error***

In general, the result of any measurement of physical quantity must include both the value itself and its error. The result is usually quoted in the form

$\pm\Delta X = X_0 - X_{measured}$

where X_0 is the best estimate of what we believe is a true value of the physical quantity and ΔX is the estimate of absolute error (uncertainty). ΔX indicates the reliability of the measurement, but the quality of the measurement also depends on the value of X_0.

- ***Fractional Error***

Fractional error is defined as $\frac{\Delta X}{X_0}$

Fractional error can be also represented in percentile form: $\frac{\Delta X}{X_0} \times 100$

Ohm's law states that current is directly proportional to voltage and inversely proportional to resistance. Ohm's law is given in the following formula:

$$I = \frac{V}{R}$$

where I is current in amperes (A), V is voltage in volts (V), and R is resistance in ohms

Kirchhoff's Voltage Law:

Kirchhoff's voltage law is a fundamental circuit law that states that the algebraic sum of all the voltages around a single closed path is zero or, in other words, the sum of the voltage drops equals the total source voltage.

Kirchhoff's voltage law applied to a series circuit is illustrated in Figure 6. For this case, Kirchhoff's voltage law can be expressed by following equation:

$V_S = V_1 + V_2 + V_3 + \ldots + V_n$

where the subscript n represents the number of voltage drops.

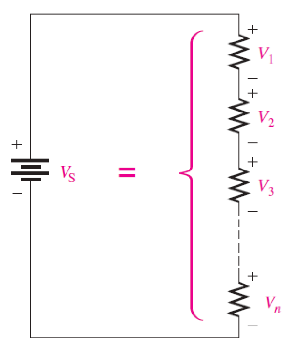


Figure 6. Sum of n voltage drops equals the source voltage.

If all the voltage drops around a closed path are added and then this total is subtracted from the source voltage, the result is zero. This result occurs because the sum of the voltage drops always equals the source voltage.

Therefore, another way of expressing Kirchhoff's voltage law in equation form is

$V_S - V_1 - V_2 - V_3 - \ldots - V_n = 0$

Oscilloscope

Oscilloscope is a measuring instrument used to extract the characteristics of circuit elements and to examine voltages and currents that change over time. It is also used to measure amplitude, frequency and phase difference in the simultaneous examination of one or more rapidly changing signals.

B. *PRELIMINARY WORK:*

1. Find out how to read the value of a resistor using color bands.
2. Find out working principles of voltmeter, ammeter and ohmmeter.
3. Find out how to use the multimeter which you purchased. Read its user manual.
4. Find out how to connect circuit components in series or parallel on a breadboard.
5. The values of resistors are given in Table 2. Write the corresponding color bands into related fields in Table 2.

Table 2

The resistance	1st color	2nd color	3rd color	4th color
820 Ω ± 10%				
2.7 kΩ ± 5%				
56 kΩ ± 20%				
47 kΩ ± 5%				
1.8 kΩ ± 20%				

6. The color bands are given in Table 3. Write the corresponding resistors into related fields in Table 3.

Table 3

1st color	2nd color	3rd color	4th color	The resistance
Brown	Green	Orange	No band	
Green	Blue	Black	Silver	
Yellow	Violet	Red	Gold	
Blue	Gray	Black	Gold	
Orange	White	Orange	No band	

7. Calculate all node voltages and mesh currents in Fig. 10 using nodal and mesh analysis and write your results into related fields in Table 4.
8. Determine voltages across each component in Fig. 10 using calculated node voltages in the previous question and write them into related fields in Table 4.
9. Show that Kirchhoff's Voltage confirms your calculations for Fig. 10.
10. Write the transfer function of the low-pass filter circuit shown in Figure 11. Using this function, calculate the amplitude of the output signal for the input signals given in Table 6.

Instructions for preliminary work report:

1. All calculations must be given in your report.
2. All the tables must be given in your report. You will write your calculations into related fields and other fields will be completed in the experiment.
3. Using ORCAD, build each circuit schematic and add them to your report with simulation results for all circuits in the order of experimental part. Give all the requested results in the experimental part as graphics and/or tables. Simulation results must be clear. Background must be white and data trace width must be in proper thickness.
4. Using TinkerCAD, build each circuit schematic and add them to your report with the results for all circuits in the order of experimental part.

This preliminary work report is of prime importance to check your results in experiment.

C. *EXPERIMENTAL PART:*

1. Build the circuit given in Fig. 10, measure voltages and currents given in Table 5 using a multimeter and write them in related places in Table 5. Write expressions for the generated and dissipated powers and calculate them.

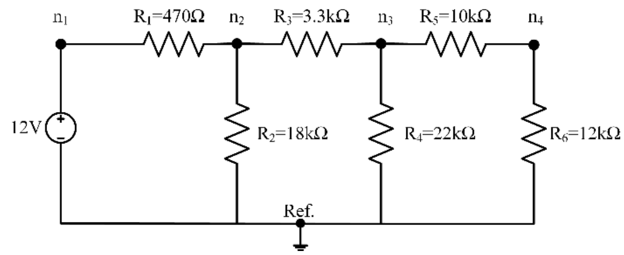


Figure 10.

Table 4

	$V(n_1)$ (V)	$V(n_2)$ (V)	$V(n_3)$ (V)	$V(n_4)$ (V)	$V(R_1)$ (V)	$V(R_2)$ (V)	$V(R_3)$ (V)	$V(R_4)$ (V)	$V(R_5)$ (V)	$V(R_6)$ (V)	$I(R_1)$ (A)	$I(R_3)$ (A)	$I(R_5)$ (A)
Calculated													
Measured													

Table 5

Generated power:	
Dissipated power:	

2. Build the circuit given in Figure 11. Apply a sinusoidal signal (Vpp=10V) from the signal generator. Measure the output voltage on the 1μF capacitor according to changing frequency values given in Table 6 by using an oscilloscope and write the results in related places in Table 7.

