

ECE 3110
LABORATORY MANUAL
VER 1.9

Apoorva Kapadia
D J Dumin
Department of Electrical and Computer Engineering
Clemson University
Clemson, SC, 29634

Version 1.6, July 2023 – Apoorva Kapadia and Praladh Paniyil

CONTENTS

CONTENTS	ii
Notes about the Course	iii
Syllabus Example.....	vi
Lab Schedule Example.....	viii
SAFETY	ix
Standard Component Values	x
Graph Paper	xii
LABORATORY REPORTS	xiv
Pre-Lab Reports	xv
Formal Lab Reports.....	xvii
LABORATORY EXPERIMENTS.....	xxii
Introduction – Laboratory Demonstration.....	1
Experiment #1 – Diode Characteristics.....	7
Experiment #2 – Power Supply Operation.....	17
Experiment #3 – Power Supply Design	27
Experiment #4 – Diode Clippers and Clampers	32
Experiment #5 – Bipolar Junction Transistor Characteristics	43
Experiment #6 – BJT Common-Emitter Circuit Bias	53
Experiment #7 – BJT Common-Emitter Circuit Voltage Gain	60
Experiment #8 – BJT Common-Emitter Design I	68
Experiment #9 – BJT Common-Emitter Amplifier Design II.....	74
Experiment #10 – Field Effect Transistors.....	79
Experiment #11 – FET Bias and Amplification	87
Experiment #12 – Basic Logic Circuits	94
Appendix A: Summary on Report Writing	100
Appendix B: Tektronix Type 576 Curve Tracer Introduction.....	101
Appendix C: Tektronix Type 576 Curve Tracer Operating Manual.....	108
<i>Steps for acquiring diode forward I-V characteristics</i>	<i>108</i>
<i>Steps for acquiring diode forward and reverse I-V characteristics at the same time.....</i>	<i>109</i>
<i>Steps for acquiring I_C vs. V_{CE} characteristics for a BJT.....</i>	<i>111</i>
<i>Steps for acquiring I_D vs. V_{DS} characteristics for a FET</i>	<i>112</i>
Appendix D: Hameg HM6042 Curve Tracer Manual.....	114

Notes about the Course

Electronics I consists of a theoretical module ECE 320 and a practical module ECE 3110. These courses run concurrently and the aim of the manual is to introduce students to laboratory procedure including data recording and report writing. The experiments were developed to expand on the material covered in lectures and to experimentally demonstrate the validity of principles presented in ECE 320 classes. The manual outlines 12 experiments and instructors will select the most important topics for experimental confirmation.

NOTE: Learning occurs differently for different students and no one approach is 100% effective. **Laboratory work is an effective teaching tool and it is important to realize that it stands alone. There is no plan or need to have a lecture on a subject prior to a lab.** A lab experience and a lecture should reinforce learning, but the order in which the learning takes place should not significantly affect the process.

It is this instructor's opinion that the students would better understand the fundamentals of electronics and the use of active devices in electronic circuits if the students were able to build circuits and see for themselves that the principles presented in the text are real, or pretty close to reality. It was also felt that the students would become aware of the limitations of the analytical approach to active circuit analysis by testing circuits and comparing their results with analytical solutions and circuit simulations. Most analytical approaches to circuit analysis use linear analysis techniques and these techniques are not always valid for non-linear electronics devices, or are valid only over a limited range of currents and voltages.

The component parameters presented in text books are usually nominal values, valid for most circuits. By measuring actual device parameters and comparing these values with values in the text, the students should be able to obtain a feel for the magnitudes of the device parameters used in the circuits. Experiments in the ECE 3110 labs use discrete devices and some parameters (for example, diode forward current) are often much larger than the values you would measure for similar devices on an integrated circuit. The reason is that integrated circuits contain devices which are physically much smaller and hence they are unable to handle as much power as the larger discrete components.

Most of the experiments require simulation of the circuits. The simulations discussed in the experiments assume the students are familiar with B2 Spice (Beige Bag simulation program with integrated circuit emphasis). **It is important that students simulate their circuits before coming to the lab.** These simulations will be extremely important in the design of experiments: it will be impossible to complete the lab without having done the simulations and calculations based on these simulations before the lab. Most of the experiments ask that you compare the results of the simulations and the results of the analytical analysis with the measured results, to see how the three techniques compare. It is important to gain an understanding of the strengths and limitations of linear circuit analysis techniques taught in class.

It is extremely important for students to read the labs prior to coming to the laboratory. In many cases you will be asked to bring calculated component values and/or simulation results prior to starting the lab in order to guide you in selecting component values for your practical circuit. You will need this information in order to perform and complete the labs. You should also try to become familiar with standard component values, since your calculations will often

call for resistor or capacitor values which are non-standard and you will have to choose from standard components to build your circuits.

The experiments listed in the contents may not be assigned in the order presented. In particular the design experiments may be presented later in the course to let classes get ahead of the labs. The use of these labs later in the course will allow students to become more familiar with electronic circuits and devices before attempting circuit design. The design experiments are also designed as single-student exercises, to test students' individual laboratory skill development. The design experiments should be assigned as one-hour lab sessions and may be used in place of a final exam for this lab.

During the first year of introduction of this lab, the students were asked to complete an experimental evaluation sheet following each experiment. The students were invited to comment on ways to improve the experiment just completed. Students were also encouraged to propose topics they felt would benefit through experimental confirmation of principles discussed in class. Many of these ideas have been incorporated in the manual. Any suggestions from current students for ways to improve the lab learning experience are always greatly appreciated.

Several pages in this manual are reproduced from Hameg EQUIPMENT MANUALS. This material is copyrighted by Hameg and has been reproduced with permission of Hameg. The authors thank Hameg for permission to use this material.

In anticipation that the textbook for ECE320 may change from time to time, the lab manual attempts to remain independent of a specific textbook. Thus in many places short descriptions of principles involved are included and an attempt has been made to define the symbols used in this text. Unfortunately, nomenclature and symbols are not always standard in printed texts on electronic circuits. At the end of each experiment is a 1-page check list. The check list should be used by students to ensure they have covered the most important parts of the experiment.

Examples of lab reports and pre-lab reports have been inserted before the first experiment. These examples are provided to help students learn some of the techniques used for effective technical communication.

The author would like to thank Sally Surver for assistance in typing some of the original copies of experiments found in this lab manual. John Adjaye's help in designing, writing, and performing the experiments was invaluable. The author would also like to thank Robert Palazzo, who wrote the check lists and helped with revisions during the first year. Omer Oralkan helped with writing a lab report example. The author also acknowledges all lab instructors and undergraduates who have contributed to revisions of the manual.

D J Dumin
May 1999

Version 1.1

With the introduction of a complete set of new equipment for the lab, updating experiments and software changes, the lab manual has undergone a major revision. Whilst the general wording and theme is the same, the present author took the liberty of editing many sections and paragraphs. In the interests of keeping the volume to a minimum I have made editorial changes throughout, some paragraphs are verbatim from the first manual whilst others reflect style and

emphasis differences. It is hoped that this new version will build on the excellent foundation provided by Dr Dumin and his students. My apologies if my edits have misconstrued any ideas or concepts attributed to the original “author”. As a work in progress, I expect the manual will undergo periodic revisions to keep it up to date.

My thanks to Daniel Damjanovic for suggestions and editing the original manual and to Trish Nigro who retyped the complete revised manual.

K F Poole
Jan 2004

Version 1.2

The manual has undergone a comprehensive review over the past year. Thanks to David Epting who has tracked and corrected the errors in version 1.1 and suggested many improvements throughout the manual.

Thanks to Janet Bean for preparing the 2005 version 1.2 laboratory manual.

K. F. Poole
Jan 2005

Version 1.3

The manual has been modified to accommodate new equipment – NI-ELVIS workstation and the Tektronix Type 576 Curve Tracer. Additional reformatting and corrections by Dr J. E Harriss.

E. Iyasere
May 2010

Version 1.4

The manual has undergone an extensive review and many revisions have been made to clarify and correct the lessons throughout the experiments. The manual still favors the Hameg curve tracer; a clear and balanced treatment using the Tektronix curve tracer needs to follow.

Nishant Gupta
J. E. Harriss
January 2011

Version 1.5

Correct newly identified errors. Clarify sections that students identified as troublesome. Eliminate confusing R_B references in Experiments 6 through 9.

Nishant Gupta
J. E. Harriss
July 2011

Version 1.6

Updated to reflect new component values for a new transistor and for the AD2.

Praladh Paniyil
Apoorva Kapadia
July 2023

Syllabus Example

ECE 3110 Electrical Engineering Lab Syllabus

Instructor Information:

Name :
Email :
Phone :
Office :
Office Hours :

Course Coordinator:

Name : Dr. xxxxx xxxxx
Email : xxxxx@clemson.edu
Office : xxx Riggs Hall
Phone : 656-xxxx

Course Section Information:

Semester : Fall 2010
Section : 004
Time : 2:00 – 4:00 TH
Room : Riggs Hall 200C

Materials Required:

ECE 320 Text
ECE 3110 Lab Manual
(download from http://www.clemson.edu/ces/departments/ece/resources/lab_manuals.html)
IEEE Lab Kit
Scientific Calculator
PC with B2 Spice circuit simulation software (B2SPICE V 5.0)
(download from <http://www.beigebag.com/demos.htm#v5>)
Lab Notebook

Attendance Policy:

Students are required to attend all lab sessions. A student who misses a lab will receive a grade of zero for the lab and any associated reports. No make up labs will be given. The lowest pre-lab report score and post-lab report score will be dropped in computing final averages.

Goals and Objectives:

The goal of this laboratory is to study electronics through experimentation. Upon completion of this course, students should be able to use standard laboratory equipment to analyze the behavior of basic electronic devices and to design and construct simple circuits containing these devices.

Lab Teams:

Most of the labs will consume the full two-hour lab period and will be performed by two-person lab teams. Some labs will require only one hour and will be performed by each team member sequentially and independently during half of the corresponding lab period.

Homework Policy:

Every lab requires preparation prior to performing the experiments. Most of the labs require SPICE simulations, and some of the labs require calculations of design parameters before

beginning the experiments. Students are required to perform this preparatory work prior to coming to the lab. In order to fulfill the written communication component of the course, students are required to turn in pre-lab reports written in a proposal format prior to performing the laboratory work.

Grading:

Final grades will be calculated according to the following weights:

PRE-LAB REPORT	20 %
POST-LAB GRADE	30 %
LAB REPORTS	50 %

Pre-lab reports:

Each pre-lab report is due at the beginning of the lab period. The required report format is found in the lab manual. Each student is required to keep a copy of all pre-labs submitted. The pre-lab report should be used as the basis of the lab report, which will be written sometime after completion of the lab. In certain cases, an instructor may require the student to make corrections to simulations in the pre-lab reports, which will be due the following lab meeting. A 10-point deduction will be taken on every resubmission. Note: If you have not done your pre-lab when due, a grade of 50% will be assigned for the post lab grade as well!!!

Post-lab grades:

Upon completion of each laboratory, the instructor will verify each student's lab notebook for correctness and experimental completion. Also, an individual oral question or a short quiz will be given and a grade will be assigned.

Lab reports:

Students are required to turn in two lab reports for the semester. An opportunity to turn in a rough draft will be given with no grade assignment. An electronic copy is required for final submissions of all lab reports. Lab reports will be assigned randomly to each student. The first report will be assigned upon completion of Lab 3 and the second report will be assigned upon completion of Lab 7. The lab reports are due at the beginning of the lab period on a date determined by the instructor. The required report format is found in the manual.

Changes to the lab syllabus:

The instructor reserves the right to make changes to this syllabus during the semester. Students will be given adequate notice in the lab of any changes.

Academic Honesty: "As members of the Clemson community, we have inherited Thomas Green Clemson's vision of this institution as a 'high seminary of learning.' Fundamental to this vision is a mutual commitment to truthfulness, honor, and responsibility, without which we cannot earn the trust and respect of others. Furthermore, we recognize that academic dishonesty detracts from the value of a Clemson degree. Therefore, we shall not tolerate lying, cheating, or stealing in any form."

Lab Schedule Example

Lab Schedule (dates will be announced in each section)

- Lab 0: Laboratory Introduction (Attendance mandatory);
- Lab 1: Exp #1 - Semiconductor Diode Characteristics (2-person lab);
- Lab 2: Exp #2 - Power Supply Operation (2-person lab);
- Lab 3: Exp #4 - Diode Clippers and Clampers (2-person lab);
- Lab 4: Exp #3 - Power Supply Design (one hour; 1-person lab);
- Lab 5: Exp #5 - Bipolar Junction Transistor Characteristics (2-person lab);
- Lab 6: Exp #6 - BJT Common Emitter Circuit Bias (2-person lab);
- Lab 7: Exp #7 - BJT Common Emitter Circuit Voltage Gain (2-person lab);
- Lab 8: Exp #8 - BJT Common Emitter Amplifier Design I (2-person lab);
- Lab 9: Exp #9 - BJT Common Emitter Amplifier Design II (one hour; 1-person Lab)
- Lab 10: Exp #11 - Field Effect Transistors (2-person lab);
- Lab 11: Exp #12 – Basic Logic circuits (2-person lab);

**ELECTRONICS 1
ECE 3110**

SAFETY

Safety is always an important topic whenever laboratory work is being considered, and it is certainly true in the case of ECE 3110 labs. Safety is important.

The experiments in the laboratory use low voltages and low currents. However, the lab equipment is powered by the 110V, 60Hz, line voltage. Be careful with the line voltages. Do not touch exposed prongs on the equipment plugs when connecting the equipment to the lines.

Take care when using power supplies, which may be low voltage but can supply currents in the ampere range. Shorting such a supply can lead to a serious burn as high currents arc and can ignite flammable material. This is precisely why a car battery needs to be treated with respect. The hundreds of amps a battery can supply are sufficient to cause serious burns.

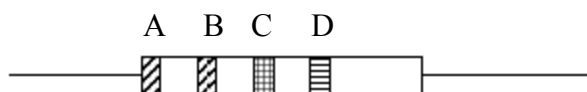
The equipment is heavy enough to be generally stable on the bench. Be sure to keep the equipment away from the edges of the benches to avoid having a piece of equipment fall off the bench. Besides endangering people who might be struck, falling equipment endangers everyone in vicinity by stressing the power cords, possibly causing a line short or live fault on the equipment, not to mention damage to the expensive lab equipment. In general electronic equipment does not survive harsh treatment.

The capacitors furnished in your lab kits are electrolytic capacitors with positive and negative terminals. Be sure to always connect the positively marked terminal to the most positive terminal in your circuit. An excess negative voltage applied to these capacitors can cause the device to overheat and explode.

The curve tracers can apply voltages as high as 200 V to a device. There is an interlock forcing the user to cover the device when applying these voltages. Do not attempt to override this safety feature when using the curve tracer.

Standard Component Values

In many of the experiments you will be asked to use standard resistor or capacitor values which are closest to those you calculated. The values were chosen to fit the tolerance and eliminate overlap. Most resistors are color coded. Figure SC-1 provides the color codes for standard resistor values to help you quickly select the values you need in your experiments. Since the capacitors used in the ECE 3110 experiments are for coupling or bypass purposes, values are not as critical and hence the color coding is not included.



A = first significant figure
 B = second significant figure
 C = decimal multiplier
 D = tolerance

Black	= 0	decimal multiplier = $10^0 = 1$	
Brown	= 1	decimal multiplier = $10^1 = 10$	
Red	= 2	decimal multiplier = $10^2 = 100$	
Orange	= 3	decimal multiplier = $10^3 = 1000$	
Yellow	= 4	decimal multiplier = $10^4 = 10000$	
Green	= 5	decimal multiplier = $10^5 = 100000$	
Blue	= 6	decimal multiplier = $10^6 = 1000000$	
Violet	= 7	decimal multiplier = $10^7 = 10000000$	
Gray	= 8	decimal multiplier = $10^8 = 100000000$	
White	= 9	*	
Gold	= *	decimal multiplier = $10^{-1} = 0.1$	tolerance = $\pm 5\%$
Silver	= *	decimal multiplier = $10^{-2} = 0.01$	tolerance = $\pm 10\%$
No color	= *	decimal multiplier = *	tolerance = $\pm 20\%$

Figure SC-1: Resistor color coding

The values of carbon resistors are guaranteed by the manufacturer to be within a certain tolerance, usually 5%, 10%, or 20% of the standard value. Perhaps most common is $\pm 10\%$. Table SC-1 shows the standard values for the $\pm 10\%$ tolerance.

<u>Standard value</u>	<u>Band 1</u>	<u>Band 2</u>	<u>Band 3</u>	<u>Band 4</u>
10	Brown	Black	Decade multiplier	Silver
12	Brown	Red	Decade multiplier	Silver
15	Brown	Green	Decade multiplier	Silver
18	Brown	Gray	Decade multiplier	Silver
22	Red	Red	Decade multiplier	Silver
27	Red	Violet	Decade multiplier	Silver
33	Orange	Orange	Decade multiplier	Silver
39	Orange	White	Decade multiplier	Silver
47	Yellow	Violet	Decade multiplier	Silver
56	Green	Blue	Decade multiplier	Silver
68	Blue	Gray	Decade multiplier	Silver
82	Gray	Red	Decade multiplier	Silver

Table SC-1: Standard values for carbon resistors with $\pm 10\%$ tolerance

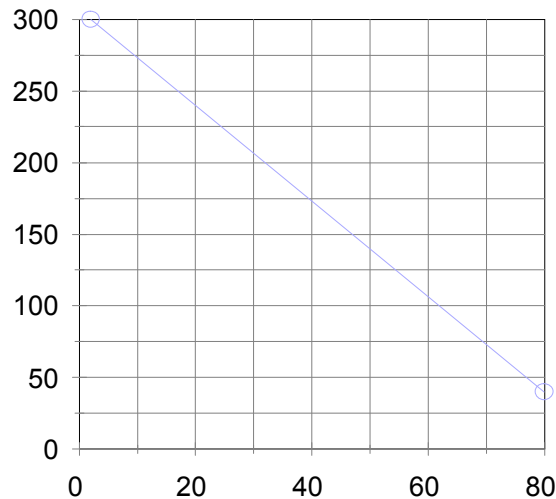
As an example of how to use this information, a 680Ω resistor with $\pm 10\%$ tolerance is guaranteed to have a value between 612Ω and 748Ω . The color code is

<u>Band 1</u>	<u>Band 2</u>	<u>Band 3</u>	<u>Band 4</u>		<u>Resistor Value</u>	<u>Tolerance</u>
Blue (6)	Gray (8)	Brown ($\times 10^1$)	Silver (10%)		$= 68 \times 10^1 = 680$	± 68

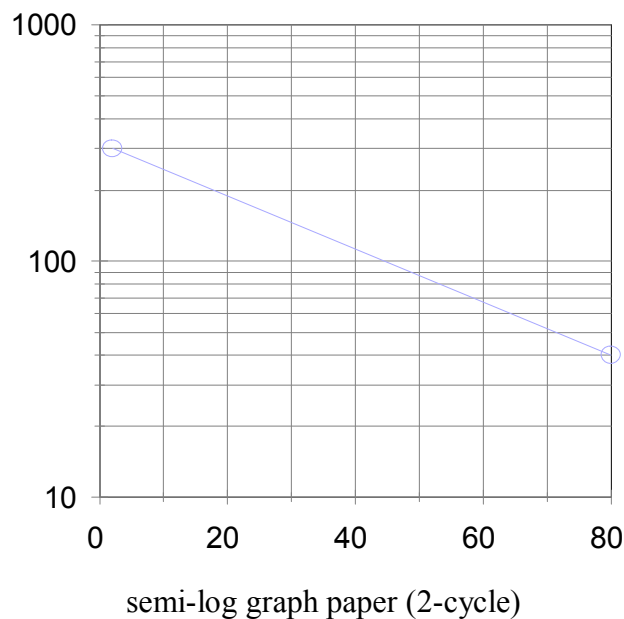
Graph Paper

Graphs of data may take many forms, whether plotted on traditional graph paper or using computer software. Three fundamental graph types to show the relationships between a dependent and an independent variable are *linear*, *semi-log*, and *log-log* plots.

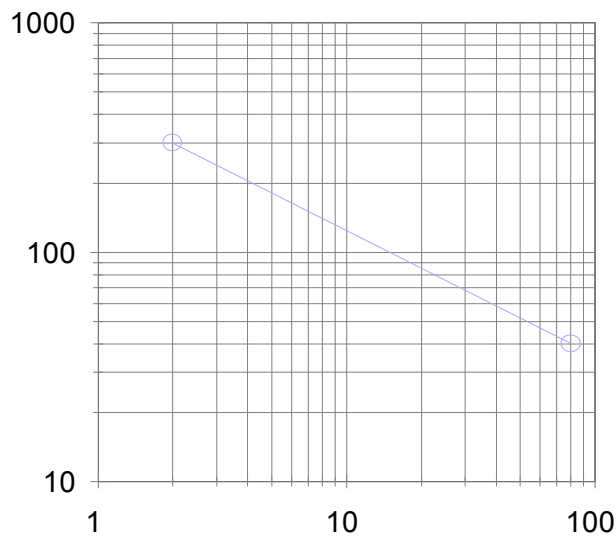
Linear graph paper has uniform spacing along the horizontal (“X”) axis and uniform spacing along the vertical (“Y”) axis, although the scales along the two axes do not need to be the same.



Semi-log graph paper is linear on one axis and logarithmic on the other axis:



Log-log graph paper has a logarithmic scale along both axes:



log-log graph (2-cycle by 2-cycle)

Choose the style that best reveals the relationships between the variables.

Note:

- A straight line on linear paper has the relationship $y=mx+b$.
 $m = (y_1 - y_2) / (x_1 - x_2)$
 $b = y$ when $x=0$
- A straight line on semi-log paper (if x-axis is linear) has the relationship $\log(y)=mx+b$
 $m = (\log(y_1) - \log(y_2)) / (x_1 - x_2)$
 $b = \log(y)$ when $x=0$
- A straight line on log-log paper has the relationship $\log(y)=m \cdot \log(x)+b$
 $m = (\log(y_1) - \log(y_2)) / (\log(x_1) - \log(x_2))$
 $b = \log(y)$ when $\log(x)=0$

LABORATORY REPORTS

Engineers are most effective if they can clearly communicate their ideas and developments to others, both other engineers and their managers. For this reason, writing and documenting are essential aspects of an engineer's job. Engineers spend over 60 percent of their time documenting their work and communicating the results to others. Many engineering students do not realize the importance of this documentation and communication process and have difficulties in their first job documenting their work. Engineers in the workplace are evaluated on their communication skills, which include both the quality and sometimes the quantity of their publications and technical reports.

In this class you are required to prepare pre-lab reports and to submit two formal lab reports for the semester. Pre-lab reports assist you in preparing for the labs by forcing you to organize your thoughts and understand the task ahead. The formal lab reports communicate the results of your work.

The lab report is as important as the work done in the lab, because unless you can communicate the results of your work, the work has little usefulness. Furthermore, the lab report reinforces the material that was learned in the lab and helps you develop effective technical communication skills. Development of both oral and written technical communication skills is one of the most important things you can learn as an undergraduate student.

Pre-Lab Reports

Almost all of the labs that are performed in ECE 3110 require a pre-lab report. The pre-lab report is an important part of the lab. The pre-lab report helps you get ready for the lab and gives you some experience in writing proposals. It assures that you have, at least, given some consideration to the type of experiment that you will be performing. The pre-lab report is in the form of a proposal, a statement you would give to your supervisor, outlining what you want to do, why you want to do it, and some estimates of what results you expect to obtain.

These pre-lab reports are to be short proposals of what you intend to do in the lab. They should be limited to two typewritten pages of text and two pages of supporting documentation, usually SPICE simulations of your anticipated results. A sample pre-lab report is shown below. Feel free to modify this format if you think you could communicate more information using a different format or if you think a different format would help you prepare for the lab.

General Report Guidelines:

- No more than 2 pages of written text.
- No more than 2 pages of supporting documentation, schematics, tables, or results of SPICE simulations.

Title: The title will often be the same as the title given in the lab manual. The title is centered on the top half of the page and is written in bold type, all capitals, followed by your name, lab section, and date.

EXAMPLE:

DIODE CIRCUITS

John O. Student, ECE 3110(4), January 32, 1997

Proposal: This part should be a short and to-the-point proposal of what you expect to do and what you expect to accomplish during the lab. You can get a lot of this information from the introduction part of the experiment in your lab manual. You generally don't need to spend too much space defining all of the terms you use in the pre-lab report, since you will be referring to your lab manual.

EXAMPLE:

Proposal: It is proposed that the current-voltage (I-V) characteristics of an electronic widget be measured and compared with the theoretically predicted I-V characteristics. Both the input and output characteristics will be measured. The input characteristic is expected to have an I-V characteristic of the form

$$I_{in} = V_{in} (R_1 + R_2)^2$$

The output characteristic is expected to be of the form

$$I_{\text{out}} = K I_{\text{in}}$$

where K is a constant proportional to the emitter capacitance. All of the above terms are defined in the Lab Manual under Lab #4, Diode Circuits. The input and output characteristics will be measured using values of $100\ \Omega \leq R_1 \leq 200\ \Omega$, $1\ \Omega \leq R_2 \leq 1\ \text{M}\Omega$, and $10\ \text{pF} \leq C_E \leq 100\ \mu\text{F}$. SPICE simulations will be performed on the widget covering the ranges of resistance and capacitance described above and the measured characteristics will be compared with the SPICE simulations.

Experimental: This part of the pre-lab should describe how you expect to make your measurements, including what equipment will be needed. As in the proposal part, you should be able to get most of this information from your lab manual.

EXAMPLE:

Experimental: The widget input and output characteristics will be measured using a curve tracer. In this experiment, R_1 will be initially set at $100\ \Omega$ and R_2 and C_E will be independently varied in decade steps. R_1 will then be incremented in $10\ \Omega$ steps to $200\ \Omega$, while R_2 and C_E are kept at their optimum values.

Anticipated Results: This part of the pre-lab report will contain any anticipated results, including any results of SPICE simulations.

EXAMPLE:

Anticipated Results: The input current will be plotted as a function of input voltage on log-log paper. The slope of the I-V characteristic should be a straight line with slope = 2, for all values of R_1 and R_2 . The constant K will be determined from the output characteristics. The anticipated value of K is $0.01\ C_E$.

On the next two pages the SPICE simulations of the widget characteristics are shown. Provide SPICE output, circuit diagram, and netlist.

Formal Lab Reports

The lab report is as important as work done in the lab. The lab report reinforces the material that was learned in the lab and helps you develop effective technical communication skills. Development of both oral and written technical communication skills is one of the most important things you can learn as an undergraduate student.

Engineers are most effective if they can clearly communicate their ideas and developments to others, both other engineers and their managers. For this reason, writing and documenting are essential aspects of an engineer's job. Engineers spend over 60 percent of their time documenting their work and communicating the results to others. Many engineering students do not realize the importance of this documentation and communication process and have difficulties in their first job documenting their work. Engineers in the workplace are evaluated on their communication skills, which include both the quality and sometimes the quantity of their publications and technical reports.

The lab reports you submit in ECE 3110 should conform to the guidelines given below. Each report should be a self-contained document and should be no longer than necessary to present the required information. Each report is to be typed using a word processor. Proper spelling and grammar are to be used throughout. Figures, drawings, charts, and tables should be added where they are needed and should contain understandable labels, including units for the axes. When plotting B vs. A, B is the dependent variable and is plotted on the y-axis; A is the independent variable and is plotted on the x-axis. The figure should appear in the text as soon after mention of the figure as possible, but not in the middle of a paragraph. The figure should refer to the main text and should not stand alone. Except for raw data, all figures should be computer-drawn using either a plotting program such as Cricket Graph or a spreadsheet program such as Excel. A stand-alone figure may be more confusing than no figure at all. All pages should be consecutively numbered. Sign your report on the cover page. This signature shows that you take responsibility for what is contained in the report. Reports are due at the start of the class after the lab has been performed. The details of the report are given below.

General Report Guidelines:

A technical report is expected to contain the following items or subsections:

- Title page,
- Table of contents,
- Abstract,
- Introduction,
- Theoretical discussion or background,
- Experimental procedure and methodology and experimental results,
- Discussion of results,
- Conclusion or summary,
- Acknowledgements,
- Appendices,
- References.

