Birzeit University

Faculty of Engineering & Technology

ENEE3102

Electronics Laboratory

FIRST SEMESTER 2020 - 2021

Table of Contents

Lab Safety Instructions and Rules
Electrical & Computer Systems Engineering Department

General Behavior

- Never work in the laboratory alone, always have another qualified person in the area do not use any equipment unless you are trained and approved as a user by your instructor or staff. Ask questions if you are unsure of how to operate something.
- Perform only those experiments authorized by the instructor. Never do anything in the laboratory that is not called for in the laboratory procedures or by your instructor. Carefully follow all instructions, both written and oral. Unauthorized experiments are prohibited.
- Don't eat, drink, or smoke, in the laboratory
- Please don’t yell, scream, or make any sudden loud noises that could startle others who are concentrating on their work.
- When you are done with your experiment or project, all components must be dismantled and returned to proper locations.
- Dress properly during all laboratory activities. Long hair, dangling jewelry, and loose or baggy clothing are a hazard in the laboratory. Long hair must be tied back and dangling jewelry and loose or baggy clothing must be secured.
- Keep aisles clear and maintain unobstructed access to all exits, fire extinguishers, electrical panels, and eyewashes.

First Aid & fire

- First aid equipment is available in the lab, ask your instructor about the nearest kit.
- Fire extinguisher are available in the lab, ask your instructor about the nearest one to your lab.

Electricity

- Do not handle electrical equipment while wearing damp clothing (particularly wet shoes) or while skin surfaces are damp.
- Never bend or kink the power cord on an instrument, as this can crack the insulation, thereby introducing the danger of electrical shocks or burns.
- Know where the stop button, main switch or other device for stopping the apparatus is located

Machines and moving parts

- In order to avoid the possibility of injuries, it is important that the students be aware of their surroundings and pay attention to all instructions.

- Deal with caution with rotating machines, fans pumps compressors, motors etc. don't touch any of the rotating parts; shafts, or blades.
- Read and understand operation instructions before turning on the machines, do not turn machine till you instructed by the instructor or the technician.

Hot surfaces and burns

- Do not touch hot surfaces; hot plates boilers, heating elements machines etc.

Introduction

The Electronics Laboratory is one of the most important Laboratories that engineering students will take since it will enhance the theoretical knowledge gained in classes through a series of experiments.

In addition to practical experiments, students will have to prepare for the lab through simulation and calculation of the circuits under test. Students will work in groups to enhance communication and team work skills, they are also required to write a report to illustrate and interpret results and draw conclusions and observations.
In most experiments prior knowledge of the theoretical material is assumed.

Objective

- To test and debug various electronic components and circuits including transient, steady state and frequency response
- To perform characterization tests on Diodes, BJTs, FETs, Op-amps and voltage regulators and oscillators.
- To perform tests to verify operation of diode based circuits such as rectifiers, clippers, clampers, multipliers and zener diodes
- To perform tests to verify operation and performance of BJT and FET amplifier circuits
- To perform tests to verify operation and performance of Op-amps and voltage regulators and oscillators.
- To Report on experiments and to develop necessary skills to communicate experiment findings in a scientific and precise way

Laboratory Instructions

- Each Student should prepare for the lab by reviewing the theoretical background and simulating the circuits under test using Pspice and submit the material of the prelab before the start of the laboratory session
- Students will work in groups of 2-3 students maximum
- Each group should prepare a report and submit it at the beginning of next lab session
- Reports should be original and contain the basic required elements detailed below, any copy from any source will result in a zero grade and proper academic punishment
- During the laboratory session it is required to have an experimental setup checked and approved by the instructor before starting data collection

- Data sheet should be signed by the instructor before you leave the laboratory, otherwise your report will not be accepted
- Smoking, eating, drinking and use of cell phones is not allowed during the laboratory session

Report Writing

An experiment report is an important tool to communicate the experiment results and findings to others and it should be organized and written in a way to provide information about the experiment and its results and conclusions. The report should contain calculations and explanations of the results and a neat and clear way
The report should contain the following parts:

Cover page

It should contain the experiment name, date and name of experimenters.

Abstract

This section provides a brief summary explaining the aim of the experiment, the methods used and the main results

Theory

This section should include any relevant theory along with mathematical formulas, the following should be considered when writing the theory section:

- Avoid copying from lab manual
- Summarize the theory in your own words
- Explain symbols in the mathematical formulas

Procedure

This section describes in detail the way the experiment was conducted. This is very important so that anyone who reads it should be able to re-produce the experiment and its results. In this section what was measured and how it was measured should be provided.

Data and Calculations

Measurement data should be recorded in clear and readable fashion, the data should be provided in tables when possible and the following notes should be considered:

- Data should be written in ink and signed by the TA or Instructor.
- Data should have units

- Calculations should be performed to get the required quantities from measured ones
- Simulations results should be included where applicable

Conclusions

This section presents the final result with the uncertainty associated with it. The conclusion should be based on evidence and does not reflect how the experimenter feels about the experiment. The conclusion should contain answers to the following questions:

- Is the result acceptable?
- What is the behavior of graphs/plots?
- What are the possible sources of error?
- Were there any major experimental complications?
- How the result can be improved in the future if the experiment is repeated?

Report Grading Guidelines

The laboratory report grade will depend on the following:

- Is the report written well and in good English?
- Does the theory contain the necessary illustrative figures? Are these figures meaningful and clear?
- Do data and calculated quantities have correct units?
- Are calculations made correctly?
- Does the report contain all information to reproduce the experiment?
- Is the result correct and consistent with what is expected?
- Are the graphs complete, correct and properly labeled (title, axis labels)?
- Does the conclusion show insight and indicate ways for future improvements in the experiment?
- Are all elements of the report included?
- Is the report submitted on time?

Experiment#1

Introduction to Simulation

Objectives:

In this experiment a review of basic simulation tools and types will be conducted.

Equipment:

1. Pspice Student Edition

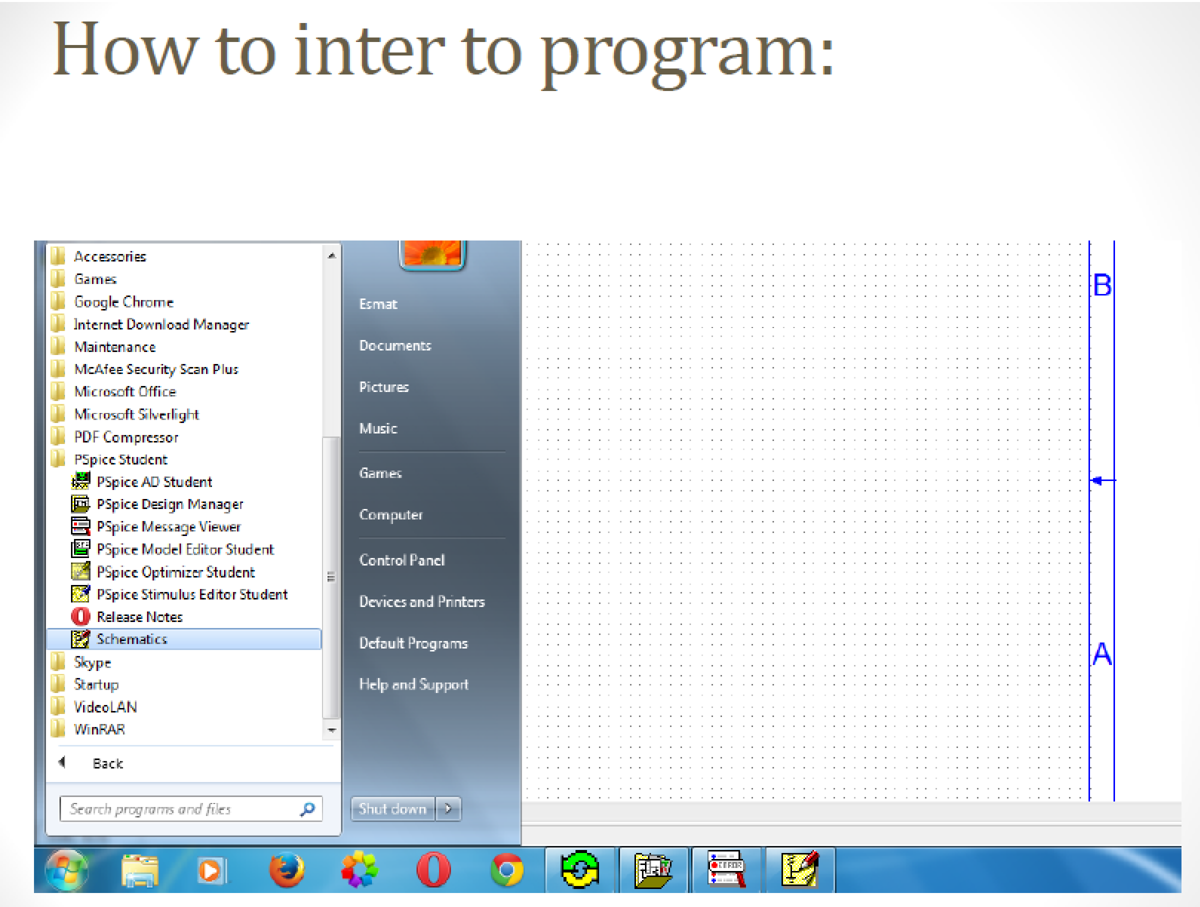
Overview:

A review of Pspice simulation will be done through a tutorial that covers the following topics:

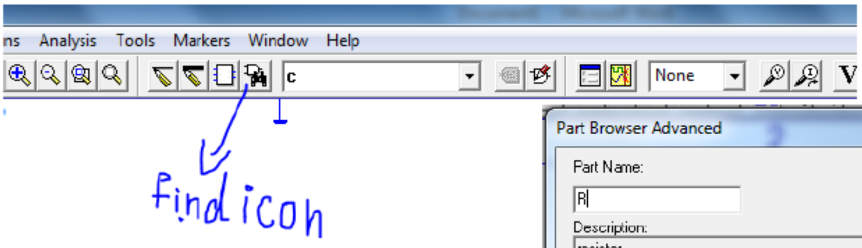
1. Using Schematic capture to draw the required schematic and setting component values and changing model parameters for an instance of component.
2. Bias point analysis and measuring of bias point currents and voltages
3. Transient analysis for time domain and step response analysis, setting up simulation time and step ceiling.
4. Ac sweep for frequency response , plotting of magnitude and phase response.
5. Parametric analysis and dc sweep.
6. Using probe, and copying the results into word .