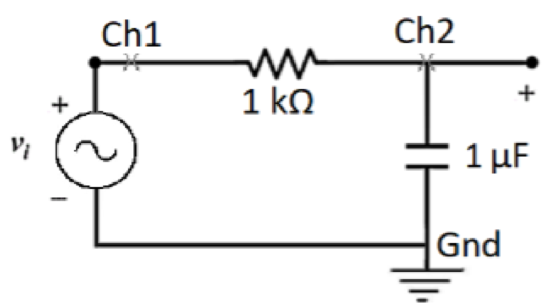


Figure 11

Table 6

V_{in} (V)	f_{in} (Hz)	V_{out}	Av
10	20		
10	40		
10	80		
10	160		
10	320		
10	640		
10	1280		
10	2560		

EXPERIMENT 2: DIODE CHARACTERISTICS & DIODE APPLICATIONS (CLIPPER CIRCUITS)

Objectives

The objective of Experiment 2 is to learn current-voltage (I–V) characteristics of diodes and the working principle of clipper circuits.

Components Required:

- **Diodes:** 2x1N4001, 1xLED and 2x5.6V zener diode
- **Resistors:** 2x1kΩ
- Breadboard, jumpers, multimeter

Preliminary Work:

1. Study the characteristics of the diodes (frequency ranges, maximum conduction currents, breakdown voltages).
2. Analyze each circuit theoretically.
3. Perform simulations of the part of experimental work in OrCAD.
   a) **Part 1:** Plot current–voltage (**ID–VD**) characteristic of the diodes) and add them to your report.
   - You will perform DC sweep analysis. You can find an example in the link below for your simulations.
     https://www.youtube.com/watch?v=WEio1P1oA_g

   b) **Part 2:** Setup the circuits given in Figure 4 and Figure 5 in OrCAD. Use 1N4001 diode model. Set frequency to 1kHz and plot the input and output voltages on the same plot pane for two time periods. Repeat the simulation for V1= 0 and V1=12V for Figure 4. Repeat the simulation for voltage values given in a, b,c for the circuit given in Figure 5. Observe how the output changes. Explain how the circuits work.

4. Perform simulations of all of the circuits in the experimental work using TinkerCAD.

5. Fill Table 1, draw the characteristic curves in Figure 2 and Figure 3 according to your simulation results and add them to your report .

Experimental Work:

Part 1: Diode Characteristics

1. Setup the circuit given in Figure 1 for the component of 1N4001. Apply eight different input voltages and write down the measured voltage values (V1 & V2) to Table 1.
2. Repeat part 1 for the other components named as LED and 5.6V Zener. Draw the outputs on Figure 2 and Figure 3.

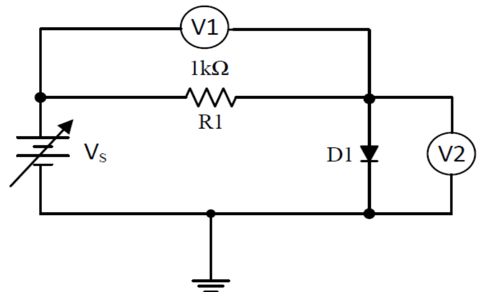


Figure 1

Table 1

Vs	1N4001		Vs	LED		Vs	Zener	
	V1	V2		V1	V2		V1	V2
0.2V			0.2			-7V		
0.4V			0.5V			-6V		
0.6V			0.8V			-4V		
0.7V			1V			-1V		
0.9V			1.5V			0V		
1.5V			2V			0.4V		
2V			2.5V			0.7V		
3V			3V			1V		

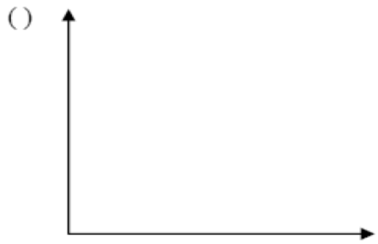


Figure 2

Figure 3

Part 2: Diode Applications-Clipper Circuits

1. Build the circuit given in Fig. 4. Connect a 10**Vp** sinusoidal voltage source (f=1kHz) and draw the output that you observe on the oscilloscope for V1=0V. Observe how the output changes if V1 is tuned to 5V and 12V.

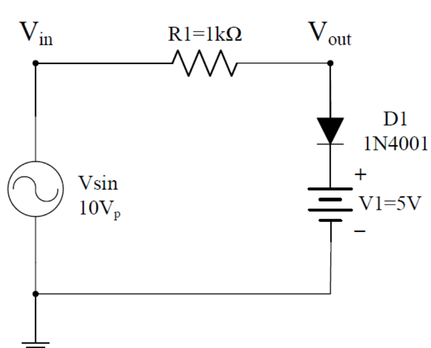


Figure 4

2. Build the circuit in Fig. 5. Draw the output that you observe on the oscilloscope for the voltages of V_1 and V_2 given below.

   a. V_1=5V and V_2=5V
   b. V_1=2.5V and V_2=5V
   c. V_1=5V and V_2=2.5V

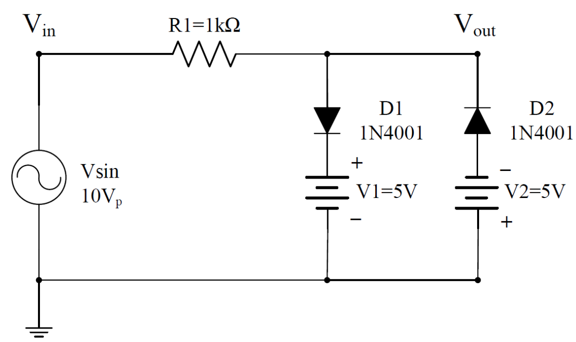


Figure 5

EXPERIMENT 3: DIODE APPLICATIONS PART CLAMPER & RECTIFIER CIRCUITS

Objectives

The objectives of Experiment 3 are

1. To understand clamper circuits.
2. To understand rectifier circuits.

Components Required:

- 1N4148, 1N4001, 1kΩ, 100kΩ, 1µF, 100µF, LEDs (4 in two different colors).