You may find some reports have one or more of these sections removed.

Title page: This page is the first page of the report and should act as the cover page. The title will often be the same as the title given in the lab manual. The title is centered on the top half of the page and is written in bold type, all capitals. Centered on the bottom half of the page is your name, the name of your lab partner(s), the course name, the course title, the instructor or the name of the institution for whom the report is being prepared.

EXAMPLE: The typing for this page should be larger than shown below.

DIODE CIRCUITS

John Q. Student

Lab Partner, A. Wiley Pardner

January 32, 1997

ECE 3110

Electrical Engineering Laboratory III

Prepared for: Any-Old Grad. Student

Abstract: An abstract is a short and to-the-point statement of what was done, how it was done, the results, and the conclusions. The abstract gives the reader enough information to determine if the reader wants to read the full report.

EXAMPLE:

ABSTRACT

The current-voltage characteristics of an electronic widget were measured. The widget characteristics were compared with theoretical predictions. It was found that the internal resistance increased from 1 ohm to 10,000 ohm when the voltage was raised from 1 V to 1.1 V. The external resistance dropped from 1 ohm to 0.1 ohm over the same voltage range. The control factor, m , for the widget varied between 1.1 and 1.3 over the full input voltage range. The device tested in the lab was well characterized by the ideal widget equation using a form factor of 2.2.

Introduction: The introduction should explain the background of the work. It should put the experimental work into perspective and should lead the reader into the subject matter. It should have at least one sentence explaining why the work was undertaken. It should end with one or two sentences describing the general experimental approach and results.

EXAMPLE:

INTRODUCTION

A fidget can be combined with resistors and capacitors to change the wave shape of an ac signal. The position of the fidget in the circuit, with respect to the resistors and capacitors, causes the output wave shape to change. The resistor and capacitor act as a smoothing filter. Four basic fidget/resistor/capacitor circuits were studied. The resistor and capacitor values were changed over two orders of magnitude to make the RC time constant much smaller and much larger than the period of the input signal. The output wave shapes were studied as a function of resistor and capacitor values. It was found that the piecewise linear model of the fidget could adequately explain the output wave shapes.

Theoretical discussion or background: This section is used to develop the theoretical aspects of the experiment. Any relevant theory from class or from the lab manual can be used.

EXAMPLE:

BACKGROUND

When an RC network is placed across the output of a sinusoidal signal, the wave shape is filtered. The output wave shape is the input wave shape multiplied by a constant related to the RC time constant of the network and is given by

$$V_0 = V_{in} e^{-(t/RC)} \quad (1)$$

where V_{in} = the input voltage, etc. (Define all terms)

(Continue with enough discussion to explain the general theoretical ideas, but don't put in so much material as to make this section overly long)

Experimental Procedure and Methodology and Experimental Results: This section is for explanation of apparatus, circuit configuration, and procedures used in this experiment. The title will probably change from experiment-to-experiment. You can put drawings and circuit diagrams into this section.

EXAMPLE:

EXPERIMENTAL PROCEDURE AND RESULTS

The circuit shown in Figure 1 was constructed. The quiescent emitter-to-ground voltage and the collector-to-ground voltage were measured as R_3 was systematically varied from 10 k Ω to 100 k Ω in 10 k Ω steps. The output voltage was simultaneously measured using an oscilloscope. At no time during this measurement did the output voltage wave shape become distorted. The measured values of VEO and VCO have been shown in Figure 2, along with the derived value of VCE. The collector voltage, VCE, was independent of variations in R_3 except when R_3 was

varied from $43,445\Omega$ to $43,447\Omega$. For these variations in R_3 the collector voltage dropped by almost two orders of magnitude.

Continue to describe each of your other measurements.

Include all data here and any preliminary discussion of the results.

Discussion of experimental results: This section is probably the most important part of the report. The data should be presented in a reduced form. Usually figures are the easiest form to present data, but tables or lists can be used, where appropriate. Do not assume that the reader knows what you are talking about. Be descriptive. Include sample calculations along with your calculated data. Do not present data or graphs without explanation. Be sure to compare your measured results with those that you expected. Here is where theory meets reality. If the results agree with the theory show how. If the results do not agree with the predicted values, try to explain why you think they are different or where any errors in data-taking could have occurred. Explain any anomalous data.

EXAMPLE:

DISCUSSION OF EXPERIMENTAL RESULTS

The theory predicted that the output current could be written as

$$I_0 = I_{in} e^{(V_{in}/V_T)}$$

where, define terms here. By plotting the measured values of I_0 vs V_{in} , on a semi-log graph, as shown in Figure 6, it is seen that the output current is proportional to the exponential of the input voltage. Noting that the output current increases by a factor of 10 every time the term (V_{in}/V_T) increases by 2.3, it was possible to determine the value of V_T . V_T was found to be constant and equal to 4.4 V. This value of V_T is approximately 2x larger than expected. The difference was probably due to the fact that V_T is a function of voltage and we only measured V_T over a small range of voltages.

The value of V_2 was determined to be

$$V_2 = V_1^m$$

By plotting V_2 vs. V_1 on a log-log graph, the slope, m , was determined to be 2. This value of m agrees well with the expected value of 2.0.

Continue to compare all of your results with your expected results, including any comparisons with SPICE simulations. For every SPICE simulation, include a net list or circuit diagram and include the plots of the SPICE outputs, not the tables of node values.

Conclusion-Summary: This section should provide closure to your report. Conclusions should be based on the information described in the report. The conclusions may not exactly match the lab's objectives, but make sure your conclusions are supported by your data. Any advantages

and/or limitations of the information presented here should be included. You may want to include any personal observation that may not be reflected in the data, e.g., the problems encountered while using a particular instrument or when performing a particular step in the experiment.

EXAMPLE:

CONCLUSIONS

The current-voltage relationships governing operation of a super-duper-widget were measured over the range of input voltages from 0 V to 0.001 V. The output current was found to be proportional to the input voltage while the input current was exponentially proportional to the output voltage. These input-output relationships were those predicted by the text. If the input voltage had been varied over a somewhat larger range, these relationships may not have been found.

Acknowledgements: This section is used to acknowledge any technical or financial aid that was received in support of this work. You should state who your lab partner was in this section.

Appendices: This section should contain any miscellaneous calculations, any mathematical derivations or proofs, and any computer programs or SPICE simulations.

References:

This section contains all bibliographical work cited in the report. Usually you will reference your textbook and lab manual here. The format of these is

1. Author, *Title of reference*, page numbers, who published, where published, when published.
 1. B. Grob, *Basic Electronics*, pg. 43 to 435, McGraw Hill, New York, NY, 1943.

LABORATORY EXPERIMENTS

ELECTRONICS I ECE 3110

Introduction – Laboratory Demonstration

PURPOSE:

This laboratory session is intended to introduce you to the lab, B2 Spice, and some of the instruments that will be used during the semester. In this session you will learn to use a curve tracer to measure the characteristic curves of transistors in your lab kit. You will also learn to simulate those characteristic curves using B2 Spice software. You will begin to learn the terminology that will be used throughout the semester in this lab and in Electronics I.

LAB KIT:

For this lab and subsequent labs, you will need an IEEE ECE lab kit. If you do not have an IEEE ECE Lab kit, you should make arrangements to purchase one through the IEEE Student Branch as soon as possible. Some additional devices may be supplied by the lab instructor as needed. **Always bring your lab kit to class with you.**

B2 Spice:

You will use B2 Spice to simulate many of your experiments. All lab PCs are loaded with B2 Spice v5. You can obtain a personal copy from Beige Bag software. A light version is available free and a full version costs less than a textbook with special student discounts provided for bulk purchases.

***LTspice**

You are required to have a copy of ~~B2 Spice (v4 or v5)~~ on your laptops and are **strongly advised to purchase the full version 5** (current student price is \$60 through Beige Bag); this will be of benefit throughout your degree program. Version 5 is required for you to download and use the full database of parts. Under extenuating circumstances, e.g., laptop in for repair, you may contact your TA to gain access to ECE 3110 lab computers. Lab computer usage is restricted to emergency situations.

The printer is connected to a single desktop computer in the lab. It is, thus, necessary to bring a portable drive if you want to print out any results, or need help debugging circuits.

The web address is for B2 Spice software is: <http://www.beigebag.com/>

Student pricing for B2Spice software is at: http://www.beigebag.com/pricing_other.htm

LTspice web address: <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>

TERMINOLOGY:

Below are a few common terms used in this chapter. Others will be defined as we go along.

BIP	Bipolar
DUT	Device under Test
FET	Field Effect Transistor
I-V	Current-Voltage
I_b or I_B	Base current of the transistor

*** For this lab we will be using LTspice for circuit simulations. Every mention of B2Spice should be replaced with LTspice. Kindly note this trivial change throughout the lab manual.**

EQUIPMENT:

For this laboratory session you will need the following:

- a. Hameg HM 6042 curve tracer or Tektronix Type 576 curve tracer
- b. IEEE ECE lab kit
- c. PC with B2 Spice loaded

The curve tracer automatically plots the current-voltage characteristics of two- and three-terminal devices. The curve tracer automatically applies the independent voltages or currents to the appropriate terminals and automatically measures the dependent currents and voltages that result.

Two types of curve tracers are provided for work stations in this laboratory: the Hameg HM 6042 and the Tektronix type 576. **Appendix B** includes front-panel photographs for the Tektronix curve tracer and basic setup instructions for it. **Appendix C** includes specific procedures for conducting some of the basic tests using the Tektronix type 576 curve tracer. **Appendix D** shows a copy of the Hameg curve tracer equipment manual (by permission of Hameg).

You should have read the equipment manual given to you before coming to lab. Failure to do so might result in your not being able to finish this lab session within the allotted time. **It is your responsibility to read the equipment manual and you should bring it to each lab session.** For this lab session your lab instructor will help you get started.

Outline for measuring with the curve tracer:

(In the following paragraphs, the numbers in parentheses are reference numbers on the Hameg HM6042 curve tracer manual's diagram of the unit's front panel.)

- Set up the curve tracer and select the parameters you want to measure.
- Choose the type of device under test (DUT) and connect to the correct terminals.
- Set the current, voltage, and power maximum ranges. Use V_{\max} (21), I_{\max} (19) and P_{\max} (23) to set the X-axis, Y-axis, and power dissipation limits, respectively.
- In the BIP (bipolar) mode the rotary knob (8) enables the user to set the base current, I_B . In the FET mode the knob adjusts the gate voltage V_G .
- It is generally a good idea to start new device measurements with the minimum P_{\max} setting.

Device parameters are measured and displayed on the instrument. The exact point on the curve is indicated by the cursor position. h-parameters and various DC parameters are available and are chosen by pressing the buttons marked $\blacktriangleleft\blacktriangleright$ (5).

PRE-LAB:

Review the portion of the curve tracer manual that accompanies your lab manual. Download a free student version of B2 Spice or purchase your own full copy (recommended) and install it on your PC.

TIPS FOR BUILDING CIRCUITS:

- Before starting to create the circuit, collect and sort out all the components (capacitors, resistors, diodes, probes, etc.) required in the lab. Doing so reduces the chance of using wrong components, especially resistors, which would produce wrong results.
- When building the circuit, try to mimic the layout of the lab manual's circuit diagram on the breadboard. It will help in finding wrong connections.
- Minimize the number of connecting wires, to reduce the chance of loose connections.
- Use standard resistors instead of the resistance box, when possible.

EXPERIMENTAL:

PART I: CURVE TRACER

1) Transistor characteristics (see sec 4.1, p15, Hameg Manual)

- Place a silicon NPN transistor 2N3904 or 2N2222 in the DUT socket. The TO-92 transistor package has the connections to E B C (Emitter, Base, Collector) from left to right with the flat portion of the case down, and looking from the top of the case.



- Set the conditions** as follows: (Good starting conditions for most devices)

Type	[NPN/BIP]
V_{MAX}	[10 volts]
I_{MAX}	[20 mA]
P_{MAX}	[0.4 W]

- Display the Curves:** Press the DUT switch (14) to display the curves. Notice that the curve tracer allows you to display five I-V characteristics at once. In this case, **each I-V characteristic corresponds to one particular setting of I_B** . Use the buttons marked \longleftrightarrow to select I_B . Pressing the CURSOR key moves the cursor from one curve to another. Move the cursor to the maximum curve.

d) Measure I-V Characteristics:

- Using the FUNCTION key, select MAX. Rotate the knob to display the family of curves and set the maximum curve to $I_B = 45\mu A$.
- Using the FUNCTION key, select MIN and set the minimum curve to $I_B = 5\mu A$.
- By adjusting the MIN/MAX settings, you have set the I_B -step to $10\mu A$.
- Using the FUNCTION key, select $\blacktriangleleft\blacktriangleright$. Rotating the control knob with the $\blacktriangleleft\blacktriangleright$ function selected moves the cursor along a specific curve.
- Measure some points (I_B , I_C , and V_C) on the transistor characteristics by looking at the display LCD and using the buttons marked \longleftrightarrow .

The importance of using the curve tracer to measure transistor characteristics will become evident when transistor circuits are discussed in ECE 320. You will need to determine experimentally the I-V characteristics of your bipolar transistors at the start of several of the experiments you will perform. Your transistor characteristic should resemble the characteristic shown in Figure 0.1. **Sketch this transistor characteristic using the values you recorded.**

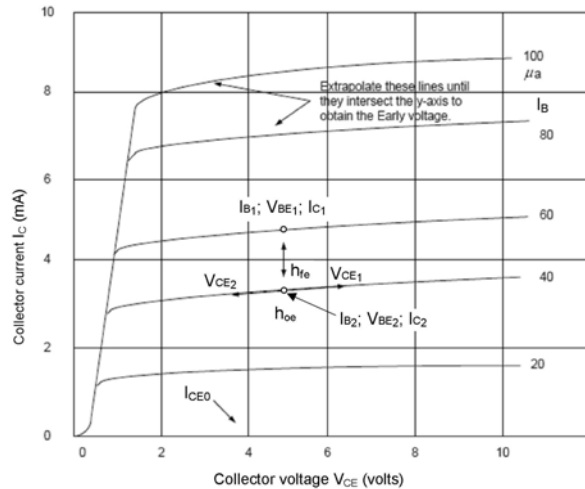


Figure 0.1: Typical I-V characteristic curves

e) **Measure Characteristics in Inverted Mode:**

- Press the DUT button to turn off power to your transistor.
- Reverse the transistor in the socket. That is, place the collector in the emitter contact and place the emitter in the collector contact. This transistor configuration is called the **inverse active mode** or **inverted mode**.
- Press the DUT button to turn on power to your transistor.
- Measure the transistor characteristics using the same settings used above. The collector current should be quite low.
- Reduce I_{MAX} to 2mA and increase maximum and minimum I_B settings by a factor of 10 using the MAX and MIN functions.
- Sketch this transistor characteristic, recording several values to note on your sketch.
- Press DUT to turn off power to the transistor.

PART 2: CIRCUIT SIMULATION

B2 Spice is a fully featured mixed-mode simulator with powerful and innovative features that make it the simulator of choice for power SPICE users. Features include a parts-chooser window, workspace editor, importing and exporting to Eagle PCB as well as other PCB programs, and animated schematics. All lab PCs are loaded with B2 Spice v5. The demo is based on B2 Spice v5 but is similar to v4.

The **schematic and output** for the circuit to obtain the transistor characteristics are given below, in Figure 0.2 and Figure 0.3.

1. Open B2 Spice v5. The Analog Schematic window opens.
2. Select ► **Common Parts** ► **Choose Part**.
3. In the **Name** box enter ► 2N3904
4. Hit the **Find** button.
5. Select **2N3904** and then click select device; this will attach a schematic symbol to the cursor. You can place it on the Schematic page by clicking on the page.
6. Continue building your circuit by repeating Step 2 through Step 5. Often the part you want may appear directly in the Common Parts list; just select it and place it on the schematic window (for example, ground, current source, voltage source)
7. Connect all parts with a wire by first selecting the “\” button.
8. Add meters to observe voltages and currents (voltmeters and ammeters).
9. **SAVE** your circuit.
10. Click on the **Tests** Tab.
11. Select **DC Sweep** by clicking the Setup button.
12. Select the **Sweep** Tab.
13. Select V_1 from the **Source** menu. Select “0” for the **Start Value**, “5” for the **End Value** and “100m” for the **Step Value**.
14. Click “**Set up Sweeps**”. Select “ I_1 ” from the list window. Enable “**Sweep**”. Select “**Linear**” from **Interval Type** menu. Select “ 5μ ” for the **Start Value**, “ 45μ ” for the **End Value** and “ 10μ ” for the **Step Value**. Click **Accept Changes**. Click **OK**. Enable “graph” for the 2D output.
15. Click **Run**.

NOTE: It is important that you use the actual device you want to simulate and not a GENERIC component.

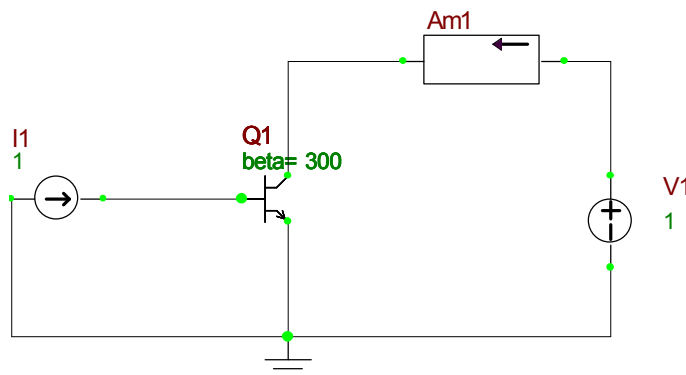


Figure 0.2: B2 Spice Circuit

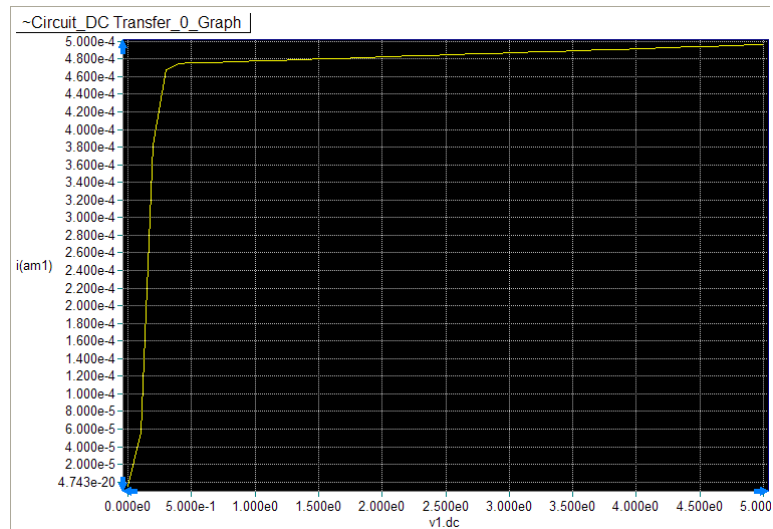


Figure 0.3: Output Information.

LAB REPORT:

There is no formal lab report required for this session. You should assure yourself that you know how to use:

- i. The curve tracer to obtain the I-V characteristics of a BIP device.
- ii. B2 Spice to simulate a circuit

ECE 3110-EXPERIMENT 0 CHECK LIST

- 1) Curve Tracer
Obtain NPN transistor characteristics.
- 2) SPICE
Obtain simulation results
- 3) Conclusion
Compare SPICE and curve tracer outputs.

**ELECTRONICS I
ECE 3110**

Experiment #1 – Diode Characteristics

PURPOSE:

The purpose of this experiment is to acquaint the student with the operation of semiconductor diodes. You will use a curve tracer to obtain the current-voltage (I-V) characteristics of a silicon diode. From these characteristics, you will determine several diode parameters including the **dynamic resistance**, $r_f = r_d$; the **diode forward resistance**, $R_F = R_D$; the **cut-in voltage**, V_γ ; the **forward diode ideality factor**, n ; and the **breakdown voltage**, V_{BR} . 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. You will also compare the dc operation of a diode in a circuit with both the calculated and simulated operation.

PRE-LAB:

Review the INTRODUCTION section below. Simulate the diode characteristic using B2 Spice for comparison with experimentally measured results. Determine r_d , V_γ , and n for the 1N4004 diode in your diode characteristic plot. Simulate the circuit shown in Figure 1.5 for the resistor (R) values shown using a DC sweep test (sweep V_{in} and R simultaneously; see Part 2 of previous lab for help).

EQUIPMENT:

For this experiment, you will need:

- a) Silicon diodes – 1N4004 or equivalent, and 1N4744. These diodes are all silicon diodes with different breakdown voltages and different power handling capabilities. The 1N4744 is a Zener Diode and has the lowest breakdown voltage. It should be used when attempting to obtain the reverse breakdown voltage. The 1N4004 diode will be used extensively in circuits in other experiments.
- b) A curve tracer
- c) NI ELVIS II workstation
- d) IEEE ECE lab kit
- e) B2 Spice simulation program

INTRODUCTION:

Diode Structure:

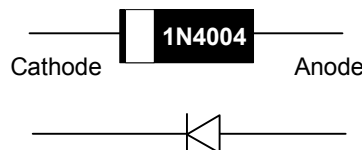


Figure 1.1

Figure 1.1 shows the physical and schematic circuit symbol of the diode. The band on the diode and the bar on the left of the circuit symbol represent the cathode (n-type material) and

must be noted. The p-type material (the anode) in the diode is located to the right. The circuit symbol of the diode is an arrow showing forward bias, when the p-side is positive with respect to the n-side, and the direction of the arrow represents the direction of large current flow.

Ideal Diode Equation:

The relationship between the diode current and voltage is given by the diode equation

$$I_D = I_S \left(e^{V_D/nV_T} - 1 \right) \quad (1.1)$$

The terms in Equation (1.1) are defined as follows:

I_D = the diode current (amperes).

V_D = the voltage across the diode (volts).

I_S = the reverse saturation current or the reverse leakage current (amperes).

I_S is a function of the diode material, the doping densities on the p-side and n-side of the diode, the geometry of the diode, the applied voltage, and temperature. I_S is usually of the order of $1\mu\text{A}$ to 1mA for a germanium diode at room temperature and of the order of $1\text{pA} = 10^{-12}\text{A}$ for a silicon diode at room temperature. I_S increases as the temperature rises.

$V_T = kT/q$ = the thermal equivalent voltage = 0.0258 V at room temperature

where

$q = 1.6 \times 10^{-19}\text{ Coulombs}$ = the electric charge,

$k = 1.38 \times 10^{-23}\text{ J/K}$ = Boltzmann's constant,

and

T = absolute temperature (Kelvin) [room temperature = 300 K].

n = the ideality factor or the emission coefficient.

The Ideality Factor (n):

The ideality factor, n , depends on the type of semiconductor material used in the diode, the manufacturing process, the forward voltage, and the temperature. Its value generally varies between 1 and 2. For voltages less than about 0.5 V , $n \sim 2$; for higher voltages, $n \sim 1$. (Based on experimental measurements, at higher voltages, typically $1.15 \leq n \leq 1.2$.)

The ideality factor, n , can readily be found by plotting the diode forward current on a logarithmic axis vs. the diode voltage on a linear axis.

Equation (1.1) indicates that an increase in current I_D by a factor of 10 represents an increase in $\exp(V_D/nV_T)$ by a factor of 10, as long as $\exp(V_D/nV_T) \gg 1$. If ΔV_D is the change in voltage required to produce a factor-of-10 change in the current, then

$$\Delta V_D / nV_T = \ln(10) = 2.30$$

or,

$$\Delta V_D = 2.30 \cdot n \cdot V_T = 2.30 \cdot n \cdot 0.0258\text{ volts} = n \cdot 0.0593\text{V} = n \cdot 59.3\text{ mV}$$

And so,

$$n = \Delta V_D / 59.3\text{mV} \approx \Delta V_D / 60\text{mV} \quad (1.2)$$

To find n , it is only necessary to find the amount of voltage needed to increase the diode current by a factor of 10 and use Equation (1.2).

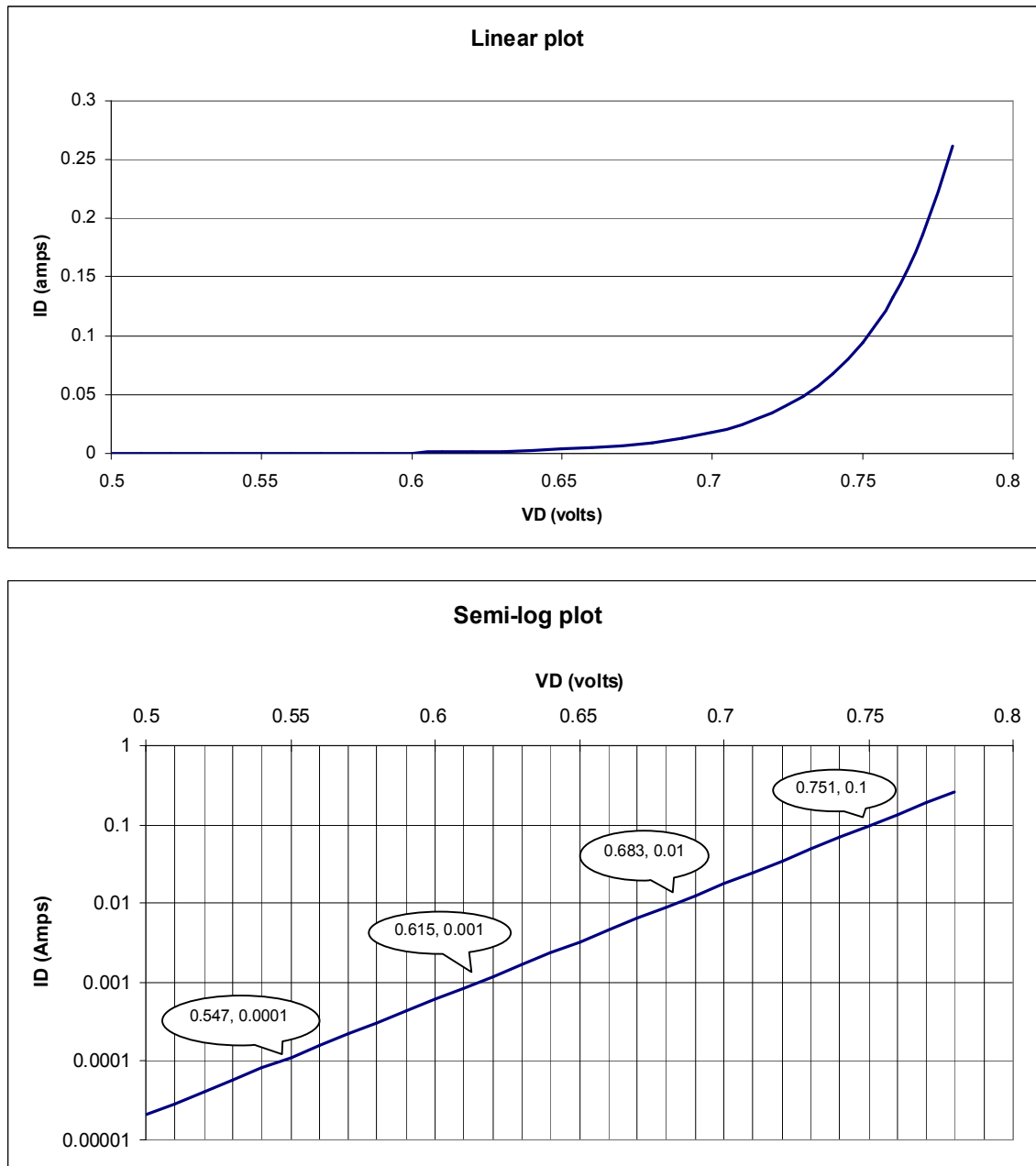


Figure 1.2: Graphs of the same forward diode current I_D vs diode voltage V_D as (a) Linear plot and (b) Semi-log plot

Figure 1.2 shows example graphs of the forward diode current I_D vs diode voltage V_D as (a) Linear plot and (b) Semi-log plot. To calculate the ideality factor n , create the semi-log plot for

the diode's data. Draw a straight line through adjacent points, then read off coordinates where the current I_D increases by powers of 10 (e.g., 0.0001, 0.001, 0.01, 0.1, ...), as illustrated in Figure 1.2. Calculate ΔV_D , the amount of voltage needed to increase the diode current by a factor of 10, and then divide by 59.3mV (or 60 mV) to calculate n :

$$n = \frac{0.751 - 0.683}{0.0593} = 1.147.$$

(*Advanced note:* The ideality factor is a measure of how close the diode matches “ideal” behavior. If the ideality factor is different from 1, it indicates either that there are unusual recombination mechanisms taking place within the diode or that the recombination is changing in magnitude. Thus, the ideality factor is a powerful tool for examining the recombination in a device.)

