INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500 043

Lab Manual:

ELECTRONIC DEVICES AND CIRCUITS LABORATORY(AECC05)

Prepared by

AJITHA G(IARE10052)

ELECTRONICS AND COMMUNICATION ENGINEERING
INSTITUTE OF AERONAUTICAL ENGINEERING

December 3, 2021

Contents

INTRODUCTION

1.1 Introduction

This course is intended to enhance the learning experience of the student in topics encountered in AECB09. In this lab, gives the general awareness of the various components, equipment's and other details required to carry out the prescribed experiments without much difficulty to the students. students are expected to gain experience in using the basic electronic devices used in electronics and communication engineering and in interpreting the results of measurement operations in terms of the concepts introduced in the Electronic Devices and Circuits course. How the student performs in the lab depends on his/her preparation, participation, and teamwork. Each team member must participate in all aspects of the lab to insure a thorough understanding of the equipment and concepts. The student, lab assistant, and faculty coordinator all have certain responsibilities toward successful completion of the lab's goals and objectives.

1.1.1 Student Responsibilities

1. The student is expected tocome prepared for each lab.Lab preparation includes understanding the labexperiment from the lab manual and reading the related textbook material.

2. Students have to write the allotted experiment for that particular week in the work sheets given and carry them to the Lab. In case of any questions or problems with the preparation, students can contact the Faculty Teaching the Lab course, but in a timely manner.

3. Students have to be in formal dress code, wear shoes and lab coat for the Laboratory Class.

4. After the demonstration of experiment by the faculty, student has to perform the experiment individually. They have to note down the observations in the observation Tables drawn in work sheets, do the calculations and analyze the results.

5. Active participation by each student in lab activities is expected. The student is expected to ask the Faculty any questions they may have related to the experiment.

6. The student should remain alert and use commonsense while performing the lab experiment.They are also responsible for keeping a professional and accurate record of the labexperiments in the files provided.

1.1.2 Responsibilities of Faculty Teaching the Lab Course

The Faculty shall be completely familiar with each labprior to the laboratory. He/She shall provide the students with details regarding the syllabus and safety review during the first week.Lab experiments should be checked in advance to make sure that everything is in working order.The Faculty should demonstrate and explain the experiment and answer any questions posed by the students.Faculty have to supervise the students while they perform the lab experiments. The

Faculty is expected to evaluate the lab worksheets and grade them based on their practical skills and understanding of the experiment by taking Viva Voce. Evaluation of work sheets has to be done in a fair and timely manner to enable the students, for uploading them online through their CMS login within the stipulated time.

1.1.3 Laboratory In-charge Responsibilities

The Laboratory In-charge should ensure that the laboratory is properly equipped, i.e., the Faculty teaching the lab receive any equipment/components necessary to perform the experiments.He/She is responsible for ensuring that all the necessary equipment for the lab is available and in working condition. The Laboratory In-charge is responsible for resolving any problems that are identified by the teaching Faculty or the students.

1.1.4 Course Coordinator Responsibilities

The course coordinator is responsible fo rmaking any necessary corrections in Course Description and lab manual. He/She has to ensure that it is continually updated and available to the students in the CMS learning Portal.

1.2 Lab Policy and Grading

The student should understand the following policy:

ATTENDANCE: Attendance is mandatory as per the academic regulations.

LAB RECORD's: The student must:

1. Thoroughly read the prelab of corresponding experiment before entering to the lab.
2. Thoroughly read the prelab of corresponding experiment before entering to the lab.
3. Perform the experiment and record the observations in the worksheets.
4. Analyze the resultsand get the work sheets evaluated by the Faculty.
5. Upload the evaluated reports online from CMS LOGIN within the stipulated time.

Grading Policy:

The final grade of this course is awarded using the criterion detailed in the academic regulations. A large portion of the student's grade is determined in the comprehensive final exam of the Laboratory course (SEE PRACTICALS),resulting in a requirement of understanding the concepts and procedure of each lab experiment for successful completion of the lab course.

Pre-Requistes and Co-Requisties:

The lab course is to be taken during the same semester as AECB06, but receives a separate grade.

1.3 Course Goals and Objectives

The Electronic Devices and Circuits laboratory is designed to provide the student with the knowledge to use basic of electronic devices and techniques with proficiency. These techniques are designed to complement the concepts introduced in AECB09. In addition, the student should learn how to record experimental results effectively and present these results in a written report using lab worksheet. In addition, the student should learn how to record experimental results effectively and present these results in a written report.

More explicitly, the class objectives are:

1. The engineering skills using breadboard circuit design with electronic devices and components.
2. The behavior and characteristics of basic electronic devices and semiconductors.
3. The basic electronic devices necessary for construct the analog and digital circuits.

1.4 Use of Laboratory Instruments

One of the major goals of this lab is to familiarize the student with the proper equipment and techniques for making electronic circuit measurements. Some understanding of the lab instruments is necessary to avoid personal or equipment damage. By understanding the device's purpose and following a few simple rules, costly mistakes can be avoided. Most of the instrumentation used in this laboratory is implemented through Analog Discovery kit, multisim, breadboard, and circuit analysis system. In general, all devices have physical limits. These limits are specified by the device manufacturer and are referred to as the device rating. The ratings are usually expressed in terms of voltage limits, current limits, or power limits. It is up to the engineer to make sure that in device operation, these ratings (limit values) are not exceeded. The following rules provide a guideline for instrument protection.

The following rules provide a guideline for instrument protection.

1.4.1 Instrument Protection Rules

1. Setup multisim analog devices edition 13.0 software in personal computer.
2. Be sure the multisim analog devices edition 13.0 software is installed properly.
3. Connect the circuit in proper way i.e positive and negative terminals of function generator and bode plotter and CRO.
4. Never connect an ammeter across a voltage source. Only connect ammeters in series with loads. An ammeter is a low-resistance device that, if connected in parallel, will short out most components and usually destroy the ammeter or its protecting fuse.
5. Do not exceed the voltage and current ratings of instruments or other circuit elements.
6. Be sure the fuse and circuit breakers are of suitable value. When connecting electrical elements to make up a network in the laboratory, it is easy to lose track of various points in the network and accidentally connect a wire to the wrong place. A procedure to follow that helps to avoid this is to connect the main series part of the network first, then go back and add the elements in parallel. As an element is added, place a small check by it on your circuit diagram. Then go back and verify all connections before turning on the power. One day someone's life may depend upon your making sure that all has been done correctly.

1.5 Data Recording and Reports

1.5.1 The Laboratory Worksheets

Students must record their experimental values in the provided tables in this laboratory manual and reproduce them in the lab worksheets. Worksheets are integral to recording the methodology and results of an experiment. Make plots of data and sketches when these are appropriate in the recording and analysis of observations. Note that the data collected will be an accurate and permanent record of the data obtained during the experiment and the analysis of the results.

1.5.2 The Laboratory Files/Reports

Reports are the primary means of communicating your experience and conclusions to other professionals. In this course you will use the lab report to inform your teaching faculty of Lab about what you did and what you have learned from the experience. Engineering results are meaningless unless they can be communicated to others. You will be directed by your teaching faculty of Lab to prepare a lab report on a few selected lab experiments during the semester.

1.5.3 order of the Lab worksheet

1. COVER PAGE - Cover page must include lab name and number, your name, and the date the lab was performed.
2. Objective: Clearly state the experiment objective in your own words.
3. Prelab: Indicate the required knowledge for performing the concerned experiment
4. Equipment used: Indicate which equipment was used in performing the experiment.
5. Background: Give the brief information about corresponding lab experiment
6. Procedure: Describe the experimental set up and procedure to perform the experiment
7. Result: After completion of experiment, provide a summary of what was learned from this part of the laboratory experiment.

LAB-1 ORIENTATION

2.1 Introduction

In the first lab period, the students should become familiar with the location of equipment and components in the lab, the course requirements, and the teaching instructor. Students should also make sure that they have all of the co-requisites and pre-requisites for the course at this time.

2.2 Objective

To familiarize the students with the lab facilities, equipment, standard operating procedures, lab safety, and the course requirements.

2.3 Prelab Preparation:

Read the Introduction and Appendix A, of this manual. Download and install the “WaveForms” and “Multisim Analog Devices “ software on your personal computer, available here.

2.4 Equipment needed

AECB09 lab manual.

2.5 Procedure

1. During the first laboratory period, the faculty will provide the students with a general idea of what is expected from them in this course. Each student will receive a copy of the syllabus, stating the instructor’s contact information. In addition, the instructor will review the safety concepts of the course.

2. During this period, the faculty will briefly review the equipment which will be used throughout the semester. The location of instruments, equipment, and components (e.g. resistors, capacitors, connecting wiring) will be indicated. The guidelines for instrument use will be reviewed.

LAB-2 pn DIODE CHARACTERISTICS

3.1 Introduction

The purpose of this experiment is to acquaint the student with the operation of semiconductor diodes. From these characteristics, you will determine several diode parameters including the dynamic forward and reverse resistances and cut-in voltage. All of these terms are defined below. You will find that most of these parameters depend on the current at which that parameter is measured.

3.2 Objective

By the end of this lab, the student should learn Experimental determination of junction diode characteristics.

3.3 Prelab Preparation

Theoretical background of diode and V-I characteristics in forward and reverse bias mode.

3.4 Equipment needed

1. Analog Discovery 2 Instrument.
2. Multisim software.
3. Personal computer.
4. Diode (1N4007)
5. Resistors -1k ohm

3.5 Background

A pn junction diode conducts only in one direction. The V-I characteristics of the diode are curve between voltage across the diode and current flowing through the diode. When external voltage is zero, circuit is open and the potential barrier does not allow the current to flow. Therefore, the circuit current is zero. When p type (Anode) is connected to +ve terminal and n- type (cathode) is connected to –ve terminal of the supply voltage is known as forward bias. The potential barrier is reduced when diode is in the forward biased condition. At some forward voltage, the potential barrier altogether eliminated and current starts flowing through the diode and also in the circuit. Then diode is said to be in ON state. The current increases with increasing forward voltage. When N-type (cathode) is connected to +ve terminal and P-type (Anode) is connected – ve terminal of the supply voltage is known as reverse bias and the potential barrier across the junction increases. Therefore, the junction resistance becomes very high and a very small current (reverse saturation current) flows in the circuit. Then diode is said to be in OFF state. The reverse bias current is due to minority charge carriers.

Cut-in Voltage (v_γ) : The characteristics shown in Figure 3.1 are the combination of the forward and reverse characteristics. Appreciable conduction occurs from around 0.4V to 0.7V for silicon and from around 0.2V to 0.4V for germanium at room temperature. The value of v_γ is a function of the current at which v_γ is measured. This point is discussed below and is one of the concepts you should master from this experiment. If the applied voltage exceeds v_γ, the diode current increases rapidly.

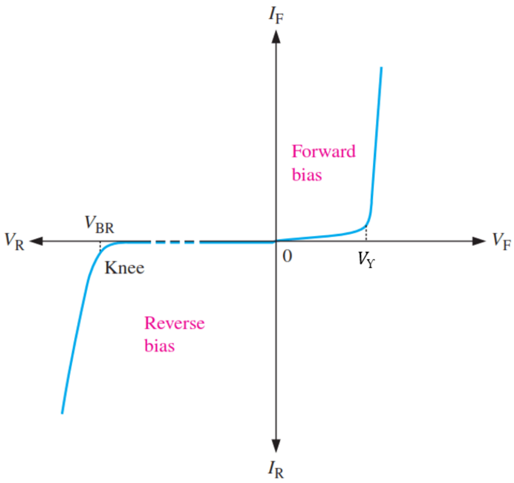

Figure 3.1: Determining the dc resistance of a diode at a particular operating point from the characteristics

DC or Static Resistance It is defined as the ratio of the voltage to the current, V/I, in the forward bias characteristics of the PN junction diode. In the forward bias characteristics of the diode as shown in Figure 3.2, the d.c. or static resistance (RF) at the operating point can be determined by using the corresponding levels of voltage V and current I,

$$\text{i.e.} \boldsymbol{R_f} = \frac{\boldsymbol{v_f}}{\boldsymbol{i_f}}$$

Here, the D.C. resistance is independent of the shape of the characteristics in the region surrounding the point of interest. The D.C. resistance levels at the knee and below will be greater than the resistance levels obtained for the characteristics above the knee. Hence, the D.C. resistance will be low when the diode current is high. As the static resistance varies widely with V and I.

AC or dynamic resistance: It is defined as the reciprocal of the slope of the volt-ampere characteristics.

$$\boldsymbol{r_f} = \frac{\Delta \boldsymbol{V_f}}{\Delta \boldsymbol{I_f}}$$

3.6 Procedure

1. Connections are made as per Figure 3.2.

2. For forward bias, the RPS +ve is connected to the anode of the diode and RPS –ve is connected to the cathode of the diode .

3. Switch on the power supply and increase the input voltage (supply voltage) in steps of 0.1V
4. Note down the corresponding current flowing through the diode and voltage across the diode for each and every step of the input voltage.
5. The reading of voltage and current are tabulated.
6. Graph is plotted between voltage (V_f) on X-axis and current (I_f) on Y-axis.

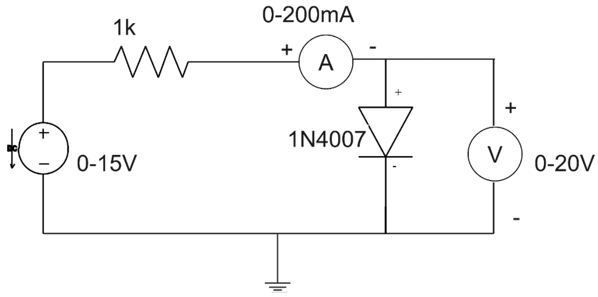

Figure 3.2: diode as forwars bias

Table 3.1: Observation Table

Input Voltage V_S **(V)**	**Diode Forward Voltage** V_f **(V)**	**Diode Forward Current** I_f**(mA)**

3.6.1 Reverse bias:

1. Connections are made as per Figure 3.3.
2. For reverse bias, the RPS +ve is connected to the cathode of the diode and RPS –ve is Connected to the anode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in steps of 0.1V
4. Note down the corresponding current flowing through the diode and voltage across the diode for each and every step of the input voltage.
5. The reading of voltage and current are tabulated.
6. Graph is plotted between voltage (V_r) on X-axis and current (I_r) on Y-axis.