Preliminary Work:

1. Study clamper and rectifier circuits from your lecture notes and textbook.
2. **Analyze each circuit theoretically.**
3. Perform simulations of the part of experimental work in ORCAD and add them to your report. All diodes are 1N4001 except for in Figure 1. Apply 1kHz Vsin voltage.

a) (Figure 1)
   - ☐ Simulate the circuit and plot the **input and the output voltages on the same plot pane** for three periods (Use white background in plot pane and change the graphic line thickness to be legible).
   - ☐ Explain how the circuit works.

b) (Figure 2 & Figure 3)
   - ☐ Choose R1=1k, C1=1µF. Apply 1kHz Vsin voltage.
   - ☐ Firstly, observe the output signal when the capacitor is not connected.
   - ☐ Secondly, observe it when the capacitor is connected.
   - ☐ Simulate the same circuit when R1=100k and C1=100µF.
   - ☐ Give simulation results in your homework for each case.
   - ☐ Plot the **input and the output voltages on the same plot pane** for three periods (Use white background in plot pane and change the graphic line thickness to be legible).
   - ☐ Explain how the circuits work.

c) Perform simulations of all of the circuits in the experimental work using TinkerCAD.

Experimental Work:

Clamper circuits:

1. Build the circuit in Figure 1. Draw the output that you observe on the oscilloscope for V_1=5V.

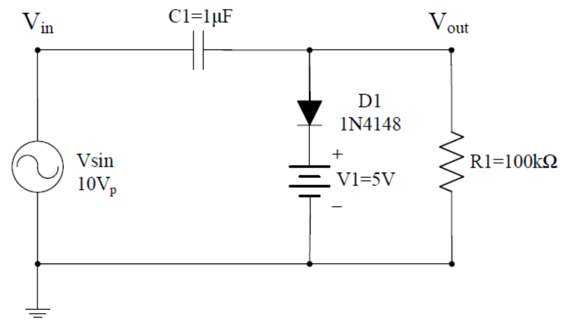


Figure 1

Rectifier circuits:

2. Build the one-way rectifier given in Figure 2, observe the output signal when the capacitor is connected and when it is not. Choose R1=1kΩ, C1=1µF, and R1=100kΩ and C1=100µF. Examine the operation of the circuit and record the results. Diode is 1N4001.

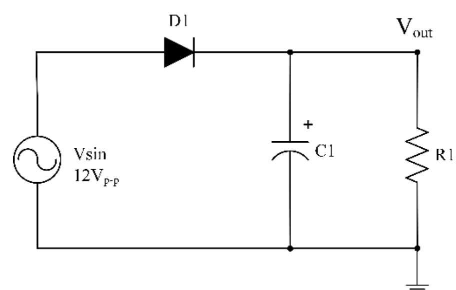


Figure 2

3. Build the bridge type rectifier given in Figure 3, observe the output signal when the capacitor is connected and when it is not. R1=1kΩ and C1=1µF. Examine the operation of the circuit and record the results. Comment on the difference with the other circuits.

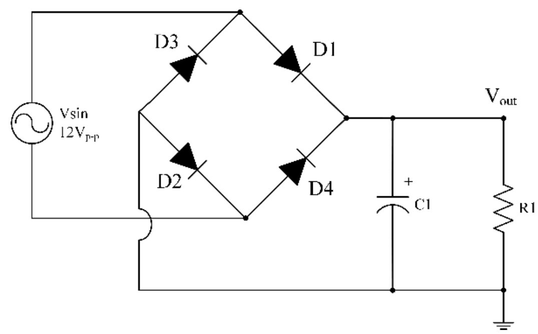


Figure 3

EXPERIMENT 4: MOSFET CHARACTERISTICS

Objectives

The objective of this experiment is to obtain DC characteristics of a MOSFET and to learn the operation of a MOSFET practically.

Components Required:

DC Voltage Source

Oscilloscope

Multimeter

Transistor: BS108 transistor

Resistors: 1kΩ

Preliminary Work:

1. Find and examine the datasheet of the BS108 transistor on the Internet. Write the critical information for the experiment. Which leg of the MOSFET corresponds to which of G, D, S. Using the datasheet, find this information and add it to your report.
2. Explain how we perform a robustness test of a MOSFET using a multimeter.
3. Research how to extract and use the MOSFET's input and output characteristics. Give brief information about this in the report.
4. Can a MOSFET be used as a switch? Explain how it can be used as a switch.
5. What is the meaning of 'threshold voltage (V_{th})' for the MOSFETs?
6. Setup the circuit given in Figure 1 in OrCAD. You can refer to Figure 2 to understand how to add the MOSFET in the circuit. Plot input characteristic of the transistor (VGS-ID). You can select the analysis type as 'DC Sweep' and options as 'Primary Sweep' with 'Secondary Sweep'. Primary Sweep input voltage (VGS) from 0 to 10V in steps of 0.1V. Secondary Sweep input voltage (VDS) from 0 to 2V in steps of 0.5V. Specify critical points on the plot and **comment about your simulation result.**
7. Setup the circuit given in Figure 1 in OrCAD. You can refer to Figure 2 to understand how to add the MOSFET in the circuit. Plot output characteristic of the transistor (VDS-ID). You can select the analysis type as 'DC Sweep' and options as 'Primary Sweep' with 'Secondary Sweep'. Primary Sweep input voltage (VDS) from 0 to 10V in steps of 0.1V. Secondary Sweep input voltage (VGS) from 0 to 2V in steps of 0.5V. Specify critical points on the plot and **comment about your simulation result**.
8. Perform simulations of all of the circuits in the experimental work using TinkerCAD.