Cut-in Voltage V_γ :

A sketch of a diode characteristic, as it would be measured on a curve tracer, is shown in Figure 1.3. The curve tracer only measures the forward I-V or the reverse I-V characteristic in any one sweep. The characteristics shown in Figure 1.4 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.

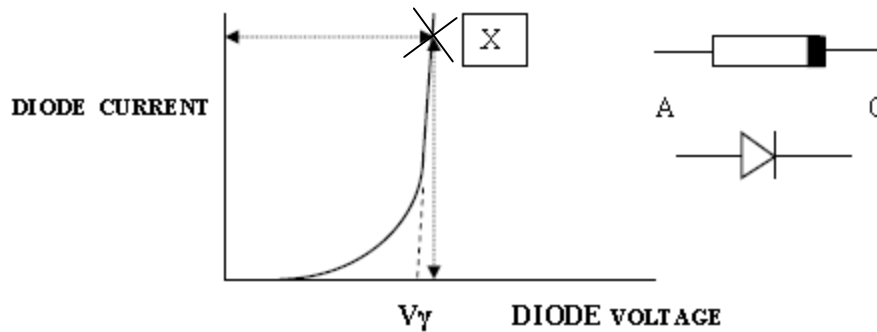


Figure 1.3: Diode forward I-V characteristic showing the definition of V_γ

The complete diode characteristic is shown in Figure 1.4, piecing together the forward-biased data and the reverse-biased data. Note that the scales of +V and -V may differ by a factor of 100, and while +I may be mA or A, -I is likely to be μ A or nA.

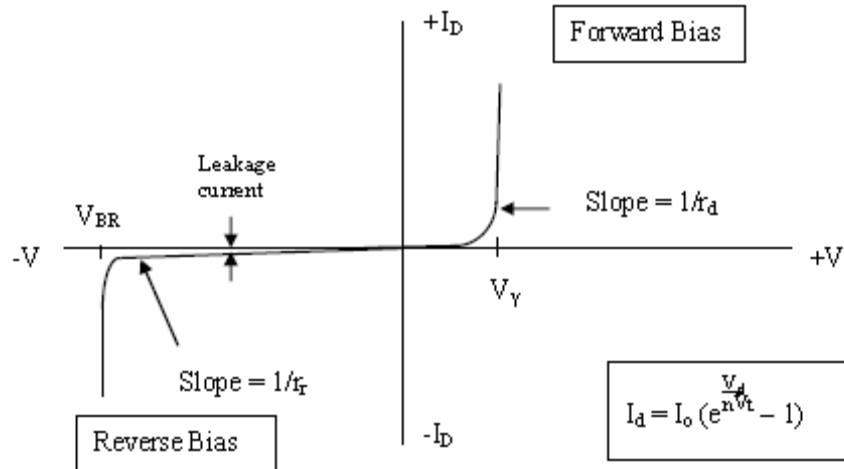


Figure 1.4: Forward and reverse diode I-V characteristics

Diode Current and Diode Saturation Current:

If the diode is operated in the forward-bias region at room temperature ($27^\circ\text{C} = 300\text{ K}$), the exponential first term in the brackets in Equation (1.1) dominates and the diode current equation is given approximately by

$$I_D = I_s e^{V_D/nV_T} \quad (1.3)$$

The current for forward bias is an exponential function of the applied voltage, V_D .

If the diode is reverse-biased, only the small reverse current (the reverse saturation current or the reverse leakage current), $-I_s$, flows. This current flows as long as the applied reverse voltage does not exceed the diode breakdown voltage, V_{BR} . If the reverse voltage exceeds V_{BR} , a large amount of current flows and the diode may be destroyed if there is not enough series resistance to limit the diode current. In silicon diodes, I_s may be very small and V_{BR} may be very large. Both of these values may be immeasurable on the curve tracers for diodes like the 1N4004.

Diode Resistance

Three diode resistances are commonly calculated:

- DC or Static forward resistance, R_F or R_D
- AC or Dynamic forward resistance, r_f or r_d
- Reverse resistance, r_r

Another diode resistance, R_S , is also mentioned. R_S refers to the sum of the diode's contact resistance, lead resistance, and internal diode resistance. It appears in B2 Spice simulations.

DC or Static forward resistance, R_F or R_D , is the total voltage drop across the diode divided by the current flowing through the diode, just as one would calculate using Ohm's Law. It includes contact resistance, lead resistance, material resistance, and the resistance of the p and n regions of the diode.

$$R_F = \frac{V_D}{I_D}.$$

AC or Dynamic forward resistance, r_f or r_d : In practice we don't often use the static forward resistance; more important is the dynamic or AC resistance, which is the opposition offered by the diode to *changing* current. It is calculated by the ratio [change in voltage across the diode] / [the resulting change in current through diode] at the operating voltage, V_D . That is, r_d is the *reciprocal of the slope* of diode current versus voltage at the operating point.

$$r_f \equiv r_d = \frac{1}{\Delta I_D / \Delta V_D} = \frac{\Delta V_D}{\Delta I_D} = \frac{\text{Change in voltage}}{\text{Resulting change in current}}$$

Applying the diode equation and differentiating, we find the dynamic forward resistance is given by

$$r_d = \frac{dV_D}{dI_D} = \frac{1}{dI_D/dV_D} = n \frac{V_T}{I_D}. \quad (1.4)$$

Owing to the nonlinear shape of the forward characteristic, the value of AC resistance of a diode is in the range of 1 to 25 ohms. Usually it is smaller than DC resistance of the diode.

Reverse Resistance, r_r : When a diode is reverse biased, besides forward resistance, it also possesses another resistance known as reverse resistance. It can be either DC or AC depending upon whether the reverse bias is direct or alternating voltage. Ideally, the reverse resistance of the diode is infinite. However, in actual practice, the reverse resistance is never infinite, due to the existence of leakage current in a reverse-biased diode.

The **reverse resistance, r_r** , is given by the reciprocal of the slope of the reverse characteristic, prior to breakdown (see Figure 1.4).

Junction Capacitance of Diode (C_j):

The space-charge region (or depletion region) of the diode is a region that contains very few holes or electrons and lies between the n-type semiconductor and the p-type semiconductor inside of the diode. The space-charge region of the diode approximates a parallel-plate capacitor, with the value of the capacitance determined by the applied voltage. Using this approximation, the capacitance of the space-charge region is approximately given by

$$C_j = \frac{eA}{w} = C_{jo} \left(1 - \frac{V_D}{V_{bi}} \right) - \frac{1}{n} \quad (1.5)$$

where

e = permittivity of silicon (10^{-12} F/cm),

A = cross-sectional area of the diode (cm^2),

w = width of the depletion region (cm),

$n = 2$ for a step junction,

V_{bi} = built-in voltage, and

C_{jo} = the junction capacitance at $V_D = 0$ V.

The junction capacitance is inversely proportional to w . As the reverse-bias voltage increases, the space-charge region widens, approximately as the square root of the applied voltage, and, thus, the capacitance decreases. This variation in w causes the diode to behave as a voltage-controlled capacitor with a capacitance that varies inversely with the square root of the applied voltage. If the diode had a very large cross-sectional area, the capacitance of the space-charge region as a function of the reverse voltage could be measured on an impedance bridge. Since the diodes in your lab kits are generally relatively small, it is very difficult to measure the voltage variation of the diode capacitance. Usually only a large area diode, like the power diode in your lab kits, is large enough to be used to measure the diode capacitance using the impedance bridges in the lab. Due to the large forward diode current, it is usually possible to measure the diode capacitance only for voltages less than V_γ .

EXPERIMENT:

PART I: MEASUREMENT OF DIODE CHARACTERISTICS

A. Forward I-V Characteristic

Procedure

1. Use the curve tracer to obtain the forward characteristics of the silicon **1N4004** diode.
 - Connect diode cathode and anode terminals to terminals E and C, respectively, and set switch BIP/FET to FET).
 - Set the voltage axis to $V_{\max} = 2$ V.
 - Begin your measurements with $I_{\max} = 2$ mA.
 - Use $P_{\max} = 0.4$ W.
 - Press the DUT button to obtain the curve.
 - If the curve is flickering, use the function key to select ◀▶, and move the control knob counter-clockwise until the flickering stops.
 - Using the cursor key, take readings from the I-V characteristic for values of current close to those in the first column of Table 1.1, to enable you to plot accurately the I-V characteristic both on linear graph paper and on logarithmic graph paper. **Increase the I_{\max} by a factor of 10 between measurements once you cannot read a higher value on the given curve.** Tabulate your results below.

TABLE 1.1: Table of Current, voltage, forward resistance, dynamic resistance

I_D (A)	Your Value of I_D (A)	V_D (volts)	$R_F (\Omega) = V_D/I_D$	$r_d (\Omega) = nV_T/I_D$
30 μ A				
100 μ A				
200 μ A				
400 μ A				
1mA				
2mA				
6mA				
14mA				
30mA				
60mA				
100mA				
150mA				

2. Compute R_F and r_d using the values you recorded and record in the table above. See Introduction for the definitions of R_F and r_d .

B. Reverse I-V Characteristic

You should use the **1N4744** diode (Zener Diode) for this part of the experiment. If you use the 1N4004 diode, your breakdown voltage will be more negative than -100 V and diode breakdown cannot be seen on lab curve tracer. Also, the reverse leakage current will be very low for all of the silicon diodes in your lab kit, and such extremely low current cannot be measured with curve tracer.

Procedure

1. Press the DUT button to turn off the power to the diode. Reverse the voltage polarity on the diode by turning the diode around in the socket and set I_{\max} to 2mA, V_{\max} to 40V, and P_{\max} to 0.4W. Press the DUT button to obtain the reverse characteristics.
2. Sketch the reverse I-V characteristic as accurately as possible up to the breakdown voltage, noting the breakdown voltage on your plot.

PART II: SIMPLE DIODE CIRCUITS

You should have simulated the diode-resistor circuit shown in Figure 1.5 with the values of R given below.

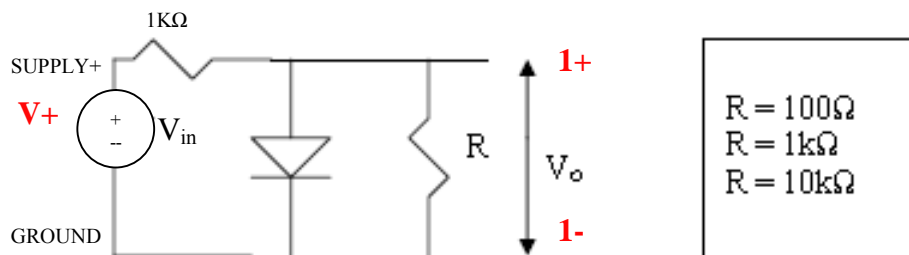


Figure 1.5: Simple Diode Circuit

Procedure

- Build the circuit shown in Figure 1.5 on the NI-ELVIS breadboard with $R = 100\Omega$. Use the 1N4001 or the 1N4004 diode.
- Use the digital multimeter (DMM) DC Voltage [V=] function to measure the output voltage, V_o , using the V Ω and COM banana jacks.
- Using the VPS front panel, Click “Run” and vary the voltage SUPPLY+ from 0 to 5V in increments of 0.5V and record the output voltage. If you are proficient with the ELVIS you can use the VPS sweep function with the following settings: Start 0V, Step 0.5V, Stop 5V, Interval 5000ms. In this case, Click “Sweep” instead of “Run”.
- Repeat steps a to c for the other values of R given in Figure 1.5 and complete Table 1.2, below.

Table 1.2: Measured V_o for circuit in Figure 1.5

V_{in} (volts)	V_o (volts)		
	$R = 100\Omega$	$R = 1k\Omega$	$R = 10k\Omega$
0.0			
0.5			
1.0			
1.5			
2.0			
2.5			
3.0			
3.5			
4.0			
4.5			
5.0			

LAB REPORT:**PART I**

- Plot I_D vs. V_D on linear graph paper (or using software of your choice).
- Determine V_γ directly from the plotted graph and record your result. Determine how V_γ depends on the diode current and voltage levels.
- Determine the slope from the linear plot of the forward I-V characteristic as a function of the diode current, as shown in Figure 1.4, using the data previously taken in Table 1.1. Note that the inverse of the slope, or r_d , you measure is a function of where you choose the diode current. Obtain at least eight (8) slope data points.
- Plot R_F and r_d from Table 1.1 as a function of diode current. How do R_F and r_d compare?
- Create a semi-log graph of I_D (log scale) vs. V_D . Determine n from the slope. Remember that on a semi-log plot where I is plotted as a function of V_D , an exponential function is a

straight line. Use Equation (1.2) to determine n . Note that if $n = 1$, the current will increase by 1 decade for every 0.060V of V_D , and if $n = 2$, the current will increase by 1 decade for every 0.120V of V_D . Extrapolate the current to $V_D = 0V$ and determine the value of I_S .

PART II

1. What is the effect of the diode on V_o ?
2. Derive an expression for V_o as a function of V_{in} using the piecewise-linear model of the diode comprised of the values of r_d and V_γ you calculated from the linear I-V characteristic plot.
3. Compare the measured V_o with the simulated output voltage and the calculated output voltage from step II.2. Comment on any differences. Remember that the B2 Spice simulation is a simulation and is only as good as the parameters you use to describe the diode. Your experimental results are reality. You should be comparing the voltages you measured with the values you calculated using the measured diode characteristics.

ECE 3110 EXPERIMENT 1 - CHECK LIST

1. Diode Characteristics

- a. Forward Characteristics ($V_{MAX} = 2V$)
 - i. Obtain forward diode characteristics on curve tracer.
 - ii. Record linear and log current data points using cursor.
 - iii. Plot I-V characteristic (linear and semi-log).
 - iv. Determine r_d as a function of diode current.
 - v. Determine R_F as a function of diode current.
 - vi. Determine V_γ . How does V_γ change with diode current?
- b. Reverse Characteristics
 - i. Attempt to obtain reverse diode characteristics. Set $V_{MAX} = 40 V$.
 - ii. Is the breakdown voltage greater than 100 V?

2. Simple Diode Circuits

- a. Build Circuit
- b. Measure output (V_o) as a function of input voltage for $R = 100\Omega$, $1k\Omega$, and $10k\Omega$.
- c. Derive an expression for V_o and V_{in} using piecewise model.
- d. Compare measured results of V_o with simulated results and calculated output voltage.

**ELECTRONICS I
ECE 3110**

Experiment #2 – Power Supply Operation

PURPOSE:

This laboratory session acquaints you with the operation of a diode power supply. You will study the operation of **half-wave and full-wave rectifiers, and the effect of smoothing filters**. You will also learn about DC voltage (V_{dc}), the ripple factor (RF), ripple voltage (V_r), and root mean square voltages (V_{rms} and $V_r(rms)$) of a power supply.

PRE-LAB:

You should simulate the circuits shown in Figures 2.3 and 2.5 with the different resistor and capacitor values that you will use in the experiment. You should be prepared to compare your simulations with your measurements after you have built your circuits.

EXPERIMENT:

- (a) Diodes (1N4001 or 1N4004)
- (b) Resistors
- (c) Capacitors
- (d) NI ELVIS II Workstation

INTRODUCTION:

The diode can be used to change the wave shape of an incoming signal. When used as a rectifier, the asymmetrical properties of the diode's current-voltage characteristics can be used to convert an ac signal into a dc signal. The rectification can either be a half-wave or full-wave.

Half-Wave Rectifier

Figure 2.1(a) shows a basic half-wave diode rectifier circuit. During the positive half-cycle of the input voltage, the diode is forward-biased for all instantaneous voltages greater than the diode cut-in voltage, V_γ . Current flowing through the diode during the positive half-cycle produces approximately a half sine wave of voltages across the load resistor, as shown in the lower part of Figure 2.1(b). To simplify our discussions, we will assume that the diode is ideal and that the peak input voltage is always much larger than the V_γ of the diode. Hence, we assume that the zero of the rectified voltage coincides with the zero of the input voltage. On the negative half-cycle of the input voltage, the diode is reverse-biased. Ignoring the reverse leakage current of the diode, the load current drops to zero, resulting in zero load voltage (output voltage), as shown in Figure 2.1(b). Thus, the diode circuit has rectified the input ac voltage, converting the ac voltage to a dc voltage.

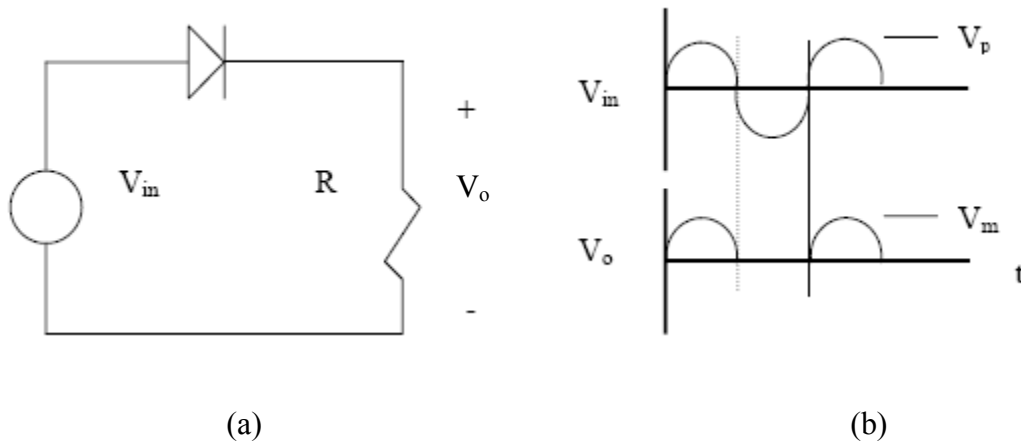


Figure 2.1: A half-wave rectifier

The average or dc value of this simple half-wave rectified signal, V_{dc} , is given by

$$V_{dc} = \left(\frac{1}{T} \right) \int_0^{T/2} V_m \sin \left(\frac{2\pi t}{T} \right) dt = \frac{V_m}{\pi} = 0.318 V_m \quad (2.1)$$

Here V_m is the peak value of the rectified signal.

The average voltage is called the dc voltage because this voltage is what a dc voltmeter connected across the load resistor would read. Hence, if $V_m = 10$ V and the diode is ideal, a dc voltmeter across the load resistor would read 3.18 V.

Full-Wave Rectifier

Figure 2.2(a) shows a full-wave bridge rectifier with a load resistor R_L and an input sine wave derived from a transformer. During the positive half-cycle of the input voltage, diodes D2 and D3 are forward biased and diodes D1 and D4 are reverse biased. Therefore, terminal A is positive and terminal B is negative, as shown in Figure 2.2(b). During the negative half-cycle, illustrated in Figure 2.2(c), diodes D1 and D4 conduct, and again terminal A is positive and terminal B is negative. Thus, on either half-cycle, the load voltage has the same polarity and the load current is in the same direction, no matter which pair of diodes is conducting. The full-wave rectified signal is shown in Figure 2.2(d), with the V_o being the output voltage.

Since the area under the curve of the full-wave rectified signal is twice that of the half-wave rectified signal, the average or dc value of the full-wave rectified signal, V_{dc} , is twice that of the half-wave rectifier.

$$V_{dc} = \frac{2V_m}{\pi} = 0.636 V_m \quad (2.2)$$

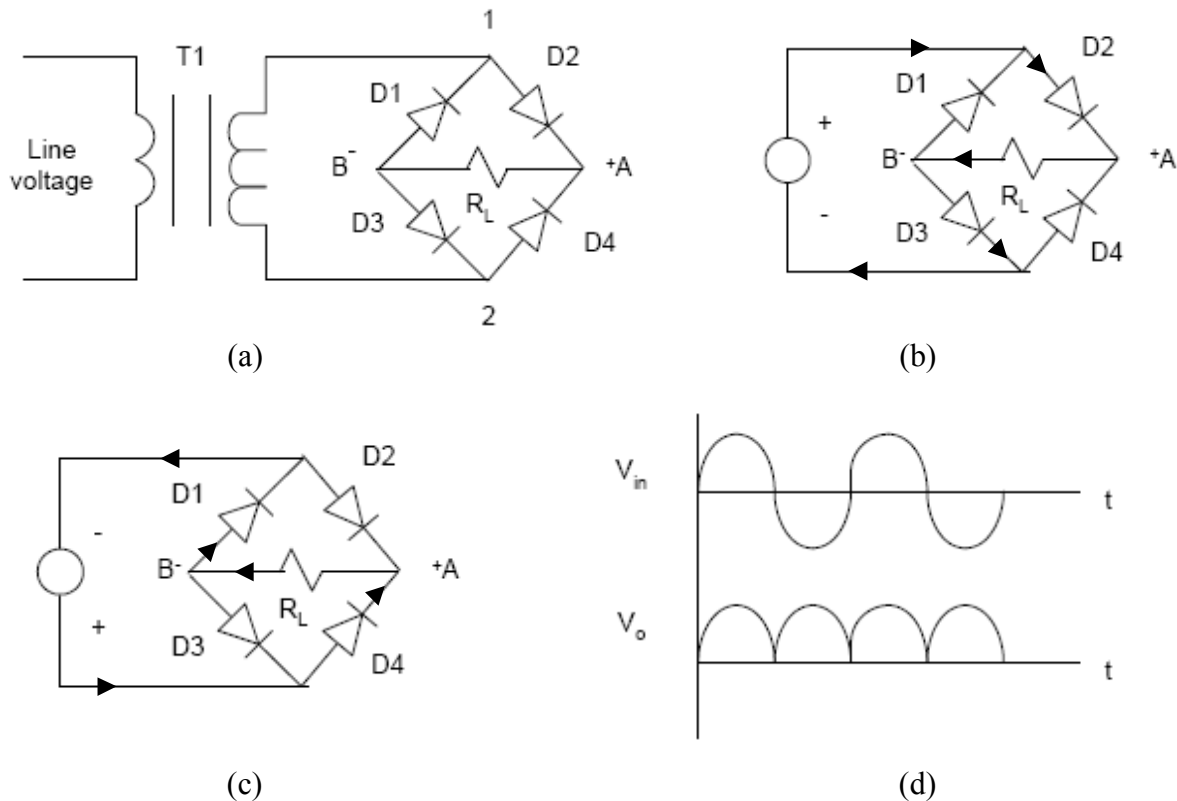


Figure 2.2: Full wave bridge rectifier circuit and waveforms

Filtering

The rectifier circuits discussed above provide a **pulsating** dc voltage at the output. **These pulsations are known as "ripple"**. The uses for this kind of output are limited to charging batteries, running dc motors, and a few other applications where a constant dc voltage is not necessary. For most electronic circuits, however, a constant dc voltage similar to that from a battery is required. To convert a half-wave or full-wave rectified voltage with ripple into a more constant dc voltage, a smoothing filter must be used at the rectifier's output.

A popular smoothing filter is the capacitor-resistor filter, which consists of a single capacitor in parallel with the load resistor. Figure 2.3 shows such a filter connected to the output of a half-wave rectifier. The output wave shape of the filtered half-wave rectifier is similar to that shown in Figure 2.4, assuming that the time constant of the $R_L C$ filter is comparable to the period of the input voltage.

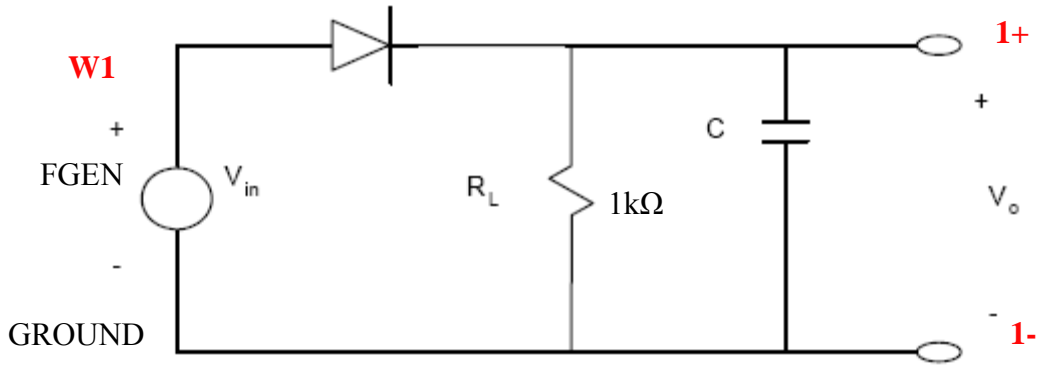


Figure 2.3: Rectifier circuit with an RC smoothing filter

Notice that the output wave shape still has ripple, but the ripple is now saw-tooth or triangular shaped, and its variation is much less than that of the unfiltered pulses. The difference between the maximum and minimum of the filtered voltage is known as the **Ripple Voltage** (V_r). In Figure 2.4, this voltage is labeled ΔV . Thus, we have

$$V_r \equiv \Delta V = V_m - V_{\min}$$

where

V_m = peak value of the rectified signal (smaller than the V_{in} , due to V_γ and R_s),

V_{\min} = the minimum of the filtered voltage. V_{\min} increases as the ripple voltage decreases.

The design equation for selecting this capacitor is

$$\frac{V_r}{V_m} = \frac{T}{R_L C} = \frac{1}{f_p R_L C} \quad (2.3)$$

where

R_L = load resistance

I_L = load current

C = filter capacitance

$T = 1/f_p$ = the period of the rectified wave.

For a half-wave rectifier, f_p is the frequency of the input voltage. For a full-wave rectifier, f_p is twice the frequency of the input voltage. The output of the filtered voltage for a full-wave rectifier is shown in Figure 2.4.

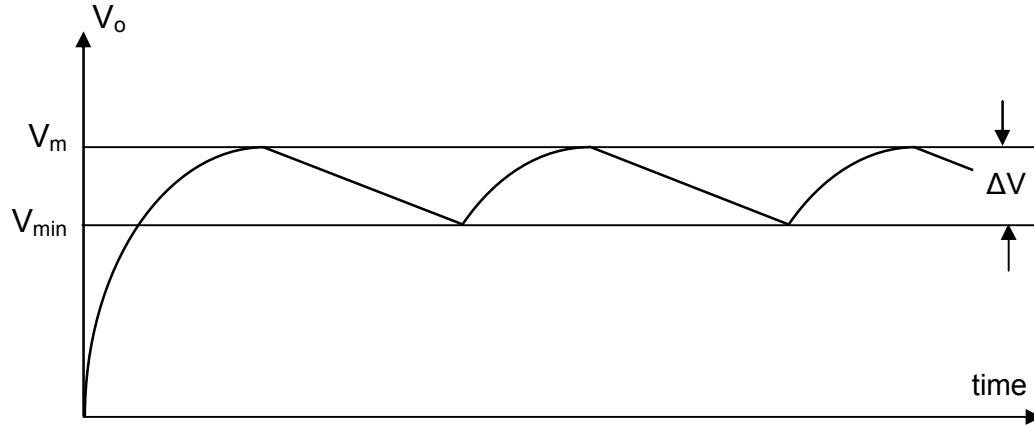


Figure 2.4: Output wave shape from a full-wave filtered rectifier.

From Figure 2.4, we see that $\Delta V (=V_r)$ determines the amount of ripple in the output signal. From Equation (2.3), we see that for the ripple voltage V_r to be small, the $R_L C$ time constant must be large. In other words, the ripple can be reduced by increasing the discharging time constant $R_L C$. Hence, increasing either C or R_L will reduce the ripple voltage. It should be noted that the resistor R_L is usually inside a commercial power supply and any external load connected to the power supply is in parallel with R_L and acts both to lower the total load resistance and to increase the ripple. This is why an audible hum is often heard from power supplies when the external load resistance drops to a very low value.