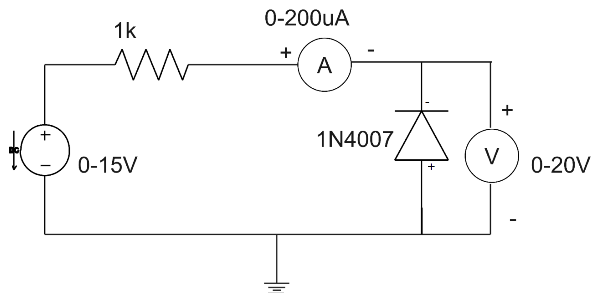

Figure 3.3: diode as reverse bias

Table 3.2: Observation Table

Input Voltage V_S **(V)**	**Diode Reverse Voltage** V_r **(V)**	**Diode Reverse Current** $I_r(\mu A)$

3.7 Further Probing Experiments

1. Plot the V-I Characteristics of germanium diode and find the cut in voltage.
2. Design diode acts as switch and plot the switching times of diode.

LAB-3 ZENER DIODE CHARACTERISTICS AND VOLTAGE REGULATOR

4.1 Introduction

The purpose of this experiment is to acquaint the student with the operation of Zener diodes. From these characteristics, you will determine Zener diode parameters i.e. breakdown voltage. The basic function of Zener diode is to maintain a specific voltage across its terminals within given limits of line or load change. Typically it is used for providing a stable reference voltage for use in power supplies and other equipment.

4.2 Objective

By the end of this lab, the student should learn
1. Experimental determination of Zener diode characteristics.
2. Experimental determination of Zener diode as voltage regulator

4.3 Prelab Preparation

Theoretical background of Zener diode and its characteristics and working of voltage regulator

4.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
Zener Diode
Resistors -1k ohm
Decade Resistance Box

4.5 Background

A Zener diode is heavily doped pn junction diode, specially made to operate in the break down region. A pn junction diode normally does not conduct when reverse biased. But if the reverse bias is increased, at a particular voltage it starts conducting heavily. This voltage is called Break down Voltage. High current through the diode can permanently damage the device.
To avoid high current, we connect a resistor in series with Zener diode. Once the diode starts conducting it maintains almost constant voltage across the terminals whatever may be the current through it, i.e., it has very low dynamic resistance. It is used in voltage regulators. The Zener diode mainly operates in reverse biased condition. We use Zener diodes for voltage

regulation and voltage stabilization. The critical parameter of this type of diodes is the Zener breakdown voltage. The Zener breakdown voltage is the minimum reverse biased voltage below which the diode blocks the reverse current through it and above which it causes a significant amount of reverse bias current to flow through it. Once the reverse voltage reaches the Zener breakdown voltage, the voltage across the device remains constant at that level. Hence we can use Zener diode for voltage regulation. The graph of voltage verses current of a diode is called its characteristic. Below you can see the characteristic

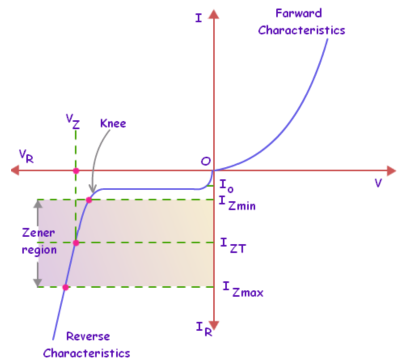

Figure 4.1: V-I characteristics of ideal Zener Diode

Here $\boldsymbol{V_z}$ is the Zener breakdown voltage. We are going to learn about an experiment to find out the Zener breakdown voltage and draw the characteristics of Zener diode.

ZENER BREAKDOWN VOLTAGE:

Draw a tangent on the reverse Bias Characteristic of the Zener Diode starting from the Knee and touching most of the points of the curve. The point where the tangent intersects the X-axis is the Zener Breakdown Voltage.
A voltage regulator is an electronic circuit that provides a stable DC voltage independent of the load current, temperature and AC line voltage variations. The voltage across the zener will remain steady at its breakdown voltage for all the values of zener current as long as the current remains in the breakdown region. Hence a regulated DC output voltage $\boldsymbol{V_0}$=$\boldsymbol{V_z}$ is obtained across $\boldsymbol{R_L}$, whenever the input voltage remains within a minimum and maximum voltage.

4.6 Procedure

4.6.1 Forward bias Characteristics:

1. Connections are made as per Figure 4.2
2. Vary the regulated power supply voltage in steps.
3. The Zener forward current ($\boldsymbol{I_F}$), and the Zener forward voltage ($\boldsymbol{V_F}$) are observed and then noted in the tabular form
4. A graph is plotted between Zener Forward current ($\boldsymbol{I_F}$) on Y-axis and the Zener Forward voltage ($\boldsymbol{I_F}$) on X-axis.

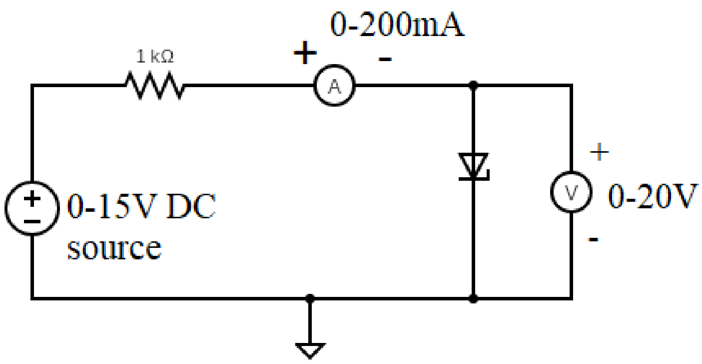

Figure 4.2: Circuit Diagram for Zener diode as forward bias

Table 4.1: Observation Table

Input Voltage V_S **(V)**	**Zener Forward Voltage** V_f **(V)**	**Zener Forward Current** I_f**(mA)**

4.6.2 Reverse bias characteristics:

1. Connections are made as per Figure 4.3
2. Vary the regulated power supply voltage in steps.
3. The Zener Reverse current (Ir), and the Zener reverse voltage (Vr) are observed and then noted in the tabular form
4. A graph is plotted between Zener Reverse current (Ir) on Y-axis and the reverse voltage (Vr) on X-axis.

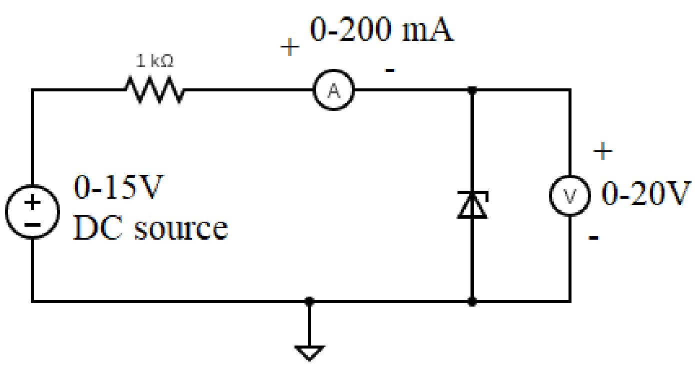

Figure 4.3: Circuit Diagram for Zener diode as reverse bias

Table 4.2: Observation Table

Input Voltage V_S **(V)**	**Zener Reverse Voltage** V_r **(V)**	**Zener Reverse Current** I_r**(mA)**

4.6.3 Zener as voltage regulator:

1. Connections are made as per the Figure 4.4.
2. The load is placed in no load condition and the zener voltage (Vz), Zener current (lz), load current (IL) are measured.
3. Connections are made as per the circuit diagram. Place the load in full load condition
4. Note down the zener voltage and current and the above steps are repeated by decreasing the Value of the load in steps.
5. All the readings are tabulated.
6. The percentage regulation is calculated using the below formula.

$$\%\text{Regulation} = (\boldsymbol{V_{NL}} - \boldsymbol{V_{FL}})\ /\boldsymbol{V_{FL}} \text{ X } 100$$

where

$\boldsymbol{V_{NL}}$ = Voltage across the diode, when no load is connected

$\boldsymbol{V_{FL}}$ = Voltage across the diode, when load is connected.

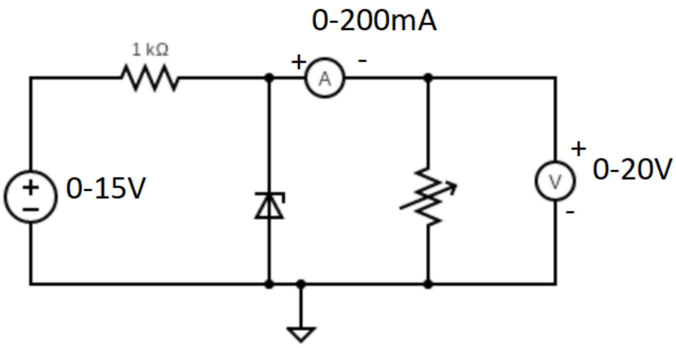

Figure 4.4: Circuit Diagram for Zener diode as voltage regulator

Table 4.3: observation table

Load Resistance R_L **(ohms)**	**Diode Load Voltage** V_{FL} **(V)**	**Diode Load Current** I_{FL}**(mA)**	**%Regulation**

Table 4.4: observation table

Input Voltage V_S **(V)**	**Zener Load Voltage** V_{FL} **(V)**	**Zener Load Current** I_{FL}**(mA)**	**%Regulation**

4.7 Further Probing Experiments

1. Design a zener voltage regulator circuit to drive a load of 6V, 100 mW from an unregulated input supply of Vmin = 8V, Vmax = 12V using a 6V zener diode. .
2. Design square wave generator using zener diode
3. Design for a Zener Transistor series voltage regulator circuit to drive a load of 6V, 1w, from a supply of 10V with a ±3V ripple voltage

LAB-4 HALF WAVE RECTIFIER WITH/WITHOUT FILTER

5.1 Introduction

The purpose of this experiment is to acquaint the student with the design and operation of rectifiers with and without filter using pn diode. From these characteristics, you will determine rectifier efficiency, ripple factor and Peak inverse voltage.

5.2 Objective

1. To design a half-wave rectifier circuit and analyse its output.
2. To analyse the rectifier output using a capacitor in shunt as a filter

5.3 Prelab

Theoretical background of rectifiers and filters

5.4 Equipment needed

Analog Discovery 2 Instrument.
Multisim software.
Personal computer.
Diode (IN4007)
Resistors -1k ohm
Capacitors- 1uF, 10uF and 100uF
Decade Resistance Box

5.5 Background

Rectifier is an electronic circuit, which offers low resistance in one direction and high resistance in opposite direction. Rectifiers are used to convert AC voltages and DC voltages and currents. The output of a rectifier without filter is pulsating in nature. It consists of a DC component and unwanted ripple (ac) component. These ripple components can be removed by use of filters. The magnitude of DC output voltage may vary with the variation of either the input AC voltage or with load current. There are two types of rectifier circuits, one is half wave rectifier and other is full wave rectifier.

Half Wave rectifier:

A half wave rectifier conducts current during the positive half cycles of AC supply. The negative half cycles of AC supply are suppressed i.e. no current is conducted and no voltage appears across the load. Therefore current always flow in one direction (i.e. DC) through the load after every half cycle. Efficiency of the half wave rectifier is given as the ratio of DC power output to the applied input AC power.

$$\eta = 0.406$$

Capacitor Filter: It consists of a capacitor in parallel with the load. It has high resistance to DC and has low capacitance reactance. The capacitor charges up to the peak value of the input voltage and tries to maintain this value as the full wave input drops to zero. An inexpensive filters for high loads is found in capacitor filters.

5.6 Procedure

1. Connect the circuit as shown in Figure 5.1.
2. Adjust the load resistance, R_L to 500 ohms, and note down the readings of input and output voltages through oscilloscope.
3. Note the readings of dc current, dc voltage and ac voltage.
4. Now, change the resistance the load resistance, R_L to 1 K ohms and repeat the procedure as above. Also repeat for 10 K ohms.

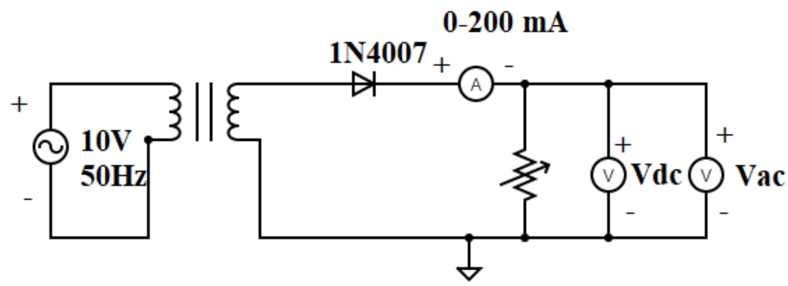

Figure 5.1: Circuit Diagram for Half Wave Rectifier without Filter

Table 5.1: Observation table

Load Resistance R_L	Input Voltage Vm	Output Voltage V0	I_{DC}	V_{DC}	V_{AC}	Ripple Factor

Halfwave rectifier with filter:

1. Connect the circuit as shown in Figure 5.2.
2. Assume C=100 microfarads

3. Adjust the load resistance, R_L to 500 ohms, and note down the readings of input and output voltages through oscilloscope.

4. Note the readings of dc current, dc voltage and ac voltage.

5. Now, change the resistance the load resistance, R_L to 1 K ohms and repeat the procedure as above. Also repeat for 10 K ohms.