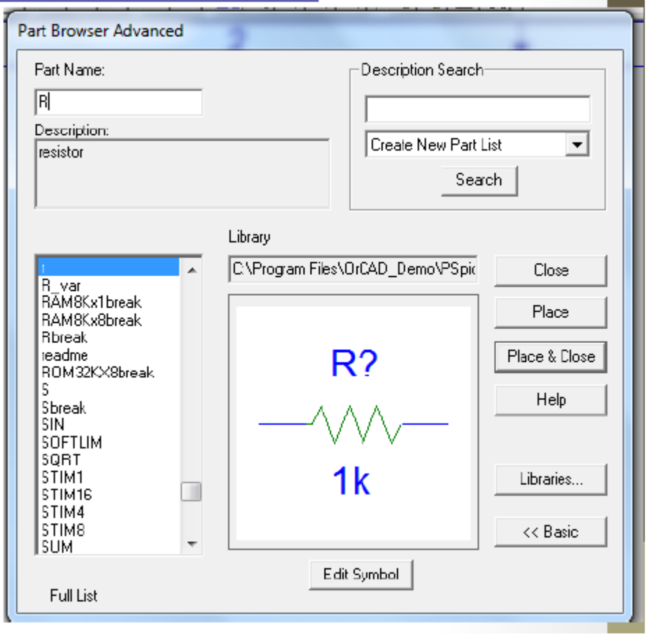
Experiment Procedure

1. Using Schematic capture to draw the required schematic and setting component values and changing model parameters for an instance of component.



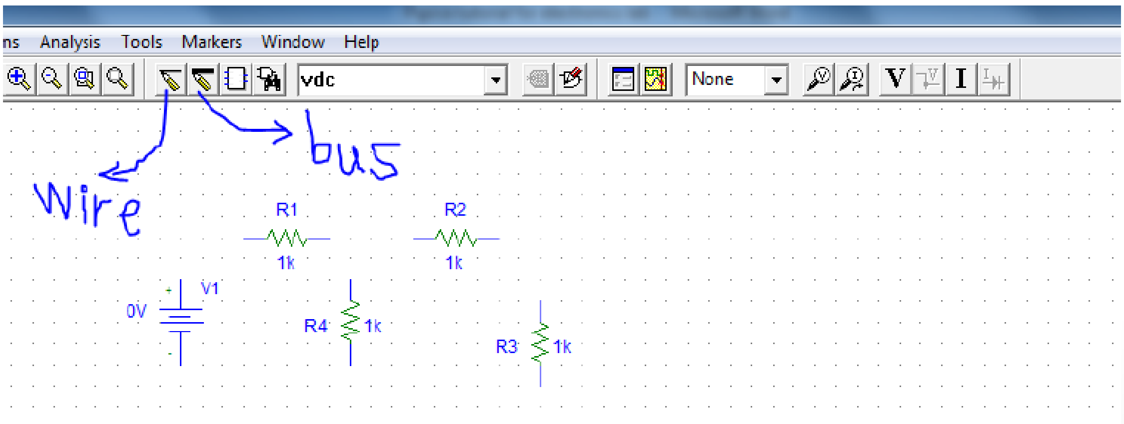
How to get and place components:



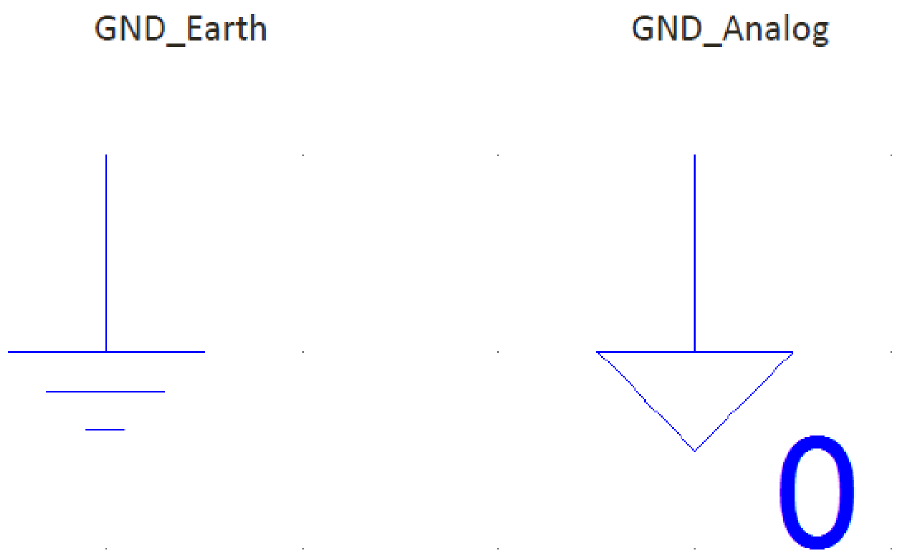


Then you have to click on place& close Icon to place the component in the circuit.

Drawing wires (CTRL+W):

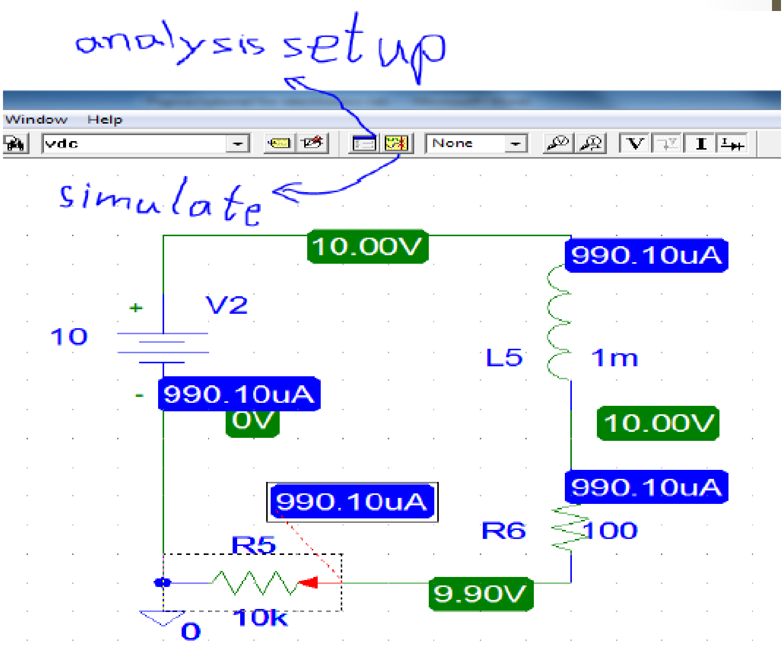


Grounding

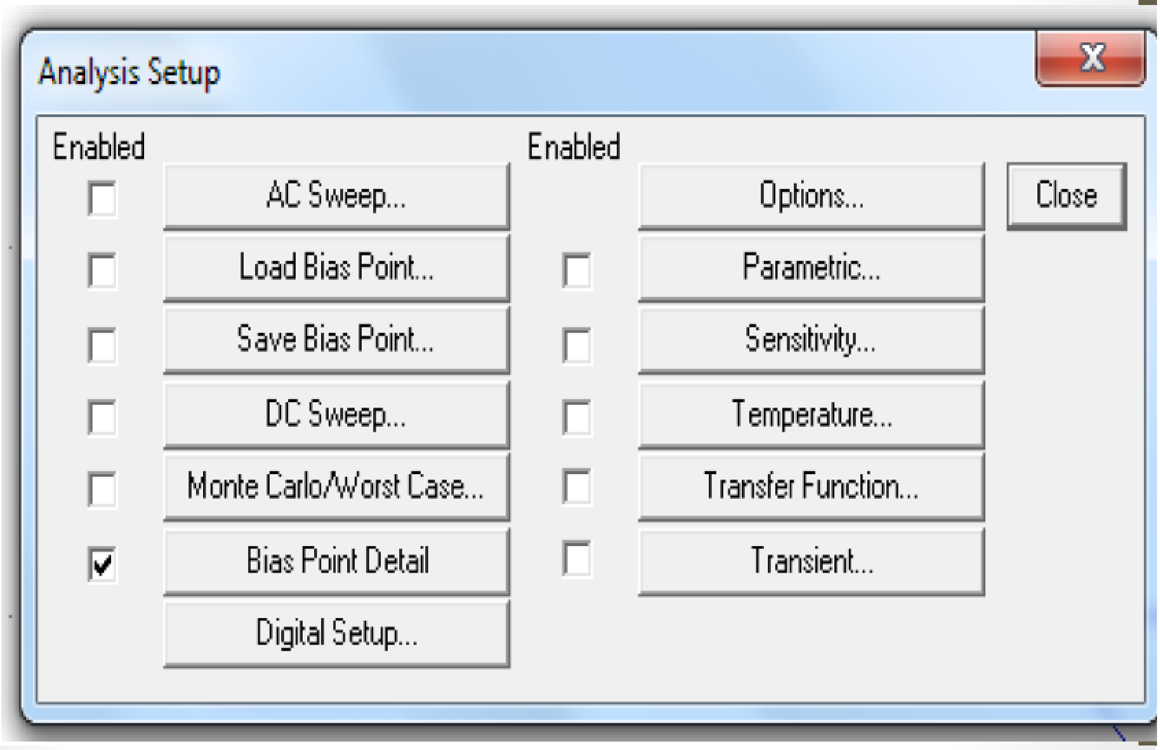


Types of simulation :