Two figures of merit for power supplies are the **ripple voltage**, V_r , and the **ripple factor**, **RF**. V_r has already been defined. RF is defined as

$$RF = V_r(\text{rms}) / V_{dc}. \quad (2.4)$$

$V_r(\text{rms})$ is the RMS value of the ripple voltage. The value of $V_r(\text{rms})$ can be calculated for various input wave shapes. For a complicated wave shape, such as that shown in Figure 2.4, the value of $V_r(\text{rms})$ is calculated as if the filtered, rectified wave were a triangular wave, for which

$$V_r(\text{rms}) = \frac{V_m - V_{\min}}{2\sqrt{3}} = \frac{V_r}{2\sqrt{3}}. \quad (2.5)$$

It is important to note that V_r and $V_r(\text{rms})$ are not the same.

Earlier we talked about the average or dc voltage, V_{dc} , for unfiltered, rectified supplies. Recall that for a sinusoidal input,

$$V_{dc} = V_m/\pi = 0.318 V_m \quad \text{for an unfiltered half-wave rectifier}$$

and

$$V_{dc} = 2V_m/\pi = 0.636 V_m \quad \text{for an unfiltered full-wave rectifier.}$$

The average (dc) voltage will lie between V_m and V_{\min} . Therefore, for the filtered, triangular wave shape shown in Figure 2.4, a better value of V_{dc} would be given by

$$V_{dc} = V_m - \frac{V_r}{2} = \frac{V_m + V_{\min}}{2}, \quad (2.6)$$

or equivalently,

$$V_{dc} = V_m - \frac{V_m}{2f_p R_L C} = V_m \left(1 - \frac{1}{2f_p R_L C} \right) = V_r \left(f_p R_L C - \frac{1}{2} \right). \quad (2.7)$$

EXPERIMENTAL:

PART 1: HALF-WAVE RECTIFIER

1. Build the circuit shown in Figure 2.3 using the NI-ELVIS function generator (FGEN) as the voltage source V_{in} . The connections are FGEN (pin #33) and GROUND (either pin 49 or 53) on the left side of the NI-ELVIS breadboard. **Leave the capacitor C out of the circuit for now.** When adding or removing components, ALWAYS "STOP" THE ELVIS INSTRUMENTS AND TURN OFF THE PROTOTYPING BOARD.
 - a. Open the function generator's instrument panel. YOUR PROTOTYPING BOARD MUST BE TURNED ON FOR APPROPRIATE FUNCTION OF THE INSTRUMENT PANEL.
 - b. Using the FGEN instrument panel, set the frequency of the function generator to 60 Hz.
 - c. Click "Run".
 - d. Enable "Manual Mode" on the function generator.
 - e. Adjust the output voltage of the function generator using the manual amplitude control knob (located at the bottom right corner of the workstation) until the voltage is $8V_{p-p}$.
 - f. Verify the function generator settings by connecting the positive clip of the BNC cable to the FGEN terminal and the negative clip to GROUND terminal. Then, connect BNC connector to SCOPE CH 0 of the oscilloscope (BNC input located at upper left corner of workstation).
 - g. Open SCOPE instrument panel.
 - i. Set the Trigger type to "Edge" or "Immediate".
 - ii. Ensure that Channel 0 is Enabled, Coupling is set to "DC", and Source is set to "SCOPE CH 0"
 - iii. Click "Run"
 - iv. Adjust the function generator amplitude as needed to ensure that $V_{p-p} = 8V$
 - v. Click "Stop".

- h. Connect SCOPE CH 1 of the oscilloscope across the resistor R_L . Ensure that ground leads of SCOPE CH 0 and 1 are both connected to ground.
 - i. Ensure that Channel 1 is Enabled, Coupling is set to “DC”, and Source is set to “SCOPE CH 1”
 - ii. Click “Run” to measure V_{in} and V_o .
2. Make an accurate sketch of the input and output waveforms on the same graph paper, with the output waveform superimposed on the input waveform. Enable Cursors by clicking the “Cursors On” box and use the cursors to record V_m and V_{min} by switching the cursor(s) to CH1. Press STOP and adjust time/div to improve accuracy when taking cursor measurements. Record the values of V_m and V_{min} in Table 2.1.
3. Connect a $10\mu\text{F}$ capacitor in parallel with R_L . Make an accurate sketch of the new output waveform on the same graph as used in step 2, but label the new waveform. Record V_m and V_{min} using the cursors.
4. Repeat step 3 for each of the following values of capacitors: $22\mu\text{F}$ and $47\mu\text{F}$. You can draw your output wave shapes on the same graph used in step 2, labeling each waveform. Record V_m and V_{min} for each different resistor/capacitor pair.

Table 2.1: RF at various C when $R_L = 1\text{k}$

$R_L=1\text{k}\Omega$; C (μf)	V_m	V_{min}	V_r	$V_r(\text{rms})$	V_{dc}	Ripple Factor
No Capacitor						
10						
22						
47						

5. Repeat steps 2-3 for $R_L = 470\Omega$, $10\text{ k}\Omega$, $100\text{ k}\Omega$. Start a new graph for each resistor case to keep your graphs of the waveforms from becoming crowded. Record V_m and V_{min} for each case and enter values on Table 2.2.

Table 2.2: RF at various R_L when $C = 10\mu\text{F}$

C= $10\mu\text{f}$; $R_L(\Omega)$	V_m	V_{min}	V_r	$V_r(\text{rms})$	V_{dc}	Ripple Factor
470						
10k						
100k						

6. The two extreme cases of filtering are the filter with the lowest values of R_L and C and the filter with the highest values of R_L and C. Sketch, on one graph, the output wave shapes for these two extreme cases; that is, for $R_L = 470\Omega$, $C = 10\mu\text{F}$ and $R_L = 100\text{k}\Omega$, $C = 47\mu\text{F}$. Record V_m and V_{min} for these two cases. Enter your results in Table 2.3 Part 1.

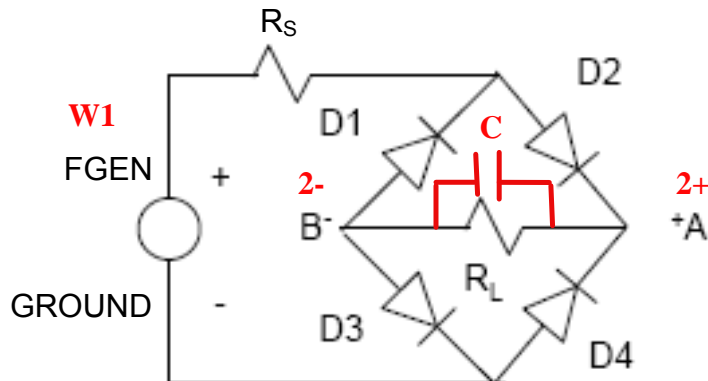
Table 2.3: Extreme Cases of R_L and C

	Part 1: Half Wave Rectifier		Part2: Full Wave Rectifier	
	$R_L = 470\ \Omega$ $C = 10\ \mu\text{F}$	$R_L = 100\ \text{k}\Omega$ $C = 47\ \mu\text{F}$	$R_L = 470\ \Omega$ $C = 10\ \mu\text{F}$	$R_L = 100\ \text{k}\Omega$ $C = 47\ \mu\text{F}$
V_m				
V_{\min}				
V_r				
$V_r(\text{rms})$				
V_{dc}				
Ripple Factor				

- Determine V_{dc} , V_r , $V_r(\text{rms})$, and ripple factor RF for each case in steps 3, 4, and 5.
- Use $R_L = 100\text{k}\Omega$, $C = 47\mu\text{F}$. Note the output wave form. Put a $1\ \text{k}\Omega$ resistor in parallel with R_L and note the change in the output wave shape.

PART II: FULL-WAVE RECTIFIER & POWER SUPPLY

- Build the circuit shown in Figure 2.5. Have your laboratory instructor inspect your circuit before turning on the function generator. Use $R_L = R_S = 1\ \text{k}\Omega$.

Figure 2.5: Full-wave rectifier with voltage-dropping series resistor R_S

- Set the function generator output by connecting the function generator terminals to CH0 of the oscilloscope (BNC CH0 input located in upper left corner of workstation). Positive clip of BNC cable will be connected to FGEN and negative clip will be connected GROUND.
 - Set the function generator's frequency to 60Hz.
 - Click "Run".
 - Enable "Manual Mode" on the function generator.

- d. Adjust the output voltage of the function generator using the manual amplitude control knob (located at the bottom right corner of the workstation) until the voltage is $12V_{p-p}$.
3. Measure the voltage across R_L by connecting nodes A and B to terminals AI0+ and AI0- (located on the top left corner), respectively. **DO NOT USE BNC INPUT OF CH1.** (This is necessary to avoid oscilloscope grounding errors.)
 - a. Ensure that Channel 1 is Enabled, Coupling is set to “DC”, and Source is set to “AI 0”.
 - b. Click “Run” to measure V_{in} and V_o .
4. Make an accurate sketch of the both waveforms. Record V_m and V_{min} of the output voltage using the cursor.
5. Put a $10\mu F$ capacitor in parallel with R_L . Repeat step 4.
6. Repeat step 4 for $C = 22\mu F$ and $47\mu F$. Record V_m and V_{min} in Table 2.4 for each configuration from steps 4, 5, and 6, and compute $V_r(rms)$, V_{dc} , and ripple factor RF.

Table 2.4: RF at various C when $R_L = 1k$

$R_L=1k\Omega$; C (μF)	V_m	V_{min}	V_r	$V_r(rms)$	V_{dc}	Ripple Factor
No Capacitor						
10						
22						
47						

7. With $C = 10\mu F$, make an accurate sketch of the input and output waveforms for $R_L = 470\Omega$, $10k\Omega$, $100k\Omega$. Record V_m and V_{min} in Table 2.5 and compute V_r , $V_r(rms)$, V_{dc} , and ripple factor RF.

Table 2.5: RF at various R_L when $C = 10\mu F$

C= $10\mu F$; $R_L(\Omega)$	V_m	V_{min}	V_r	$V_r(rms)$	V_{dc}	Ripple Factor
470						
10k						
100k						

8. Sketch the waveforms for the two extreme filters, as you did for the half-wave rectifier. Record V_m and V_{min} . Enter these values in Table 2.3 Part 2 and compute V_r , $V_r(rms)$, V_{dc} , and ripple factor RF.

LAB REPORT:

1. Discuss how the filter changed the shape of the output wave.
2. Compare measured RF with calculated RF in Equation (2.3) for each case of R_L and C.

3. Compare measured data with simulated data. How are they different?

ECE 3110 EXPERIMENT 2 – CHECK LIST

1. Half-Wave Rectifier: Build circuit
 - a. Vary Capacitor
 - i. $R_L = 1\text{ k}\Omega$.
 - ii. Sketch input and output waveforms when capacitor is not connected and for $C = 10\mu\text{F}$, $22\mu\text{F}$, $47\mu\text{F}$.
 - iii. Record V_m , V_{\min} , V_r , V_{dc} , $V_r(\text{rms})$, and RF for each case.
 - b. Vary Resistor
 - i. $C = 10\mu\text{F}$.
 - ii. Sketch input and output waveforms for $R_L = 470\Omega$, $1\text{ k}\Omega$, $10\text{ k}\Omega$, $100\text{ k}\Omega$.
 - iii. Record V_m , V_{\min} , V_r , V_{dc} , $V_r(\text{rms})$, and RF for each R_L .
 - c. Sketch output waveforms for $R_L=470\Omega$, $C=10\mu\text{F}$ and for $R_L=100\text{ k}\Omega$, $C=47\mu\text{F}$.
2. Full-Wave Rectifier: Build circuit. Have instructor inspect circuit before starting function generator
 - a. Vary Capacitor
 - i. $R_L = 1\text{ k}\Omega$.
 - ii. Sketch input and output waveforms when capacitor is not connected and for $C = 10\mu\text{F}$, $22\mu\text{F}$, $47\mu\text{F}$.
 - iii. Record V_m , V_{\min} , V_r , V_{dc} , $V_r(\text{rms})$, and RF for each case.
 - b. Vary Resistor
 - i. $C = 10\mu\text{F}$.
 - ii. Sketch input and output waveforms for $R_L = 470\Omega$, $1\text{ k}\Omega$, $10\text{ k}\Omega$, $100\text{ k}\Omega$.
 - iii. Record V_m , V_{\min} , V_r , V_{dc} , $V_r(\text{rms})$, and RF for each R_L .
 - c. Sketch output waveforms for $R_L=470\Omega$, $C=10\mu\text{F}$ and for $R_L=100\text{ k}\Omega$, $C=47\mu\text{F}$.

**ELECTRONICS I
ECE 3110**

Experiment #3 – Power Supply Design

PURPOSE:

The purpose of this laboratory session is to have you use the knowledge gained in the diode experiments to design a dc power supply to meet certain specifications. **THIS LAB IS TO BE PERFORMED BY STUDENTS ACTING INDIVIDUALLY. DO YOUR DESIGN AND SIMULATIONS BEFORE THE LAB.**

PRE-LAB:

After Experiment 2, you should be familiar with the operation of DC power supplies and you should be able to apply this knowledge to the design of a power supply.

- **Design a DC power supply** that meets specifications detailed in the “Design Problem Statement”. You can select either a 22 μ F or 47 μ F capacitor for your design.
- After completing your design, simulate your design using B2 Spice before coming to the lab. You should verify that your simulated power supply meets specifications.

For help with PRE-LAB calculations, a “Design Calculation” procedure is shown following the “Design Problem Statement”.

EQUIPMENT:

Use components as your design requires. The 1N4004 diodes in plastic packages will conduct over 500mA before overheating. Your design will probably be less susceptible to burn-out if you use these diodes.

INTRODUCTION:

Experiment 2 introduced the fundamental aspects of rectifying a sinusoidal ac voltage to obtain a dc voltage. Experiment 2 further described the application of an RC filter to reduce ripple voltage, thereby obtaining a more nearly constant dc voltage for use in power supplies. In this experiment you will apply those concepts to design and build a dc power supply meeting specified operational requirements. This introduction summarizes many equations from Experiment 2 that will be useful for designing the power supply.

Figure 3.1 illustrates the essential nature of the dc waveform resulting from rectifying a sinusoidal ac voltage and applying an RC filter.

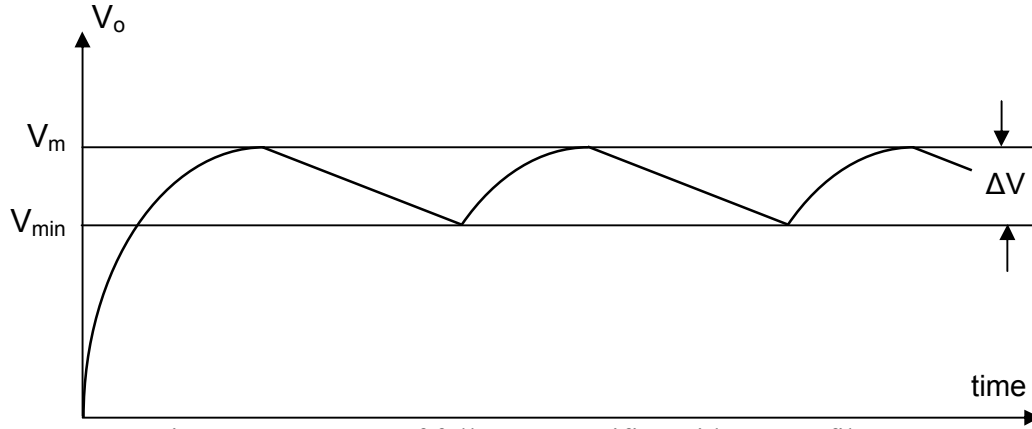


Figure 3.1: Output of full-wave rectifier with an RC filter

V_m = the maximum voltage of the rectified waveform

V_{\min} = the minimum voltage of the rectified waveform

$$\Delta V = V_m - V_{\min}$$

The cyclic variation in the waveform is called **ripple**, and we define the **ripple voltage** V_r :

$$V_r \equiv \Delta V = V_m - V_{\min} \quad (3.1)$$

Recall the following from Experiment 2,

$$\frac{V_r}{V_m} = \frac{T}{R_L C} = \frac{1}{f_p R_L C} \quad (3.2)$$

from which we find

$$R_L C = \frac{V_m}{V_r f_p} \quad (3.3)$$

and

$$V_m = V_r \cdot f_p \cdot R_L \cdot C \quad (3.4)$$

where

R_L = load resistance

I_L = load current

C = filter capacitance

$T = 1/f_p$ = the period of the rectified wave.

For a half-wave rectifier, the ripple frequency f_p is the frequency of the input voltage. For a full-wave rectifier, f_p is twice the frequency of the input voltage. Thus, for a power supply using a transformer attached to the standard electrical system in the United States:

$f_p = 60$ Hz for a half-wave rectifier

$f_p = 120$ Hz for a full-wave rectifier

The root mean square (rms) ripple voltage, $V_r(\text{rms})$, **for a rectified sinusoid with filtered saw-tooth or triangular ripple** (such as in Figure 3.1) is given by

$$V_r(\text{rms}) = \frac{V_r}{2\sqrt{3}} = \frac{(V_m - V_{\min})}{2\sqrt{3}} \quad (3.5)$$

Average or dc voltage
$$V_{dc} = V_m - \frac{V_r}{2} = \frac{(V_m + V_{\min})}{2} \quad (3.6)$$

Substituting Equation 3.4,

$$V_{dc} = V_r \left(f_p R_L C - \frac{1}{2} \right) = \frac{V_r}{2} (2f_p R_L C - 1) \quad (3.6a)$$

Ripple Factor, RF
$$RF = \frac{V_r(\text{rms})}{V_{dc}} \quad (3.7)$$

NOTE: All voltages are those across the load resistor and not those at the output of the rectifier.

Substituting Equations (3.5) and (3.6a) into Equation (3.7), we find **for a rectified sinusoid with filtered saw-tooth or triangular ripple**

$$RF = \frac{1}{\sqrt{3}} \cdot \frac{1}{(2f_p R_L C - 1)} \quad (3.7a)$$

DESIGN PROBLEM STATEMENT:

Solve Equation (3.7a) for R_L as a function of RF, f_p , and C. Show your solution and the resulting equation.

Design a bridge-type, full-wave rectified, dc power supply (as in Figure 2.5) using a filter capacitor (either 22 μ F or 47 μ F). The maximum output voltage V_m of the rectifier is to be at least $V_{in} - 2V_\gamma$, with a ripple factor, RF, less than 0.05. The voltage, V_{in} , is the voltage supplied to your rectifier circuit after the voltage drop across the resistor R_S .

Use the NI-ELVIS workstation to build and test the power supply. Use a 12V_{P-P}, 60 Hz signal from the function generator as your input. Because of the power dissipation limits of the 1N4004 diodes, you should use a 1 k Ω resistor (R_S) in series with the function generator output to drive your rectifier circuit.

- Perform the calculations to design the circuit before simulating the circuit.
- **Simulate** and **measure** the voltage V_{in} .
- Over what range of load resistor values will your circuit meet your design specifications?

NOTE: You need to measure the function generator's output and the voltage after the 1k Ω series resistance in order to obtain the magnitude of V_{in} . Be sure to include the 1k Ω series resistor in your B2 Spice simulations. Be sure to see how the 1k Ω series resistor distorts the

input wave to the bridge rectifier circuit. **WHEN MEASURING THE OUTPUT VOLTAGE ACROSS THE LOAD RESISTOR, V_o , DO NOT USE SCOPE CH0 OR SCOPE CH1. INSTEAD, USE THE ANALOG INPUT CHANNEL, AI 0±.**

Design Calculation:

1. Solve Equation (3.7a) for R_L as a function of R_F , f_p , and C .
2. Substitute into your equation the value of your capacitor ($C=22\mu\text{F}$ or $C=47\mu\text{F}$), the ripple frequency $f_p = 120\text{Hz}$, and the Ripple Factor R_F from the design statement.
3. Calculate R_L .

EXPERIMENT:

Build the power supply circuit you designed in the lab.

DO NOT BRING A POWER SUPPLY BUILT OUTSIDE OF CLASS TO THIS LAB, JUST BRING YOUR CALCULATIONS AND SIMULATIONS. YOU WILL BE WORKING INDIVIDUALLY ON THIS EXPERIMENT.

NOTE: After connecting your circuit, HAVE THE LAB INSTRUCTOR CHECK THE CIRCUIT FOR PROPER DIODE CONNECTIONS BEFORE YOU START THE FUNCTION GENERATOR. Also, REMEMBER TO MOVE BOTH THE POSITIVE AND NEGATIVE TERMINALS OF THE OSCILLOSCOPE WHEN MEASURING THE INPUT AND OUTPUT VOLTAGES.

Verify your design by measuring your circuit's input voltage, output voltage, and ripple factor. Let your lab instructor verify your results. Show your lab instructor your calculations and simulations showing the range of load resistors over which your design meets the specifications.

LAB REPORT:

You should come to the lab with your completed design calculations and B2 Spice simulations. You must show your design calculations, your simulations, and the results of your experimental circuit to the lab instructor before you leave the lab. After you have shown your circuit to your lab instructor, explain how the minimum value of R_L that satisfies your design would be affected if C were $1/2$ of the value you used.

You will complete the Lab Summary in the lab. This summary will contain all of the measurements you make on your circuit, including the effects of changing the load resistor and filter capacitor.

Name: _____

ECE 3110 Experiment 3- Lab Summary TO

BE TURNED-IN TO LAB INSTRUCTOR

Objective:

Design a bridge-type, full-wave rectifier, DC power supply with $RF < 0.05$. The supply will be driven by a function generator (FGEN) with a sinusoidal waveform of $12V_{p-p}$ at 60Hz. The maximum size for the filter capacitor is $47 \mu F$. You must use a $1 k\Omega$ resistor in series with the bridge to limit the current.

Get instructor to check circuit before starting the function generator.

Complete this form with data from your lab experiment — not from simulation — and turn it in along with your simulation output and all calculations.

Record the following data from the oscilloscope.

Peak input voltage (after the $1k\Omega$ resistor), $V_{in} =$ _____ V.

Maximum output voltage, $V_m =$ _____ V.

Minimum output voltage, $V_{min} =$ _____ V.

Use the measured values above in the following calculations.

$V_r(rms) = (V_m - V_{min}) / (2\sqrt{3}) =$ _____ V(rms).

$V_{dc} = V_m - [(V_m - V_{min}) / 2] =$ _____ V.

Ripple factor, $RF = V_r(rms) / V_{dc} =$ _____.

Answer the following two questions based on your design calculations.

What is the value of the smallest resistor R_L for which the filter will meet the design specifications? _____ Ω ?

If the maximum size of the filter capacitor were reduced by one-half, the smallest resistor for which the filter would meet the design specifications would be _____ Ω .

**ELECTRONICS I
ECE 3110**

Experiment #4 – Diode Clippers and Clampers

PURPOSE:

The purpose of this experiment is to study the use of diodes in wave-shaping (clipper) circuits and in level-shifting (clamper) circuits.

PRE-LAB:

The circuits to be tested and simulated are several clipper circuits and a clamper circuit. The clippers use a diode, resistor, and voltage source in 8 possible locations and orientations in a circuit. You should be familiar with the basic operation of clipper and clamper circuits as discussed in the text. You should be able to produce the output wave shapes shown for the circuits in Figures 4.1 through 4.7 below. You should simulate the circuits shown below in Figure 4.4(a), Figure 4.5(a), Figure 4.6 and Figure 4.7 using B2 Spice. (Assume $R_L = 1k\Omega$, $V_{SIN} = 8V_{p-p}$ and $V_B = 2V$)

You will need to compare your measured circuits with these specific circuit simulations.

EQUIPMENT:

For this experiment, you will need:

- a. NI ELVIS Workstation
- b. Diodes (IN4004 or equivalent)
- c. Resistors and Capacitors

INTRODUCTION:

Clippers

It is frequently necessary to modify the shape of various waveforms for use in instrumentation, controls, computation, and communications. Wave shaping is often achieved by relatively simple combinations of diodes, resistors, and voltage sources. Such circuits are called clippers, limiters, amplitude selectors, or slicers. Clipper circuits are primarily used to prevent a waveform from exceeding a particular limit, either positive or negative. For example, one may need to limit a power supply's output voltage so it does not exceed +5 V. The most widely used wave shaping circuit is the rectifier, which you have previously studied.

Figure 4.1 shows a **positive clipper circuit**. As indicated, the output voltage has the entire positive half-cycles clipped off. The circuit works as follows: During the positive half-cycle of the input voltage, the diode turns on. For an ideal diode, the output voltage is zero. For an actual diode the output voltage is equal to V_γ , the cut-in voltage of the diode.

During the negative half-cycle, the diode is reverse-biased and can be approximated by an open circuit. In many clippers, the load resistor, R_L , is much larger than the series resistor, R . In which case, essentially all of the negative half-cycle voltage appears at the output through

voltage-divider action. If R_L and R are comparable, then on the negative half-cycle, the output voltage would be given by

$$V_o = V_{p1} = V_p \cdot (R_L / (R_L + R)).$$

Since the first V_γ volts are used to begin conduction in the diode, the output signal is clipped near V_γ , rather than at 0V. If the diode polarity is reversed, the result is a negative clipper that removes the negative half cycle. In this case, the clipping levels occur near $-V_\gamma$.

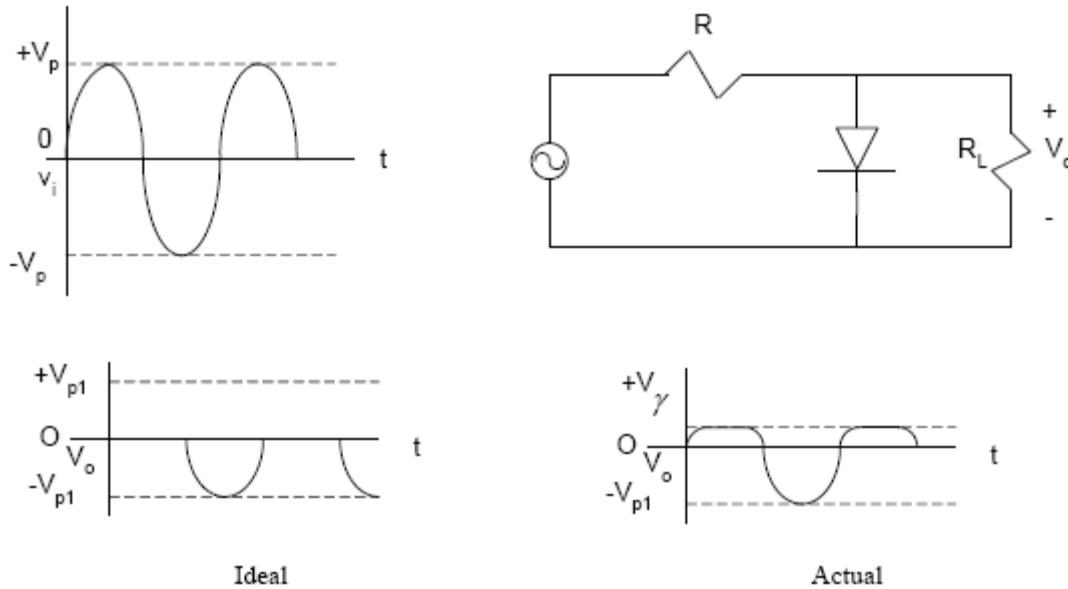


Figure 4.1: A positive clipper circuit: (a) Sinusoidal input to clipper circuit; (b) A positive clipper circuit; (c) Output of ideal positive clipper circuit; and (d) Output of actual positive clipper circuit

If a constant voltage source is placed in series with the diode shown in Figure 4.1(b), the result is a **biased positive clipper**, as shown in Figure 4.2(b). When the input voltage is greater than $V+V_\gamma$, the diode is forward biased and the output voltage is held at $(V+V_\gamma)$ volts (assuming $R_L \gg R$). When the input voltage is less than $V+V_\gamma$, the diode becomes an open circuit and the circuit acts as a voltage divider. R_L is usually much greater than R , in which case, essentially all of the input voltage appears at the output. If both the diode and battery polarities are reversed, a biased negative clipper results, with the output clipped near $-(V+V_\gamma)$ volts.