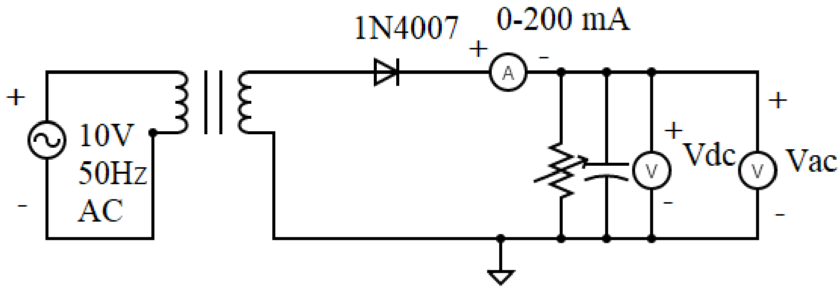

Figure 5.2: Circuit Diagram for Half Wave Rectifier with Filter

Table 5.2: Observation table

Load Resistance R_L	**Input Voltage Vm**	**Output Voltage V0**	I_{DC}	V_{DC}	V_{AC}	**Ripple Factor**

5.7 Further Probing Experiments

1. Design half wave rectifier with an applied input a.c. power is 100 watts, and it is to deliver an output power is 40 watts.

2. Design half wave rectifier with an a.c. supply of 230 V is applied through a transformer of turn ratio 10 : 1. Observe the output d.c. voltage, peak inverse voltage and identify dc output voltage if transformer turns ratio changed to 20:1.

LAB-5 FULL WAVE RECTIFIER WITH/WITHOUT FILTER

6.1 Introduction

The purpose of this experiment is to acquaint the student with the design and operation of rectifiers with and without filter using pn diode. From these characteristics, you will determine rectifier efficiency, ripple factor and Peak inverse voltage.

6.2 Objective

1. To design a Full-wave rectifier circuit and analyse its output.
2. To analyse the rectifier output using a capacitor in shunt as a filter

6.3 Prelab

Theoretical background of rectifiers and filters

6.4 Equipment needed

Analog Discovery 2 Instrument.
Multisim software.
Personal computer.
Diode (IN4007)
Resistors -1k ohm
Capacitors- 1uF, 10uF and 100uF
Decade Resistance Box

6.5 Background

Rectifier is an electronic circuit, which offers low resistance in one direction and high resistance in opposite direction. Rectifiers are used to convert AC voltages and DC voltages and currents. The output of a rectifier without filter is pulsating in nature. It consists of a DC component and unwanted ripple (ac) component. These ripple components can be removed by use of filters. The magnitude of DC output voltage may vary with the variation of either the input AC voltage or with load current. There are two types of rectifier circuits, one is half wave rectifier and other is full wave rectifier.

Full Wave rectifier: A full wave rectifier is a circuit, which converts AC voltages into a pulsating DC voltage using both half cycle of the applied AC voltages. These are classified into center-tapped full wave rectifier and bridge rectifier. Efficiency of the full wave rectifier is given by

$$\boldsymbol{\eta} = 0.812$$

Capacitor Filter: It consists of a capacitor in parallel with the load. It has high resistance to DC and has low capacitance reactance. The capacitor charges up to the peak value of the input voltage and tries to maintain this value as the full wave input drops to zero. An inexpensive filters for high loads is found in capacitor filters.

6.6 Procedure

1. Connect the circuit as shown in Figure 6.1.
2. Adjust the load resistance, R_L to 500 ohms, and note down the readings of input and output voltages through oscilloscope.
3. Note the readings of dc current, dc voltage and ac voltage.
4. Now, change the resistance the load resistance, R_L to 1 K ohms and repeat the procedure as above. Also repeat for 10 K ohms.

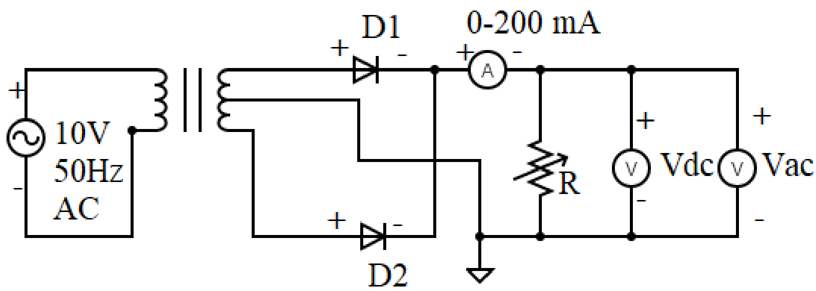

Figure 6.1: Circuit Diagram for Full Wave Rectifier without Filter

Table 6.1: Observation table

Load Resistance R_L	**Input Voltage Vm**	**Output Voltage V0**	I_{DC}	V_{DC}	V_{AC}	**Ripple Factor**

Full wave rectifier with filter:

1. Connect the circuit as shown in Figure 6.2.
2. assume C=100 microfarads
3. Adjust the load resistance, R_L to 500 ohms, and note down the readings of input and output voltages through oscilloscope.
4. Note the readings of dc current, dc voltage and ac voltage.
5. Now, change the resistance the load resistance, R_L to 1 K ohms and repeat the procedure as above. Also repeat for 10 K ohms.

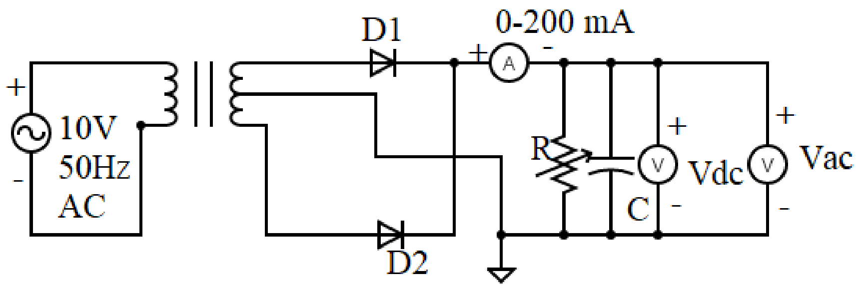

Figure 6.2: Circuit Diagram for Full Wave Rectifier with Filter

Table 6.2: Observation table

Load Resistance $\boldsymbol{R_L}$	**Input Voltage Vm**	**Output Voltage V0**	I_{DC}	V_{DC}	V_{AC}	**Ripple Factor**

6.7 Further Probing Experiments

1. Design a full wave rectifier with step down transformer and center tapped transformer. Justify the operation.

2. Design Full wave rectifier with capacitive filter using 10uF and 1uF. Observe the ripple factor.

LAB-6 TRANSISTOR CB CHARACTERISTICS

7.1 Introduction

The purpose of this experiment is to introduce you to the bipolar junction transistor (BJT) characteristics that describe the common-base configuration of operation. Both the output and input characteristics of an npn silicon transistor will be measured.

7.2 Objective

To plot the input and output characteristics of a transistor in CB Configuration and to compute the h – parameters.

7.3 Prelab Preparation

Read the text that describe the bipolar transistor operation and the transistor input and output characteristics. You should concentrate on understanding what is meant by a transistor input or output characteristic and the h-parameters in the small-signal transistor model.

7.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
BJT (BC107)
Resistors - 470 ohms
Ammeter- 0-200mA
Voltmeter- 0-20V

7.5 Background

A Bipolar Junction Transistor, or BJT is a three terminal device having two PN-junctions connected together in series. Each terminal is given a name to identify it and these are known as the Emitter (E), Base (B) and Collector (C).
There are two basic types of bipolar transistor construction, NPN and PNP, which basically describes the physical arrangement of the P type and N-type semiconductor materials from which they are made.
Bipolar Transistors are current amplifying or current regulating devices that control the amount of current flowing through them in proportion to the amount of biasing current applied to their base terminal. The principle of operation of the two transistor types NPN and PNP, is exactly the same the only difference being in the biasing (base current) and the polarity of the power supply for each type.

Transistor configurations

There are three possible configurations possible when a transistor is connected in a circuit. (a) Common base, (b) Common emitter (c) Common collector. The behavior of a transistor can be represented by d.c. current-voltage (I-V) curves, called the static characteristic curves of the device. The two important characteristics of a transistor are: (i) Input characteristics, (ii) Output characteristics. These characteristics give information about various transistor parameters, e.g. input and out dynamic resistance, current amplification.

Common Base Transistor Characteristics: In common base configuration, the base is made common to both input and output as shown in its circuit diagram.

Input Characteristics: The input characteristics is obtained by plotting a curve between $\boldsymbol{I_E}$ and $\boldsymbol{V_{EB}}$ keeping voltage $\boldsymbol{V_{CB}}$ constant. This is very similar to that of a forward-biased diode and the slope of the plot at a given operating point gives information about its input dynamic resistance.

Input Dynamic Resistance ($\boldsymbol{r_i}$): This is defined as the ratio of change in base emitter voltage ($\triangle \boldsymbol{V_{EB}}$) to the resulting change in emitter current ($\triangle \boldsymbol{I_E}$) at constant collector-emitter voltage ($\boldsymbol{V_{CB}}$). This is dynamic as its value varies with the operating current in the transistor.

Output Characteristics: The output characteristic curves are plotted between $\boldsymbol{I_C}$ and $\boldsymbol{V_{CB}}$, keeping $\boldsymbol{I_E}$ constant. The output characteristics are controlled by the input characteristics. Since $\boldsymbol{I_C}$ changes with $\boldsymbol{I_E}$, there will be different output characteristics corresponding to different values of $\boldsymbol{I_E}$. These curves are almost horizontal. This shows that the output dynamic resistance, defined below, is very high.

Output Dynamic Resistance ($\boldsymbol{r_0}$): This is defined as the ratio of change in collector-base voltage ($\triangle \boldsymbol{V_{CB}}$) to the change in collector current ($\triangle \boldsymbol{I_C}$) at a constant emitter current $\boldsymbol{I_E}$.

Current amplification factor ($\boldsymbol{\alpha}$): This is defined as the ratio of the change in collector current to the change in emitter current at a constant collector-base voltage ($\boldsymbol{V_{CB}}$) when the transistor is in active state.

7.6 Procedure

Input characteristics:

1. Connect the circuit as per the given Figure 7.1.
2. For plotting the input characteristics the output voltage $\boldsymbol{V_{CB}}$ is kept constant at 4V and for different values of $\boldsymbol{V_{BE}}$, note down the values of $\boldsymbol{I_E}$.
3. Repeat the above step by keeping $\boldsymbol{V_{CB}}$ at 8V.
4. Tabulate all the readings.
5. Plot the graph between VBE on x-axis and $\boldsymbol{I_E}$ on y-axis for constant $\boldsymbol{V_{CB}}$.

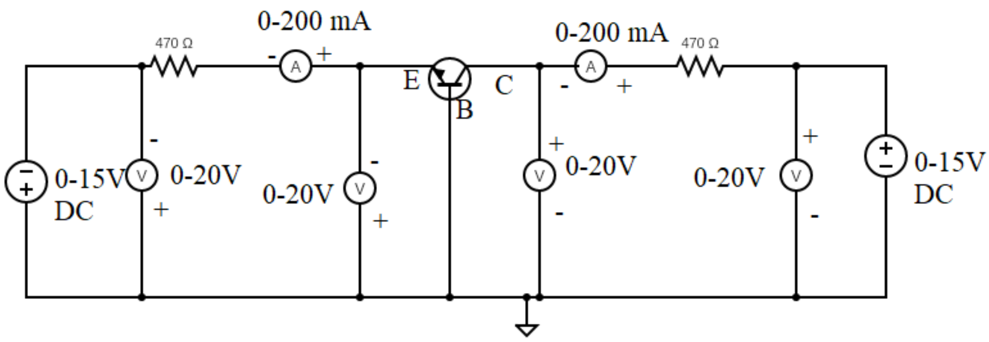

Figure 7.1: Circuit Diagram for transistor as common base configuration

Table 7.1: Observation table

V_{CB}=-4V		V_{CB}= -8V	
V_{EB} (Volts)	I_E (milli Amps)	V_{EB} (Volts)	I_E (milli Amps)

Output Characteristics:

1. Connect the circuit as per the given Figure 7.1.
2. For plotting the output characteristics the input current I_E is kept constant at 4mA and for different values of V_{CB}, note down the values of I_C.
3. Repeat the above step by keeping I_E at 8mA.
4. Tabulate the all the readings.
5. Plot the graph between V_{CB} on x-axis and I_C on y-axis for constant I_E.

Table 7.2: Observation table

I_E=4mA		I_E= 8mA	
V_{CB} (Volts)	I_C (milli Amps)	V_{CB} (Volts)	I_C (milli Amps)

7.7 Further Probing Experiments

1. Describe, based on your observations, the I-V curves of npn transistor. At approximately what collector-emitter voltage (V_{CE}) does the transition from saturation to active region occur?

2. Demonstrate the characteristics of Common base PNP transistor to determine the h parameters.

LAB-7 TRANSISTOR CE CHARACTERISTICS

8.1 Introduction

The purpose of this experiment is to introduce you to the bipolar junction transistor (BJT) characteristics that describe the common-emitter configuration of operation. Both the output and input characteristics of an npn silicon transistor will be measured.

8.2 Objective

To plot the input and output characteristics of a transistor in CE Configuration and to compute the h – parameters.

8.3 Prelab Preparation

Read the text that describe the bipolar transistor operation and the transistor input and output characteristics. You should concentrate on understanding what is meant by a transistor input or output characteristic and the h-parameters in the small-signal transistor model.

8.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
BJT (BC107)
Resistors - 100k ohms and 470 ohms
Ammeter- 0-200mA and 0-200uA
Voltmeter- 0-20V

8.5 Background

A Bipolar Junction Transistor, or BJT is a three terminal device having two PN-junctions connected together in series. Each terminal is given a name to identify it and these are known as the Emitter (E), Base (B) and Collector (C).
There are two basic types of bipolar transistor construction, NPN and PNP, which basically describes the physical arrangement of the P type and N-type semiconductor materials from which they are made. Bipolar Transistors are current amplifying or current regulating devices that control the amount of current flowing through them in proportion to the amount of biasing current applied to their base terminal. The principle of operation of the two transistor types NPN and PNP, is exactly the same the only difference being in the biasing (base current) and the polarity of the power supply for each type.