- Add Table 1 and Table 2 to your report.
- The link below may help you to simulate the circuit:

https://www.youtube.com/watch?v=4wv3CUQE3Q0

Experimental:

Set up the circuit given in Figure 1.

1. Set the VDS source to 2V.
- Then, determine the threshold voltage of VGS (Vth). **Use an oscilloscope** for this purpose.
- Connect the oscilloscope probes between the drain terminal and ground.
- Add Vmax to the oscilloscope screen. Remove other parameters from the oscilloscope screen.
- Increase the VGS value from zero to 2 (Look at Table 1 for step size).
- Fill in Table 1 for the Vmax values you obtained.
- Determine Vth using the measured VR voltages (see Table 1).
2. Set the VGS source to 0.3V.
- Connect the oscilloscope probes' resistor terminals.
- Add Vmax to the oscilloscope screen. Remove other parameters from the oscilloscope screen.
- Increase the VDS value from zero to 10 (Look at Table 2 for step size).
- Fill Table 2 helping with Vmax values you obtained. (Attention:You should write current values instead of voltage values)
- Repeat the same steps for VGS=0.7V and VGS=1V and fill Table 2.
- Draw the output characteristic of the MOSFET using the obtained values from **VGS=1V.**

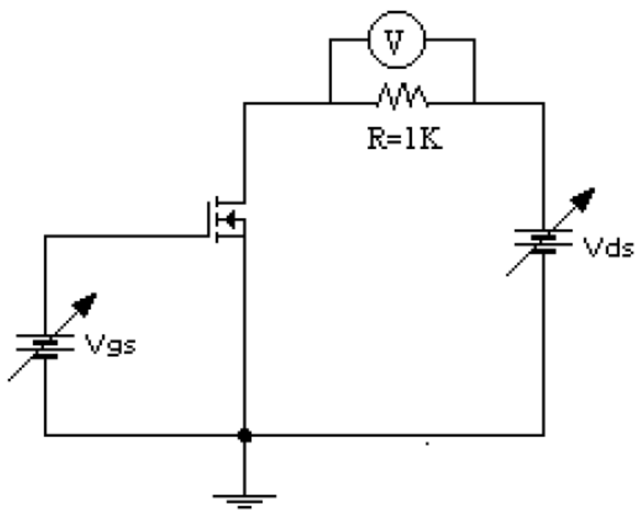


Figure 1

VGS	***VD***
0 V	
0.1 V	
0.2 V	
0.3 V	
0.5 V	
0.7V	
0.8 V	
1 V	

Table 1

Vth =

Comments about Vth:

VDS	***VGS=0.3V Ids***	***VGS =0.5V Ids***	***VGS=0.7V Ids***	***VGS =1V Ids***	***VGS =1.5V Ids***
0 V					
0.1 V					
0.2 V					
0.3 V					
0.5 V					
0.7V					
1 V					
2 V					
4 V					
6 V					
8 V					
10 V					

Table 2

Comments about output characteristic of the MOSFET:

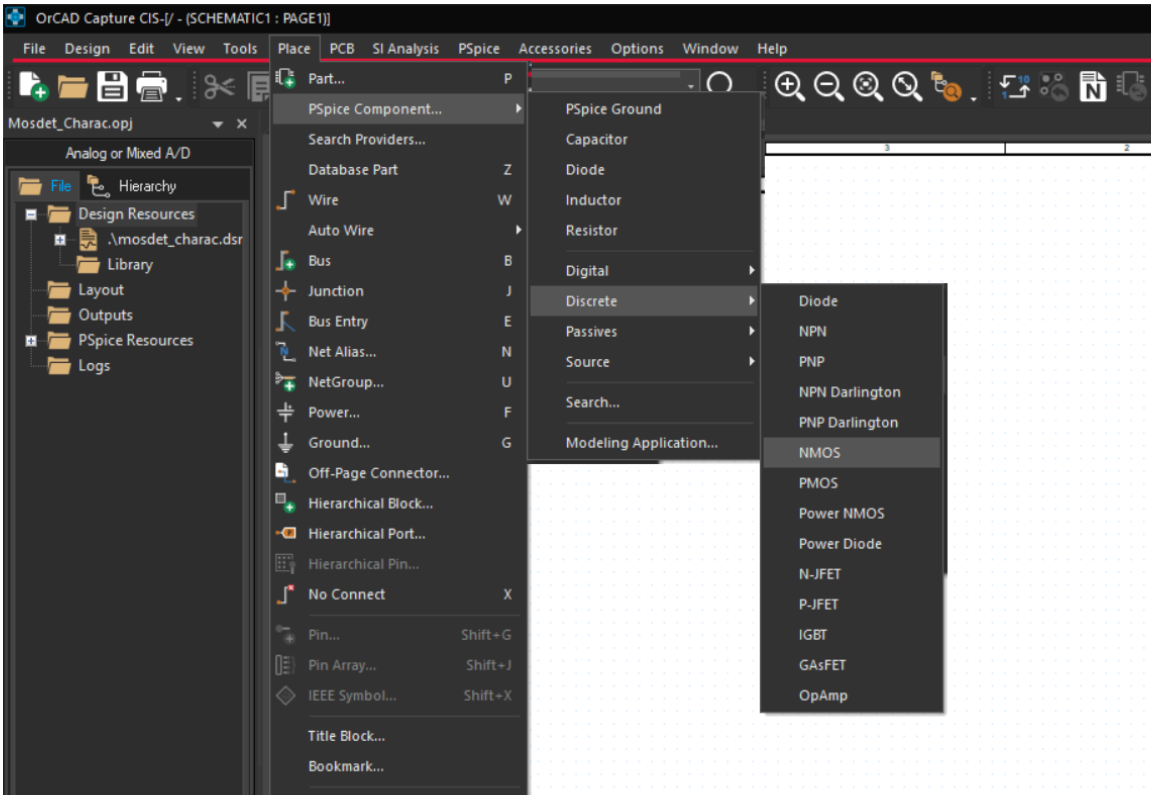

OrCAD Capture CIS-[/ - (SCHEMATIC1 : PAGE1)]
File
Design
Edit
View
Tools
Place
PCB
SI Analysis
PSpice
Accessories
Options
Window
Help
Mosdet_Charac.opj
Analog or Mixed A/D
File
Hierarchy
Design Resources
.\mosdet_charac.dsn
Library
Layout
Outputs
PSpice Resources
Logs
Part... P
PSpice Component...
Search Providers...
Database Part Z
Wire W
Auto Wire
Bus B
Junction J
Bus Entry E
Net Alias... N
NetGroup... U
Power... F
Ground... G
Off-Page Connector...
Hierarchical Block...
Hierarchical Port...
Hierarchical Pin...
No Connect X
Pin... Shift+G
Pin Array... Shift+J
IEEE Symbol... Shift+X
Title Block...
Bookmark...
PSpice Ground
Capacitor
Diode
Inductor
Resistor
Digital
Discrete
Passives
Source
Search...
Modeling Application...
Diode
NPN
PNP
NPN Darlington
PNP Darlington
NMOS
PMOS
Power NMOS
Power Diode
N-JFET
P-JFET
IGBT
GAsFET
OpAmp

Figure 2

EXPERIMENT 5: MOSFET AMPLIFIER

Objectives

The objective of this experiment is to obtain and learn the gain characteristic of MOSFET amplifier circuits practically.

Components Required:

DC Voltage Source

Signal Generator

Multimeter

Transistor: BS108 transistor

Resistors: 220Ω, 1kΩ, 10kΩ, 330kΩ, 560Ω, 1MΩ

Capacitor: 1µF (x3)

Preliminary Work:

- ☐ ***Theoretical:***

1. Study MOSFET amplifier circuits, gain, frequency characteristics, and explain how the frequency value affects the characteristics. Give the information about that in your report. Explain how the MOSFET works in the low and high frequency regions.

2. Analyze the circuit given in Figure 1 and find the gain for 100 kHz input signal (You can find a short video about MOSFET' AC analysis in the link- https://www.youtube.com/watch?v=dLLN4fLsXb0). Use the parameters given in 'Notes 1'for calculations.
   - Do dc analysis of the circuit to find the bias point of the circuit. Insert the Id value into the equation below in order to find gm.

   $$g_m = \sqrt{2k_n \frac{W}{L} I_D}$$

   - Calculate AC voltage gain. You may ignore the r_0 resistance in the MOSFET in the AC equivalent circuit.
   - Fill the part of 'Theoretical Value' in Table 1 according to your calculations.

- ☐ ***Simulation***

1. Set up the circuit given in Figure 1 in OrCAD. You can refer to Figure 3 to understand how to add the MOSFET in the circuit. Use the PSpice model given in 'Note 2' below. (Right click on the Mosfet and select the 'Edit Pspice Model' and then you can find a pane like in Figure 2. You must paste the model parameters given in 'Note 2' in the pane. Then click the 'Save Library'.)

Use AC Voltage Source for the input signal. Run AC simulation from 100 Hz to 100 MHz (100 MEG). Set type of sweep as "decade" with 101 point per decade. Plot the AC voltage gain (*vout* / *vin*). What is the midband voltage gain as a ratio (not in dB)?

3. Change the input signal with a sinusoidal voltage source that has 240mV amplitude at **100kHz**. Run transient simulation. Plot the input and output signals for 5 periods of the signal. What is the gain based on the transient simulation? Does it agree with the AC gain in the question (3) and question (2)?
4. Determine if there is a phase difference between the input and output signal.
5. Design how to set the circuit in Figure 1 on the breadboard. Add your drawing to your report.
6. Set up the circuit given in Figure 1 in TincerCAD and add the results to the report.
7. Before coming to experiment, check that you have the components required.

Experimental:

- Set up the circuit given in Figure 1.
- Apply the sinus signal to the input so that the Vpp value is 240 mV and the frequency value is 100 kHz.
- Measure the VG, VD, VS and ID values. Fill in Table 1.
- Determine the gain.
- Find the cut-off (corner) frequency points of the MOSFET by changing the frequency values of the input signal according to Table 2. Fill in Table 2 according to the values you obtained.