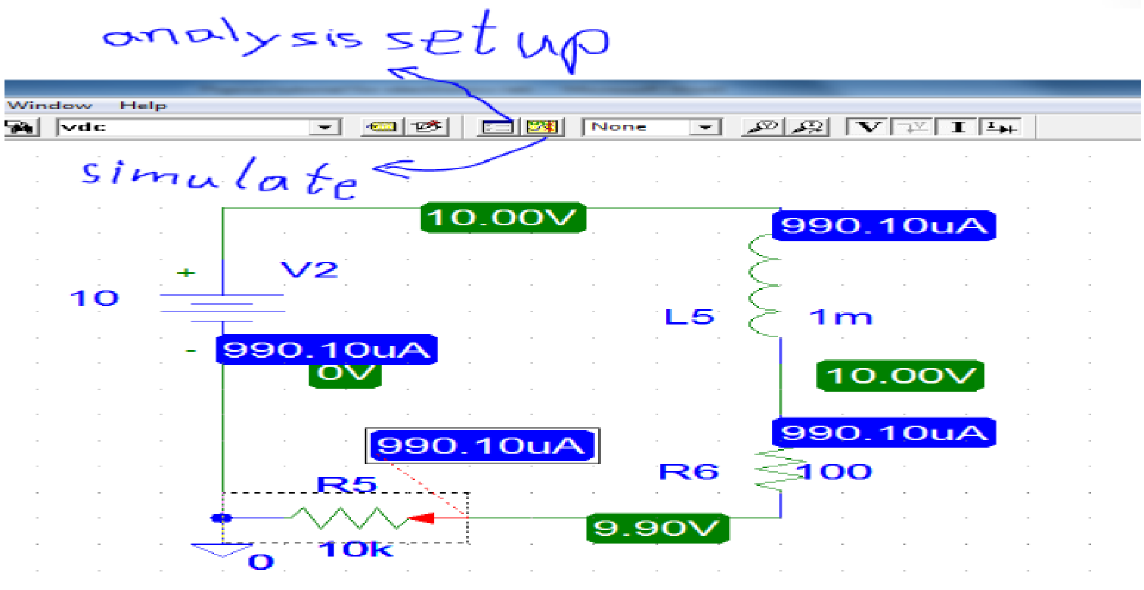
***NOTE: Before simulation you have to save your schematic.



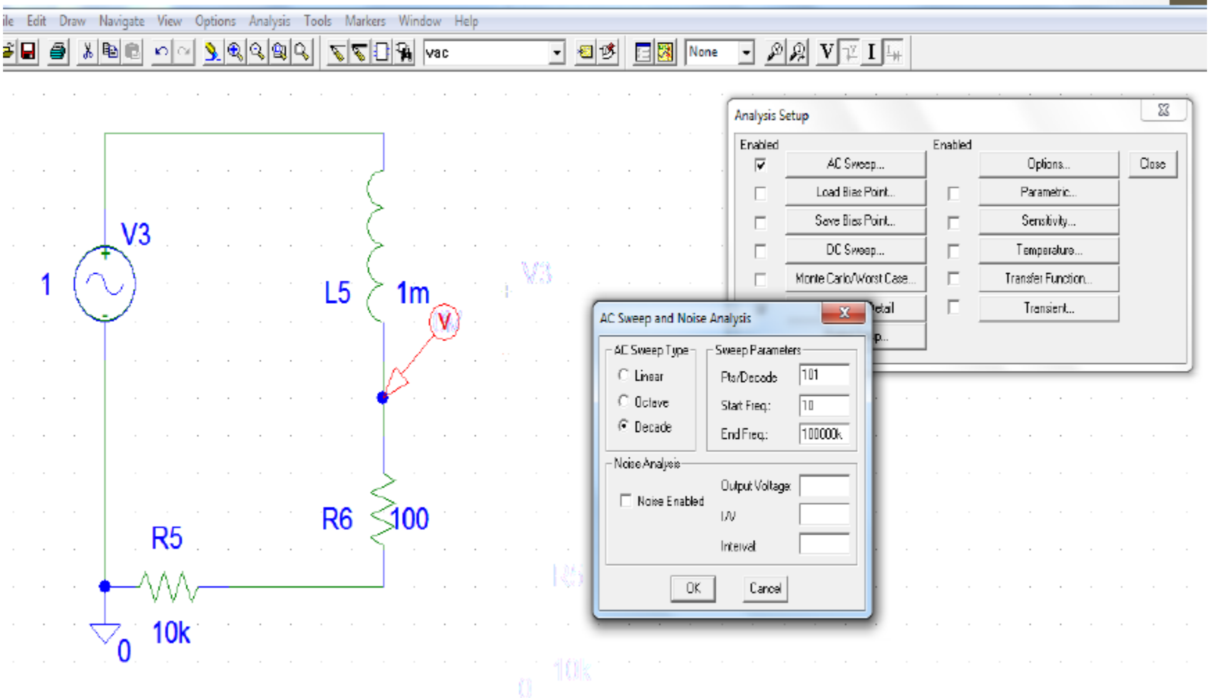
Types of simulation :



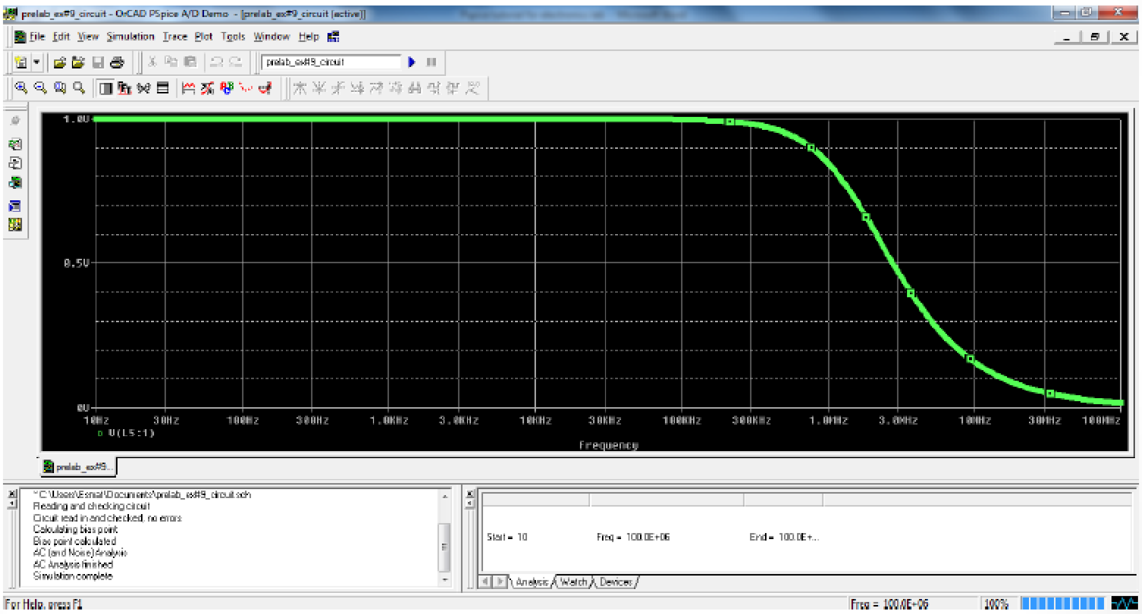
1-Bias Point Detail: this is used in DC analysis at a certain point of input only.



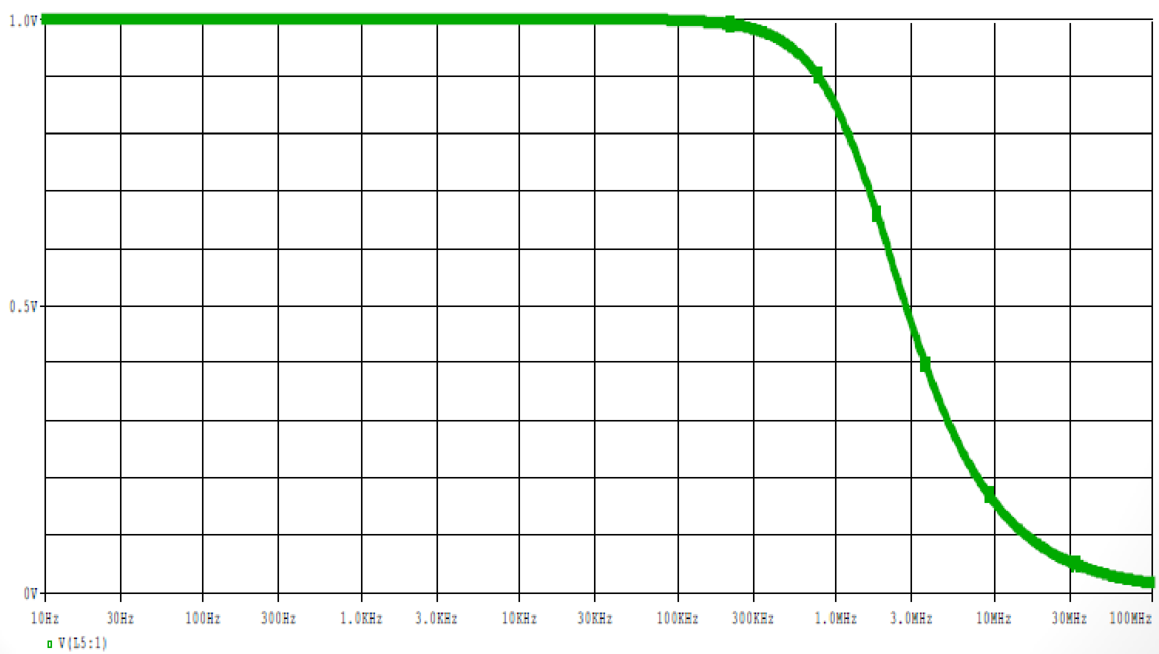
2-AC Sweep: this type used to plot the response in the frequency domain.