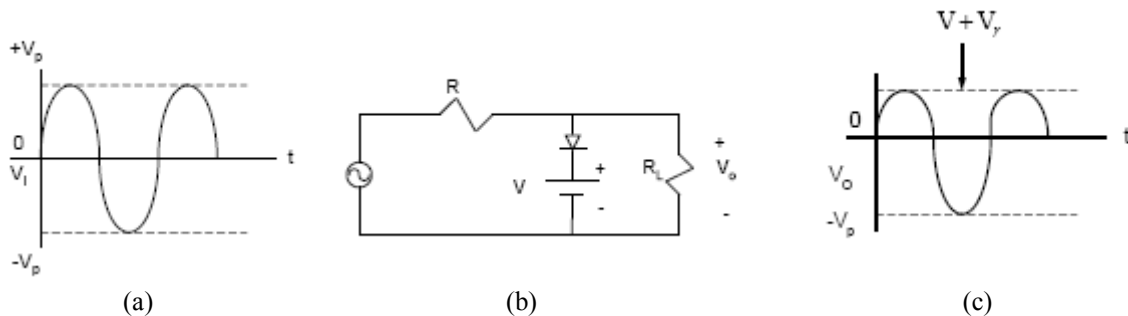


Figure 4.2: Biased positive clipper circuit: (a) Input to clipped circuit; (b) Biased positive clipper circuit; and (c) Output of positive clipper circuit

Clampers

In certain instances, it may be desirable to keep the output waveform essentially unchanged, but modify its dc level to some required value. This can be done by the use of diodes, resistors, capacitors, and voltage sources. Such circuits are known as clampers. For example, if the input voltage signal swings from -10V to $+10\text{V}$, a positive dc clamber can produce an output that keeps the signal wave shape intact but swings the voltage from 0V to $+20\text{V}$. TV receivers use a dc clamber to add a dc voltage to the video signal. Here the dc clamber is usually called a dc restorer.

In Figure 4.3(b) a positive dc clamber is shown. The clamber operates as follows: During the negative half-cycle of the input voltage, the diode turns on as illustrated in Figure 4.4(a). At the negative peak, the capacitor charges up to V_p with the polarity shown and the output voltage is zero. As the voltage grows beyond the negative peak, the diode shuts off as shown in Figure 4.4(b).

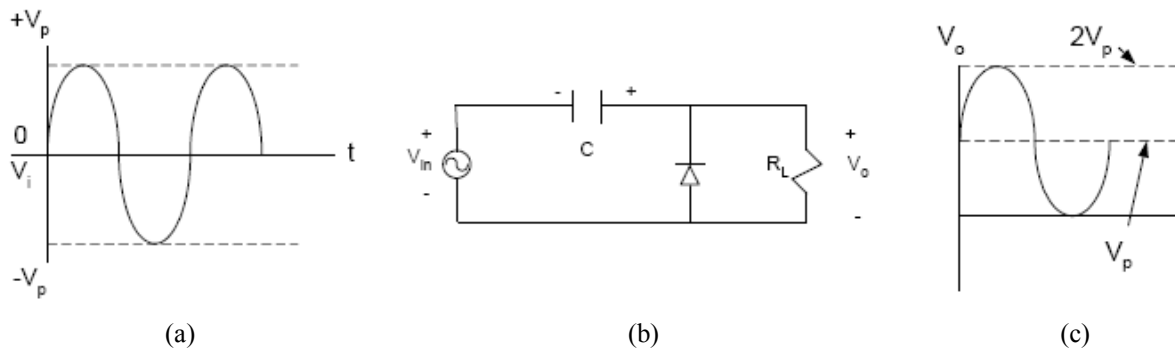


Figure 4.3: Positive dc clamber: (a) Sinusoidal input to positive dc clamber; (b) Positive dc clamber; and (c) Clamped sinusoidal output

The capacitor retains the voltage for a short time. The $R_L C$ time constant is deliberately made much larger than the period, T , of the input signal. Hence, the capacitor remains almost fully charged during the entire off time of the diode. The capacitor thus acts like a battery of V_p volts and now only passes the ac signal, which rides on top of V_p . The output voltage signal, therefore, consists of the input signal riding on a dc voltage of $+V_p$ volts.

Since the diode drops V_γ volts when conducting, the capacitor voltage does not quite reach $+V_p$ volts. For this reason, the dc clamping is not perfect, and the negative peaks are at $-V_\gamma$ as shown in Figure 4.4(e).

When the polarity of the diode in Figure 4.3(b) is reversed, the polarity of the capacitor voltage reverses also, and the circuit becomes a negative dc clamber. Ideally, the output voltage consists of the input voltage riding on a dc voltage of $-V_p$ volts. If the diode is considered non-ideal, then the output will consist of the input signal riding on a dc voltage of $-(V_p - V_\gamma)$ volts, and the positive peaks will occur at V_γ volts.

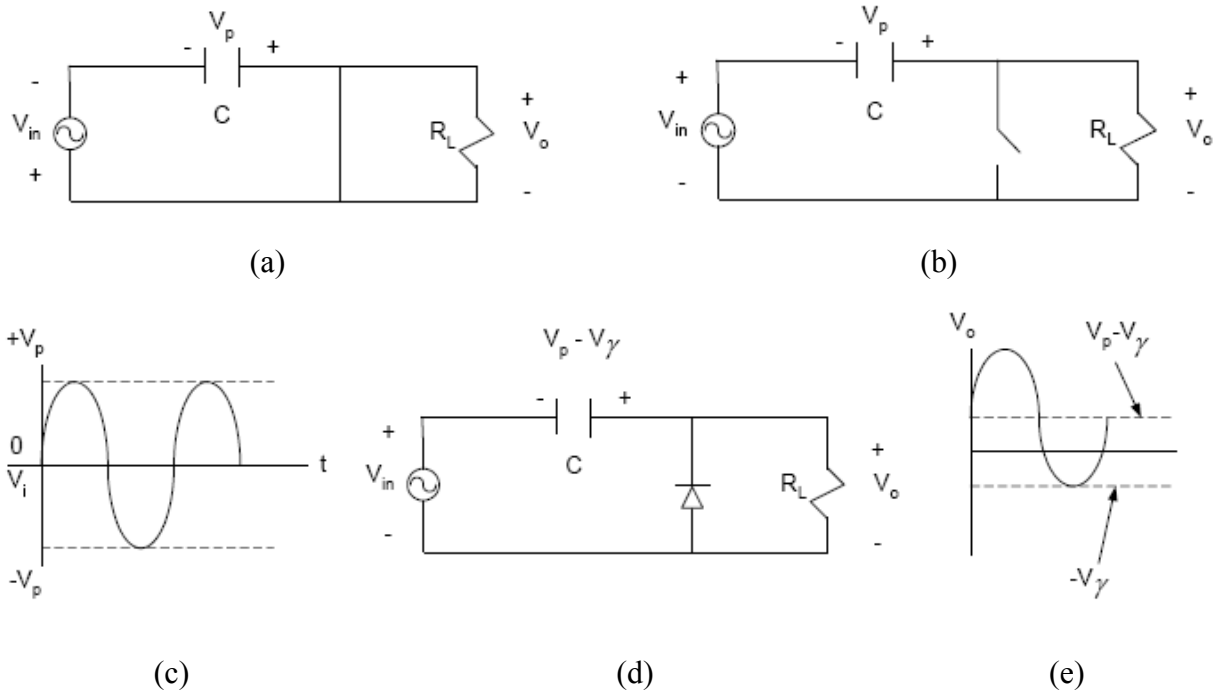


Figure 4.4: Positive clamping circuit operation

From the above discussions, it can be seen that when the diode points upward, a positive dc clamper results. When the diode points downward, the circuit is a negative dc clamper. The clamping value can be modified by putting a voltage source V_B in series with the diode, shifting the peak voltage to $(\pm V_\gamma \pm V_B)$, depending on the sign of V_B and the polarity of the diode.

EXPERIMENT:

PART I: IDEAL CLIPPING CIRCUITS

NOTE: Set $V_{in} = 8V_{p-p}$ at 1kHz with 0V DC offset and $R = 1k\Omega$ for all circuits.

(FGEN setup was covered in Experiment #2, PART I: Half-Wave Rectifier, step 1.)

1. Clipping Circuit 1

- Connect the circuit shown in Figure 4.5(a) using the SUPPLY+ power supply as V_B . Set the SUPPLY+ voltage to 0V using the Variable Power Supply (VPS) front panel.
- Measure V_{in} and V_o using the oscilloscope. Make an accurate sketch of the input and output waveforms on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the input voltage at which clipping occurs.
- Set V_B (SUPPLY+) = 2V and repeat step b.

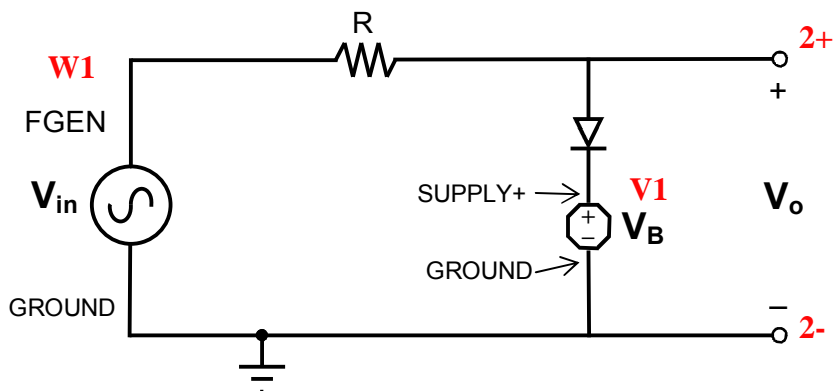


Figure 4.5(a): Clipping Circuit 1

2. Clipping Circuit 2

- Connect the circuit shown in Figure 4.5(b) using the SUPPLY+ power supply as V_B . Set the SUPPLY+ voltage to 0V.
- Measure V_{in} and V_o using the oscilloscope. Make an accurate sketch of the input and output waveforms on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the input voltage at which clipping occurs.
- Set V_B (SUPPLY+) = 2V and repeat step b.

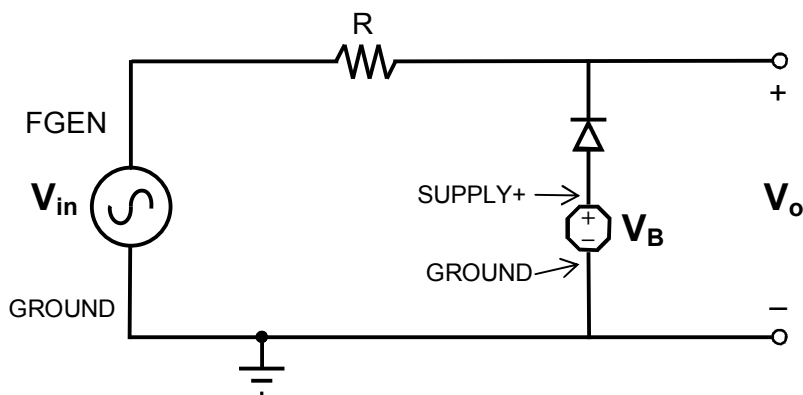


Figure 4.5(b): Clipping Circuit 2

3. Clipping Circuit 3

- Connect the circuit shown in Figure 4.5(c) using the SUPPLY– power supply as $-V_B$. Set the SUPPLY– voltage to 0V.
- Measure V_{in} and V_o using the oscilloscope. Make an accurate sketch of the input and output waveforms on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the input voltage at which clipping occurs.
- Set $V_B = 2V$ (SUPPLY– = -2V) and repeat step b.

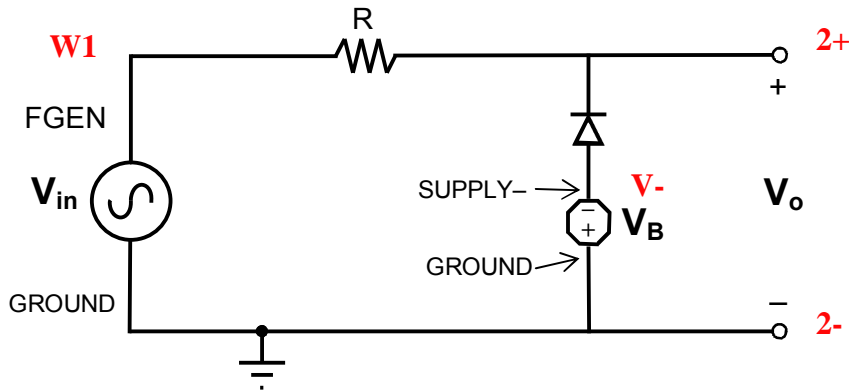


Figure 4.5(c): Clipping Circuit 3

4. Clipping Circuit 4

- Connect the circuit shown in Figure 4.5(d) using the SUPPLY– power supply as $-V_B$. Set the SUPPLY– voltage to 0V.
- Measure V_{in} and V_o using the oscilloscope. Make an accurate sketch of the input and output waveforms on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the input voltage at which clipping occurs.
- Set $V_B = 2V$ (SUPPLY– = -2V) and repeat step b.

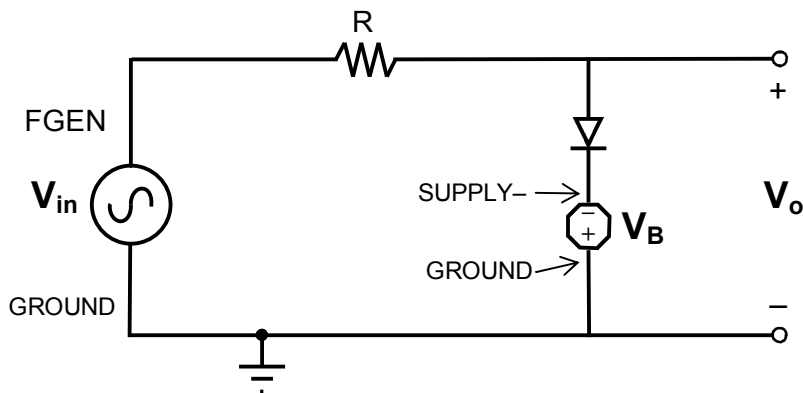


Figure 4.5(d): Clipping Circuit 4

PART II: SERIES-BIASED CLIPPING CIRCUITS

NOTE: The Function Generator (FGEN) in the following circuits will provide both the V_{SIN} and V_B voltages. DO NOT USE SUPPLY+ OR SUPPLY– IN THE CIRCUITS.

$R = 1k\Omega$ for all circuits.

1. Series-Biased Clipping Circuit 1

- Connect the circuit shown in Figure 4.6(a) using the function generator to supply both V_{SIN} and V_B . Set the AMPLITUDE voltage to $8V_{\text{P-P}}$. Set the frequency to 1kHz. **Set the DC offset to 0V.**
- Measure V_{SIN} and V_o using the oscilloscope using SCOPE CH0 and CH1, respectively. **Set the coupling on CH0 and CH1 to DC.** Click “Autoscale”. Make an accurate sketch of V_{SIN} and V_o on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the value of V_{SIN} at which clipping occurs. Use the cursors as needed.
- Set the DC offset of the function generator to ~~2V~~ **1V** (which is same as ~~$V_B = 2V$~~ **$V_B = 1V$**) and repeat step b.

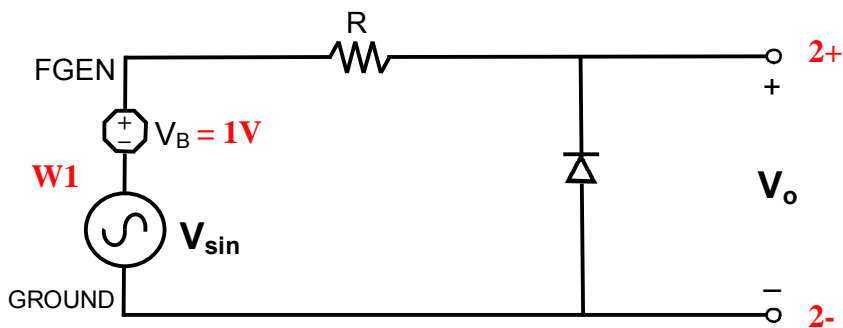


Figure 4.6(a): Series-Biased Clipping Circuit 1

2. Series-Biased Clipping Circuit 2

- Connect the circuit shown in Figure 4.6(b) using the function generator to supply both V_{SIN} and V_B . Set the AMPLITUDE voltage to $8V_{\text{P-P}}$. Set the frequency to 1kHz. Set the DC offset to 0V.
- Measure V_{SIN} and V_o using the oscilloscope using SCOPE CH0 and CH1, respectively. Set the coupling on CH0 and CH1 to DC. Click “Autoscale”. Make an accurate sketch of V_{SIN} and V_o on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the value of V_{SIN} at which clipping occurs. Use the cursors as needed.
- Set the DC offset of the function generator to ~~2V~~ **-1V** (~~$V_B = 2V$~~ **$V_B = -1V$**) and repeat step b.

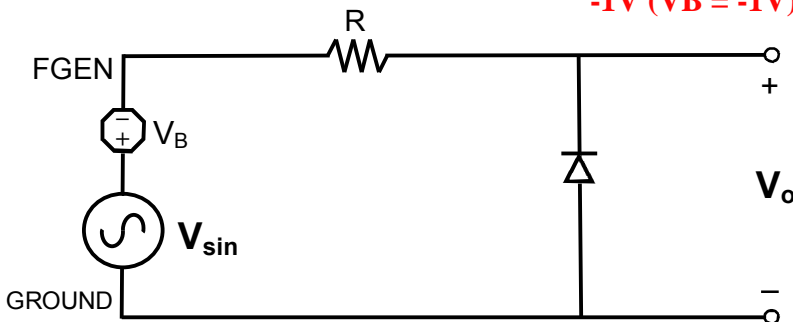


Figure 4.6(b): Series-Biased Clipping Circuit 2

3. Series-Biased Clipping Circuit 3

- Connect the circuit shown in Figure 4.6(c) using the function generator to supply both V_{SIN} and V_B . Set the AMPLITUDE voltage to $8V_{\text{P-P}}$. Set the frequency to 1kHz. Set the DC offset to 0V.
- Measure V_{SIN} and V_o using the oscilloscope using SCOPE CH0 and CH1, respectively. Set the coupling on CH0 and CH1 to DC. Click “Autoscale”. Make an accurate sketch of V_{SIN} and V_o on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the value of V_{SIN} at which clipping occurs. Use the cursors as needed.
- Set the DC offset of the function generator to ~~2V ($V_B = 2V$)~~ **1V ($V_B = 1V$)** and repeat step b.

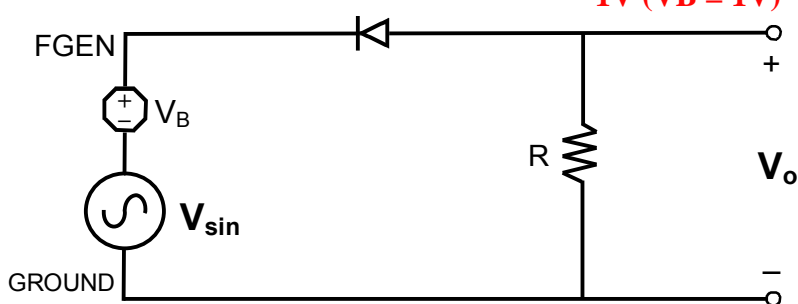


Figure 4.6(c): Series-Biased Clipping Circuit 3

4. Series-Biased Clipping Circuit 4

- Connect the circuit shown in Figure 4.6(d) using the function generator to supply both V_{SIN} and V_B . Set the AMPLITUDE voltage to $8V_{\text{P-P}}$. Set the frequency to 1kHz. Set the DC offset to 0V.
- Measure V_{SIN} and V_o using the oscilloscope using SCOPE CH0 and CH1, respectively. Set the coupling on CH0 and CH1 to DC. Make an accurate sketch of V_{SIN} and V_o on the same graph, making note of the peak values of V_o (minimum V_o and maximum V_o) and the value of V_{SIN} at which clipping occurs. Use the cursors as needed.
- Set the DC offset of the function generator to ~~-2V ($V_B = -2V$)~~ **-1V ($V_B = -1V$)** and repeat step b.

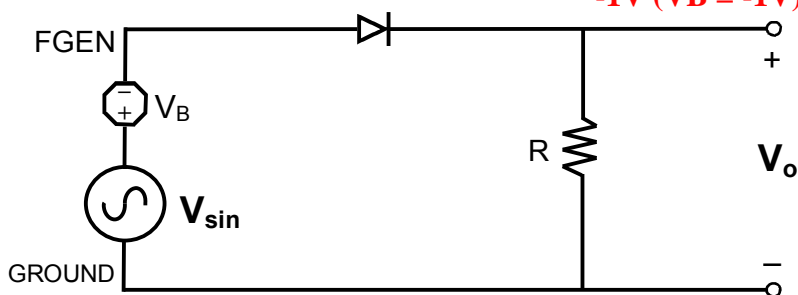


Figure 4.6(d): Series-Biased Clipping Circuit 4

PART III: PARALLEL-BIASED CLIPPER

It is possible to clip wave forms at two different voltages. The circuit shown in Figure 4.7 clips the waveform ideally at V_{B1} (SUPPLY+ = 2V) and V_{B2} (SUPPLY- = -2V). Make an accurate sketch of the input and output voltage waveforms for the circuit in Figure 4.7, noting the peaks of the output waveform and the input voltage at which clipping occurs. What would happen to the output wave shape if either V_{B1} or V_{B2} were changed continuously from 1V to 2V?

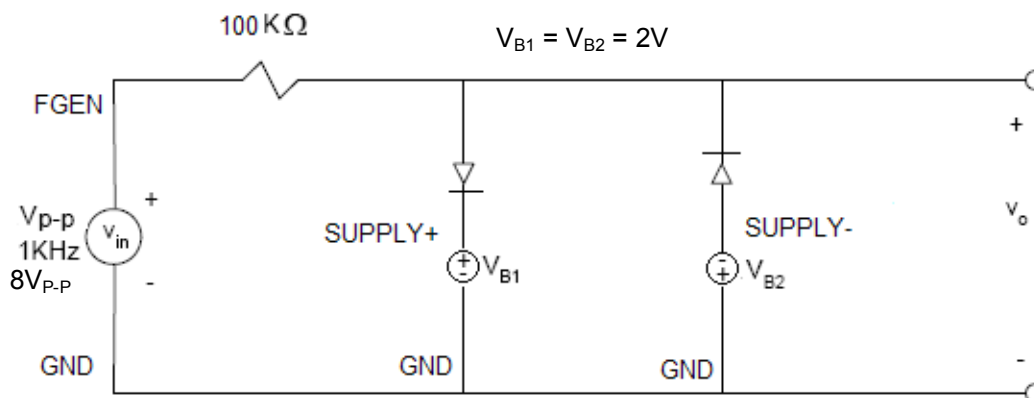


Figure 4.7: Parallel biased clipping circuit

PART IV: CLAMPERS

Build the circuit shown in Figure 4.8 and make an accurate sketch of the input and output waveforms for a $8V_{p-p}$ sine wave input voltage for each of the following conditions. **Each condition should be relative to the circuit as shown in Figure 4.8.** Be sure to record the peak values of both waveforms on your sketch. A capacitor larger than $22\ \mu F$ will work in this circuit. **Set the coupling of both oscilloscope channels to DC.**

1. Circuit as shown with V_{DC} (or SUPPLY+) = 2 volt.
2. Reverse polarity of diode and of capacitor.
3. Diode as shown with the battery reversed. (**Replace SUPPLY+ with SUPPLY-**)
4. Reverse polarity of diode, of capacitor, and of battery. (**Replace SUPPLY+ with SUPPLY-**)
5. Continuously vary SUPPLY+/SUPPLY- to see how the waveform is shifted in each case.

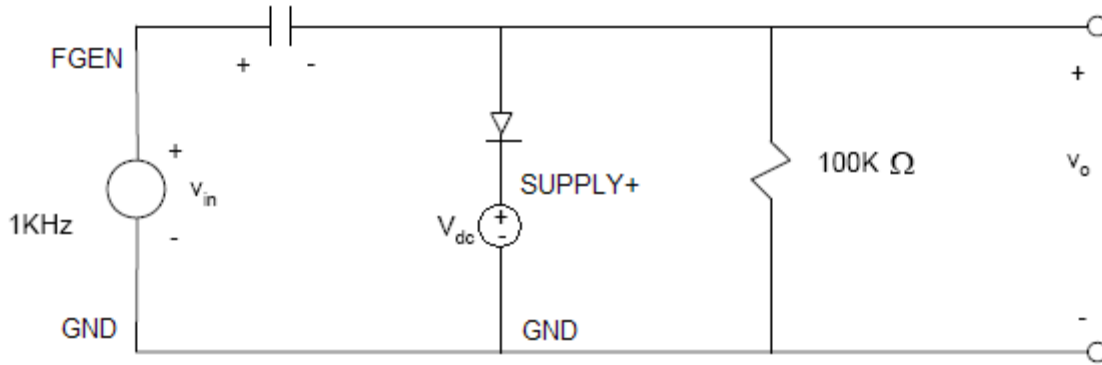


Figure 4.8: Clamping circuit

LAB REPORT:

1. Calculate the clipped voltage levels using circuit analysis for Fig. 4.5(a), 4.6(a), 4.7, and 4.8. Assume diode cut-in voltage $V_\gamma = 0.7\text{V}$ and neglect diode forward resistance r_f .
2. Make a table comparing the measured, calculated, and simulated clipped voltage levels for the circuits in Figures 4.5(a), 4.6(a), 4.7, and 4.8 (i.e. % difference).
3. Explain differences between measured, calculated, and simulated data. What are the assumptions made in calculated and simulated data? Are the assumptions valid? What are the advantages and disadvantages to three approaches?

ECE 3110 EXPERIMENT 4 - CHECK LIST

PRE-LAB:

Simulate the circuits in Figures 4.5(a) ($V_B = 0$ & 2V), 4.6(a) ($V_B = 0$ & 2V), 4.7, and 4.8 (as shown).

EXPERIMENT:

1. Ideal Clipping Circuits — Build 4 circuits (Figures 4.5 a-d)
 - a. $V_{in} = 8\text{V}_{\text{P-P}}$, 1kHz , $V_B = 0$ & 2V .
 - b. Sketch the input and output waveforms.
 - c. Record the voltage at which clipping occurs.
2. Series-Biased Clippers — Build 4 circuits (Figures 4.6 a-d)
 - a. $V_{sin} = 8\text{V}_{\text{P-P}}$, 1 kHz , $V_B = 0$ & 2V .
 - b. Sketch the input and output waveforms.
 - c. Record the voltage at which clipping occurs.
3. Parallel-Biased Clipper — Build circuit (Figure 4.7)
 - a. $V_{in} = 8\text{V}_{\text{P-P}}$, 1kHz , V_{B1} and $V_{B2} = 2\text{V}$.

- b. Sketch the input and output waveforms.
 - c. Record the voltage at which clipping occurs.
- 4. Clamper — Build circuit (Figure 4.8)
 - a. $V_{in} = 8V_{P-P}$, **1kHz**, $V_{dc} = 2V$.
 - b. Sketch the input and output waveforms.

**ELECTRONICS I
ECE 3110**

Experiment #5 – Bipolar Junction Transistor Characteristics

PURPOSE:

This experiment is designed 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.

PRE-LAB:

Read the sections in 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.

Make sure to bring a piece of semi-log graph paper and a piece of linear graph paper (or you may plot the characteristics in Excel).

EQUIPMENT:

For the experiment you will require:

- a) A silicon transistor (NPN) 2N3904 ~~or 2N2222~~
- b) A curve tracer
- c) Semi-log graph paper
- d) Linear graph paper

You will need a few sheets of notebook paper to record your readings. You should have your copy of the text with you while performing the experiment. The portion of the experiment devoted to measuring the h-parameters of the output characteristics is also covered in the portion of the Hameg manual in Appendix D.

INTRODUCTION:

A Bipolar Junction Transistor (BJT) is a three-terminal semiconductor device capable of amplifying an ac signal. The three terminals are called the emitter, the base, and the collector. The device is made up three “layers” of p-type and n-type semiconductor material. BJTs consist of a thin base layer (either P- or N-type) sandwiched between two layers of the opposite type material. Thus, BJTs are either NPN or PNP. They are somewhat like two interconnected, back-to-back diodes, with two diode junctions. The two types of BJTs are illustrated schematically in Figure 5.1.