Transistor configurations: There are three possible configurations possible when a transistor is connected in a circuit. (a) Common base, (b) Common emitter (c) Common collector. The behavior of a transistor can be represented by d.c. current-voltage (I-V) curves, called the static characteristic curves of the device. The two important characteristics of a transistor are: (i) Input characteristics, (ii) Output characteristics. These characteristics give information about various transistor parameters, e.g. input and out dynamic resistance, current amplification.

Common Emitter Transistor Characteristics: In a common emitter configuration, emitter is common to both input and output as shown in its circuit diagram.

Input Characteristics: The variation of the base current IB with the base-emitter voltage VBE keeping the collector-emitter voltage VCE fixed, gives the input characteristic in CE mode.

Input Dynamic Resistance (r_i): This is defined as the ratio of change in base emitter voltage (ΔV_{BE}) to the resulting change in emitter current (ΔI_B) at constant collector-emitter voltage (V_{CE}). This is dynamic as its value varies with the operating current in the transistor.

Output Characteristics: The variation of the collector current I_C with the collector-emitter voltage V_{CE} is called the output characteristic. The plot of I_C versus V_{CE} for different fixed values of I_B gives one output characteristic. Since the collector current changes with the base current, there will be different output characteristics corresponding to different values of I_B.

Output Dynamic Resistance (r_0): This is defined as the ratio of change in collector-base voltage (ΔV_{CE}) to the change in collector current (ΔI_C) at a constant base current I_B.

Current amplification factor (β): TThis is defined as the ratio of the change in collector current to the change in base current at a constant collector-emitter voltage when the transistor is in active state.

8.6 Procedure

Input characteristics:

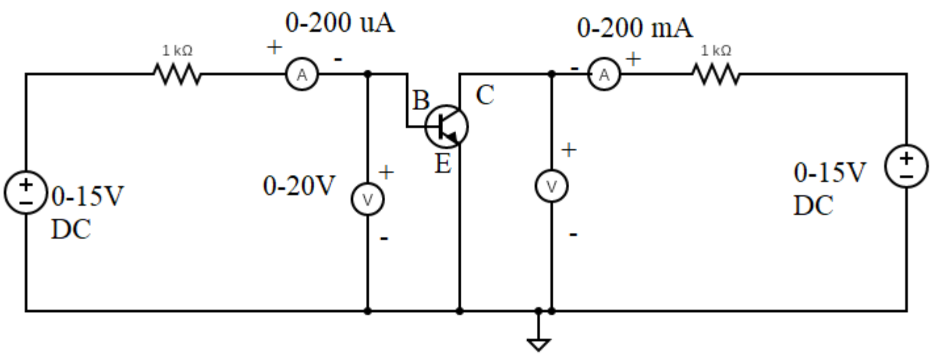

Figure 8.1: Circuit Diagram for transistor as common emitter configuration

1. Connect the circuit as per the given Figure 8.1.

2. For plotting the input characteristics the output voltage $\boldsymbol{V_{CE}}$ is kept constant at 4V and for different values of $\boldsymbol{V_{BE}}$, note down the values of $\boldsymbol{I_B}$.

3. Repeat the above step by keeping $\boldsymbol{V_{CE}}$ at 8V.

4. Tabulate all the readings.

5. Plot the graph between $\boldsymbol{V_{BE}}$ on x-axis and $\boldsymbol{I_B}$ on y-axis for constant $\boldsymbol{V_{CE}}$.

Table 8.1: Observation table

V_{CE}=2V		**V_{CE}= 6V**	
V_{BE} (Volts)	**I_B (μAmps)**	**V_{BE} (Volts)**	**I_B (μAmps)**

Output Characteristics:

1. Connect the circuit as per the given Figure 8.1.

2. For plotting the output characteristics the input current $\boldsymbol{I_B}$ is kept constant at 20$\boldsymbol{\mu}$A and for different values of $\boldsymbol{V_{CE}}$, note down the values of $\boldsymbol{I_C}$.

3. Repeat the above step by keeping $\boldsymbol{I_B}$ at 40$\boldsymbol{\mu}$A.

4. Tabulate the all the readings.

5. Plot the graph between $\boldsymbol{V_{CE}}$ on x-axis and $\boldsymbol{I_C}$ on y-axis for constant $\boldsymbol{I_B}$.

Table 8.2: Observation table

I_B=20 μamps		**I_B= 40 μAmps**	
V_{CE} (Volts)	**I_C (milli Amps)**	**V_{CE} (Volts)**	**I_C (milli Amps)**

8.7 Further Probing Experiments

1. Design a Sustainable Relay Driving Circuit Using BJT.

2. Design an electronic switch using CE configuration

LAB-8 FREQUENCY RESPONSE OF CE AMPLIFIER

9.1 Introduction

The purpose of this experiment is to give the student individual experience in designing an amplifier to meet certain design specifications. You will design a BJT CE amplifier to meet a specified voltage gain $\boldsymbol{A_v}$, and output impedance $\boldsymbol{R_O}$.

9.2 Objective

1. To design a common emitter transistor (NPN) amplifier circuit.
2. To obtain the frequency response curve of the amplifier and to determine the mid frequency gain, $\boldsymbol{A_{mid}}$, lower and higher cutoff frequency of the amplifier circuit.

9.3 Prelab Preparation

Design the circuit shown in circuit diagram by calculating the values of R1, R2, R3 and R4. BC107 NPN transistor is to be used. In this design the bias voltage between emitter and collector should be 5V DC. The quiescent collector current is 1mA. The design should provide maximum possible voltage swing at the amplifier output.

9.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
BJT (BC107)
Resistors - 2.2k, 22k, 1.2k, 3.6k, 220 ohms
Capacitors – 1uF, 100uF

9.5 Background

A Bipolar Junction Transistor, or BJT is a three terminal device having two PN-junctions connected together in series. Each terminal is given a name to identify it and these are known as the Emitter (E), Base (B) and Collector (C). There are two basic types of bipolar transistor construction, NPN and PNP, which basically describes the physical arrangement of the P type and N-type semiconductor materials from which they are made. Bipolar Transistors are current amplifying or current regulating devices that control the amount of current flowing through them in proportion to the amount of biasing current applied to their base terminal. The principle of operation of the two transistor types NPN and PNP, is exactly the same the only difference being in the biasing (base current) and the polarity of the power supply for each type. The most common circuit configuration for an NPN transistor is that of the Common Emitter amplifier and that a family of curves known commonly as the output characteristics curves, relates the Collector current (I_C), to the output or Collector voltage (V_{CE}), for different values of Base current (I_B). All types of transistor amplifiers operate using AC signal inputs which alternate between a positive value and a negative value. Presetting the amplifier circuit to operate between these two maximum or peak values is achieved using a process known as Biasing. Biasing is very important in amplifier design as it establishes the correct operating point of the transistor amplifier ready to receive signals, thereby reducing any distortion to the output signal. The single stage common emitter amplifier circuit shown below uses what is commonly called "Voltage Divider Biasing". The Base voltage (V_B) can be easily calculated using the simple voltage divider formula below:

$$V_b = \frac{V_{CC}R_2}{R_1+R_2}$$

Thus the base voltage is fixed by biasing and independent of base current provided the current in the divider circuit is large compared to the base current. Thus assuming $I_B \approx 0$, one can do the approximate analysis of the voltage divider network without using the transistor gain,β, in the calculation. Note that the approximate approach can be applied with a high degree of accuracy when the following condition is satisfied. i.e.

$$\beta R_E \geq 10R_2$$

Coupling and Bypass Capacitors:

In CE amplifier circuits, capacitors C1 and C2 are used as coupling Capacitors to separate the AC signals from the DC biasing voltage. The capacitors will only pass AC signals and block any DC component. Thus they allow coupling of the AC signal into an amplifier stage without disturbing its Q point. The output AC signal is then superimposed on the biasing of the following stages. Also a bypass capacitor, CE is included in the emitter leg circuit. This capacitor is an open circuit component for DC bias, meaning that the biasing currents and voltages are not affected by the addition of the capacitor maintaining a good Q-point stability. However, this bypass capacitor acts as a short circuit path across the emitter resistor at high frequency signals increasing the voltage gain to its maximum. Generally, the value of the bypass capacitor, CE is chosen to provide a reactance of at most, 1/10th the value of R_E at the lowest operating signal frequency.

Amplifier Operation: Once the Q-point is fixed through DC bias, an AC signal is applied at the input using coupling capacitor C1. During positive half cycle of the signal V_{BE} increases leading to increased I_B. Therefore I_C increases by β times leading to decrease in the output voltage, V_{CE}. Thus the CE amplifier produces an amplified output with a phase reversal. The voltage Gain of the common emitter amplifier is equal to the ratio of the change in the output voltage to the change in the input voltage. Thus,

$$A = \frac{V_{out}}{V_{in}} = \frac{\Delta V_{CE}}{\Delta V_{BE}}$$

Frequency Response Curve: The performance of an amplifier is characterized by its frequency response curve that shows output amplitude (or, more often, voltage gain) plotted versus frequency (often in log scale). Typical plot of the voltage gain of an amplifier versus frequency is shown in the figure below. The frequency response of an amplifier can be divided into three frequency ranges. The frequency response begins with the lower frequency range designated between 0 Hz and lower cutoff frequency. At lower cutoff frequency f_L , the gain is equal to 0.707 A_{mid}. A_{mid} is a constant mid-band gain obtained from the mid-frequency range. The third, the higher frequency range covers frequency between upper cutoff frequency and above. Similarly, at higher cutoff frequency f_H, the gain is equal to 0.707 A_{mid}. Beyond this the gain decreases with frequency increases and dies off eventually.

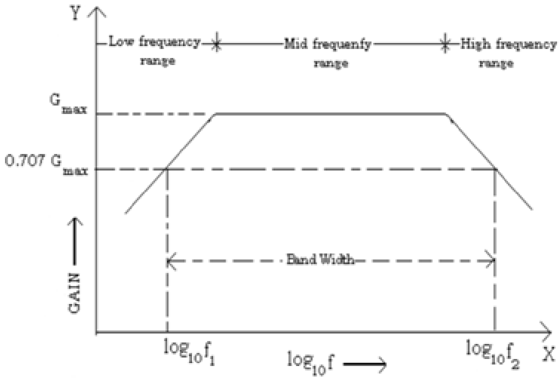

Figure 9.1: Frequency response of amplifier

9.6 Procedure

1. Connect the circuit diagram as shown in figure 9.2.
2. Adjust input signal amplitude in the function generator and observe an amplified voltage at the output without distortion.
3. By keeping input signal voltages at 50mV, vary the input signal frequency from 0 to 1MHz in steps as shown in tabular column and note the corresponding output voltages.
4. Calculate the voltage gain for each frequency. Observe the inverted output.
5. Plot the frequency response curve, i.e. voltage gain in dB versus frequency on a semi-log graph-sheet.
6. Estimate the mid-frequency gain and also the lower and higher cut off frequencies and hence the bandwidth

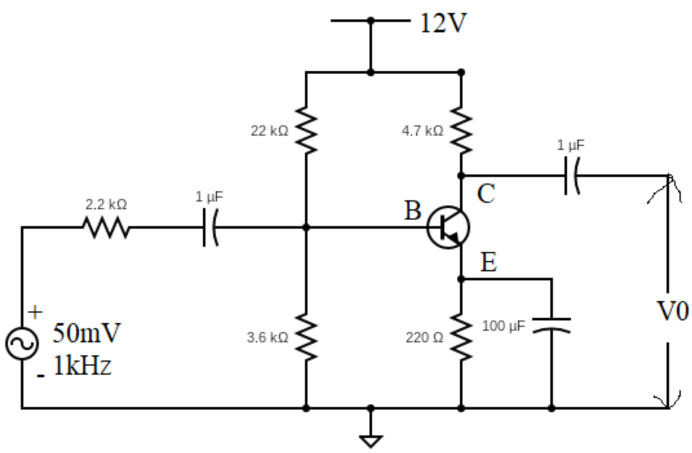

Figure 9.2: Circuit Diagram for Common Emitter amplifier

Table 9.1: Observation table

Frequency (Hz)	**Output Voltage (V)**	**Gain**	**Gain in dB**

9.7 Further Probing Experiments

1. Measure the DC voltages to make sure the BJT is in the forward active region. If it's not in forward active, adjust your resistor values to compensate.

2. Measure the voltage gain. Adjust the input signal from your wavetek to approximately 10mv amplitude, with a frequency of 100 kHz. What is the voltage swing?

LAB-9 FREQUENCY RESPONSE OF CC AMPLIFIER

10.1 Introduction

The purpose of this experiment is to give the student individual experience in designing an amplifier to meet certain design specifications. You will design a BJT CC amplifier to meet a specified voltage gain $\boldsymbol{A_v}$, and output impedance $\boldsymbol{R_O}$.

10.2 Objective

1. To design a common collector transistor (NPN) amplifier circuit.
2. To obtain the frequency response curve of the amplifier and to determine the mid frequency gain, $\boldsymbol{A_{mid}}$, lower and higher cutoff frequency of the amplifier circuit.

10.3 Prelab Preparation

Design the circuit shown in circuit diagram by calculating the values of R1, R2, R3 and R4. BC107 NPN transistor is to be used. In this design the bias voltage between emitter and collector should be 12V DC. The quiescent collector current is 1mA. The design should provide maximum possible voltage swing at the amplifier output.