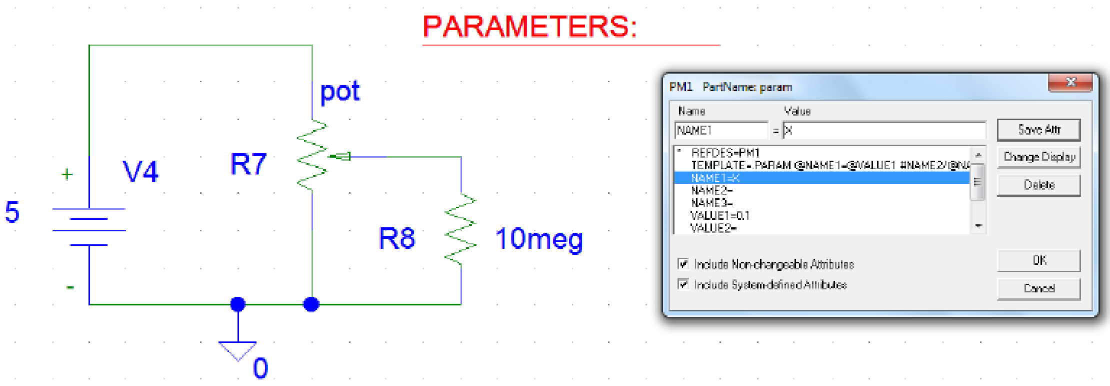
Output:



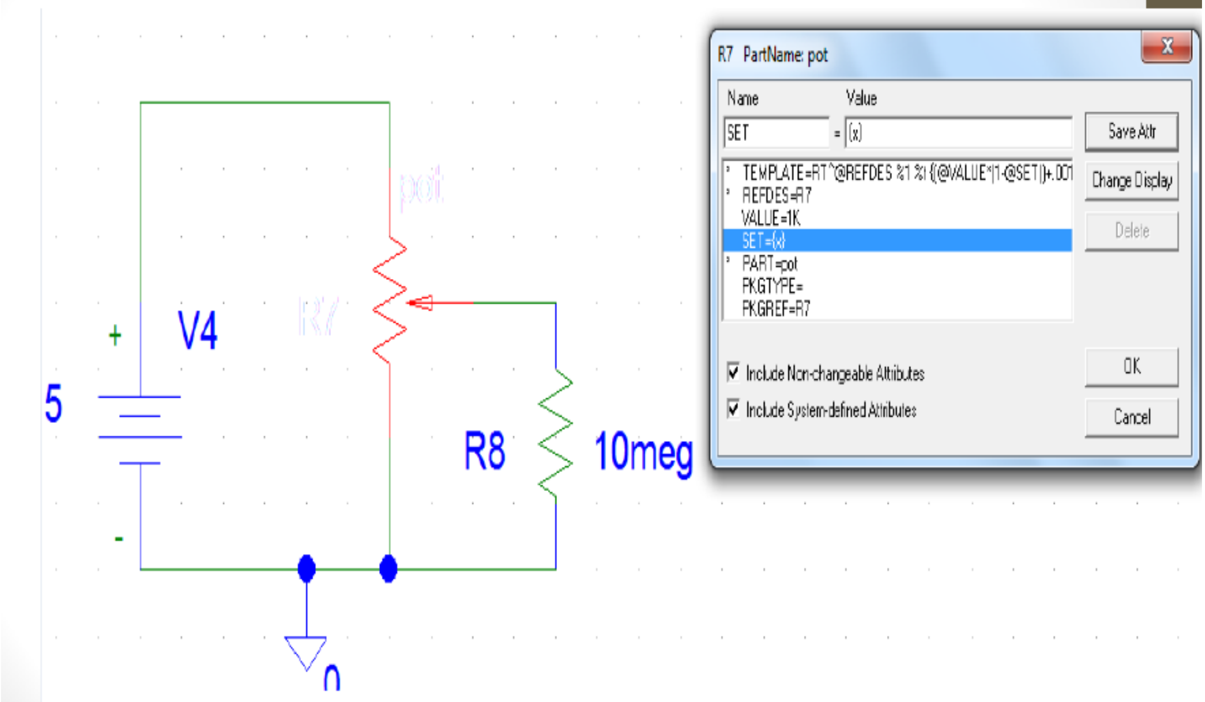
Note:To get clear output:
Click on Windows➔Copy to clipboard➔Change White to Black ➔ Ok. Then you can Pa
the output in your file in a clear view.



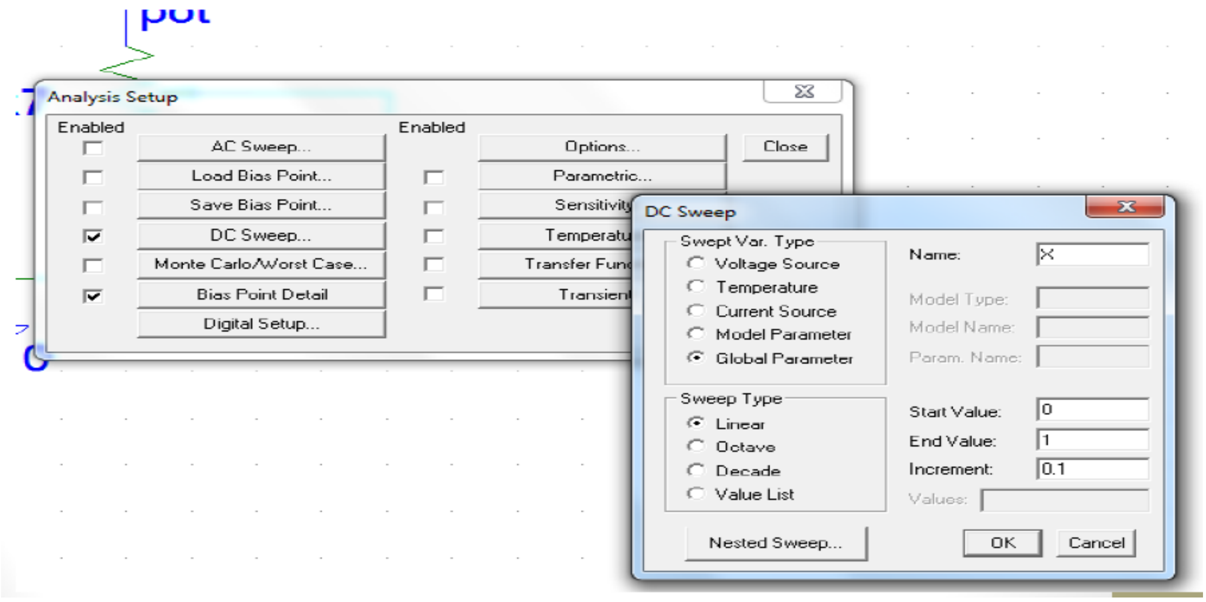
3-DC Sweep: this is used to see the change of output versus a change in one parameter of the circuit "Parameter settings".



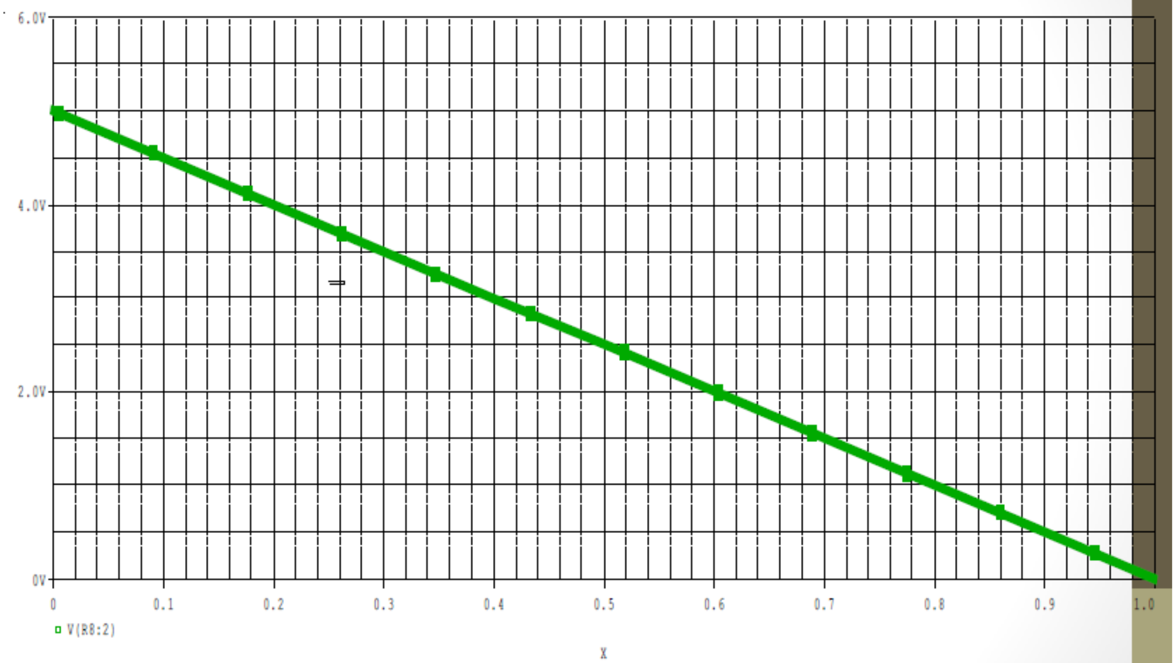
3-DC Sweep: this is used to see the change of output versus a change in one parameter of the circuit “Component settings”.