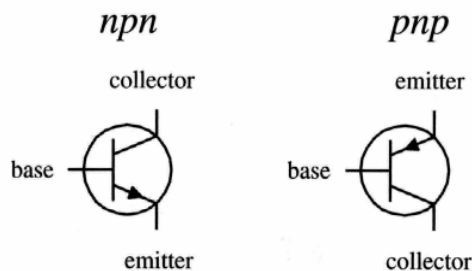


Figure 5.1 Schematic diagrams of general NPN and PNP transistors

BJTs are used to amplify current, using a small base current to control a large current between the collector and the emitter. This amplification is so important that one of the most noted parameters of transistors is the dc current gain, β , which is the ratio of collector current to base current: $I_C = \beta \cdot I_B$. In this lab you will measure several such parameters, discussed in paragraphs below.

The terminology for transistors includes a lot of subscripts. Generally, the subscripts mean:

C	→	Collector
E	→	Emitter
B	→	Base

So, V_{CE} is the collector-emitter voltage; V_{BE} is the base-emitter voltage; and I_B is the base current. One relationship to keep in mind is that I_C is always less than I_E . In fact, $I_E = I_C + I_B$.

Three configurations for connecting bipolar junction transistors are common-base, common-emitter, and common-collector. A large number of transistor circuits use the BJT connected in the common-emitter (CE) or grounded-emitter configuration. In the CE configuration, the input current and output voltage are the independent variables, while the input voltage and output current are the dependent variables.

In this experiment, the input and output characteristics of a transistor will be measured. **The input characteristics are plots of I_B versus V_{BE} at constant values of V_{CE} .** These characteristics will look like diode characteristics, particularly if the collector is shorted to the emitter and the emitter-base junction is forward biased. **Since the curve tracer is not able to extract the base current from the emitter current, the common-base input characteristics will be measured,** and I_E will be plotted as a function of V_{BE} . Since the characteristics will be measured under dc conditions, the dc currents and voltages will be specified.

The output characteristics, often called the collector characteristics, are plots of I_C versus V_{CE} at constant values of I_B and have 3 basic regions of transistor operation. These regions are the cutoff, the active, and the saturated regions. If a BJT transistor is to be used as an amplifier, it will usually be operated in the active region, where the relationship between the input current, I_B , and the output current, I_C , is nearly linear; that is, $I_C = \beta \cdot I_B$, where β is the gain. If the transistor is to be used in digital circuitry, it will be operated in the saturated or cutoff conditions and will only be in the active region when switching from one condition to the other.

In this experiment:

- You will measure and plot the input and output characteristics for the transistor used in the common-emitter mode and label the three regions of operation.

- You will determine the **dc current gain, h_{FE}**

$$h_{FE} = I_C/I_B = \beta_F \quad (=BET \text{ on Hameg}) \quad \text{with } V_{CE} \text{ at a constant voltage} \quad (5.1)$$

h_{FE} is also called β_F , the forward dc current gain. It is often simply written as β , and nominal values are quoted in transistor specifications. β is usually in the range of 10 to 500 (most often near 100) and is not constant, being affected by temperature and current.

- You will measure the four hybrid (h_{ij}) parameters:

- the **small-signal current gain, h_{fe}**

$$h_{fe} = \Delta I_C / \Delta I_B \equiv \beta_o \quad (=H21 \text{ on Hameg}) \quad \text{with } V_{CE} \text{ at a constant voltage} \quad (5.2)$$

h_{fe} is also called β_o , the small-signal current gain. Usually β_o is approximately equal β_F .

- the **input resistance, h_{ie}**

$$h_{ie} = \Delta V_{BE} / \Delta I_B \quad (=H11 \text{ on Hameg}) \quad \text{with } V_{CE} \text{ at a constant voltage} \quad (5.3)$$

h_{ie} is sometimes written as r_π

- the **output conductance, h_{oe}**

$$h_{oe} = \Delta I_C / \Delta V_{CE} \quad (=H22 \text{ on Hameg}) \quad \text{with } I_B \text{ at a constant current} \quad (5.4)$$

- the **voltage feedback ratio, h_{re}**

$$h_{re} = \Delta V_{BE} / \Delta V_{CE} \quad \text{with } I_B \text{ at a constant current} \quad (5.5)$$

These terms are defined in Figure 5.3 and Figure 5.7 as the slopes of the characteristics or the spacing between the characteristic curves.

As noted, $h_{ie} = r_\pi$. It is possible to approximate h_{ie} from the output characteristic using the relationship

$$h_{ie} = \frac{V_T}{I_{BQ}} = \frac{\beta V_T}{I_{CQ}} = \beta h_{ib} = r_\pi. \quad (5.6)$$

Here $V_T = kT/q = 26\text{mV} = 0.026\text{V}$ at room temperature, and I_{CQ} and I_{BQ} are I_C and I_B measured at the Q-point (the *quiescent point*). In most cases, the quiescent point is the operating point chosen to allow maximum symmetrical signal swing of the output waveform without clipping. When using the output characteristic to measure the value of h_{ie} , it will be necessary to decide which value of I_{CQ} or I_{BQ} is to be used.

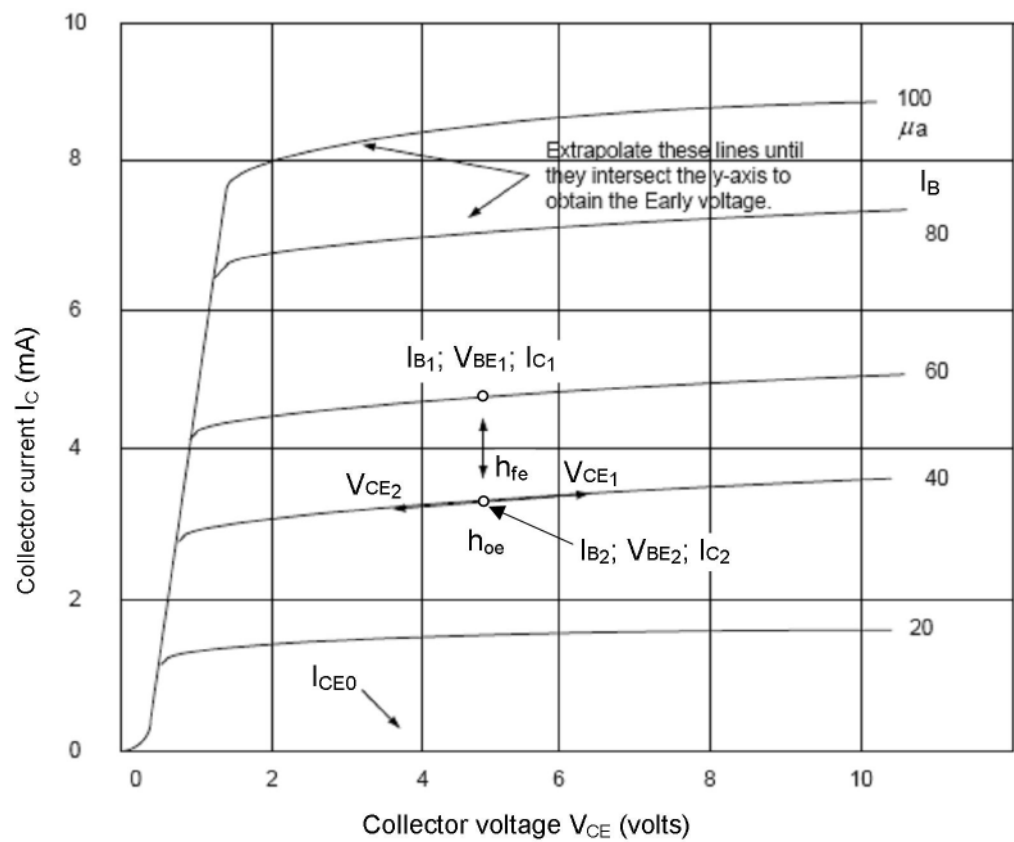


Figure 5.3: Bipolar transistor output characteristics

EXPERIMENT:

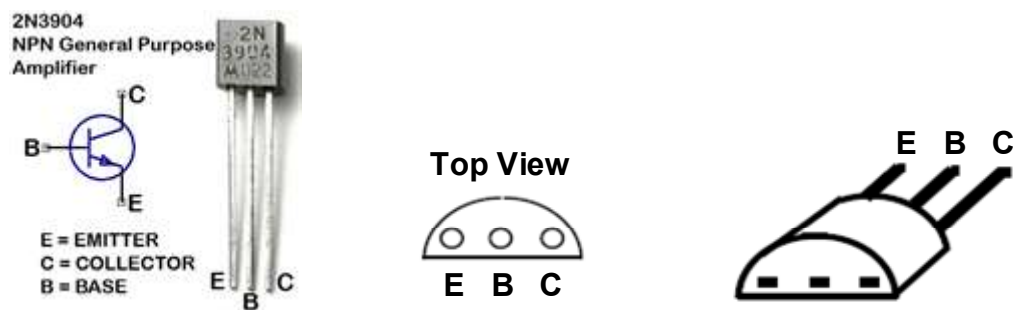


Figure 5.4: Pin diagram for BJT 2N3904

PART I: OUTPUT CHARACTERISTICS

1. Linear Region

- Figure 5.4 shows the pin configuration of the 2N3904 transistor. Use the curve tracer to obtain the output characteristics of the NPN transistor. Set menu to BIP-NPN (bipolar-NPN), $V_{\max} = 10$ volts, $I_{\max} = 2\text{mA}$, and $P_{\max} = 0.4\text{W}$.
- Adjust the curves using MIN and MAX functions until you get desired value base current I_B , as given as in Table 5.1.
- Make a copy of the characteristic curves by hand. Your curves should be similar to those shown in Figure 5.3, which is a good representation of the output characteristics in the active region.
- From the characteristics on the curve-tracer screen, record the values of I_{Bx} , I_{Cx} and V_{BEx} at corresponding V_{CE1} and V_{CE2} . Then, you can compute ΔI_C , ΔI_B , ΔV_C , h_{FE} , h_{fe} and h_{oe} in Table 5.1. Use the cursor control to help with your measurements. You can verify the calculated h_{xx} parameter by measuring them using the Hameg curve tracer, using the push button located below the LCD display.

NOTE: You will need to increase I_{\max} and P_{\max} as you increase I_B to more than $10\mu\text{A}$ ($I_{\max} = 20\text{mA}$ and $P_{\max} = 0.4\text{W}$) and $100\mu\text{A}$ ($I_{\max} = 200\text{mA}$ and $P_{\max} = 4.0\text{W}$).

- Reverse the emitter and collector leads. Set $V_{\max} = 10$ volts, $I_{\max} = 20\text{mA}$, and $P_{\max} = 0.4\text{W}$. Adjust MAX and MIN until five characteristics are displayed. Measure h_{FE} with the transistor operating in the inverse or inverse-active mode. Notice the decrease in the values of h_{FE} .

Table 5.1a: Measured Output Characteristics

N	I_B	Your value of I_B	V_{BE}	I_{C1} @ $V_{CE} = 2.5V$	I_{C2} @ $V_{CE} = 5.0V$	ΔI_{C2} = $I_{C2}(N) - I_{C2}(N-1)$	ΔI_B = $I_B(N) - I_B(N-1)$	ΔV_{BE} = $V_{BE}(N) - V_{BE}(N-1)$	h_{FE} @5V	h_{fe}
0	0.5 μ A									
1	5 μ A									
2	10 μ A									
3	50 μ A									
4	75 μ A									
5	100 μ A									
6	150 μ A									
7	200 μ A									

Table 5.1b: h_{oe} Calculations

N	$\Delta I_C = I_{C2} - I_{C1}$	$\Delta V_{CE} = V_{CE2} - V_{CE1}$	h_{oe}
0			
1			
2			
3			
4			
5			
6			
7			

2. Saturation Region

With $V_{max} = 2V$, $P_{max} = 0.4W$, and $I_{max} = 20mA$, obtain the output characteristics of the NPN transistor in *Forward Active Mode*. The value of $V_{CE}(SAT)$ is the minimum collector voltage for which it is possible to define h_{FE} as being in the linear region of operation; which implies h_{FE} will be relatively constant for increasing V_{CE} . The minimum value of $V_{CE}(SAT)$ is not clearly defined but is less than 0.5 V. Often this value is chosen to be 0.2 V in many texts. Notice that h_{FE} and h_{fe} are not clearly defined until the collector voltage exceeds $V_{CE}(SAT)$. You should verify the current gain using the instrument function keys below the LCD screen.

3. Breakdown voltages and the Early voltage

The slope of the characteristic curves in the forward-active mode is due to an effect that was first analyzed by J. M. Early. The phenomenon is generally called the *Early effect*. When the curves are extrapolated to zero current, they meet at a point on the negative voltage axis, at $V_{CE} = -V_A$, as illustrated in Figure 5.5. The voltage V_A is a positive quantity called the **Early voltage**. Typical values of V_A are in the range $50 < V_A < 300V$.

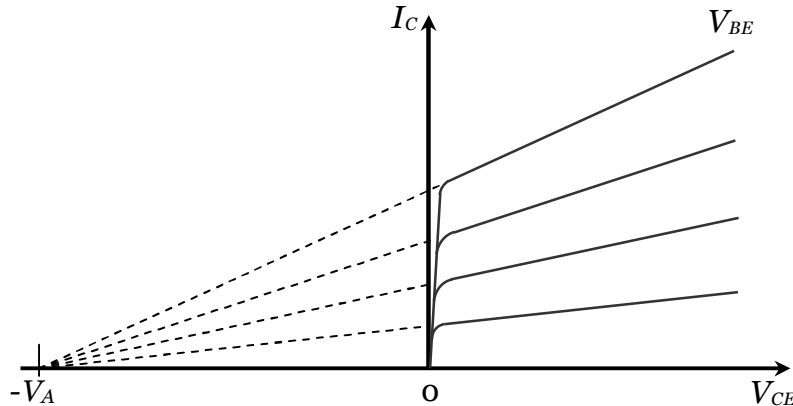


Figure 5.5: Current-Voltage characteristics for common-emitter circuit, illustrating Early voltage.

- Set V_{max} to the highest scale, $P_{max} = 0.4W$, and $I_{max} = 2mA$. Copy this characteristic and obtain the collector breakdown voltage with the base injecting current. It may not be possible to see the breakdown voltage, depending on the maximum voltage scale capable of the curve tracer. Typical output characteristics are shown below in Figure 5.6.
- Estimate the Early voltage by finding a linear fit ($y = mx + b$ or $I_C = mV_{CE} + b$) equation for one of the steepest output curves and solving for x when $y=0$ (i.e., solving for V_{CE} when $I_C=0$). (Early voltage for 2N3904 is often between 100V and 150V.)

Hint To Calculate Early Voltage:

- Adjust the MAX function until you get the top curve on the screen as almost a straight line.
- On this top curve, move the cursor far to the left on the straight-line section, and read the I_C and V_{CE} from the display; these are your I_{C1} and V_{CE1} .
- On the same curve, move the cursor far to the right on the straight-line section, and read the I_C and V_{CE} from the display; these are your I_{C2} and V_{CE2} .

- Calculate the slope m :
$$m = \frac{(I_{C2} - I_{C1})}{(V_{CE2} - V_{CE1})}$$

- Solve for the I_C intercept, b :
$$b = I_{C1} - m \cdot V_{CE1}$$

- Set $I_C = 0$ in the straight-line equation, and calculate the value of V_{CE} for the Early voltage V_A :
$$V_{CE}(0) = -b/m = -V_A.$$

- c. Insert the transistor in the inverted connection position as done in Part 1, Section 1.e. Use $V_{MAX} = 10V$ and $I_{MAX} = 2mA$. Obtain the output characteristics and determine the emitter breakdown voltage. It should come around 6.0 to 8.0V.
- d. Remove the base lead from the socket. Set the Menu to FET-NCH and measure the emitter breakdown voltage. Are the two emitter breakdown voltages the same?

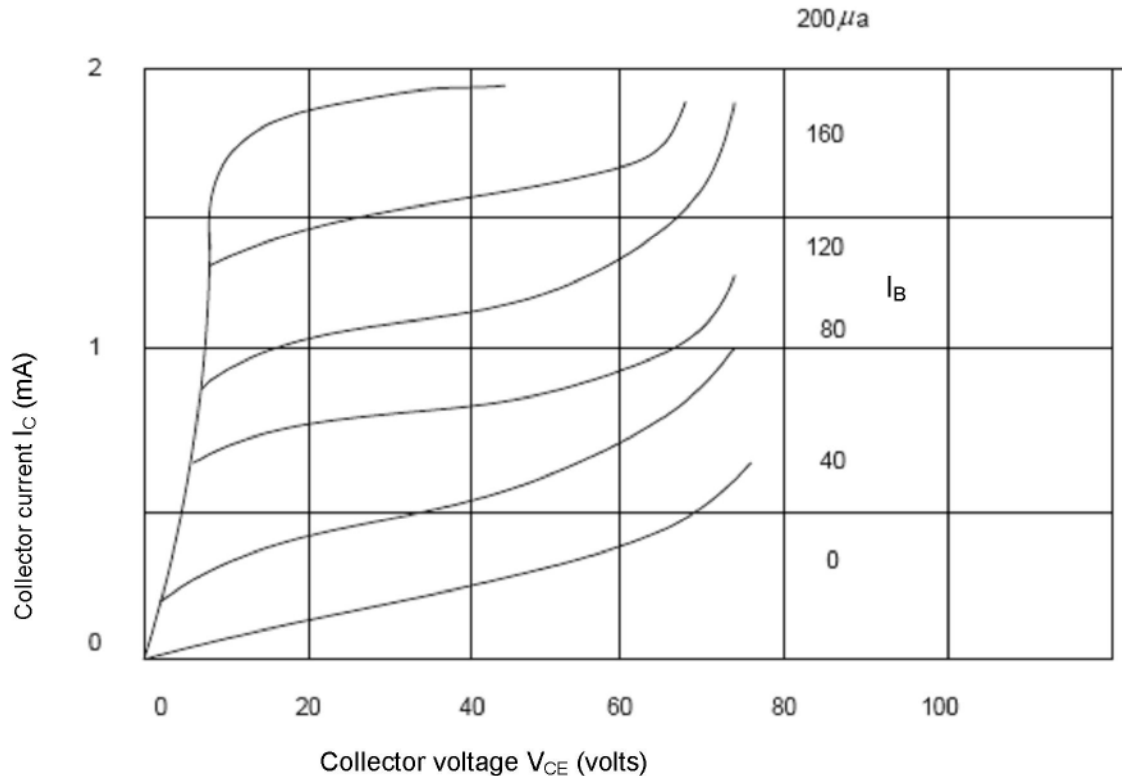


Figure 5.6: Output characteristics at high collector voltages

PART II: Input Characteristics

It is possible to use the curve tracer to obtain the input characteristics of the BJT operating in the common base (CB) configuration. You will measure the CB input characteristic and the value of h_{ib} . The value of h_{ie} can be found from

$$h_{ie} = h_{ib} \times (1 + \beta). \quad (5.7)$$

In order to make this measurement you will apply a sweep voltage to the base-emitter terminals and step the collector voltage through positive voltages.

- a. Adjust the curve tracer for FET - NCH operation and reinsert the silicon NPN transistor into the test socket. Connect the base lead to the "C" terminal and the emitter lead to the "E" terminal. Use $V_{MAX} = 2V$ and $I_{MAX} = 2mA$. The collector lead may be left "hanging" for the first part of this experiment. It is possible to use this configuration to measure part of the input characteristic of the emitter/base junction and to measure h_{ib} , as described in the next step (b). Then, by using the value of β found above, it is possible to calculate h_{ie} .

The type of characteristic that should appear on the screen is the one characteristic shown in Figure 5.7 and labeled as $V_{CE} = 0V$.

- b. Plot the curve directly on linear graph paper and measure $h_{ib} = \Delta V_{EB} / \Delta I_E = h_{ie} / (\beta + 1)$. To calculate h_{ib} , choose two farthest points on the straight line region of the curve and record V_d (for V_{EB1} and V_{EB2}) and I_d (for I_{E1} and I_{E2}) from the display. In this measurement you are directly measuring I_E which is approximately equal to I_C . Use this value of h_{ib} and β (h_{fe}) (from Table 5.1) to calculate $h_{ie} = r_{\pi}$. In this case you will record $\Delta V_{BE} / \Delta I_E$ from your tabulated values.

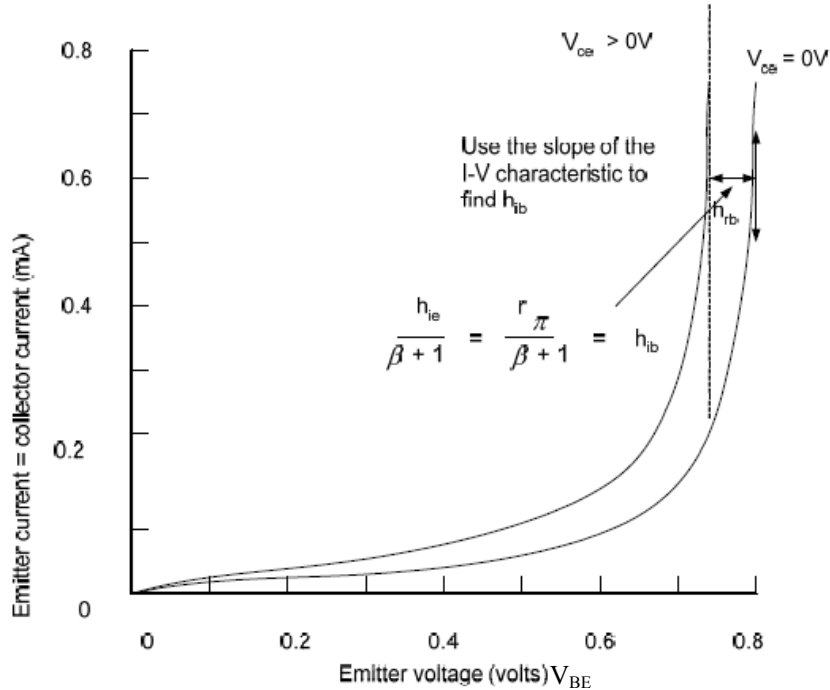


Figure 5.7: Transistor input characteristics

LAB REPORT:

Turn in a lab report containing the following:

1. An output characteristic of your transistor in the linear region of operation showing how to find h_{FE} , h_{fe} , and h_{oe} . Include relevant equations and recorded data.
2. An output characteristic of your transistor showing the saturation region and showing the range of values for $V_{CE(SAT)}$.
3. An input characteristic of your transistor showing how to find h_{ie} , h_{ib} , and h_{re} .
4. Plot h_{FE} and h_{fe} together on one graph of semi-log paper as a function of I_C .
5. Describe how h_{oe} varies with I_C .
6. Estimate the Early voltage from the high-voltage output characteristics.

ECE 3110 EXPERIMENT 5 - CHECK LIST

1. Output Characteristics

- a. Measure output characteristics in the linear region.
 - i. $V_{MAX} = 10V$, $I_B/\text{step} = 0.5, 5, 50\mu A$, adjust I_C to fit characteristics and complete Table 5.1a & b.
 - ii. Copy transistor characteristics.
 - iii. At $V_{CE} = 2.5V$ and $5V$ record I_B , I_C , h_{FE} , h_{fe} , and h_{oe} for each I_B .
- b. Measure output characteristics in the saturation region
 - i. $V_{CE} = 0.5V$.
 - ii. Copy characteristics.
 - iii. Determine $V_{CE}(\text{sat})$.

2. Input Characteristics

- a. Use curve tracer on DIODE. Connect base to “C” and emitter to “E”. Leave collector hanging.
- b. $V_{MAX} = 2V$ and $I_{MAX} = 2mA$: copy characteristic.
- c. Record several values of V_{BE} and I_E and determine the corresponding h_{ib} values.
- d. For each value of h_{ib} , determine a value for h_{ie} .

ELECTRONICS I
ECE 3110

Experiment #6 – BJT Common-Emitter Circuit Bias

PURPOSE:

This experiment is intended to illustrate how a bipolar transistor circuit can be biased for the desired DC operation and how this DC condition can be kept stable if a transistor with a different $\beta = h_{fe}$ is substituted into the circuit. The biasing of the transistor establishes the Q-point of the transistor circuit. The Q-point determines the relationship between the input waveform shape and the output waveform shape. If the Q-point is not near the middle of the load line or if the input signal is too large, it is possible for the output waveform to become distorted. The CE amplifier with a bypassed emitter resistor (i.e., with a capacitor in parallel with the emitter resistor) will be used as the test circuit to show how the Q-point can be stabilized. It will be shown that moving the Q-point along the load line changes the shape of the output waveform.

PRE-LAB: **Q point = (VCE, IC) = (2.5 V, 1mA)**

You should be familiar with the DC analysis techniques needed to find the Q-point of a common-emitter circuit. You should understand how varying the ratio of the two base resistors, R_1 and R_2 , affect the base voltage and motion along the load line. Pre-Lab simulation can be divided in two parts: (i) DC sweep and (ii) Transient analysis.

- (i) For the first part, you should perform DC sweep on the circuit shown in Figure 6.2 to find R_2 for the Q-point (~~$V_{CE}=5V$, $I_C=1mA$~~). Also, you should measure V_{CEQ} , I_{CQ} ($\approx I_{EQ}$) for $R_2 = 1k\Omega$, $5k\Omega$, $15k\Omega$, and $25k\Omega$ on the same circuit.
- (ii) In the second part of the simulation, transient analyses should be performed in B2 Spice on the circuit shown in Figure 6.1 for $R_2 = 1k\Omega$, $5k\Omega$, $15k\Omega$, and $25k\Omega$. Record the values of V_S and V_O . Assume $V_S = 0.02V_{P-P}$ at 1kHz, the transistor is a 2N3904, and all capacitors are greater than or equal to $10\mu F$. For the second part, you may use source resistance $R_S (=50\Omega)$.

NOTE: Check the DEVICE IC box in transient setup. Preferred values for transient setup:

Start Time =3.00

Stop Time =3.01

Linearize Step=100u

Step Ceiling= 10u

EQUIPMENT:

- 2 NPN transistors with different β 's, 2N3904, 2N2222, or equivalent. The transistors may have the same part number, as long as they have different β 's.
- NI ELVIS II Workstation
- 1 Curve tracer
- 1 Decade resistance box
- Resistors and capacitors as noted in the schematic.

- Linear graph paper (or plotting software like Excel or MatLab)

INTRODUCTION:

Figure 6.1 shows a schematic diagram of a BJT common-emitter (CE) amplifier circuit. BJT transistor amplifiers are frequently used in the common-emitter configuration (CE), since this design gives both a high current gain (A_i) and a high voltage gain (A_v). This experiment explores the dc and ac characteristics of the common-emitter circuit and how changing the Q-point affects circuit performance.

- You will study the movement of the Q-point on the dc load line as R_2 is varied.
- You will see how input signals can lead to distorted output signals as the Q-point changes and/or as the magnitude of the input signal changes.
- You will learn how to create a Q-point that is almost independent of the β of the transistor that is being used.

The collector current (I_C) in a BJT circuit depends on the β of the transistor as well as the transistor's temperature and the other circuit elements. Good amplifier design requires choosing biasing resistors such that the quiescent (DC) collector current remains constant, regardless of whether a transistor with a different β is being used. Selecting the proper biasing network and resistors keeps the collector current relatively constant. A key element that stabilizes the circuit to changes in the transistor β is the emitter resistor, R_E .

Using a bias network with an emitter resistor is a good way to keep I_C relatively constant if the β or temperature changes. Any increase in I_C will cause the feedback voltage drop, V_{FB} , across R_E , to increase, thus lowering the base-emitter voltage, V_{BE} , and causing I_B to decrease. This is a negative feedback effect. Therefore, any changes in β will not cause I_C to change significantly, since I_B will scale with $1/\beta$.