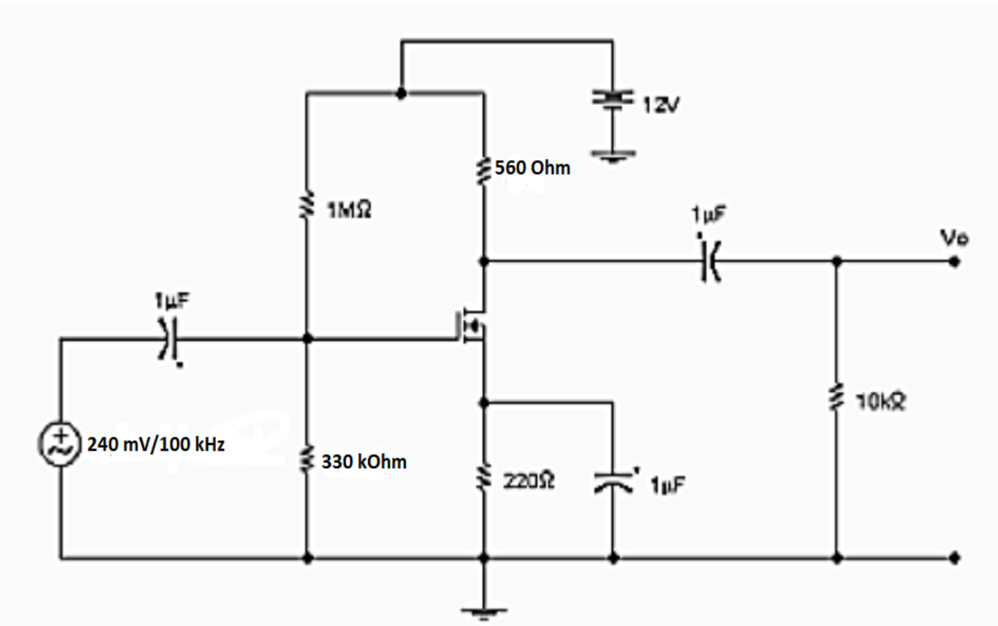


Figure 1

	Theoretical Value	**Measured Value**
VD		
VG		
VS		
ID		

Table 1

FREQUENCY (f)	**Vo**	**Vo/Vin**
5 kHz		
10 kHz		
50 kHz		
100 kHz		
200 kHz		
500 kHz		
1 MHz		
1.5 MHz		
2 MHz		
3 MHz		

Table 2

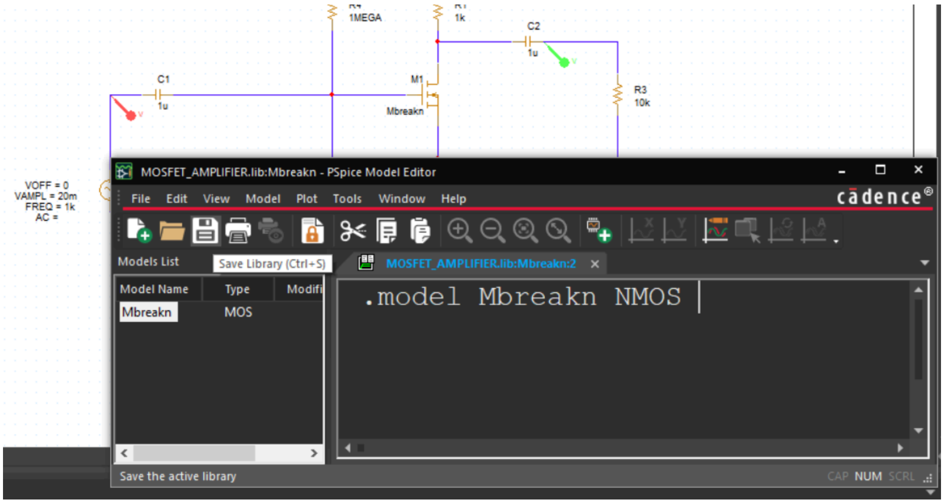


Figure 2

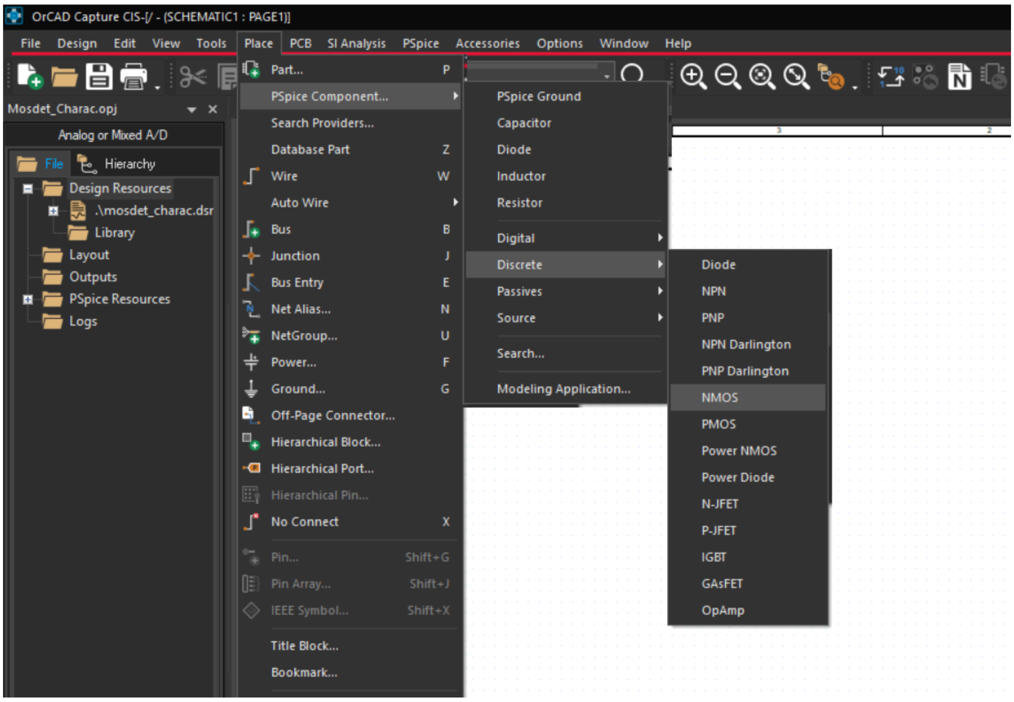


Figure 3

Note 1:

V_{TN}= 0.675V, k_n =178.5 mA/V^2, W=100u, L=100u, λ=0.02

Note 2:

.model Mbreakn NMOS VTO=0.675 KP=0.1785 W=100u L=100u LAMBDA=0.02

+CGSO=2.1u CGDO=2.1u CGBO=0

EXPERIMENT 6: BIPOLAR JUNCTION TRANSISTOR (BJT) CHARACTERISTICS

Objectives

The objective of this experiment is to obtain DC characteristics of BJT transistors and to learn the operation of bipolar transistors practically.