3-DC Sweep: this is used to see the change of output versus a change in one parameter of the circuit “analysis setup”.

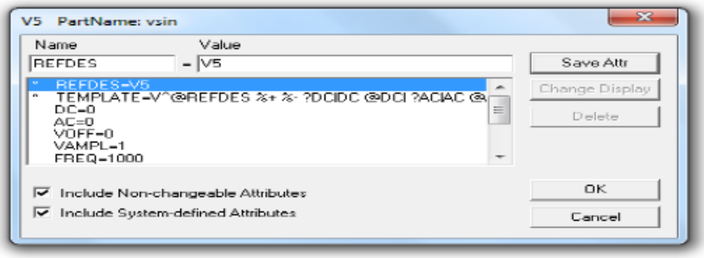


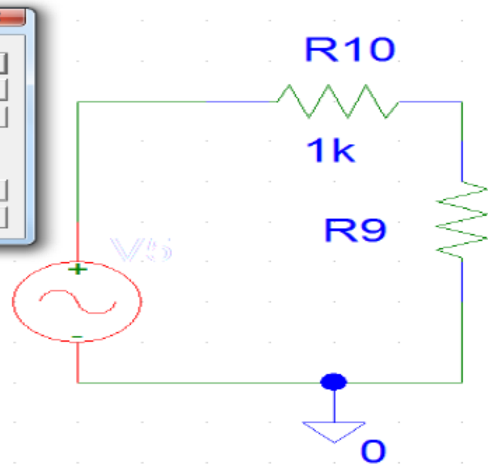
The output



4-Transient: this is used to see the output response in the time domain.

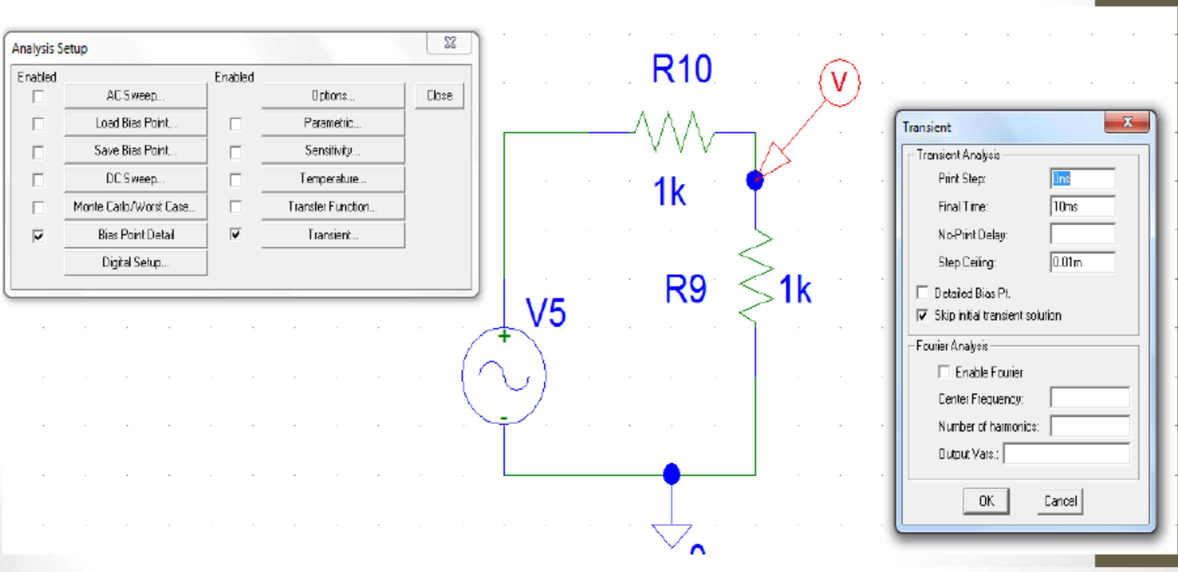
- To have a sine source you have to choose Vsin its setting are as shown below:



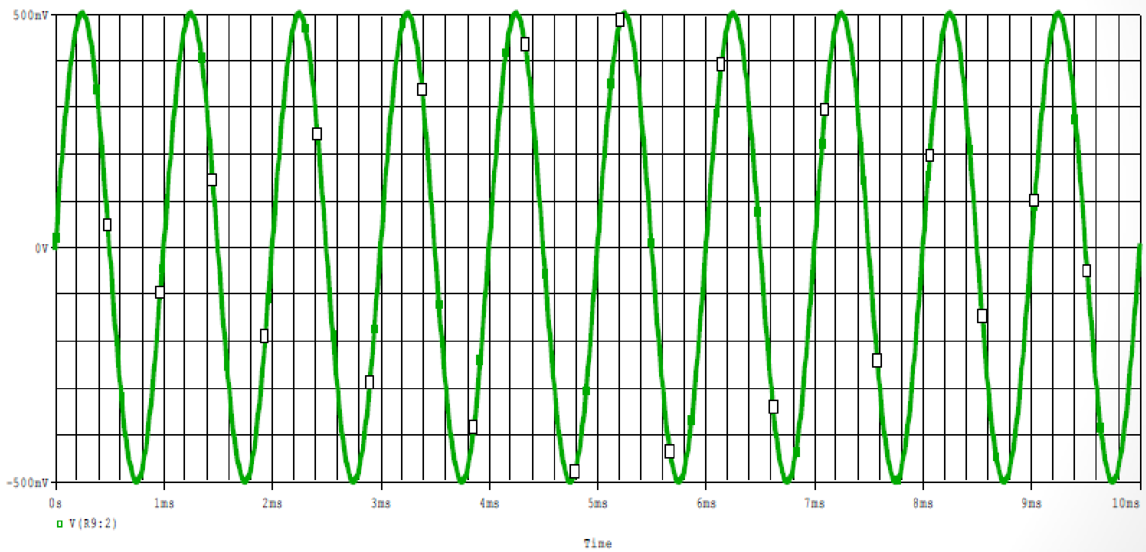


4-Transient:

- Then the simulation setup configuration should be as below:



And the output will look like this:



5-Parametric: this type of simulation used along with other type like DC sweep, AC Sweep or Transient. It is used to plot more than one plot on the same graph.

- Example:

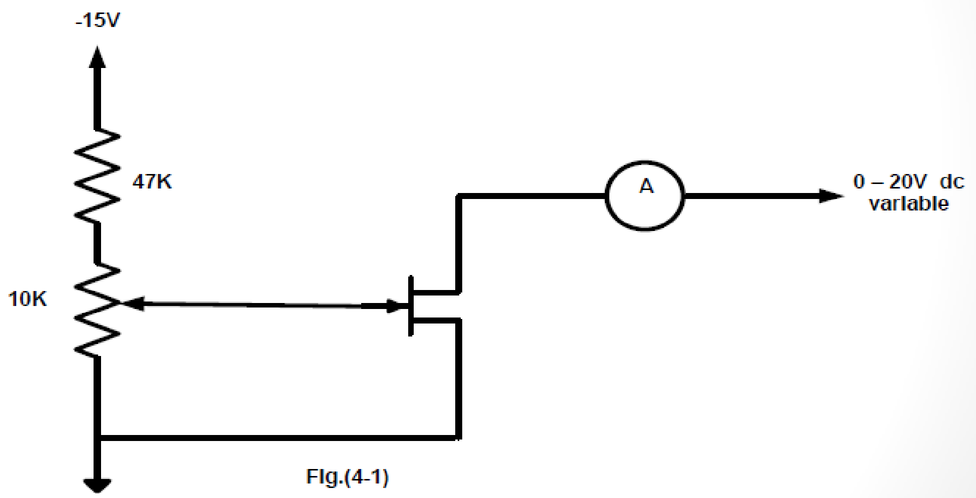
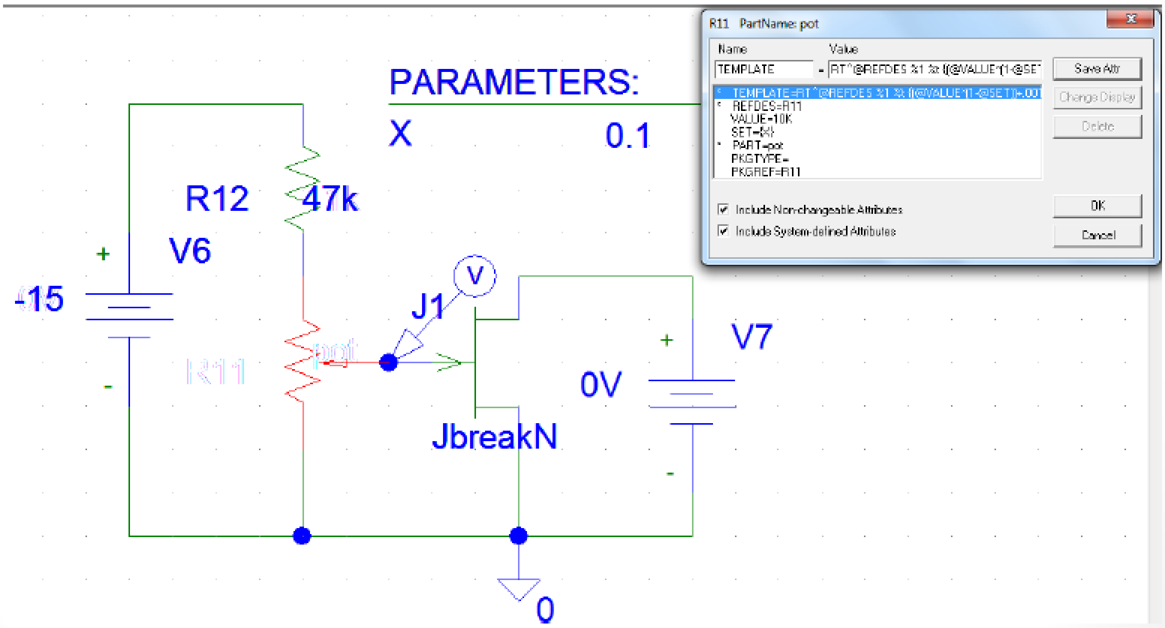