However, using an emitter resistor lowers the ac voltage gain of the circuit, since the ac component, i_c , must flow through R_E to ground. Therefore, R_E impedes i_c . The negative feedback voltage, V_{FB} across R_E "kills" the ac gain. To get around this undesired ac response and still maintain the excellent control of I_{CQ} that R_E provides, a capacitor is placed across R_E in order to short the ac current, i_c , around R_E to ground. This capacitor, C_E , is called an emitter bypass capacitor and is generally large, 10 μ F or greater. This capacitor allows most of the ac current to flow around R_E directly to ground. Note that the dc current, I_{CQ} , still flows through R_E , since the capacitor acts as an open circuit to dc, and the stabilizing effect of R_E is maintained. If R_E is physically removed from the circuit, the negative feedback effect on the gain is taken away. You will see in the next lab, Experiment #7, that without C_E , the voltage gain will be determined by the ratio of R_L and R_E , where R_L is the total ac load resistance, which the amplifier "sees". **An important principle of operation of feedback amplifiers is that the gain will be only dependent on resistor elements, not the active elements, if proper feedback is provided.**

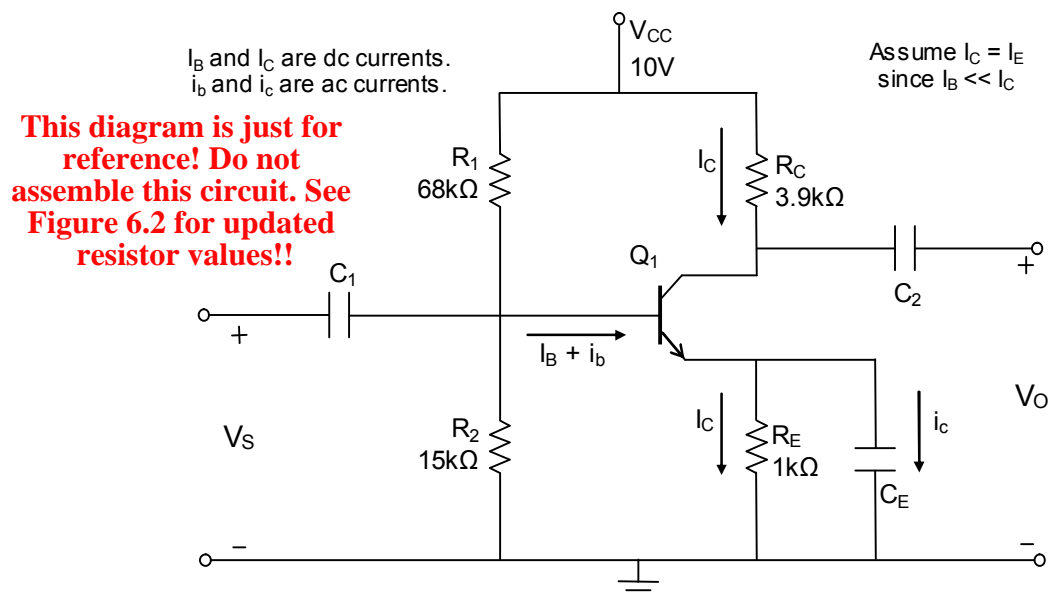


Figure 6.1: Common-emitter circuit with emitter bypass capacitor

EXPERIMENT:**PART I: Bias Measurements: The importance of the emitter resistor**

1. Use the curve tracer function keys to measure the β of both transistors at a collector current of about 1.0 mA and V_{CE} of about 5 V, i.e. Q-point $\approx (5V, 1\text{ mA})$. Please refer to Experiment 5, Part I step 1 for measuring β (h_{fe}) using the curve tracer. Make a hard copy of the transistor output characteristics of one of the transistors. **This is your primary transistor (use this transistor in Part II) for this experiment.** The other transistor is your secondary transistor. Each student will need at least one copy of the characteristics for the lab report. When copying the output characteristic, use scales of $I_{C,MAX} = 2\text{mA}$ and $V_{CE,MAX} = 10V$. Use 5 steps of base current. Draw the DC load line on the copied output characteristics. Remember that the DC load line includes the emitter resistor. The AC load line has this resistor bypassed by the emitter capacitor.
2. Set up the circuit using the secondary transistor, as shown in Figure 6.2 without connecting the capacitors or the source voltage V_S . Use a decade box for R_2 and adjust it until V_{CE} is about 2.5V. R_2 will probably be close to 16 kΩ. Measure and record V_{E0} , or the emitter-to-ground voltage using the DMM. From this voltage measurement and the value of R_E , find $I_E \approx I_C$. Also measure and record V_{CE} .
3. Replace the secondary transistor with the primary transistor and measure V_{CE} and I_E . Compare your result with the secondary transistor for same R_2 , you found in step 2.

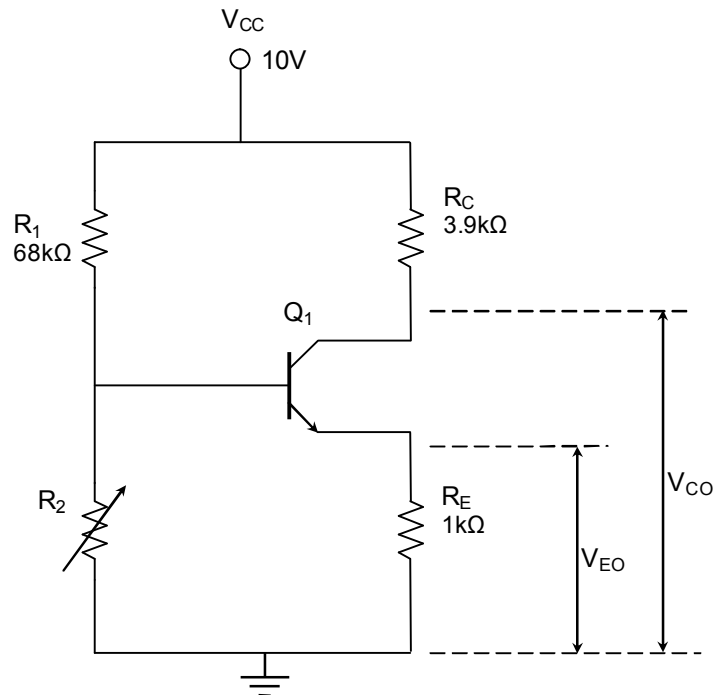


Figure 6.2: Common-emitter circuit

PART II: Q-point versus the value of the bias resistors

1. Keep the primary transistor in the circuit shown in Figure 6.2. Vary R_2 through the following values and measure and record the resulting Q-point (V_{CE} and I_C): 1kΩ, 2kΩ, 5kΩ, 8kΩ, 10kΩ, 15kΩ, 18kΩ, 20kΩ, 25kΩ, 30kΩ.

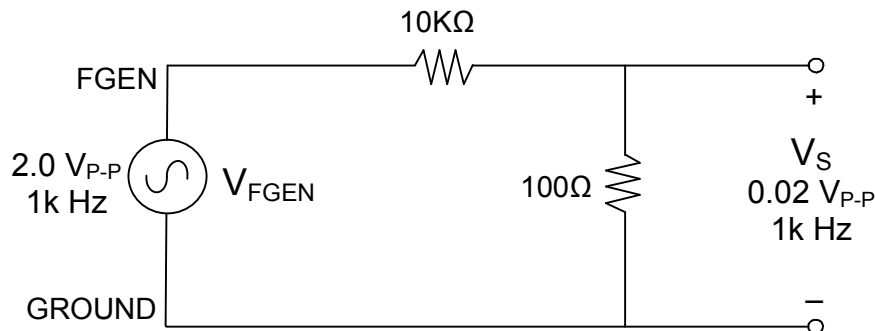
Table 6.1: Q-points measurement

R_2 (kΩ)	V_{C0}	V_{E0}	V_{CE}	I_E
1				
2				
5				
8				
15				
18				
20				
25				
30				

2. Plot the ten Q-points on your output characteristics.
3. In order to make the input signal, V_s , small enough to prevent distortion of the amplifier output, use the voltage divider circuit shown in Figure 6.3, where V_s is the voltage across

Note: $R_2 \sim 16k$ or $16.5k$ for desired Q point (2.5 V, 1mA)

the 100Ω resistor. The voltage divider circuit shown in the Figure 6.3 divides input voltage from FGGEN by 100:1. Thus, to get $V_S = 0.02V_{P-P}$, set FGGEN to provide $2.0V_{P-P}$ as input to the voltage divider. **You may find, it is not possible to see the correct amplitude of V_S on the oscilloscope**, but if you are applying $2.0V_{P-P}$, 1kHz, at the input of the voltage divider shown in Figure 6.3, you can be assured that $V_S = 0.02V_{P-P}$, 1kHz.

Figure 6.3: Voltage divider network to generate V_S

- Set up the circuit in Figure 6.4 with the capacitors (use $22\mu F$ or $47\mu F$) and voltage divider. To apply V_S to the input of the amplifier, set FGGEN to $2.0V_{P-P}$, 1kHz, as input to the voltage divider, as discussed in step 3. Make sure your input is right by connecting the oscilloscope's CH0 to the function generator. **THIS IS A VERY COMMON ERROR, RESULTING IN WRONG OR UNEXPECTED OUTPUT.** If your input voltage is too large, you will clip the top and/or bottom of your output voltage. **DO NOT MAKE GAIN MEASUREMENTS ON A CLIPPED OR DISTORTED OUTPUT WAVEFORM!!!**

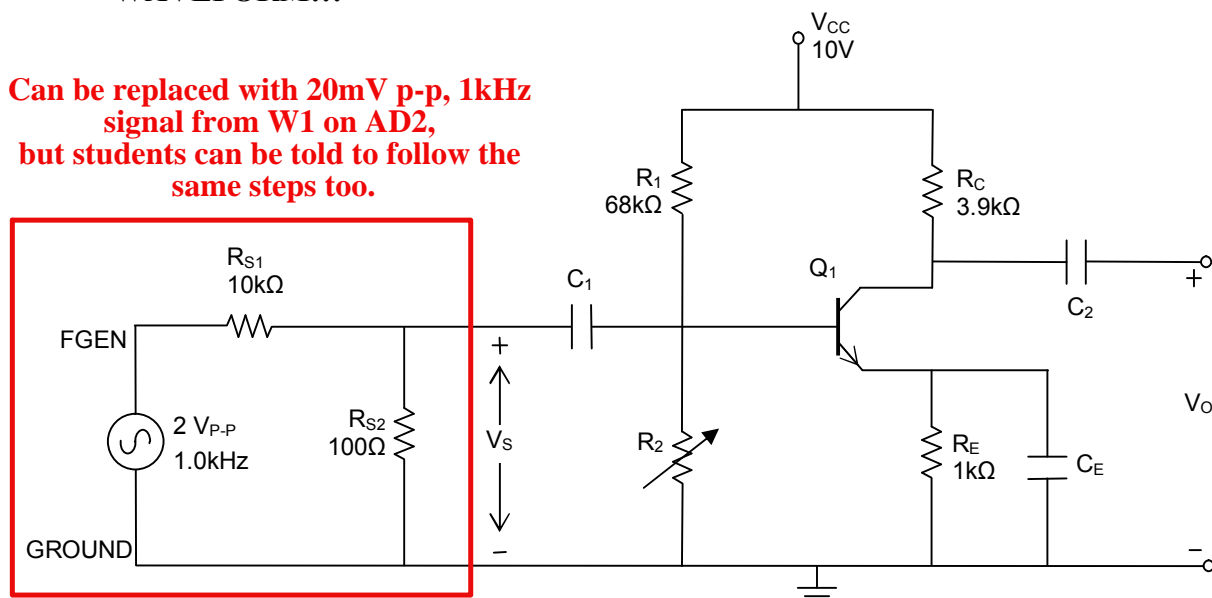


Figure 6.4: CE Amplifier with voltage divider circuit

- Use **SCOPE CH0 and CH1**, with coupling set to “AC”, to measure V_S and V_O . Because the voltage is small, V_S may appear too noisy to get an accurate measurement at

0.02V_{P-P}; that is why we rely on the voltage divider to ensure an accurate setting. Monitor the output waveform shape during all of the experiment. **Ensure that the negative leads of both channels are connected to ground during the whole experiment.**

Table 6.2: Amplified signal measurement

R ₂ (kΩ)	V _O	Shape of the waveform (e.g. sinusoidal or distorted)
1		
2		
5		
8		
15		
18		
20		
25		
30		

NOTE: DO NOT MAKE GAIN MEASUREMENTS ON A CLIPPED OR DISTORTED OUTPUT WAVEFORM! YOU WILL NEED TO RECORD WHAT RESISTOR VALUES PRODUCE DISTORTED OUTPUTS.

6. Measure the input and output voltage and the shape of the output voltage as you vary R₂ from 1 kΩ to 30 kΩ, similar to step 1, and record on Table 6.2. Note all instances (values of R₂) of output waveform signal distortion. For each value of R₂, make an accurate plot of the input and output waveforms for a sinusoidal input. Note the distortion of your output signals. You will be asked to determine how variation in R₂ moved you along the load line.
7. You will now see how the amplifier responds to changes in the magnitude of the input signal. With R₂ set at 5 kΩ slowly increase the magnitude of the input voltage and note the changes in output waveform shape. Repeat this part of the experiment with R₂ = 15kΩ, 20kΩ. Determine how the output signal becomes distorted as the input signal increases for all three values of resistor R₂. Use the changes in the output waveform shapes to verify how the Q-point changes as R₂ changes. Be sure your output waveform shapes agree with your position on the load line. Remember that the CE amplifier has a 180° voltage phase reversal. That is, increases in the base current or base voltage result in increases in the collector current and decreases in the collector voltage.

LAB REPORT:

1. In Part I, compare the measured ΔI_C with the theoretical ΔI_C when you changed transistors.
2. How did the Q-point change with R₂? How did V_O change with R₂?

3. Compare measured results with B2 Spice simulation. Include B2 Spice output plots.
4. What range of R_2 drove the transistor into saturation? ... into cutoff?

ECE 3110 EXPERIMENT 6 - CHECK LIST

1. Bias Measurements

- a. Measure and record the β of the two transistors at a Q-point = 5V, 1mA.
- b. Set-up the circuit without V_S connected.
- c. Measure and record V_{CE} and $I_E \approx I_C$
- d. Repeat 1c with the second transistor.

2. Q-point vs. Bias Resistors

- a. Mark the Q-point and draw the DC load line on the output characteristics of the primary transistor.
- b. Apply $V_S = 0.02V_{P-P}$ at 1 kHz.
- c. Use the oscilloscope to measure V_S (if possible) and V_O . Check for clipping of output on scope. **Note all instances of clipped voltages.**
- d. Repeat 2c, varying R_2 from 1k Ω to 30k Ω .
- e. Sketch the V_S and V_O for $R_2 = 2\text{ k}\Omega$, 5 k Ω , 10 k Ω , 15 k Ω , and 20 k Ω . Note how the output waveform becomes distorted.

ELECTRONICS I
ECE 3110

Experiment #7 – BJT Common-Emitter Circuit Voltage Gain

PURPOSE:

This experiment is designed to illustrate how the voltage gain of a CE bipolar transistor circuit can be changed by changing the bias of the circuit and by changing the values of the resistors in the circuit.

- The role of the emitter bypass capacitor C_E will be illustrated.
- The base resistors determine the Q-point.
- The Q-point determines the relationship between the input wave shape and the output wave shape.
- If the Q-point is not near the middle of the load line, it is possible for the output wave shape to be distorted.
- If the magnitude of the input signal is too large, it is possible for the output wave shape to be distorted and/or clipped.

The gain of the CE amplifier is a function of the transistor used in the circuit, the values of the resistors, and the presence an emitter bypass capacitor. The CE amplifier with capacitor-bypassed emitter resistance shows good gain stability to variations in transistor β . This principle will be explored.

PRE-LAB:

- You should be familiar with the dc analysis techniques needed to find the Q-point of a common-emitter circuit.
- You should understand how varying the ratio of the two base resistors, R_1 and R_2 , affect the base voltage and motion along the load line.
- You should know how R_E and R_C affect the Q-point. = (2.5V, 1mA)
- Perform B2 Spice simulations of the circuit shown in Figure 7.1, using the resistor values given and with extreme values of the resistors R_C (1k Ω & 20k Ω) and R_E (200 Ω and 10k Ω) , both with and without capacitor C_E . You should end up with ten different simulation graphs with the following resistor combinations:

- | | | |
|-----------------------|-------------------|------------------------|
| 1. $R_C = 3.9k\Omega$ | $R_E = 1k\Omega$ | with and without C_E |
| 2. $R_C = 3.9k\Omega$ | $R_E = 200\Omega$ | with and without C_E |
| 3. $R_C = 3.9k\Omega$ | $R_E = 10k\Omega$ | with and without C_E |
| 4. $R_C = 1k\Omega$ | $R_E = 1k\Omega$ | with and without C_E |
| 5. $R_C = 20k\Omega$ | $R_E = 1k\Omega$ | with and without C_E |

Assume sinusoidal input $V_S = 0.02V_{p-p}$ at 1 kHz, the transistor is a 2N3904, and all capacitors are 22 μ F or greater.

NOTE: Check the DEVICE IC box in transient setup. Preferred values for transient setup:

Start Time =3.00

Stop Time =3.01

Linearize Step=100u

Step Ceiling= 10u

EQUIPMENT:

- 1 NPN transistor, either 2N3904, 2N2222, or equivalent
- 1 Curve tracer
- 1 Decade resistance box
- NI-ELVIS workstation
- Resistors and capacitors as noted in the schematic.
- Linear graph paper or graphing software (e.g., Excel or MatLab).

INTRODUCTION:

Figure 7.1 shows a schematic diagram of a BJT common-emitter (CE) amplifier circuit. BJT transistor amplifiers are frequently used in the common-emitter configuration (CE), since this design gives both a high current gain (A_I), and a high voltage gain (A_V). Experiment #6 explored the dc and ac characteristics of the common-emitter circuit and how changing the Q-point affected circuit performance. You will study how the voltage gain of the circuit is affected by changing the values of the emitter and collector resistors. You will see how input signals can lead to distorted output signals as the gain and Q-point change.

The collector current (I_C) in a BJT depends on the β of the transistor, the transistor temperature, and the circuit elements. **Good amplifier design requires choosing biasing resistors such that the quiescent (DC) collector current remains constant, regardless of transistor β or temperature.** Selecting the proper biasing network and resistors keeps the collector current and the voltage gain relatively constant. The key elements that stabilize the circuit to changes in the transistor β are the emitter resistor, R_E , and the values of the bias resistors, R_1 and R_2 . To keep the high gain of the CE circuit, the emitter resistor is usually bypassed by a large capacitor. You will learn in ECE 321 that this capacitor affects the frequency response of the CE circuit, thus, the improvement in gain stability is obtained at a cost of reduced frequency response.

Using an emitter resistor lowers the ac voltage gain of the circuit, since the ac component, i_c , must flow through R_E to ground. Therefore, R_E impedes i_c ; the negative feedback voltage across R_E (V_{FB}) "kills" the ac gain. To get around this undesired ac response and still maintain the excellent control of I_C that R_E provides, a capacitor is placed across R_E to short the ac current, i_c , around R_E to ground. This capacitor C_E is generally large, 10 μ F or greater, and allows most of the ac current through R_E to flow to ground. Note that the dc current, I_C , still flows through R_E since the capacitor acts as an open circuit to dc and the stabilizing effect of R_E is maintained. You will see in this lab that without C_E the voltage gain is determined by the ratio of R_L and R_E , where R_L is the total ac load resistance, which the amplifier "sees".

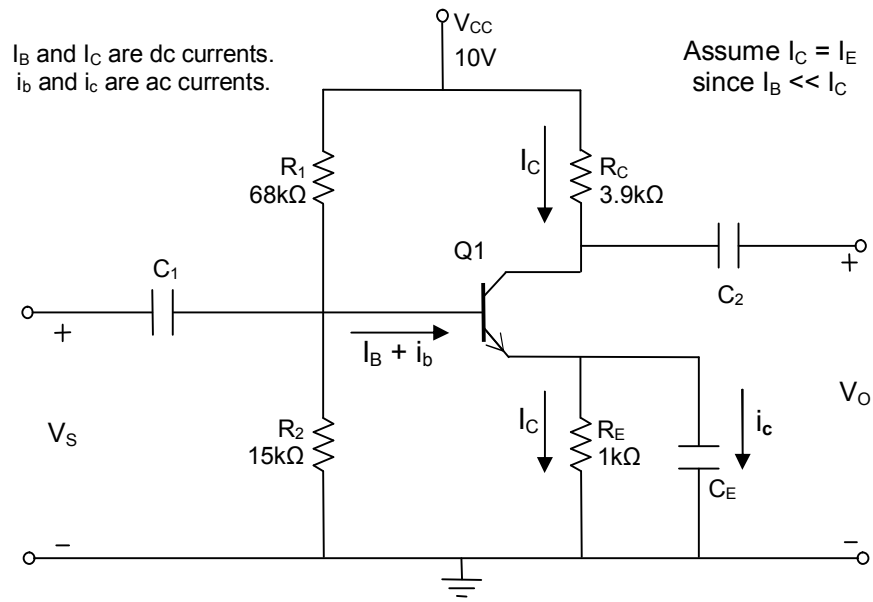


Figure 7.1: Common-emitter circuit with emitter bypass capacitor

EXPERIMENT:

PART I: Establishing the proper Q-point

(2.5V, 1mA)

1. Use the curve tracer and measure the β and V_{BE} of the transistor at a collector current of about 1.0mA and V_{CE} of about 5 V, i.e. Q-point = ~~(5 V, 1 mA)~~. Set $I_{MAX} = 20$ ma, $V_{MAX} = 10V$ to obtain 5 steps on the curve tracer. Make a copy of the transistor characteristics. You can use either the 2N3904 or 2N2222 transistor for this part of the experiment.
2. Set up the circuit as shown in Figure 7.2 with $V_{CC} = 10.0V$. Use the DMM to measure V_{C0} and V_{E0} . You can calculate $I_E (\approx I_C)$ using V_{E0} and R_E . Then, obtain the Q-point (V_{CE} and I_C). Draw your load line and Q-point on your transistor characteristic.

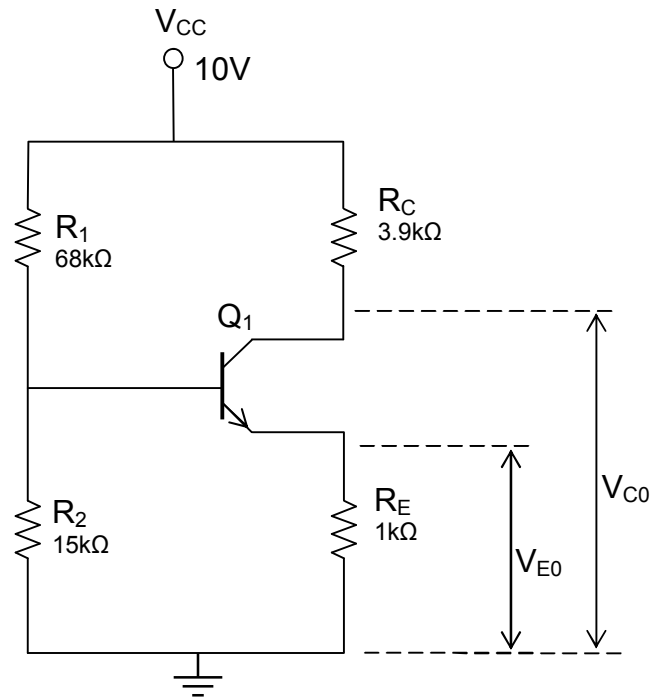


Figure 7.2: Common-emitter circuit for Q-point measurement.

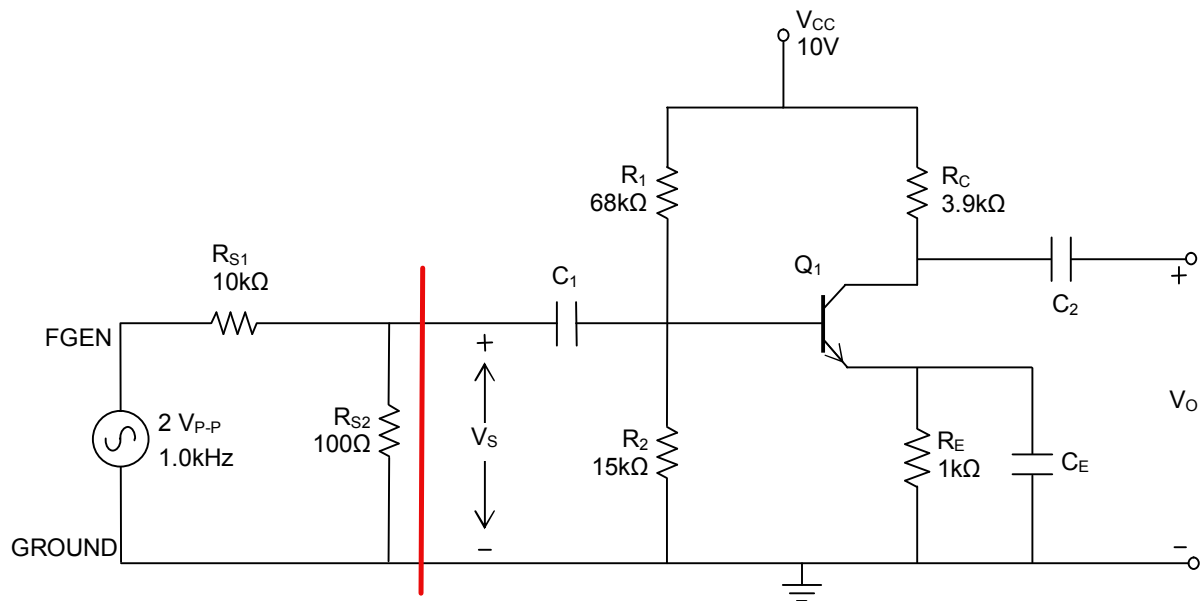


Figure 7.3: The common emitter circuit of Figure 7.2 with added capacitors and an input signal V_S provided by the function generator and the voltage divider, R_{S1} and R_{S2} .

PART II: AC Measurements

DO NOT MAKE VOLTAGE GAIN MEASUREMENTS ON A CLIPPED OR DISTORTED OUTPUT WAVEFORM!! IF YOUR OUTPUT WAVEFORM BECOMES DISTORTED, NOTE THIS CONDITION ON YOUR MEASUREMENTS AND ADJUST THE MAGNITUDE OF THE INPUT SIGNAL AND/OR RESISTORS TO OBTAIN AN UNDISTORTED OUTPUT.

1. Voltage gain versus the Collector Resistor R_C , with emitter bypass capacitor C_E

- Build the circuit shown in Figure 7.3 for ac measurements by adding the capacitors (use $22\mu\text{F}$ or $47\mu\text{F}$) and sinusoidal input of $2V_{p-p}$ 1kHz applied to the voltage divider circuit, R_{S1} and R_{S2} . Measure V_O and V_S with the oscilloscope. **It is possible that you will not be able to measure the exact value of V_S (i.e. 20mV) due to noise. Therefore, you should make sure that your FGGEN output and R_{S1} and R_{S2} are correct as shown in Figure 7.3.** If there is no distortion in the output wave shape, continue with the experiment. If the output wave shape is distorted, either reduce the magnitude of the input signal or adjust your Q-point. **Use SCOPE CH0 and CH1, and set the coupling of both channels to “AC”. Enable “Autoscale” to improve accuracy.**
- Replace R_C with the resistance decade box and set the resistance decade box to $3.9k\Omega$.
- Use the oscilloscope to measure V_O ; it should have same value as in step a. Leave the oscilloscope connected to V_O throughout this experiment and monitor the output wave shape.
- Measure V_O while R_C is varied from $1k\Omega$ to $20k\Omega$ using the resistance decade box. During this measurement the shape of V_O and its magnitude may change drastically for some values of R_C . Complete Table 7.1 with values of V_O . **If, at any time, your output wave shape becomes seriously distorted, note this on your table and do not take V_O measurement.** Also, sketch the output wave shapes.
- Determine the voltage gain, $A_V = V_O/V_S$, as a function of R_C and plot your results.

Table 7.1: V_O vs R_C

$R_C (\Omega)$	With C_E			Without C_E		
	V_{O-PP}	Output waveform shape	$A_V = \frac{V_{O-PP}}{V_{S-PP}}$	V_{O-PP}	Output waveform shape	$A_V = \frac{V_{O-PP}}{V_{S-PP}}$
1k						
3k						
6k						
9k						
15k						
20k						

2. Voltage gain versus the value of the Collector Resistor R_C , without C_E

- Remove the emitter bypass capacitor, C_E .
- Repeat all of Part II, section 1, and record your measurements in Table 7.1 columns for “Without C_E ”. If, at any time, your output wave shape becomes seriously distorted, note this on your plots and record the output wave shapes. For values of R_C that are $2k\Omega$ less than the value that produced distortion, increase the magnitude of the input signal until distortion appears in the output signal. Note the magnitude of the input signal that causes significant distortion in the output signal.