Components Required:

DC Voltage Source

Multimeter

Transistor: BC237 transistor

Resistors: 120kΩ, 1kΩ,100Ω

Preliminary Work: Find and examine the datasheet of the BC237 transistor on the Internet.

1. Research how to extract and use the transistor's transfer and output characteristics.
2. Identify the operating regions of a BJT transistor. How are the operating regions determined? Explain.
3. What are the signs (positive or negative) of I_E, I_C, I_B, V_{CB} and V_{EB} for a PNP transistor operating in the active area?
4. What are the signs (positive or negative) of I_E, I_C, I_B, V_{CB} and V_{EB} for an NPN transistor operating in the active area?
5. An NPN - BJT is used as a switch. Write the necessary conditions for BJT to be operated in saturation or in cut-off.
6. Setup the circuit given in Figure 1 in OrCAD. Use BC237 (in Zetex Library) transistor model. Plot input characteristic of the transistor (VBE–IB). You can select the analysis type as 'DC Sweep' and options as 'Primary Sweep'. Sweep input voltage (VBB) from 0 to 6V in steps of 0.1V. Specify critical points on the plot and comment about your simulation result.
7. Setup the circuit given in Figure 2 in OrCAD. Use BC237 transistor model. Plot output characteristic of the transistor (VCE–IC). You can select the analysis type as 'DC Sweep' and options just as 'Primary Sweep'. Sweep input voltage (VCC) from 0 to 15V in steps of 1V. Specify critical points on the plot and comment about your simulation result.
8. Setup the circuit given in Figure 2 in OrCAD. Use BC237 transistor model. Plot output characteristic of the transistor (VCE–IC). You can select the analysis type as 'DC Sweep' and options as 'Primary Sweep' with 'Secondary Sweep'. Primary Sweep input voltage (VCC) from 0 to 15V in steps of 1V. Secondary Sweep input voltage

(VBB) from 0 to 3V in steps of 1V. What is the relationship between IC and IB according to the results? What is the relationship between VCE and IC according to the result? Specify critical points on the plot and comment about your simulation result.

9. Set up the circuits given in Figure 1 and Figure 2 in TincerCAD and add the results to the report.

10. Add the tables (Table 1, Table 2 and Table 3) to your report.

Experimental Work:

1. Set up the circuit in Figure 1 and measure the V_B, V_C, and V_E voltages for the V_S values given in Table 1 and write them in Table 1. Calculate the values of I_B, I_C, I_E, β, V_{BE}, V_{CE} according to the measured values and write them down in Table 2.
2. Set up the circuit in Figure 2 and measure the V_B, V_C, and V_E voltages for V_{CC} values given in Table 3. Calculate the values of I_B, I_C, and V_{CE} and write them in Table 3.
3. Draw the $I_C - V_{CE}$, $I_C - I_B$, $I_B - V_{BE}$ curves using the calculated data in the tables.

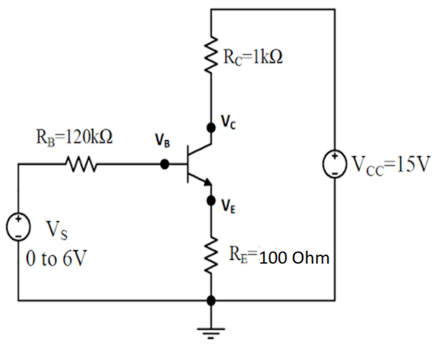


Figure 1

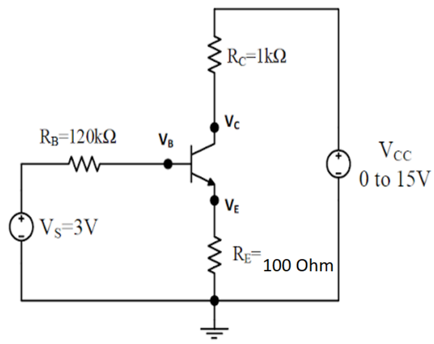


Figure 2

Table 1: Measurements according to the given V_S voltages

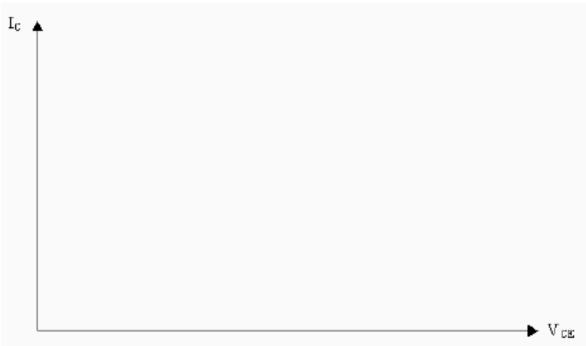
V_S (V)	V_S	V_B	V_E	V_C
0.5				
1.0				
1.5				
2.0				
2.5				
3.0				
3.5				
4.0				
4.5				
5.0				
5.5				
6.0				

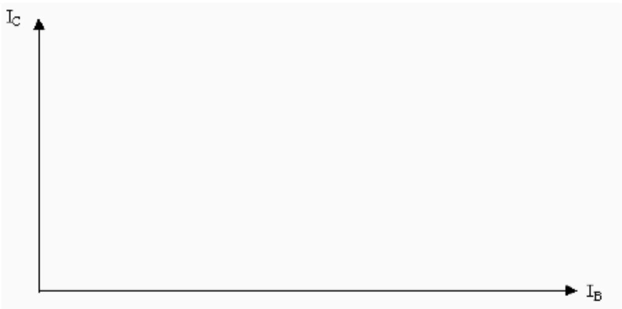
Table 2: The values below will be calculated according to the table above!