Fig.(4-1)

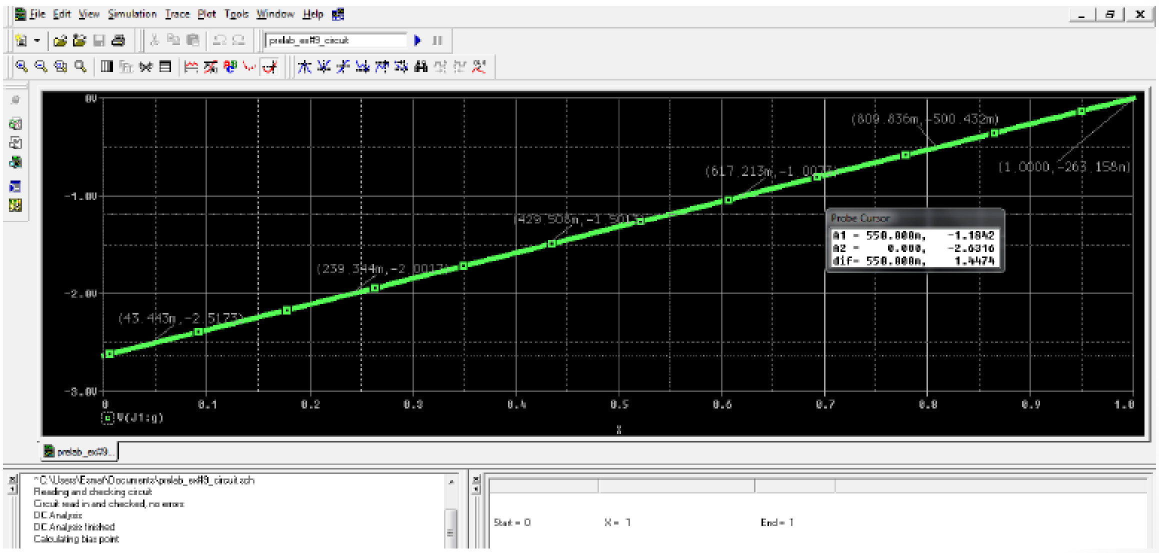
Example Cont.

	I_D(mA) for V_{DS}=(V)						
V_{GS}(V)	0	0.5	1	2	5	10	15
0							
-0.5							
-1.0							
-1.5							
-2.0							
-2.5							

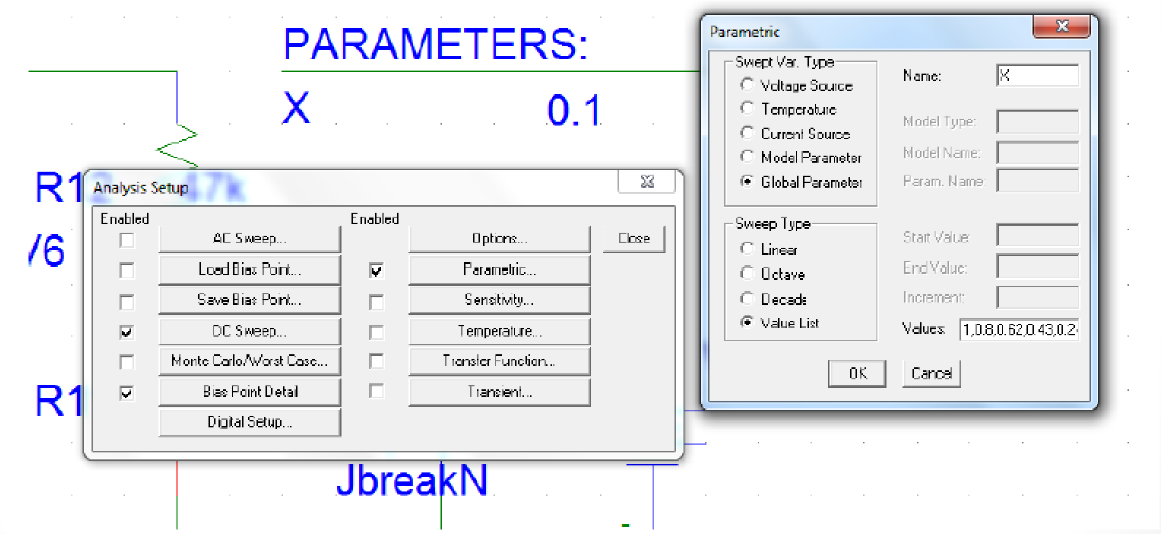
Adjusting the VGS v.s set



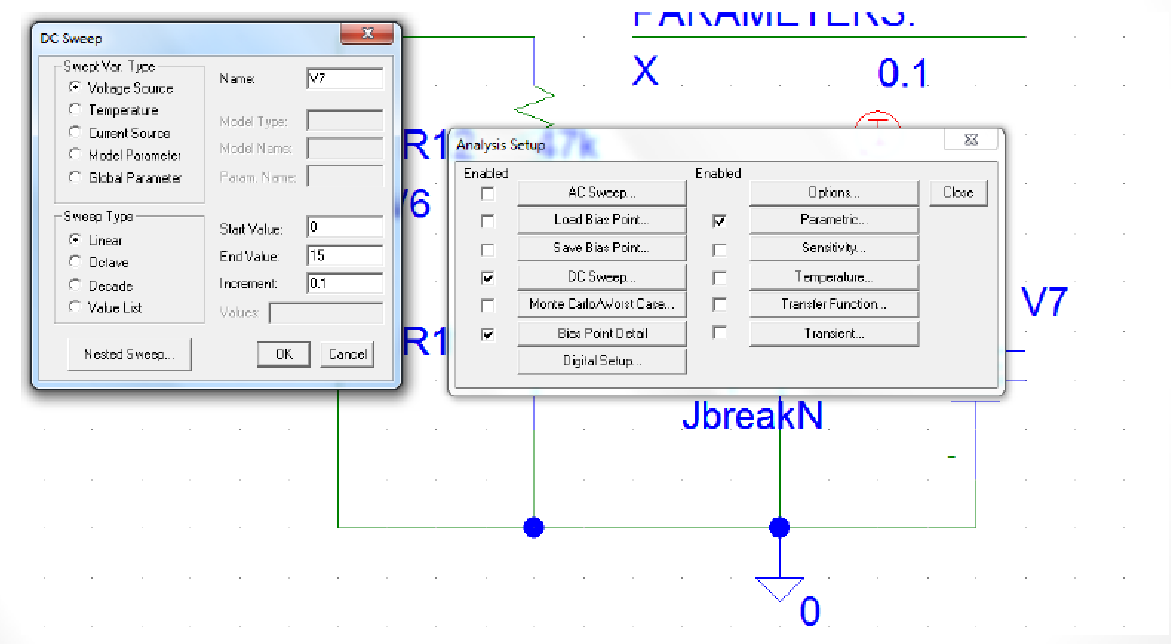
VGS v.s Set



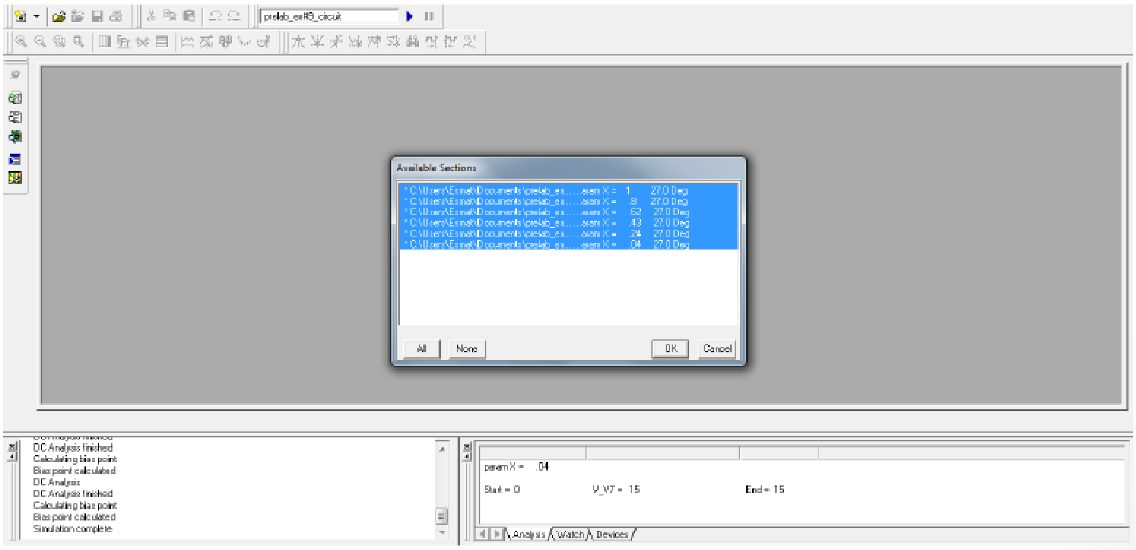
Parametric Sweep With DC sweep to plot the characteristic curves of the JFET