10.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
BJT (BC107)
Resistors - 22k, 10k, 150, 100 ohms
Capacitors – 0.1uF,0.001uF, 100uF

10.5 Background

A Bipolar Junction Transistor, or BJT is a three terminal device having two PN-junctions connected together in series. Each terminal is given a name to identify it and these are known as the Emitter (E), Base (B) and Collector (C). There are two basic types of bipolar transistor construction, NPN and PNP, which basically describes the physical arrangement of the P type and N-type semiconductor materials from which they are made. Bipolar Transistors are current amplifying or current regulating devices that control the amount of current flowing through them in proportion to the amount of biasing current applied to their base terminal. The principle of operation of the two transistor types NPN and PNP, is exactly the same the only difference being in the biasing (base current) and the polarity of the power supply for each type. In common collector amplifier as the collector resistance is made to zero, the collector is at ac ground that is why the circuit is also called as grounded - collector amplifier or this configuration is having voltage gain close to unity and hence a change in base voltage appears as an equal change across the load at the emitter, hence the name emitter follower. In other words the emitter follows the input signal. This circuit performs the function of impedance transformation over a wide range of frequencies with voltage gain close to unity. In addition to that, the emitter follower increases the output level of the signal. Since the output voltage across the emitter load can never exceed the input voltage to base, as the emitter-base junction would become back biased. Common collector state has a low output resistance, the circuit suitable to serve as buffer or isolating amplifier or couple to a load with large current demands.

Characteristics of CC amplifier:

1. Higher current gain
2. Voltage gain is approximately unity
3. Power gain approximately equal to current gain
4. No current or voltage phase shift
5. Large input resistance
6. Small output resistance

Amplifier Operation: Once the Q-point is fixed through DC bias, an AC signal is applied at the input using coupling capacitor C1. During positive half cycle of the signal $\boldsymbol{V_{BE}}$ increases leading to increased $\boldsymbol{I_B}$. Therefore $\boldsymbol{I_C}$ increases by $\boldsymbol{\beta}$ times leading to decrease in the output voltage, $\boldsymbol{V_{CE}}$. Thus the CE amplifier produces an amplified output with a phase reversal. The voltage Gain of the common emitter amplifier is equal to the ratio of the change in the output voltage to the change in the input voltage. Thus,

$$\boldsymbol{A = \frac{V_{out}}{V_{in}}}$$

Frequency Response Curve: The performance of an amplifier is characterized by its frequency response curve that shows output amplitude (or, more often, voltage gain) plotted versus frequency (often in log scale). Typical plot of the voltage gain of an amplifier versus frequency is shown in the figure below. The frequency response of an amplifier can be divided into three frequency ranges. The frequency response begins with the lower frequency range designated

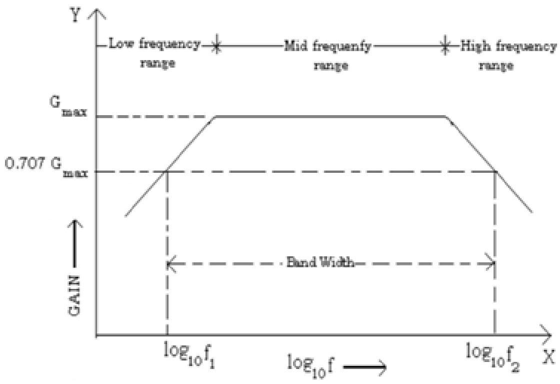

Figure 10.1: Frequency response of amplifier

between 0 Hz and lower cutoff frequency. At lower cutoff frequency $\boldsymbol{f_L}$, the gain is equal to

0.707 $\boldsymbol{A_{mid}}$. $\boldsymbol{A_{mid}}$ is a constant mid-band gain obtained from the mid-frequency range. The third, the higher frequency range covers frequency between upper cutoff frequency and above. Similarly, at higher cutoff frequency $\boldsymbol{f_H}$, the gain is equal to 0.707 $\boldsymbol{A_{mid}}$. Beyond this the gain decreases with frequency increases and dies off eventually.

10.6 Procedure

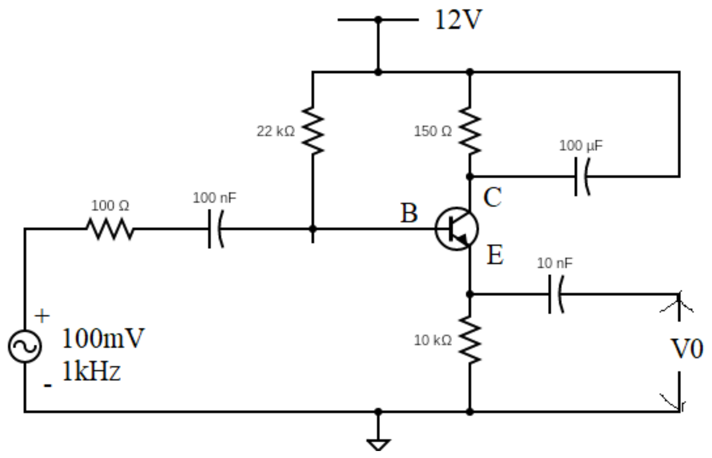

Figure 10.2: Circuit Diagram for Common Collector amplifier

1. Connect the circuit diagram as shown in figure 10.2.
2. Set Source Voltage $\boldsymbol{V_s}$ = 50mV (say) at 1 KHz frequency, using function generator.
3. Calculate the Voltage Gain by using the formula

$$\boldsymbol{A_v} = \text{Output voltage } (\boldsymbol{V_0}) \text{ / Input voltage } (\boldsymbol{V_s})$$

4. Calculate the Voltage Gain in dB by using Voltage Gain

$$\boldsymbol{A_v} \text{ (dB)} = \boldsymbol{20log_{10}(\frac{V_0}{V_s})}$$

5. Plot the Graph by taking Voltage gain (dB) on x-axis and frequency (Hz) on y-axis.
6. The Bandwidth of the amplifier is calculated from the graph using the expression,

   Bandwidth, BW=$\boldsymbol{f_2 - f_1}$ Where $\boldsymbol{f_1}$ is lower 3-dB frequency $\boldsymbol{f_2}$ is upper 3-dB frequency

7. Calculate the voltage gain for each frequency. Observe the inverted output.
8. Plothe frequency response curve, i.e. voltage gain in dB versus frequency on a semi-log graph-sheet.
9. Estimate the mid-frequency gain and also the lower and higher cut off frequencies and hence the bandwidth

Table 10.1: Observation table

Frequency (Hz)	**Output Voltage (V)**	**Gain**	**Gain in dB**

10.7 Further Probing Experiments

1. Connect the common collector amplifier circuit you designed. Set the values of capacitors $\boldsymbol{C_1}$, $\boldsymbol{C_2}$, and $\boldsymbol{C_3}$ to 1uF each. Set $\boldsymbol{R_L}$ to be 1 kΩ and the supply voltage to 15V DC. Measure the DC bias voltages on the base, emitter and the collector. Calculate the collector current. Compare the measured voltages with the design intent and calculation. Tabulate the measured versus the calculated bias voltages and current.

2. Measure the frequency response of the amplifier starting from 100 Hz. change the test frequency to cover the upper cut-off frequency of the amplifier. Throughout the measurement of the frequency response, apply low input signal levels (in the order of few milli-Volts) to ensure that the output signal is not distorted. Monitor both input and output waveforms on the oscilloscope.

LAB-10 UJT CHARACTERSTICS

11.1 Introduction

The purpose of this experiment is to acquaint the student with the operation of Uni Junction Transistor. From these characteristics, you will determine pinch-off voltage and intrinsic stand off ratio. From this experiment you can identify the negative resistance region.

11.2 Objective

1. To measure the inter base resistance and determine the emitter-base 1 pn junction diode characteristics of unijunction transistor.
2. To determine the intrinsic stand-off ratio

11.3 Prelab Preparation

Theoretical background of UJT and V-I characteristics in forward and reverse bias mode.

11.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
UJT (2N2646)
Resistors - 1k ohms
Ammeter- 0-200mA
Voltmeter - 0-20V DC

11.5 Background

A Unijunction Transistor (UJT) is an electronic semiconductor device that has only one junction. The UJT Unijunction Transistor (UJT) has three terminals an emitter (E) and two bases (B1 and B2). The base is formed by lightly doped n-type bar of silicon. Two Ohmic contacts B1 and B2 are attached at its ends. The emitter is of p-type and it is heavily doped. The resistance between B1 and B2, when the emitter is open-circuit is called inter base resistance. The original unijunction transistor, or UJT, is a simple device that is essentially a bar of N type semiconductor material into which P type material has been diffused somewhere along its length. The 2N2646 is the most commonly used version of the UJT.
The UJT is biased with a positive voltage between the two bases. This causes a potential drop along the length of the device. When the emitter voltage is driven approximately one diode voltage above the voltage at the point where the P diffusion (emitter) is, current will begin to flow from the emitter into the base region. Because the base region is very lightly doped,

the additional current (actually charges in the base region) causes (conductivity modulation) which reduces the resistance of the portion of the base between the emitter junction and the B2 terminal. This reduction in resistance means that the emitter junction is more forward biased, and so even more current is injected. Overall, the effect is a negative resistance at the emitter terminal. This is what makes the UJT useful, especially in simple oscillator circuits. When the emitter voltage reaches V_p, the current starts to increase and the emitter voltage starts to decrease. This is represented by negative slope of the characteristics which is referred to as the negative resistance region, beyond the valley point, R_{B1} reaches minimum value and this region, V_{EB} proportional to I_E.

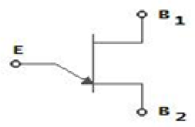

Figure 11.1: UJT Symbol

11.6 Procedure

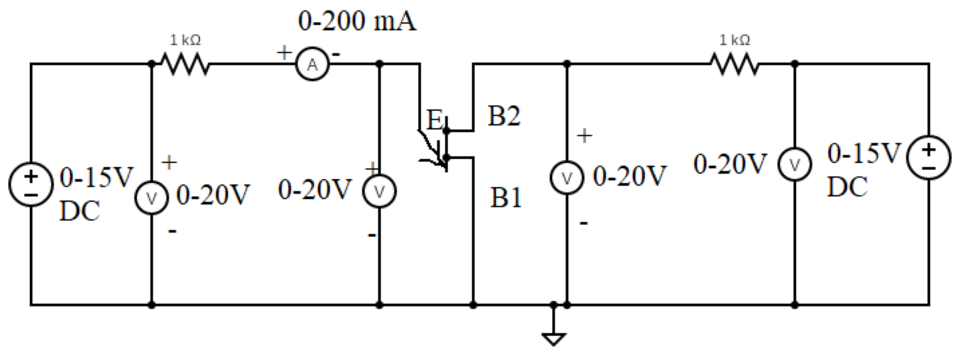

Figure 11.2: Circuit Diagram for UJT Characteristics

1. Connection is made as per given figure 11.2.
2. Output voltage is fixed at a constant level and by varying input voltage corresponding emitter current values are noted down.
3. This procedure is repeated for different values of output voltages.
4. All the readings are tabulated and Intrinsic Stand-Off ratio is calculated using

$$\eta = \frac{(V_p - V_d)}{V_{BB}}$$

5. A graph is plotted between V_{EE} and I_E for different values of V_{BE}.

Table 11.1: Observation table

V_{BB}=1V		V_{BB}=2V		V_{BB}=3V	
V_{EB}	I_E	V_{EB}	I_E	V_{EB}	I_E

11.7 Further Probing Experiments

1. Design and observe the characteristics of relaxation oscillator using Uni-Junction Transistor.
2. Design Voltage sensing with a unijunction transistor and observe the characteristics.

LAB-11 SCR CHARACTERSTICS

12.1 Introduction

The purpose of this experiment is to acquaint the student with the operation of Silicon Control rectifier. From these characteristics, you will observe how the forward break over voltage is reduced with increasing of gate current.

12.2 Objective

1. To observe and analyse the V-I Characteristics of Silicon Control Rectifier (SCR) 2. To determine holding, latching current and break over voltage of given SCR.

12.3 Prelab Preparation

Theoretical background of SCR and V-I characteristics in forward and reverse blocking mode.

12.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
SCR
Resistors – 100k and 1000k ohms
Ammeter- 0-200mA and 0-200uA
Voltmeter - 0-20V DC

12.5 Background

It is a four layer semiconductor device being alternate of p-type and n-type silicon. It consists os 3 junctions J1, J2, J3 the J1 and J3 operate in forward direction and J2 operates in reverse direction and three terminals called anode A, cathode K , and a gate G. The operation of SCR can be studied when the gate is open and when the gate is positive with respect to cathode.

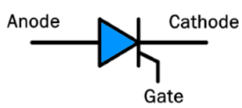

Figure 12.1: Symbol of SCR

When gate is open, no voltage is applied at the gate due to reverse bias of the junction J2 no current flows through R2 and hence SCR is at cut off. When anode voltage is increased J2 tends to breakdown.

When the gate positive, with respect to cathode J3 junction is forward biased and J2 is reverse biased .Electrons from n-type material move across junction J3 towards gate while holes from p- type material moves across junction J3 towards cathode. So gate current starts flowing, anode current increase is in extremely small current junction J2 break down and SCR conducts heavily. When gate is open the break over voltage is determined on the minimum forward voltage at which SCR conducts heavily. Now most of the supply voltage appears across the load resistance. The holding current is the maximum anode current gate being open, when break over occurs.

If a anode to cathode voltage is increased to a sufficiently large value, the reversed biased junction J2 will break, this is known as avalanche breakdown and the corresponding voltage is called forward breakdown voltage VBO. Since junctions J1 and J3 are already forward biased, there will be free movement of carriers across all three junctions, resulting in a large forward anode current. The device will then be in a conducting state or ON state. The voltage drop would be due to the ohmic drop in the four layers and it is small, typically, 1V. In the ON state, the anode current is limited by an external impedance or resistance.