3. Voltage gain versus the value of the Emitter Resistor with C_E in place

- Set $R_C = 3.9 k\Omega$ using a fixed resistor instead of the resistance decade box. Connect the emitter bypass capacitor C_E as shown Figure 7.3.
- Replace R_E with the resistance decade box and set the resistance decade box to $1k\Omega$.
- With $R_C = 3.9 k\Omega$ and C_E connected, measure V_O and V_S for R_E values from 200Ω and $10k\Omega$. Record your results in Table 7.2. **If at any time your output wave shape becomes seriously distorted, note this on your table and do not take V_O measurements. Sketch the waveforms.**
- Determine the voltage gain, $A_V = V_O/V_S$, for all values of R_E and plot your results. Take extra data points when the gain is changing rapidly or when the output wave shape is becoming distorted.
- Record the values of emitter resistor that lead to significant output waveshape distortion.

Table 7.2: V_O vs R_C

$R_E (\Omega)$	With C_E			Without C_E		
	V_{O-PP}	Output waveform shape	$A_V = \frac{V_{O-PP}}{V_{S-PP}}$	V_{O-PP}	Output waveform shape	$A_V = \frac{V_{O-PP}}{V_{S-PP}}$
200						
500						
1k						
3k						
6k						
8k						
10k						

4. Voltage gain versus the value of the Emitter Resistor, without C_E

- Remove the emitter bypass capacitor, C_E .
- Repeat all of Part II, section 3, without C_E and record results in Table 7.2 columns for “Without C_E ”.

5. Frequency Response.

Set R_E to $1k\Omega$ and connect C_E as shown in Figure 7.3. Vary the input signal frequency from 10 Hz to 50k Hz and use the oscilloscope to see how the magnitude of the output signal varies. Record your result in Table 7.3. You will study frequency response in ECE 321.

Table 7.3: Frequency response of V_{O-PP} and voltage gain A_V

Frequency (Hz)	10	100	1k	10k	25k	50k
V_{O-PP}						
A_V						

6. **Save this transistor.** Keep this transistor safe and use its β and V_{BE} for design calculations in the pre-lab for Experiments 8 and 9. Use this transistor in Experiments 8 and 9.

LAB REPORT:

- Make four plots of the voltage gains of your circuit as a function of the values of R_E and R_C with and without the emitter bypass capacitor.
- Compare your experimental data with the simulations of this circuit. How accurate were the circuit simulations? How did the voltage gains change as R_E and R_C were changed.
- Comment on the variation in voltage gain with and without the bypass capacitor in the circuit. What sacrifices are made to ensure a stable voltage gain when the capacitor was in the circuit or was out of the circuit?
- In your lab report, you should write a conclusion to this experiment explaining how changing R_C and R_E changed the gains of your circuit, with and without the emitter bypass capacitor.

ECE 3110 EXPERIMENT 7 - CHECK LIST

- Establish Q-point.
 - Measure β at Q-point = 5V, 1mA. Copy transistor characteristics, mark Q-point, and draw load line.
 - Set-up circuit without V_S and with $R_C = 3.9k\Omega$ and $R_E = 1k\Omega$. Measure and record I_C and V_{CE} .
- Voltage Gain, A_V . Set-up circuit in Figure 7.3.
 - A_V vs. R_C , with C_E
 - Monitor V_O on oscilloscope.
 - Vary R_C from $1k\Omega$ to $20k\Omega$.
 - Measure V_O and V_S on the oscilloscope for each R_C . Do not record A_V for distorted waves, but note which values of R_C cause distortion.
 - Put V_O , V_S , and A_V in a table.
 - Plot A_V vs. R_C . Specify regions of distortion as cutoff or saturation.

- B. Repeat all of Part II, A. with C_E removed.
- C. A_V vs. R_E , with C_E
 - i. Monitor V_O on oscilloscope.
 - ii. Vary R_E from $200\ \Omega$ to $10\ \text{k}\Omega$.
 - iii. Measure V_O and V_S for each R_E . Do not record A_V for distorted waves, but note which values of R_E cause distortion.
 - a. Put V_O , V_S , and A_V in a table.
 - iv. Plot A_V vs. R_E . Specify regions of distortion as cutoff or saturation.
- D. Repeat all of Part II, C, with C_E removed.

**ELECTRONICS I
ECE 3110**

Experiment #8 – BJT Common-Emitter Design I

PURPOSE:

The purpose of this laboratory is to enable you to apply the knowledge you have gained in class and in the lab concerning the BJT common-emitter (CE) amplifiers to design a BJT CE amplifier to meet certain design specifications. You will determine how the voltage gain is affected by changes in the output resistance. You will learn how to measure the input and output impedances of CE circuits.

PRE-LAB:

1. You need to know how to obtain a stable Q-point for a CE amplifier. You need to know how to calculate voltage gain for a CE amplifier. **You will need to design your amplifier before coming to the lab. If you have not designed your amplifiers before coming to the lab, you will not be able to complete this experiment in the time allotted.**
2. **You will need to know the β of your transistor and the value of V_{BE} before you begin the design of your circuit. You should have measured this in Experiment 7. (TO INSTRUCTORS, ENSURE THAT THIS TASK IS ACCOMPLISHED IN THE PREVIOUS LAB).**
3. Design the CE amplifier shown in Figure 8.1 to drive a $4.7\text{k}\Omega$ load using an npn silicon transistor and a power supply voltage of $V_{CC} = 10\text{ V}$ and $R_L = R_C$. The voltage gain of your circuit is to be $A_V = -10$. You should have already used the curve tracer to determine the β and V_{BE} of the transistor you have selected. Your design should include the specification of R_1 , R_2 , R_C , R_E , R_O , V_{CEQ} , I_{CQ} , h_{ie} , and h_{ib} . **In this part of the experiment, C_E is not used.** You must design your amplifier for maximum symmetrical output swing. **See Design Procedure following this section for the equations needed for your design.** Assume that all coupling capacitors are large enough to pass your signal at 1kHz (i.e. $22\mu\text{F}$ or greater). Assume $R_C = R_L$ for your first design.
4. Simulate your circuit design using B2 Spice transient analysis, as in prior experiments. Note that you may need to modify some resistor values so that B2 Spice gives the correct output and gain. Measure the voltage gain. Let $V_S = 0.02V_{P-P}$.
5. Repeat the measurements of the voltage gain for $R_L = 0.1R_C$, and $R_L = 10R_C$, keeping $R_C = 4.7\text{k}\Omega$, and tabulate your results.
6. **Remember to use standard resistor values in your final design. A table of standard resistor values is found in this lab manual.**

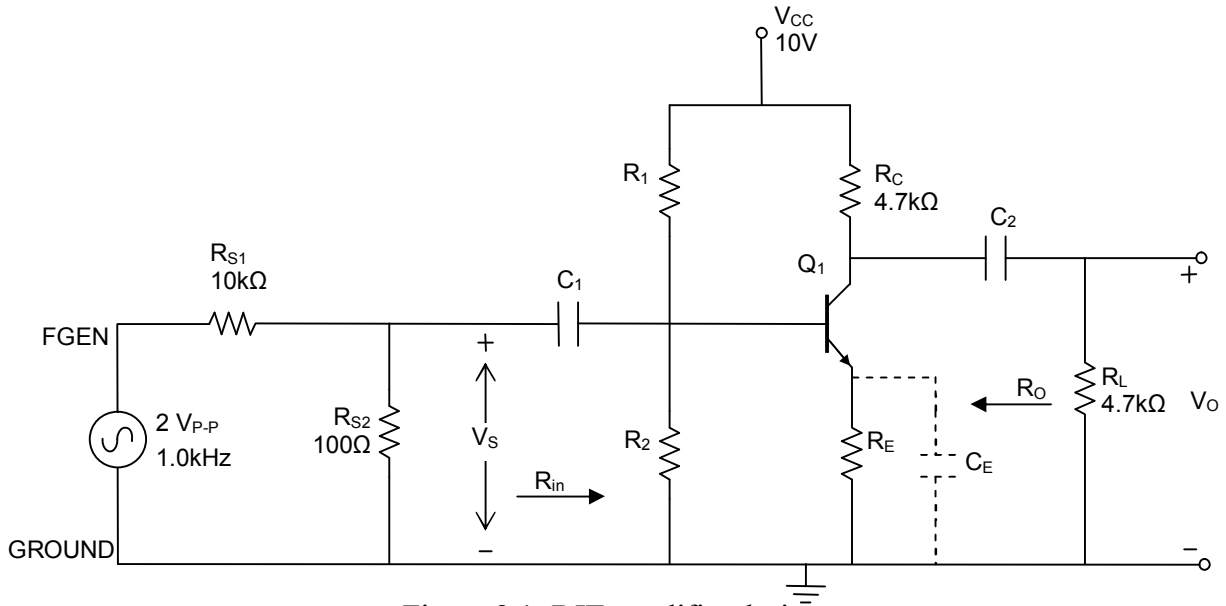


Figure 8.1: BJT amplifier design

DESIGN PROCEDURE: When an emitter bypass capacitor C_E is not used

- Estimate R_E from $A_V = -(R_L \parallel R_C) / (h_{ib} + R_E)$ for the amplifier in which emitter resistor R_E is not bypassed by a capacitor C_E .
Recognizing that h_{ib} is usually much less than R_E , we can assume $(h_{ib} + R_E) \approx R_E$ for the non-bypassed amplifier.

- Calculate R_{ac} and R_{dc}

$$R_{ac} = (R_C \parallel R_L) + R_E \quad \text{if } R_E \text{ is not bypassed.}$$

$$R_{dc} = R_C + R_E \quad \text{whether } R_E \text{ is bypassed or not.}$$

- Find I_{CQ} and V_{CEQ}

$$I_{CQ} = \frac{V_{CC}}{R_{ac} + R_{dc}}$$

$$V_{CEQ} = I_{CQ} \cdot R_{ac}$$

- Find $h_{ie} (= r_\pi)$ and h_{ib}

$$h_{ie} = \frac{V_T}{I_{BQ}} = \beta \frac{V_T}{I_{CQ}} = \beta h_{ib} = r_\pi$$

where V_T (thermal voltage) = 0.026V.

- Subtract the value of h_{ib} from the value of R_E found above to obtain the actual value of R_E that is needed in the non-bypassed amplifier. Choose the nearest standard resistor to match R_E .
- Note R_{Th} and V_{Th} are the Thévenin equivalent input resistance and voltage:

$$R_{Th} = (R_1 \parallel R_2) = \left(\frac{R_1 R_2}{R_1 + R_2} \right) \quad \text{and} \quad V_{Th} = \left(\frac{R_2}{R_1 + R_2} \right) V_{CC}$$

But we don't know R_1 and R_2 yet.

Note that the effective base resistance and effective base voltage are the same as the Thévenin values: $R_B = R_{Th}$ and $V_{BB} = V_{Th}$

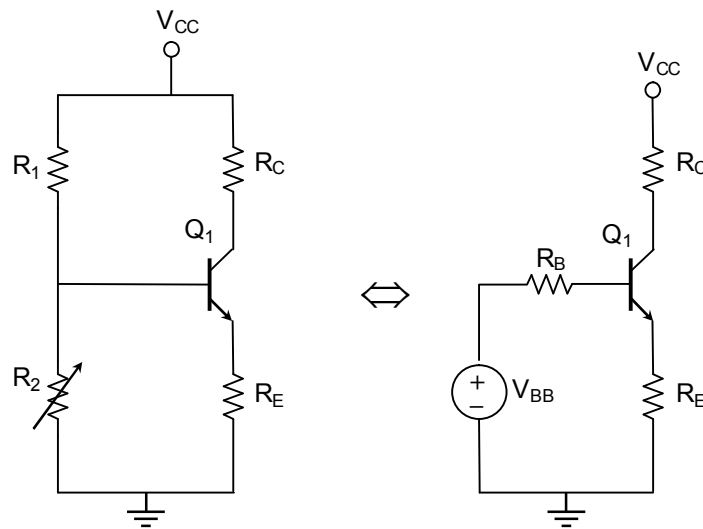


Figure 8.2: Relationship between the Thévenin-equivalent base resistance R_B and the actual circuit built with resistors R_1 and R_2 . Figure is only to illustrate relationships for calculations.

- For bias stability, choose

$$R_B = R_{Th} = 0.1 \beta R_E$$

and from Kirchhoff's voltage law, summing around the B–E loop, find V_{BB} :

$$V_{BB} = V_{Th} = V_{BE} + I_B R_B + I_C R_E$$

- Find R_1 and R_2 using the above resistor and voltage values:

$$R_2 = \frac{R_B}{\left(1 - \frac{V_{BB}}{V_{CC}} \right)} = \frac{R_{Th}}{\left(1 - \frac{V_{Th}}{V_{CC}} \right)}$$

$$R_1 = R_B \cdot \frac{V_{CC}}{V_{BB}} = R_{Th} \cdot \frac{V_{CC}}{V_{Th}}$$

EQUIPMENT:

Use equipment and components as required by your design and the design problem statement.

EXPERIMENT:

1. DC Measurements

- a. Use the curve tracer to measure the β and V_{BE} of your transistor, if you have not already done so.
- b. **Build the circuit in Figure 8.1 but without R_{S1} , R_{S2} , R_L , the capacitors, and function generator**, as was illustrated in Figure 7.2. **Use the resistance values you determined in your pre-lab and set $R_C = 4.7 \text{ K}\Omega$** . You should use the nearest standard resistance values for the resistors R_1 , R_2 , and R_E , even if the design values are non-standard. Attempt to build your circuit using standard values of resistors before you resort to using the decade resistance boxes.
- c. Measure V_{C0} and V_{E0} of your amplifier using the DMM. Then, calculate V_{CEQ} and I_{CQ} (Q-point) using V_{C0} and V_{E0} . Compare with your design values. If V_{CEQ} is not close to calculated V_{CEQ} , please read following note. Tabulate your results and show them to the lab instructor.

NOTE: If the value of V_{CEQ} is not within 5% of your calculated value, replace R_2 with a resistance decade box and adjust the resistance value until V_{CEQ} is equal to or close to your calculated value. If you used $V_{BE} = 0.7\text{V}$ in your calculations, you may find that your Q-point is very close to the saturation region and you will have to lower the value of R_2 to obtain a Q-point nearer the middle of the load line. If you use a decade resistance box for R_2 , note how sensitive the value of V_{CEQ} is to small variations in R_2 .

2. AC Analysis

- a. Connect R_{S1} , R_{S2} , R_L , the capacitors, and the function generator to your circuit, as in Figure 8.1. The coupling capacitors need to be large enough to pass an ac signal of 1 KHz (22 μF or greater). In Figure 8.1, the voltage divider using R_{S1} and R_{S2} is an easy way to keep the input voltage swing low enough to keep the transistor from being driven into either saturation or cut-off. After you have built the circuit, check your input and output wave shapes. Take appropriate action if your output wave shape is distorted from a sine wave.
- b. Measure V_O keeping the oscilloscope coupling set to “AC”. You might not be able to measure V_S (i.e., 20mV) because of noise, so you should make sure that the FGEN output is 2V_{P-P}. From V_S and the measured value of V_O , **determine A_V** and record the result in Table 8.1. How does this value of A_V compare with the one specified in the design specification?

Table 8.1: Voltage gain measurements

R_L	V_{O-PP}	A_V with C_E	A_V without C_E
$0.1 R_C$			
R_C			
$10 R_C$			

- c. Make an accurate sketch of the input and output waveforms.
- d. Place a large value emitter-bypass capacitor across R_E and repeat steps b and c. Comment on the effect of C_E on the gain of the amplifier. Compare the new value of A_V with that given by

$$A_V = -(R_L \parallel R_C) / h_{ib} = -(\beta / r_\pi) \cdot (R_L \parallel R_C) = -(R_L \parallel R_C) / h_{ib}.$$

When R_E is not bypassed the voltage gain is given by

$$A_V = -(R_L \parallel R_C) / (R_E + h_{ib}) \approx -(R_L \parallel R_C) / R_E.$$

What does this result tell you about the effect of C_E ?

- e. Repeat steps 'b' through 'd' for $R_L = 0.1R_C$ and for $R_L = 10R_C$. How does the gain vary with the R_L / R_C ratio? Compare your results with the simulations.
- f. Compare your amplifier designs with your B2 Spice simulations.

Extra credit portion:

It is possible to measure the output resistance. If you have time, measure the output impedance for the case where $R_L = R_C$.

Output Impedance: Remove R_L and C_E from the circuit. Measure V_{O-PP} . Connect a resistance decade box across the output terminals, with the resistance set at 10 k Ω . Adjust the decade box until the output voltage swing falls to half of its value before the decade box was connected across the output. The value of the resistance on the decade box will then be approximately equal to the output impedance R_O .

NOTE: CONSULT WITH YOUR INSTRUCTOR ON THE PRE-LAB FOR THE NEXT LAB BEFORE YOU LEAVE.

LAB REPORT:

- Write a neat Engineering report for this laboratory. Your report should include all of the design calculations in your pre-lab. You should compare in a table your lab results with your design specifications and with your B2 Spice simulations.
- Write a conclusion to this lab. Explain any differences between your calculated and measured values. Comment on the sensitivity of your design to the choice of Q-point. Comment on the effect of small variations in V_{BE} on your design and on your Q-point. Did your gains with and without C_E agree with what you expected?

ECE 3110 EXPERIMENT 8 - CHECK LIST

1. Measure β and V_{BE} .
2. Build circuit in Figure 8.1 using resistors calculated in your design.
Keep $V_{in} < 20mV_{p-p}$. All Capacitors 22 μ F or greater.
3. Measure V_{CE} and I_C . Compare with your design Q-point.
4. Measure V_{in} and V_o . Calculate A_v . Compare actual A_v with design spec.
5. Sketch V_{in} and V_o .
6. Insert emitter bypass capacitor. Measure V_{in} and V_o . Calculate A_v . Compare with
$$A_v = - (R_L \parallel R_C) / h_{ib} = - (\beta / r_{\pi}) \cdot (R_L \parallel R_C)$$
7. Repeat steps 2-6 for $R_L = 0.1 R_C$ and $10 R_C$. How does A_v vary with R_L/R_C ratio?
8. Optional (extra credit + 50 pts.)
9. Compare design with B2 Spice simulations.

**ELECTRONICS I
ECE 3110**

Experiment #9 – BJT Common-Emitter Amplifier Design II

PURPOSE:

The experiment is a follow-up to Experiment #6 and Experiment #7. It is intended 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, A_v , and output impedance, R_o .

THIS EXPERIMENT IS PERFORMED INDIVIDUALLY WITH A TIME LIMIT OF 1 HOUR. BUILD YOUR CIRCUIT IN THE LAB. DO NOT BRING A PREWIRED CIRCUIT TO THE LAB.

PRE-LAB:

1. Be familiar with BJT amplifier design.
2. Design a single stage, BJT CE amplifier with the specifications given below. Note that the signal source comprises the function generator and the voltage attenuator (see Figure 9.1). The output impedance (R_s) of the function generator is 50Ω . Your amplifier should also meet the following design specifications:
 - a. $A_v = -100$
 - b. $R_L = R_C = 4.7k\Omega$
 - c. Undistorted output signal with the input signal = $0.02 V_{P-P}$.

Use $R_C = R_L$ and $V_{CC} = 10 V$. Specify R_1 , R_2 , R_E , R_C , and R_L . When doing your design, calculate A_v , A_i , V_{O-PP} , V_{CEQ} , I_{CQ} , h_{ib} , and h_{ie} ($= r_\pi$). You will need to use an emitter bypass capacitor to get the desired voltage gain. Assume that all coupling and bypass capacitors are large enough to be considered ac short circuits at the design frequency of 1KHz. **A general design procedure** is given following this section. Tabulate your results. Use an input signal to the base of the transistor that is around $0.02V_{P-P}$.

NOTE: You should use same β and V_{BE} for your design calculations that you used in Experiment #8.

3. Simulate the circuit you designed using B2 Spice. You may use the transient setup you used in previous BJT amplifier experiments. However, the values of resistors given in Experiment #7 or Experiment #8 will not achieve a voltage gain of -100.

EQUIPMENT:

Use equipment and components as dictated by your design.

Design Procedure: When emitter resistor R_E is bypassed by capacitor C_E .

- Calculate R_{ac} , I_{CQ} and V_{CEQ}

$$R_{ac} = (R_C \parallel R_L)$$

At room temperature $V_T = 0.026V$. When the pn-junction is turned on, the ideality factor is about 1.15-1.20, and so $nV_T \approx 0.03V$. Thus, in the special case of a common-emitter amplifier designed for maximum symmetrical signal swing and for maximum power transfer, we have $R_C = R_L$. Therefore,

$$(R_C || R_L) = R_C/2 = R_L/2$$

$$|I_{CQ}| = \frac{nV_T |A_v|}{(R_C || R_L)} = \frac{2nV_T |A_v|}{R_C} = \frac{2nV_T |A_v|}{R_L} = \frac{0.06V |A_v|}{R_L}$$

$$V_{CEQ} = I_{CQ} R_{ac} = \frac{I_{CQ} R_C}{2} = \frac{I_{CQ} R_L}{2}$$

- Calculate h_{ie} ($=r_\pi$) and h_{ib}

$$h_{ie} = \frac{V_T}{I_{BQ}} = \beta \frac{V_T}{I_{CQ}} = \beta h_{ib} = r_\pi$$

where V_T (thermal voltage) = 0.026V.

- Calculate R_{dc}

$$R_{dc} = \frac{V_{CC} - V_T |A_v|}{I_{CQ}}$$

- Calculate R_E

$$R_E = R_{dc} - R_C = R_{dc} - R_L \quad \text{since } R_C = R_L$$

- Calculate R_B

$$R_B = 0.1 \beta R_E$$

- Calculate V_{BB}

$$V_{BB} = V_{BE} + I_{BQ} R_B + I_{CQ} R_E$$

- Calculate R_1 and R_2

$$R_1 = R_B \cdot \frac{V_{CC}}{V_{BB}}$$

$$R_2 = \frac{R_B}{\left(1 - \frac{V_{BB}}{V_{CC}}\right)}$$

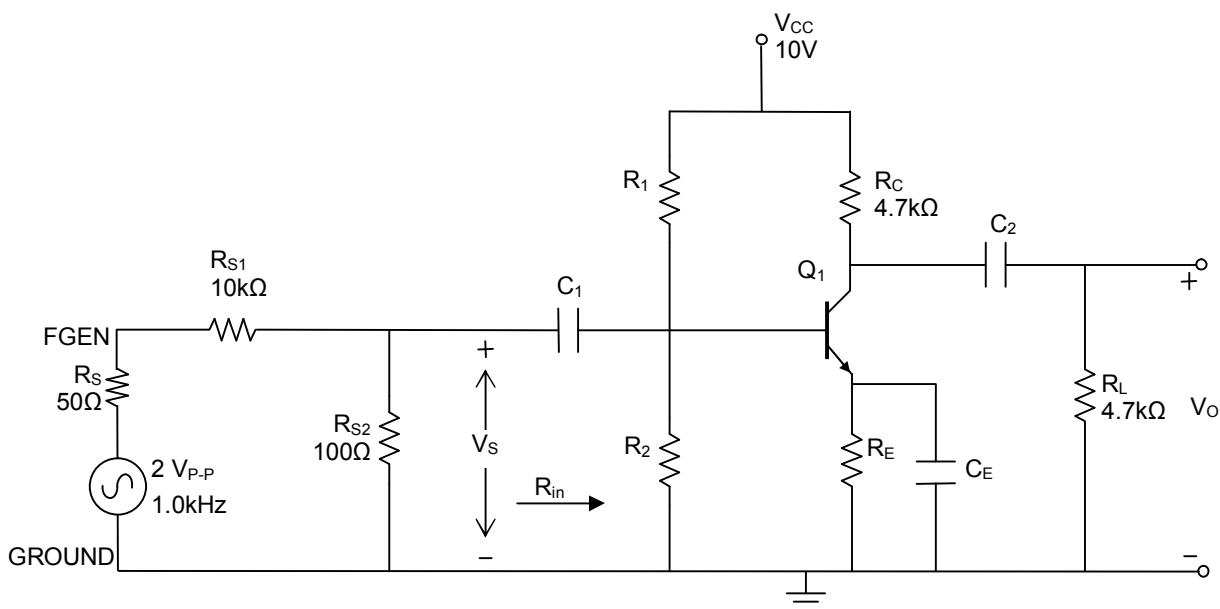


Figure 9.1: Common-emitter Amplifier

EXPERIMENTAL:**PART I: DC ANALYSIS**

1. Using the resistance values you determined from your design calculations, build your amplifier, including only the core components: R_1 , R_2 , R_C , R_E , the transistor, and V_{CC} (similar to Figure 7.2). Use the nearest standard resistance values. You can also combine resistors in parallel and in series to obtain resistance values as close to the calculated values as possible. **However, a good design will use standard resistor values.**
2. Determine the Q-point of your circuit, as in Experiments 6, 7, and 8. If the value of V_{CEQ} is not within 5% your calculated value, replace R_1 with a resistance decade box and adjust the resistance value until V_{CEQ} is equal to or close to your calculated value. Replace the resistance value on the decade box with the closest standard resistor.

PART II: AC ANALYSIS

1. In Figure 9.1, the resistor R_S , represents the 50Ω internal impedance of the signal generator. The two resistors, R_{S1} and R_{S2} , are a voltage divider, external to the signal generator. These two resistors are chosen to keep the input signal to the amplifier low enough to insure that the output wave shape is not distorted. Use an input signal of $0.02V_{P-P}$ at the base of the transistor. To get this signal, you must use a $2V_{P-P}$ output from FGGEN.
2. Complete your circuit as shown in Figure 9.1, using R_{S1} , R_{S2} , R_L , and capacitors (use $22\mu F$ or $47\mu F$). R_S is an internal resistance already in the FGGEN.
3. Apply a $0.02V_{P-P}$, 1 kHz sinusoidal input signal to your amplifier and determine the voltage gain, A_V . How does this value of A_V compare with the design specification?

Make sure the output waveform is not clipped. How does the measured value of V_{O-PP} compare with the calculated value?

4. When you are demonstrating your input and output signals to the lab instructor, be sure to show that the output signal is not clipped or clamped. Show the lab instructor what happens to the output wave shape as the magnitude of the input signal increases. If the circuit you build does not meet the design specifications, explain why. Modify the biasing resistors R_1 , R_2 , and/or R_E to meet the design specifications.

LAB REPORT:

- You must demonstrate to the lab instructor that your amplifier meets the design specifications.
- You must demonstrate to the lab instructor how the output wave shape becomes distorted as the magnitude of the input voltage increases.
- If you have designed an amplifier with maximum output voltage swing, then the distortion in the output wave shape should be symmetrical as the input signal increases in magnitude.
- If your output wave shape distorts asymmetrically as the input voltage is increased, explain whether your design is closer to cut-off or closer to saturation on the load line.

Name: _____

ECE 3110 EXPERIMENT 9

TO BE TURNED-IN TO LAB INSTRUCTOR

Objective: Design a single stage BJT CE amplifier with the following specifications:

$$A_V = -100$$

$$R_L = R_C = 4.7 \text{ k}\Omega$$

$$V_{CC} = 10\text{V}.$$

Display V_S and V_O on the oscilloscope.

Complete this form and turn it in along with your simulation output and all calculations.

 $\beta =$ _____ $V_{BE} =$ _____Calculated $R_1 =$ _____Standard resistor nearest calculated $R_1 =$ _____Calculated $R_2 =$ _____Standard resistor nearest calculated $R_2 =$ _____Calculated $R_E =$ _____Standard resistor nearest calculated $R_E =$ _____Calculated $V_{CEQ} =$ _____Measured value of $V_{CEQ} =$ _____Calculated $I_{CQ} =$ _____Measured value of $I_{CQ} =$ _____

If the measured value of V_{CEQ} is not within 5% your calculated value, replace R_1 with a resistance decade box and adjust the resistance until V_{CEQ} is as close to your calculated value as possible. Replace the resistance value on the decade box with the closest standard resistor(s), and again measure V_{CEQ} . Report those final values below:

Decade Box $R_1 =$ _____Standard resistor nearest Decade Box $R_1 =$ _____Calculated $V_{CEQ} =$ _____Final measured value of $V_{CEQ} =$ _____Calculated $I_{CQ} =$ _____Final measured value of $I_{CQ} =$ _____Measured value of $V_O =$ _____ $A_V = V_O / V_S =$ _____Display V_S and V_O on the oscilloscope and ask your instructor to verify them.