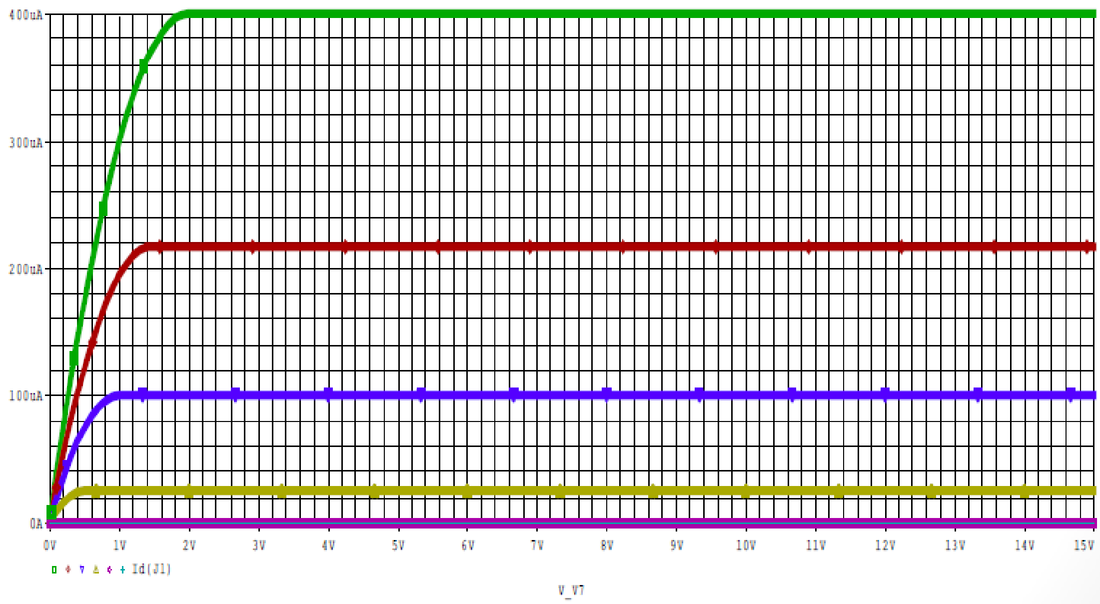
Parametric Sweep With DC sweep to plot the characteristic curves of the JFET