Latching current is the minimum anode current required to maintain the SCR in the ON state immediately after the SCR has been turned on and the gate signal has been removed. Once the SCR is turned ON, it behaves like a conducting diode and there is no control over the device. The device will continue to conduct because there is no depletion layer on the junction J2 due to the free movements of the carriers. However if the forward anode current is reduced below a level known as the holding current, a depletion layer will develop around the junction J2, due to reduced number of carriers and the SCR will be in the blocking state. Holding current is the minimum anode current required to maintain the SCR in the ON state.

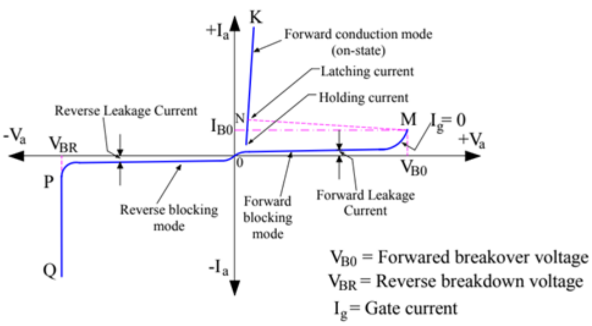

Figure 12.2: Characteristics of SCR

12.6 Procedure

1. Connection is made as per given figure 12.3
2. Set the both voltage sources to zero volts.
3. Switch on the SCR trainer kit.
4. Set the gate current of SCR at 60 μA in the ammeter by varying the gate power supply.

5. Now slowly vary the Anode voltage from 0 to 30 volts. Measure the voltage in the voltmeter, which is connected between anode and cathode.
6. Once SCR has fired for a particular gate current, note down anode to cathode voltage and down the gate current of SCR.
7. Now increase the anode to cathode supply voltage and note down the anode current.
8. Now repeat the steps 5 to 7 for gate currents 70 μA and 80 μA
9. Draw the graph between anode and cathode voltages and the anode current for various gate currents.
10. Note down the latching and holding currents from the plot.

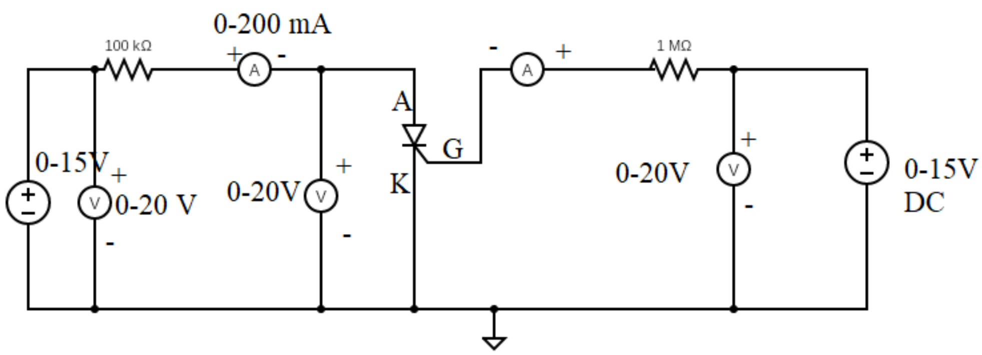

Figure 12.3: Circuit Diagram for SCR Characteristics

Table 12.1: Observation table

Gate Current =		**Gate Current =**	
Anode to Cathode Voltage	**Anode Current**	**Anode to Cathode Voltage**	**Anode Current**

12.7 Further Probing Experiments

1. Design battery charger circuit using silicon control rectifier.
2. Observe the characteristics of RC half wave and full wave Firing Circuit using silicon control rectifier.

LAB-12 FET CHARACTERSTICS

13.1 Introduction

The purpose of this experiment is to acquaint the student to measure the characteristics of a field effect transistor (FET) and to compare these characteristics with the theoretically predicted characteristics.

13.2 Objective

1. To observe the Drain and Transfer characteristics of a given FET in Common source configuration.

13.3 Prelab Preparation

You should be familiar with the discussion of field effect transistors in your text and the parameters that are used to describe the FET. You will be measuring these parameters in the lab using the analog digilent discovery kit.

13.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
FET-BFW11
Resistors – 1k and 470 ohms
Ammeter- 0-200mA
Voltmeter - 0-20V DC

13.5 Background

A FET is a three terminal device, having the characteristics of high input impedance and less noise, the Gate to Source junction of the FET s always reverse biased. In response to small applied voltage from drain to source, the n-type bar acts as sample resistor, and the drain current increases linearly with $\boldsymbol{V_{DS}}$. With increase in $\boldsymbol{I_D}$ the ohmic voltage drop between the source and the channel region reverse biases the junction and the conducting position of the channel begins to remain constant. The voltage at this instant is called "pinch of voltage". If the gate to source voltage ($\boldsymbol{V_{GS}}$) is applied in the direction to provide additional reverse bias, the pinch off voltage is decreased. In amplifier application, the FET is always used in the region beyond the pinch-off.

$$\boldsymbol{I_D = I_{DSS}(\frac{1-V_{GS}}{V_p})^2}$$

FET parameters
AC drain resistance is

$$r_d = \frac{\Delta V_{DS}}{\Delta I_D} \text{ at constant } V_{GS}$$

transconductance is

$$g_m = \frac{\Delta I_D}{\Delta V_{GS}} \text{ at constant } V_{DS}$$

amplification factor is

$$\mu = \frac{\Delta V_{DS}}{\Delta V_{GS}} \text{ at constant } I_D$$

Relation between above parameters

$$\mu = r_d * g_m$$

13.6 Procedure

1. Connection is made as per the given Figure 13.1.
2. To plot the drain characteristics, keep V_{GS} constant at 0V.
3. Vary the V_{DD} and observe the values of V_{DS} and I_D.
4. Repeat the above steps 2, 3 for different values of V_{GS} at 0.1V and 0.2V.
5. All the readings are tabulated.
6. To plot the transfer characteristics, keep V_{DS} constant at 1V.
7. Vary V_{GG} and observe the values of V_{GS} and I_D.
8. Repeat steps 6 and 7 for different values of V_{DS} at 1.5 V and 2V.
9. The readings are tabulated.
10. From drain characteristics, calculate the values of dynamic resistance (r_D)
11. From transfer characteristics, calculate the value of transconductace (g_m), and also calculate Amplification factor (μ).

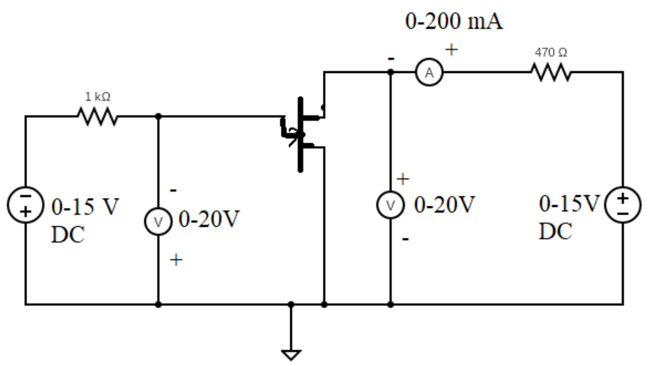

Figure 13.1: Circuit Diagram for FET Characteristics

Table 13.1: Observation table for drain characteristics

V_{GS}=0V		V_{GS}=0.1V		V_{GS}=0.2V	
V_{DS}	I_D	V_{DS}	I_D	V_{DS}	I_D

Table 13.2: Observation table for transfer characteristics

V_{DS} = 0.5 V		V_{DS} = 1 V		V_{DS} = 1.5 V	
V_{GS}	I_D	V_{GS}	I_D	V_{GS}	I_D

13.7 Further Probing Experiments

1. Obtain the transistor drain characteristics in the saturated region, by applying the V_{MAX} is 40V, I_{MAX} is 20 mA and P_{MAX} is 0.4W.

2. Junction field-effect transistors (JFETs) are normally-ON devices, the natural state of their channels being passable to electric currents. Thus, a state of cutoff will only occur on command from an external source. Explain what must be done to a JFET, specifically, to drive it into a state of cutoff.

LAB-13 FREQUENCY RESPONSE OF CS AMPLIFIER

14.1 Introduction

The purpose of this experiment is to study the use of the junction field effect transistor (JFET) as an amplifier, with attention given to the proper biasing of the circuit.

14.2 Objective

1. To obtain Frequency Response of Common Source FET amplifier.
2. To Measure the Voltage gain and Bandwidth of CS Amplifier.

14.3 Prelab Preparation

You should be familiar with biasing of the JFET using a source resistor. You should be familiar with analysis of common-source amplifiers, particularly calculation of the voltage gain. Use the linear characteristics from FET characteristics to determine the closest value $\boldsymbol{V_{GS}}$ to the desired Q-point.

14.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
FET-BFW11
Resistors – 1k and 470 ohms
Ammeter- 0-200mA
Voltmeter - 0-20V DC

14.5 Background

A field-effect transistor (FET) is a type of transistor commonly used for weak-signal amplification (for example, for amplifying wireless (signals). The device can amplify analog or digital signals. It can also switch DC or function as an oscillator. In the FET, current flows along a semiconductor path called the channel. At one end of the channel, there is an electrode called the source. At the other end of the channel, there is an electrode called the drain. The physical diameter of the channel is fixed, but its effective electrical diameter can be varied by the application of a voltage to a control electrode called the gate. Field-effect transistors exist in two major classifications. These are known as the junction FET (JFET) and the metal-oxide semiconductor FET (MOSFET). The junction FET has a channel consisting of n-type semiconductor (n-channel) or p-type semiconductor (p-channel) material, and gate is made of the opposite semiconductor type. In p-type material, electric charges are carried mainly in the form

of electron deficiencies called holes. In n-type material, the charge carriers are primarily electrons.
In a JFET, the junction is the boundary between the channel and the gate. Normally, this pn junction is reverse-biased (a DC voltage is applied to it) so that no current flows between the channel and the gate. However, under some conditions there is a small current through the junction during part of the input signal cycle. The FET has some advantages and some disadvantages relative to the bipolar transistor. Field-effect transistors are preferred for weak-signal work, for example in wireless, communications and broadcast receivers. They are also preferred in circuits and systems requiring high impedance.
An amplifier is a circuit that increase/decreases the input signal value. In this experiment you are going to investigate frequency response characteristic of a voltage amplifier circuit using the n-channel JFET device. Most amplifiers have relatively constant gain over a certain range of frequencies. This range of frequencies is called the bandwidth of the amplifier. The bandwidth for a given amplifier depends on the circuit component values, the type of active components and the dc operating point of the active component. When an amplifier is operated within its bandwidth, the current gain $\boldsymbol{A_i}$, voltage gain $\boldsymbol{A_v}$, and power gain $\boldsymbol{A_p}$ values are referred to as midband gain values. A simplified frequency-response curve that represents the relationship between amplifier gain and operating frequency is shown in Figure 14.1
As the frequency-response curve shows, the power gain of an amplifier remains relatively con-

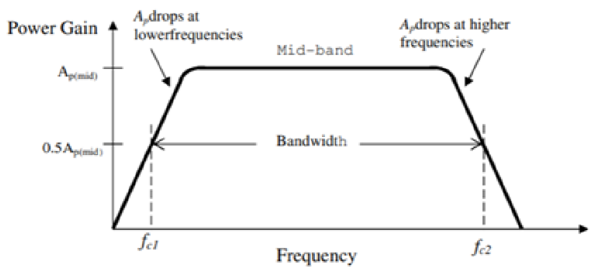

Figure 14.1: A simplified frequency Response curve

stant across a band of frequencies. When the operating frequency starts to go outside this frequency range, the gain begins to drop. Two frequencies of interest, $\boldsymbol{f_{c1}}$ and $\boldsymbol{f_{c2}}$ are the frequencies at which power gain decreases to approximately 50 percentage of $\boldsymbol{A_{p(mid)}}$. The frequencies labeled $\boldsymbol{f_{c1}}$ and $\boldsymbol{f_{c2}}$ are called the lower and upper cutoff frequencies of an amplifier, respectively. These frequencies are considered to be the bandwidth limits for the amplifier and thus bandwidth BW is given by

$$\text{BW} = \boldsymbol{f_{c2}} - \boldsymbol{f_{c1}}$$

14.6 Procedure

1. Connection is made as per the given Figure 14.2.
2. Set Source Voltage $\boldsymbol{V_s}$ = 50mV (say) at 1 KHz frequency, using function generator.
3. Keeping the input voltage constant, vary the frequency from 50Hz to 1MHz in regular steps and note down the corresponding output voltage.
4. Calculate the Voltage Gain by using the formula.

$$\boldsymbol{A_v = Outputvoltage(V_0)/Inputvoltage(V_s)}$$

5. Calculate the Voltage Gain in dB by using Voltage Gain Av (dB) = 20 log10 (Vo/Vs).

$$\boldsymbol{VoltageGainA_v(dB) = 20log_{10}(V_0/V_s)}$$

6. Plot the Graph by taking Voltage gain (dB) on x-axis and frequency (Hz) on y-axis.

7. The Bandwidth of the amplifier is calculated from the graph using the expression Bandwidth

$$\mathrm{BW} = \boldsymbol{f_2 - f_1}$$

Where $\boldsymbol{f_1}$ is lower 3-dB frequency and $\boldsymbol{f_2}$ is upper 3-dB frequency

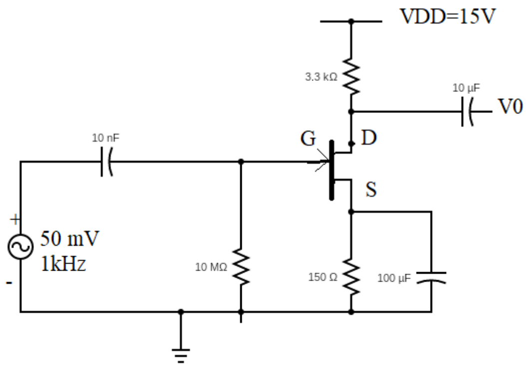

Figure 14.2: Circuit Diagram for Frequency response of CS FET amplifier

Table 14.1: Observation table

Frequency (Hz)	**Output Voltage (V)**	**Gain**	**Gain in dB**

14.7 Further Probing Experiments

1. Build and perform the CS amplifier circuit using $\boldsymbol{V_{DD}} = \boldsymbol{V_{SS}} = 5$ V. Select 50 k ohms potentiometer and adjust it to obtain 250 uA bias current. Select $\boldsymbol{R_S} = 10$k ohms.