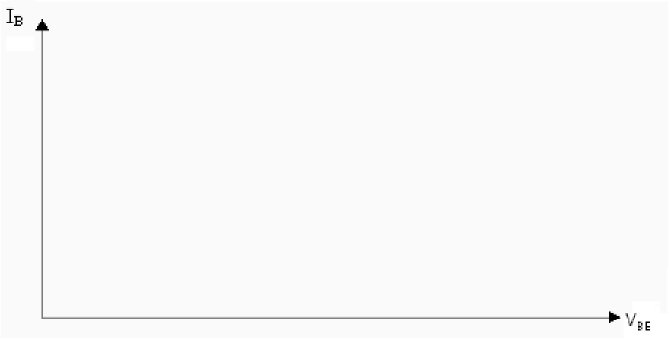
V_S (V)	I_B	I_C	I_E	β	V_{BE}	V_{CE}
0.5						
1.0						
1.5						
2.0						
2.5						
3.0						
3.5						
4.0						
4.5						
5.0						
5.5						
6.0						

Table 3: Measurements according to the given V_{CC} voltages

V_{CC}	V_B	V_E	V_C	I_B	I_C	V_{CE}
0.2						
1						
3						
5						
6						
8						
10						
15						






I_B
V_{BE}

EXPERIMENT 7: BJT AMPLIFIER

Objectives

The objective of this experiment is to obtain and learn the gain characteristic of BJT amplifier circuits practically.

Components Required:

DC Voltage Source

AC Voltage Source

Oscilloscope

Multimeter

Transistör: BC237

Resistors: 220kΩ, 1kΩ, 10kΩ, 33kΩ, 1.2kΩ, 20kΩ

Capacitor: 4.7μF (x2), 220 μF

Preliminary Work:

1. Set up the circuit given in Figure 1 in ORCAD. Use BC237 transistor model.

a. Analyze the circuit given in Figure 1 and find IB, IC, and VCE. (β_{DC} = 300)

b. Simulate the same circuit in OrCAD. Label base, emitter, and collector terminals of the transistor as B, E, and C, respectively. Perform DC simulation and provide the screenshot of the results. (Use 'Bias Point' analysis)

c. Do your simulation results justify the answers in part (a)? (IB, IC, and VCE)

d. Determine in which region the transistor operates.

2. Set up the circuit given in Figure 1 in OrCAD. Use the BC237 transistor model.

a. Find frequency response of the BC237 Transistor in ORCAD. Use AC Voltage Source for the input signal. Run AC simulation from 5 Hz to 10 MHz (10 MEG). Plot the AC voltage gain (*Vo* / *Vs*). Label the midband frequency gain (in dB) and low and high corner frequencies (the 3-dB frequencies) in the plot before importing it to your homework. What is the midband voltage gain as a ratio (not in dB)?

b. Change the input signal with a sinusoidal voltage source that has 220mV Vpp at 1kHz. Run transient simulation. Plot the input and output signals. What is the gain based on the transient simulation? Does it agree with the AC gain in part (a)?

3. Set up the circuits given in Figure 1 in TincerCAD and add the results to the report.

Experimental :

1. Setup the circuit given in Figure 1 using the BC237 transistor.
    - Set the supply voltage to 20V.
    - Apply a sine wave to the input at a frequency of 1 kHz and Vpp value is 240 mV. Connect the first channel of the oscilloscope to the signal generator (V_s) and the second channel to the output (V_o) of the circuit.
    - Measure the DC node voltages of collector, base, and emitter and write down the measured voltage values (Ic, V_C V_B & V_E) to Table 1.
    - Calculate the value of V_o/V_s.

2. Find the cut-off (corner) frequency points of the BC237 transistor by changing the frequency values of the input signal according to Table 2. Fill in Table 2 according to the values you obtained.

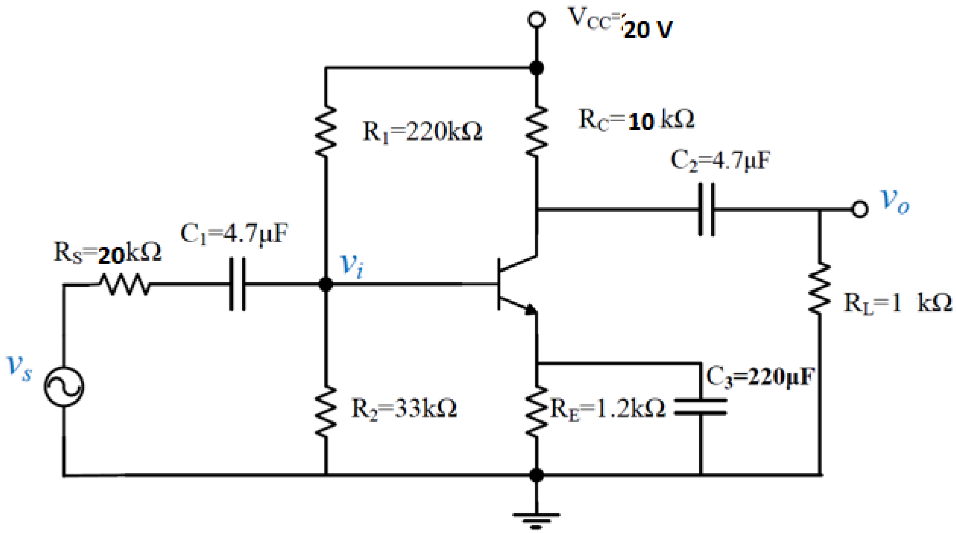


Figure 1

Table 1: DC Values

	Theoretical values	**Measured values**
V_C		
V_B		
V_E		
I_C		

Table 2 : Frequency Response of the BC237 Transistor

FREQUENCY (f)	Vo	Vo/Vs
5 Hz		
18 Hz		
40 Hz		
60 Hz		
500 Hz		
1 kHz		
5 kHz		
10 kHz		
50 kHz		
100 kHz		
170 kHz		
250 kHz		
500 kHz		