2. Obtain the frequency response of MOSFET amplifier in common source configuration.

LAB-14 FREQUENCY RESPONSE OF CD AMPLIFIER

15.1 Introduction

The purpose of this experiment is to study the use of the junction field effect transistor (JFET) as an amplifier, with attention given to the proper biasing of the circuit.

15.2 Objective

1. To obtain Frequency Response of Common Drain FET amplifier. 2. To Measure the Voltage gain and Bandwidth of CD Amplifier.

15.3 Prelab Preparation

You should be familiar with biasing of the JFET using a source resistor. You should be familiar with analysis of common-Drain amplifiers, particularly calculation of the voltage gain. Use the linear characteristics from FET characteristics to determine the closest value VGS to the desired Q-point.

15.4 Equipment needed

Analog Discovery 2 Instrument
Multisim software
Personal computer
FET-BFW11
Resistors – 1k and 470 ohms
Ammeter- 0-200mA
Voltmeter - 0-20V DC

15.5 Background

The common drain amplifier is analogous to the common collector emitter follower. The JFET version is also known as a source follower. The common drain or source follower circuit is able to provide a very high input impedance and low output impedance and is used to act as a buffer amplifier.
An amplifier is a circuit that increase/decreases the input signal value. In this experiment you are going to investigate frequency response characteristic of a voltage amplifier circuit using the n-channel JFET device. Most amplifiers have relatively constant gain over a certain range of frequencies. This range of frequencies is called the bandwidth of the amplifier. The bandwidth for a given amplifier depends on the circuit component values, the type of active components and the dc operating point of the active component. When an amplifier is operated within its bandwidth, the current gain $\boldsymbol{A_i}$, voltage gain $\boldsymbol{A_v}$, and power gain $\boldsymbol{A_p}$ values are referred to

as midband gain values. As the frequency-response curve shows, the power gain of an amplifier

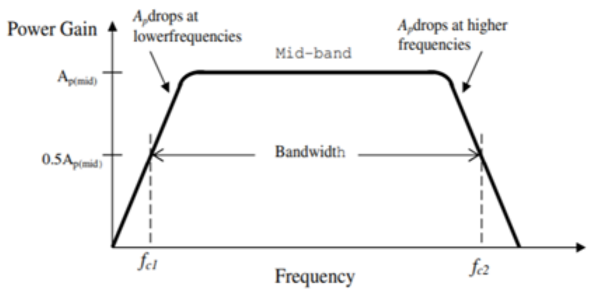

Figure 15.1: A simplified frequency Response curve

remains relatively constant across a band of frequencies. When the operating frequency starts to go outside this frequency range, the gain begins to drop. Two frequencies of interest, f_{c1} and f_{c2} are the frequencies at which power gain decreases to approximately 50 percentage of $A_{p(mid)}$. The frequencies labeled f_{c1} and f_{c2} are called the lower and upper cutoff frequencies of an amplifier, respectively. These frequencies are considered to be the bandwidth limits for the amplifier and thus bandwidth BW is given by

$$BW = f_{c2} - f_{c1}$$

15.6 Procedure

1. Connection is made as per the given Figure 15.2.
2. Set Source Voltage V_s = 100mV (say) at 1 KHz frequency, using function generator.
3. Keeping the input voltage constant, vary the frequency from 50Hz to 1MHz in regular steps and note down the corresponding output voltage.
4. Calculate the Voltage Gain by using the formula.

$$A_v = Outputvoltage(V_0)/Inputvoltage(V_s)$$

5. Calculate the Voltage Gain in dB by using Voltage Gain Av (dB) = 20 log10 (Vo/Vs).

$$VoltageGainA_v(dB) = 20log_{10}(V_0/V_s)$$

6. Plot the Graph by taking Voltage gain (dB) on x-axis and frequency (Hz) on y-axis.
7. The Bandwidth of the amplifier is calculated from the graph using the expression Bandwidth

$$BW = f_2 - f_1$$

Where f_1 is lower 3-dB frequency and f_2 is upper 3-dB frequency

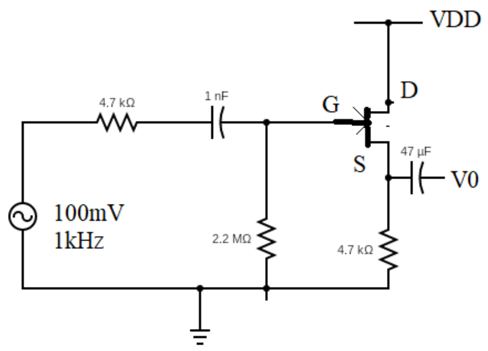

Figure 15.2: Circuit Diagram for Frequency response of CD FET amplifier

Table 15.1: Observation table

Frequency (Hz)	Output Voltage (V)	Gain	Gain in dB

15.7 Further Probing Experiments

1. Design and Plot the frequency response of single stage RC coupled amplifier using JFET
2. Design a MOSFET amplifier and plot frequency response based on the given specifications. Both the input and the output should be AC coupled.
   dual Supply Voltage is ± 5V
   Load Resistance, RL = 100 ohms
   0 to peak output swing is greaterthan or equal to 2V
   voltage gain is 50
   input resistance is 10k ohms.

Appendix-A

do's don'ts and safety

do's:

1. Proper dress code has to be maintained.
2. Students should carry observation notes and record completed in all aspects.
3. Keep bags and belongings in the allocated place.
4. Perform only appropriate experiments.
5. Maintain silence in the lab.
6. Return the experimental setup after experiment is over.
7. Get the worksheet signed before leaving the lab.
8. Switch off all the equipment's immediately after the completion of experiments.

don'ts:

1. Don't come late to the Lab.
2. Don't leave the Lab without making proper arrangement of equipment's, switch off devices and keeping the chairs properly.
3. Don't leave the lab without the permission of the faculty In -Charge.
4. Do not tamper and touch the equipment's without proper knowledge.
5. Do not communicate with other batch students.
6. Do not modify equipment settings unless instructed by lab instructor.

safety norms:

1. The lab must be equipped with fire extinguisher.
2. There should not be any overlapping of electrical wiring for the systems.
3. Monitor and CPU must be properly shielded.

Appendix-B

WORKSHOP PRACTICE

RESISTOR

A Resistor is a passive two-terminal electrical component that implements electrical resistance as a circuit element. The current through a resistor is in direct proportion to the voltage across the resistor's terminals. This relationship is represented by Ohm's law. Where 'I' is the current through the conductor in units of amperes, V is the potential difference measured across the conductor in units of volts, and R is the resistance of the conductor in units of ohms. The ratio of the voltage applied across a resistor's terminals to the intensity of current in the circuit is called its resistance, and this can be assumed to be a constant (independent of the voltage) for ordinary resistors working within their ratings.

COLOUR CODING OF RESISTOR

Color Codes are used to identify the value of resistor. The numbers to the Color are identified in the following sequence which is remembered as BBROY GREAT BRITAN VERY GOOD WIFE (BBROYGBVGW) and their assignment is listed in following table.

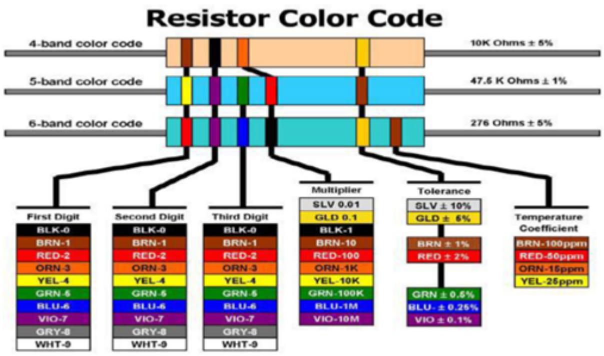

Figure 17.1: Procedure to find the value of Resistor using Color codes

CAPACITOR

A capacitor (originally known as a condenser) is a passive two-terminal electrical component used to store energy electro statically in an electric field. By contrast, batteries store energy via chemical reactions. The forms of practical capacitors vary widely, but all contain at least two electrical conductors separated by a dielectric (insulator); for example, one common construction consists of metal foils separated by a thin layer of insulating film. Capacitors are widely used as parts of electrical circuits in many common electrical devices. When there is a potential difference (voltage) across the conductors, a static electric field develops across the dielectric, causing positive charge to collect on one plate and negative charge on the other plate. Energy is stored in the electrostatic field. An ideal capacitor is characterized by a single constant value, capacitance. This is the ratio of the electric charge on each conductor to the potential difference between them. The SI unit of capacitance is the farad, which is equal to one coulomb per volt.

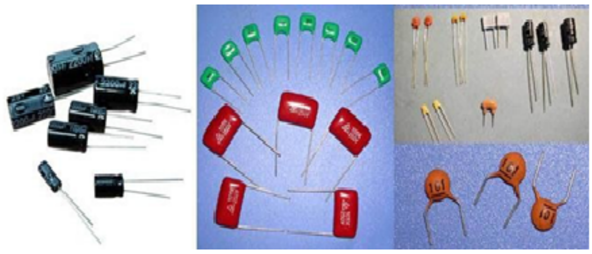

Figure 17.2: Different Capacitors

INDUCTORS

An inductor, also called a coil or reactor, is a passive two-terminal electrical component which resists changes in electric current passing through it. It consists of a conductor such as a wire, usually wound into a coil. When a current flows through it, energy is stored in a magnetic field in the coil. When the current flowing through an inductor changes, the time-varying magnetic field induces a voltage in the conductor, according to Faraday's law of electromagnetic induction, which by Lenz's law opposes the change in current that created it. Inductors, also called coils, can be a bit harder to figure out their values. If they are color coded, the resources listed for resistors can help, otherwise a good meter that can measure inductance will be needed. They are typically marked with an "L" on a circuit board.

Diode

Semiconductor pn junction diode which passes curent when forward-biased, and blocks current flow when reverse-biased. Commonly used in small-signal, rectification or high current applications.

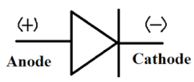

Figure 17.3: diode symbol

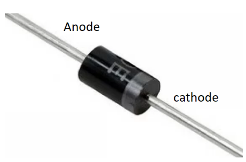

Figure 17.4: diode model

Zener diode

Zener diode used in its reverse voltage breakdown region for voltage limiting, transient suppression and regulation applications. Available in a range of reverse breakdown voltage values.

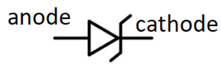

Figure 17.5: Zener diode symbol

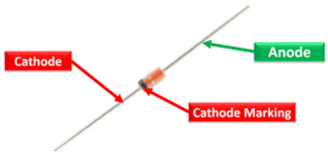

Figure 17.6: Zener diode model

Bipolor Junction Transistor

A transistor is a type of a semiconductor device that can be used to both conduct and insulate electric current or voltage. A transistor basically acts as a switch and an amplifier.The three terminals of BJT are base, emitter and collector. A very small current flowing between base and emitter can control a larger flow of current between the collector and emitter terminal. Furthermore, there are two types of BJT. These include:

PNP Transistor:

It is a type of BJT where one n-type material is introduced or placed between two p-type materials. In such a configuration, the device will control the flow of current. PNP transistor consists of 2 crystal diodes which are connected in series. The right side and left side of the diodes are known as the collector-base diode and emitter-base diode respectively.

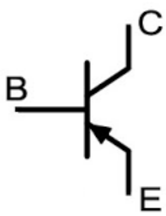

Figure 17.7: PNP Transistor symbol

NPN Transistor:

In this transistor, we will find one p-type material that is present between two n-type materials. N-P-N transistor is basically used to amplify weak signals to strong signals. In NPN transistor, the electrons move from the emitter to collector region resulting in the formation of

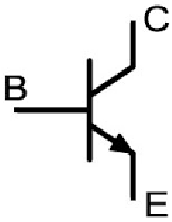

Figure 17.8: NPN Transistor symbol

current in the transistor.

MULTIMETER:

Multimeter is an electronic device that is used to make various electrical measurements, such as AC and DC voltage, AC and DC current, and resistance. It is called a Multimeter because it combines the functions of a voltmeter, ammeter, and ohmmeter. Multimeter may also have other functions, such as diode test, continuity test, transistor test, TTL logic test and frequency test. Procedure for Measurement Voltage Measurement: A.C. Voltage Measurement 1. Connect the positive (red) test lead to the 'V/mA' jack socket and the negative (black) lead to the 'COM' jack socket.
2. Set the selector switch to the desired mV D.C./D.C.V/A.C.V range.
3. Connect the test leads to the circuit to be measured.
4. Turn on the power to the circuit to be measured, the voltage value should appear on the digital display along with the voltage polarity (if reversed only).
Current Measurement:
1. Connect the positive(red) test lead to the 'V/mA' jack socket and the negative(black) lead to the 'COM' jack socket(for measurements up to 200mA). For measurements between 200mA and 10A connect the red test lead to the '10mA' socket.
2. Set the selector switch to the desired uA/mA/A range.
3. Open the circuit to be measured and connect the test leads in SERIES with the load in which current is to be measured.
4. To avoid blowing an input fuse, use the 10A jack until you are sure that the current is less than 300 mA. Turn off power to the circuit. Break the circuit. (For circuits of more than 10 amps, use a current clamp.) Put the meter in series with the circuit and turn power on. Resistance Measurement:

1. Connect the positive(red) test lead to the 'V/mA' jack socket and the negative(black) lead to the 'COM' jack socket.

2. Set the selector switch to the desired 'OHM Ω'.

3. If the resistance to be measured is part of a circuit, turn off the power and discharge all capacitors before measurement.

4. Connect the test leads to the circuit to be measured.

5. The resistance value should now appear on the digital display.

6. If the resistance to be measured is part of a circuit, turn off the power and discharge all capacitors before measurement.

Continuity Test:
This mode is used to check if two points are electrically connected. It is often used to verify connectors. If continuity exists (resistance less than 210 ohms), the beeper sounds continuously.

1. Connect the positive (red) test lead to the 'V/mA' jack socket and the negative (black) lead to the 'COM' jack socket.
2. Set the selector switch to the position.
3. Connect the test leads to two points of the circuit to be tested. If the resistence is Ohms the buzzer will sound.
4. If the resistance to be measured is part of a circuit, turn off the power and discharge all capacitors before measurement.

FUNCTION GENERATOR:

A function generator is electronic test equipment used to generate different types of waveforms over a wide range of frequencies. Function generators are capable of producing a variety of repetitive waveforms, generally sine, square and pulse waveforms.

DC VARIABLE POWER SUPPLY:

A power supply is a device that supplies electric power to an electrical load. The term is most commonly applied to electric power converters that convert one form of electrical energy to another, though it may also refer to devices that convert another form of energy (mechanical, chemical, solar) to electrical energy. A regulated power supply is one that controls the output voltage or current to a specific value; the controlled value is held nearly constant despite variations in either load current or the voltage supplied by the power supply's energy source.
A power supply may be implemented as a discrete, stand-alone device or as an integral device that is hardwired to its load. Examples of the latter case include the low voltage DC power supplies that are part of desktop computers and consumer electronics devices.
Commonly specified power supply attributes include:
1. The amount of voltage and current it can supply to its load.
2. How stable its output voltage or current is under varying line and load conditions.

ANALOG DISCOVERY 2:

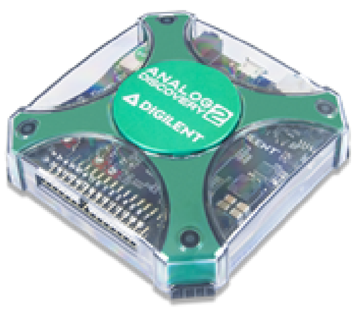

Figure 17.9: Analog Discovery 2 Kit

The Digilent Analog Discovery 2, developed in conjunction with Analog Devices, is a multi-function instrument that allows users to measure, visualize, generate, record, and control mixed signal circuits of all kinds. The low-cost Analog Discovery 2 is small enough to fit in your pocket, but powerful enough to replace a stack of lab equipment, providing engineering students, hobbyists, and electronics enthusiasts the freedom to work with analog and digital circuits in virtually any environment, in or out of the lab.

Features

The analog and digital inputs and outputs can be connected to a circuit using simple wire probes, alternatively, the Analog Discovery BNC Adapter and BNC probes can be used to connect and utilize the inputs and outputs. Driven by the free Waveforms software, the Analog Discovery 2 can be configured to work as any one of several traditional instruments, which include:

1. Two-channel oscilloscope
2. Two-channel arbitrary function generator
3. Stereo audio amplifier to drive external headphones or speakers with replicated AWG signals
4. 16-channel digital logic analyzer (3.3V CMOS, 100MS/s)
5. 16-channel pattern generator (3.3V CMOS, 100MS/s)
6. 16-channel virtual digital I/O including buttons, switches, and LEDs – perfect for logic training applications
7. Two input/output digital trigger signals for linking multiple instruments (3.3V CMOS)
8. Two programmable power supplies . The maximum available output current and power depend on the Analog Discovery 2 powering choice
9. 500mW total when powered through USB
10. 2.1W max for each supply when powered by an auxiliary supply
11. 700mA maximum current for each supply
12. Two-channel voltmeter
13. Network analyzer – Bode, Nyquist, Nichols transfer diagrams of a circuit. Range: 1Hz to 10MHz
14. Spectrum Analyzer – power spectrum and spectral measurements (noise floor, SFDR, SNR, THD, etc.)
15. Digital Bus Analyzers (SPI, I^2C, UART, Parallel, CAN)

Pinout Diagram

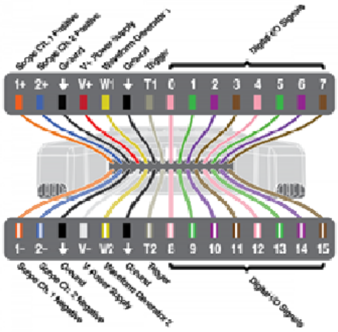

Figure 17.10: Pin diagram of analog discovery 2 Kit

Appendix-C

MULTISIM CIRCUIT SIMULATION

NI multisim is a powerful schematic capture and simulation environment that engineers, students, and professors can use to simulate electronic circuits and prototype Printed Circuit Boards (PCBs). This article shows you how to capture, simulate, and lay out your first design in Multisim.
The example circuit in the article is an amplifier circuit. This non-inverting operational amplifier configuration consists of one active component (the operational amplifier) and two passive resistor components that will be used to complete the feedback network to provide gain in this circuit.
Resistive Circuit Analysis:
Run the software by executing Multisim. The following user interface window will display. The blank region inside is the Schematics window. Multisim software integrates two programs: A

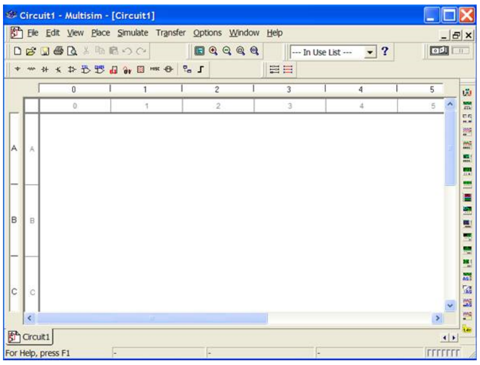

Figure 18.1: Schematic window

circuit schematics editor, and a circuit simulator. The schematics editor allows the user to build a virtual circuit by connecting various electronic components; basically, the user “draws” the schematic of the circuit to be analyzed. Let us assume that the circuit to be analyzed is the following simple voltage divider circuit. We are interested in all component voltages and currents. 1. The above circuit has three components: Two resistors and one DC voltage supply. In addition, we must specify a reference (ground) node. From the Place Menu, select Component from menu).
2. In the Component Selection Menu, set Database to Master database and Group to Sources. Next, select Power Sources. This will list a number of power source components under the Component list click on DC Power. You should see the symbol for a DC voltage source under the Symbl” area. Click “OK” to place the symbol in the Schematics window. Now, as you move the mouse cursor over the Schematics window, you will see the symbol of a DC voltage source tracking the mouse. Click the left button of the mouse to insert the component. The last inserted component can be removed if you select Undo from the Edit Menu. Alternatively,

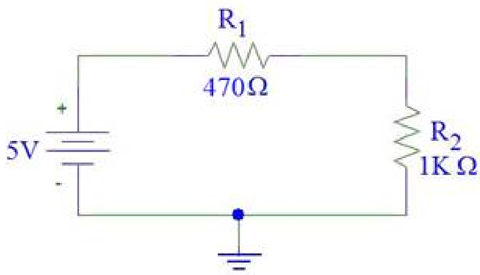

Figure 18.2: Resistive Circuit

a part can be removed if you select it by clicking its symbol (a dashed box is placed around the symbol) and then selecting Delete from the Edit Menu, or by pressing delete key on the keyboard. The default name for the first inserted DC source is V1. The second inserted source name would be V2, and so on. The default value of the source is 12V. The position of the name label (and value label) can be relocated by clicking and holding and then dragging the label. Similarly, the whole symbol can be repositioned by clicking and holding the left button of the mouse (at the center of the symbol) and then dragging.

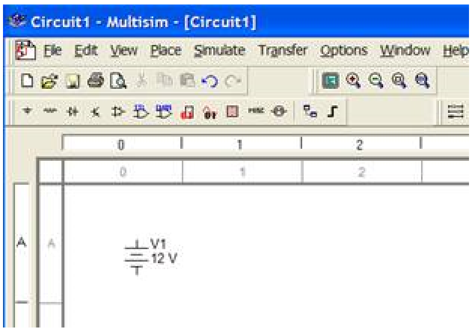

Figure 18.3: view for component placement

3. You may now edit the voltage value of the inserted DC source by double clicking its symbol. This will open the power sources window. Type the desired voltage value (5V in this case) and click OK. The name of the supply can also be edited under the Label tag in this menu.

Figure 18.4: component edit window

4. Repeat Steps 1 and 2 for the two resistors. The filter parameters in the Component Selection

window should be set to W and 5 percent for the 470W resistor, and kW and 5 percent for the 1KW resistor. The filter allows for faster location of the specific component. The inserted resistors names will automatically be set to R1 and R2, respectively.

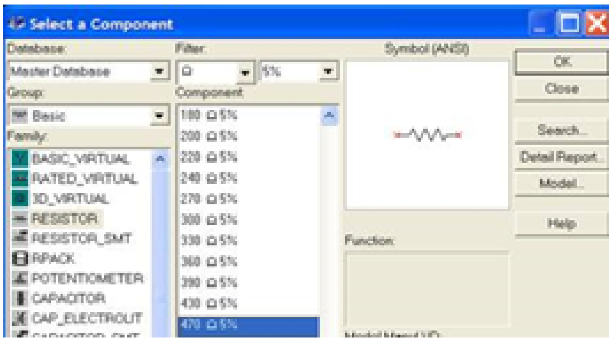

Figure 18.5: resistor component selection window

The Schematic Window should now look as shown below.

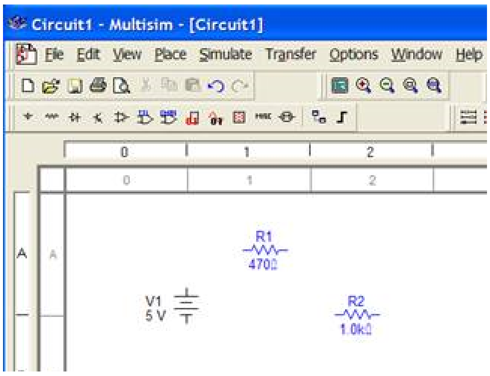

Figure 18.6: placing of all components in schematic window

5. Every simulated circuit must have a reference (ground) node. Open the Place Menu and select the "GROUND" component from the power sources family, under the sources group (refer to Steps 1 and 2). Click OK to close the menu. Click the mouse inside the Schematics window to insert the ground symbol.

6. The last step in drawing the circuit schematic is to interconnect the components. Select Wire from the Place Menu. The mouse cursor changes to a + cursor. Point the cursor to the upper tip of the DC source and click. Then, click the mouse at the left terminal of R1. This inserts a wire connecting the DC source to R1. Repeat this process to finish connecting the remaining components together, according to the given circuit topology. Right click the mouse upon completing the circuit wiring; this, deactivates the Wire Tool. Now, try dragging a component and observe how the position of the connecting wires change. A wire segment can be repositioned by dragging (left click, hold and move) it with the mouse.

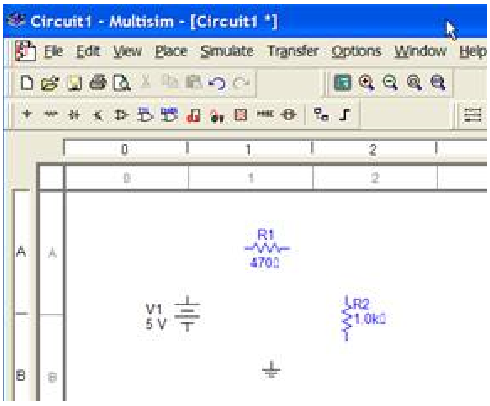

Figure 18.7: after placing the ground symbol in schematic window

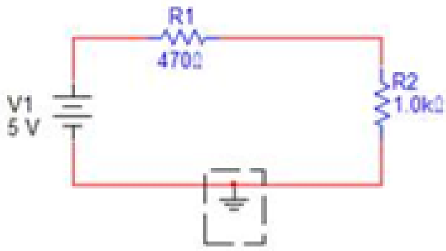

Figure 18.8: after making the connections between the components and sources in schematic window

7. A virtual Multimeter instrument is available in Multisim. you can display it by selecting "Instruments" from the View - Toolbars Menu). Toolbars can be relocated from the side to the top Menu (and vice versa) by dragging the Toolbar at its base . Double clicking the icon pastes the multimeter symbol in the circuit area. Double clicking this symbol opens the Multimeter window.The user can set the multimeter mode to Voltage, Current or Ohm measurement and to DC or AC signals. Select DC Volts as shown below. The multimeter is now a voltmeter which is set to measure DC voltages across a circuit component.

Figure 18.9: Selection of voltmeter in multimeter

Connect the Voltmeter across R2 as shown below

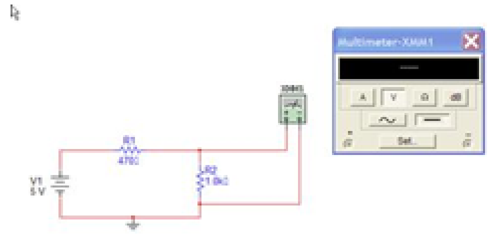

Figure 18.10: Multimeter is connected across R2 in schematic window

8. Now, you may simulate the circuit. Choose Run from the Simulate Menu.The simulation results are shown in the figure below.

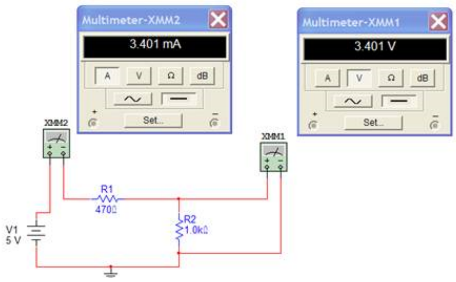

Figure 18.11: after simulation view for schematic window

Note: The simulation must be terminated (press the “0”on) before you can make changes to the existing circuit.