Output

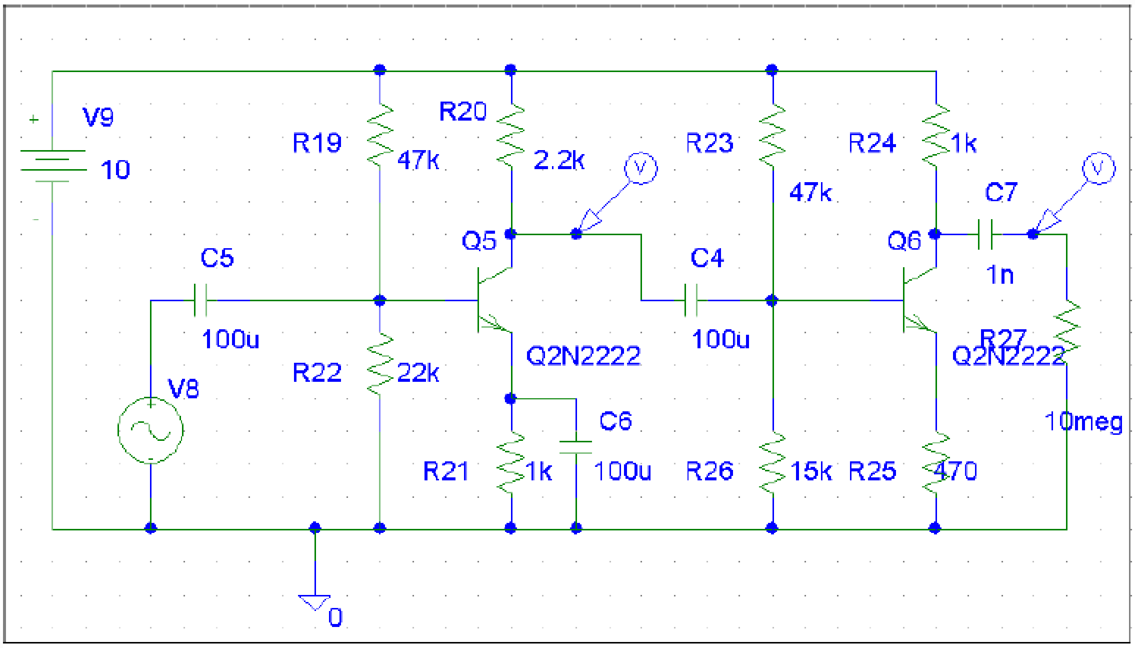


output

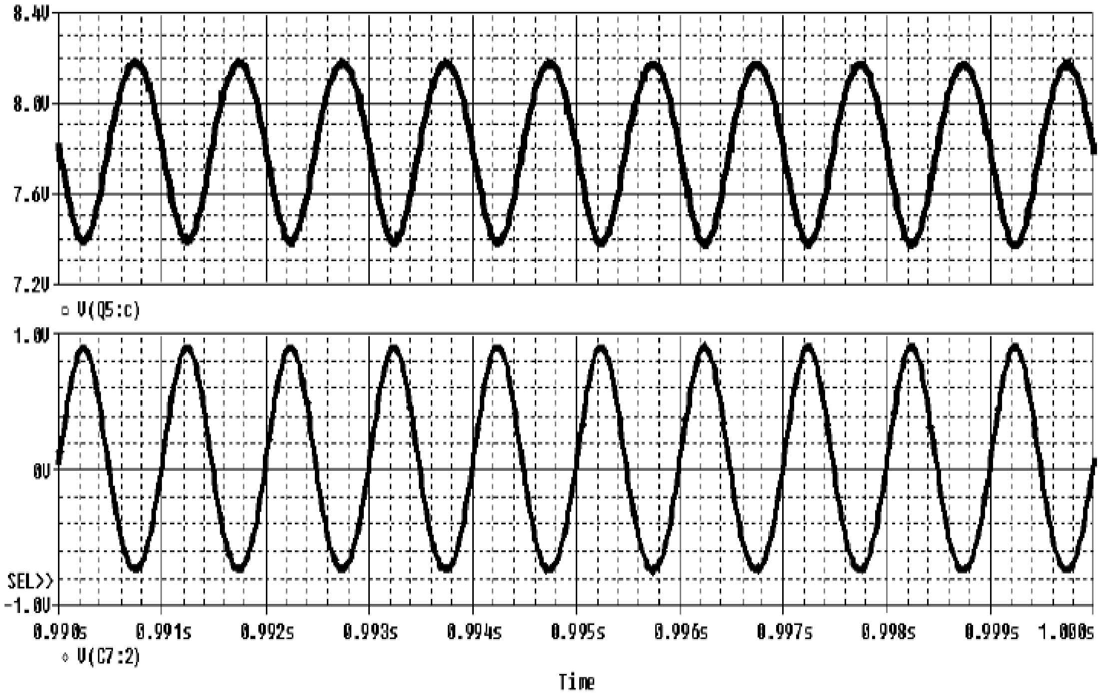


1

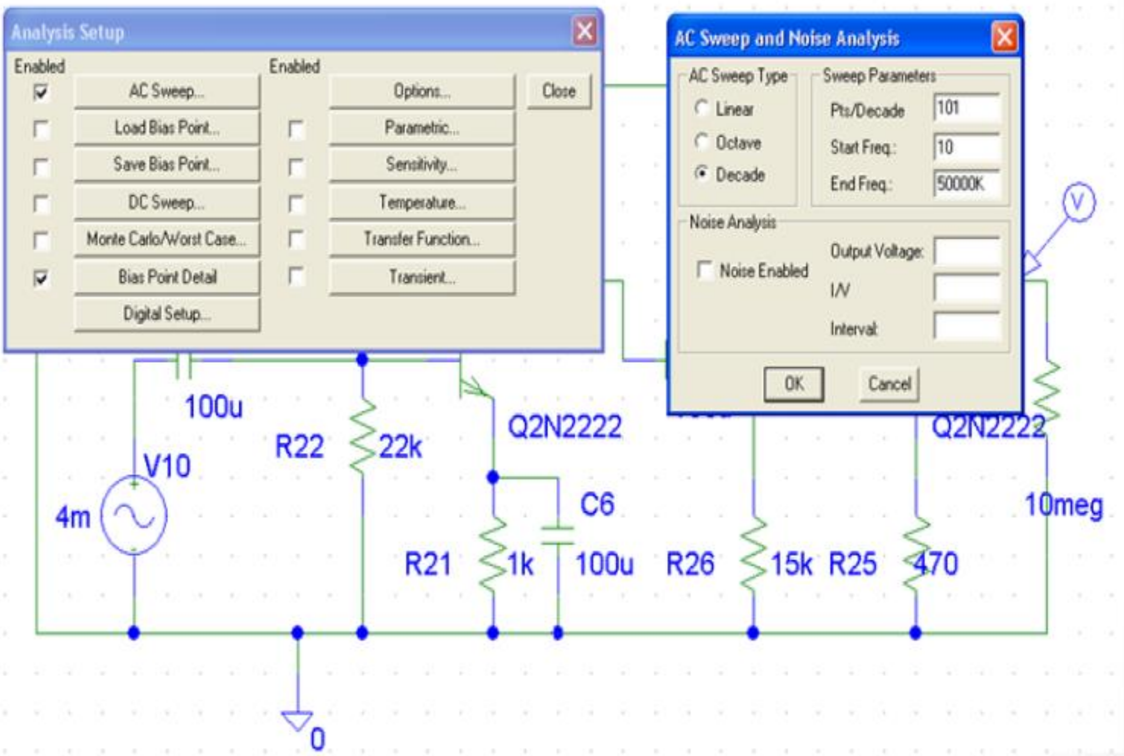
Multi stage amplifier



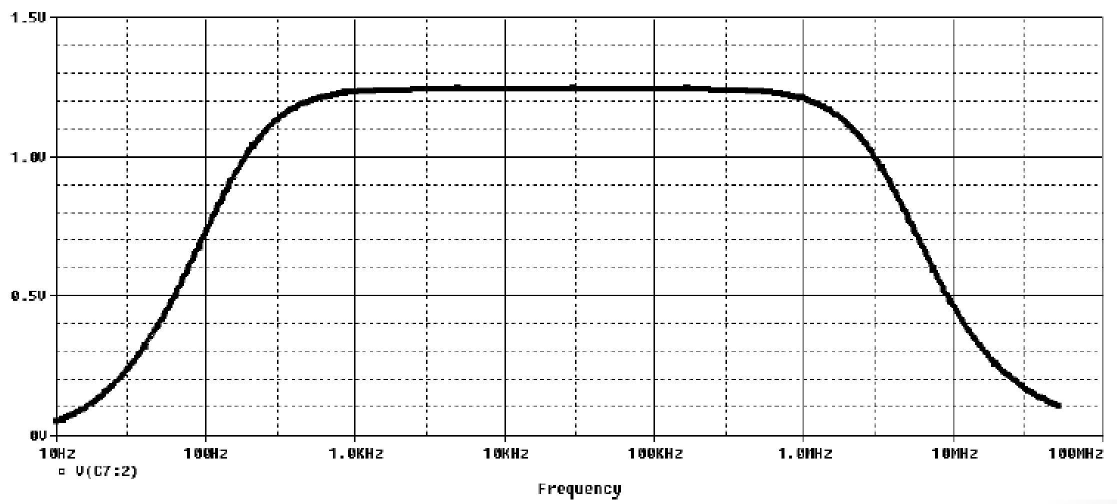
The output voltage when the input voltage is sinosiodal with 12mVp-p



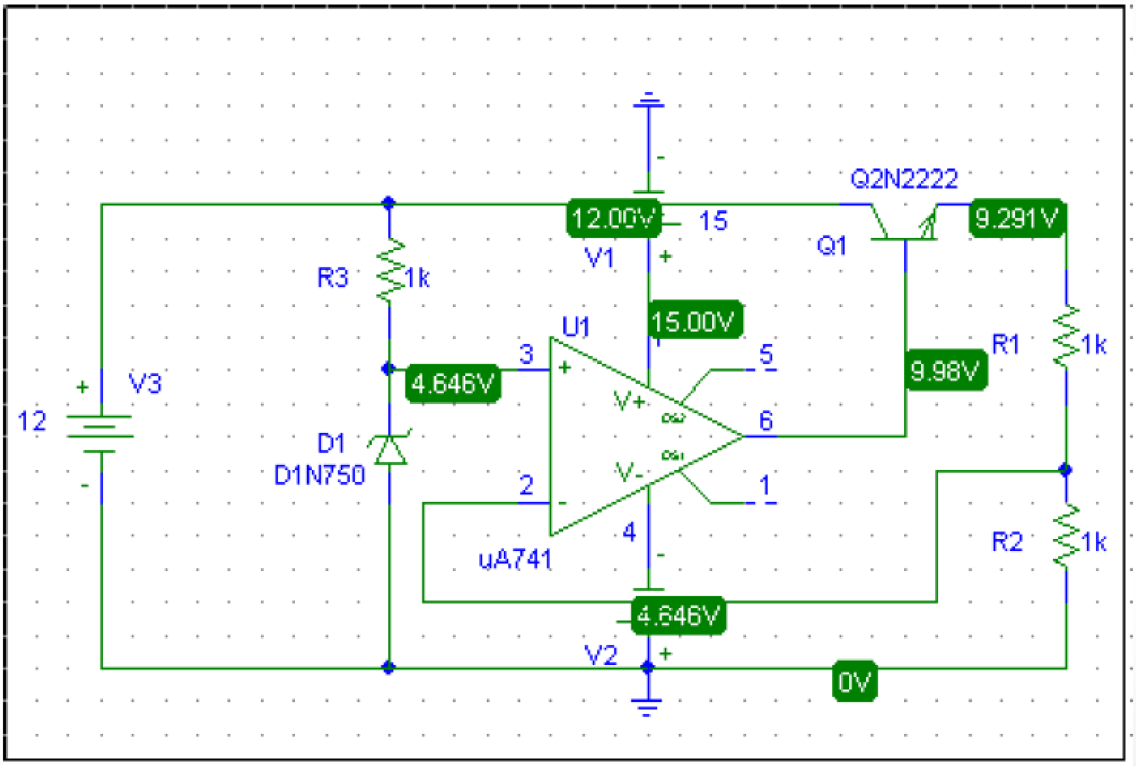
Frequency Response



Frequency response



Voltage Regulator



Experiment # 2 ENEE3102

Diode Characteristic and Applications.

Objectives:

1. To investigate the operation of PN junction, and the VI characteristics of the silicon diode .
2. To investigate some applications of the P-N junction like Rectification, Clamping and Clipping.

Pre-lab Work:

You have to apply the PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need.

Procedure:

I. DIODE CHARACTERISTICS.

1. Connect the Circuit of Fig. (2.1).

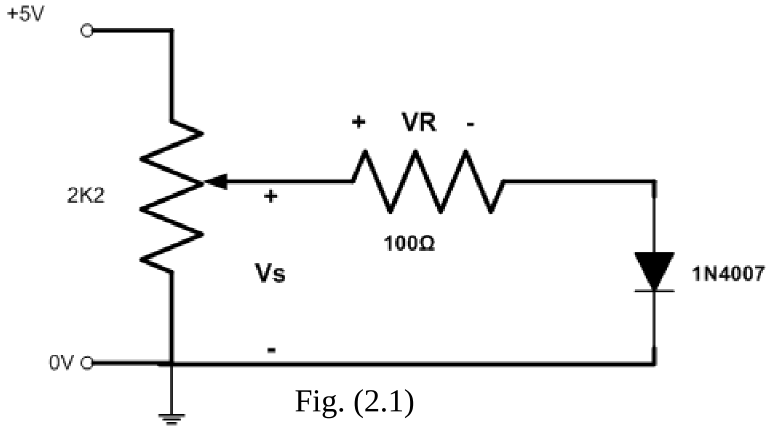


Fig. (2.1)

2. Turn the potentiometer to zero; fully anti - clockwise.
3. Switch on the power supply and adjust it to 5 volt.
4. Using potentiometer change the value of V_s from zero to 1 volt in 0.1V steps and in 0.5 steps from 1V to 3V.
5. For each setting measure the value of VR.

V_S	VR	V_D	I_D
0			
0.1			
0.2			
0.3			
0.4			
0.5			
0.6			
0.7			
0.8			
0.9			
1.0			
1.5			
2			
2.5			
3			

Table 2.1

6. Calculate V_D and I_D and enter them in the table 2.1 .
7. Draw the forward characteristics of the diode by plotting I_D versus V_D.

Questions:

- At what approximate value of V_D does the current I_D begin to rise noticeably?
- Does V_D rise much above this value for larger values of I_D?
- What happens if the diode is reversed?

II. RECTIFICATION .

A. HALF - WAVE RECTIFICATION.

1. Connect the circuit as shown in Fig.(2-3).

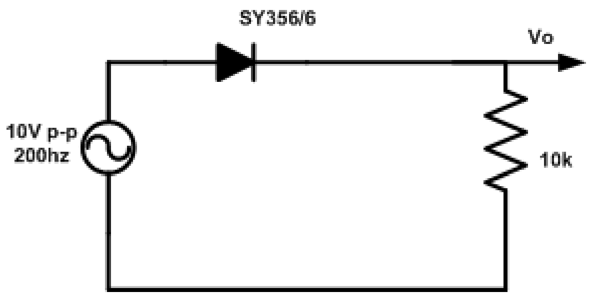


Fig. (2.3)

2. Switch on the oscilloscope and the sinusoidal supply.
3. Measure and record time T(the period) and peak voltage V_{pk} for the output voltage.
4. Measure the dc and ac components of the output voltage using DVM and compare your dc value with the theoretical value.
5. Reverse the Diode and observe the output voltage

Questions☹

- Is V_{pk} nearly equal to the peak voltage of the supply.
- Why will V_{pk} not be exactly equal to the source peak voltage ?
- How much will it differ?
- How could you obtain a negative voltage relative to zero?

6. Now add a capacitor of 2.2µF to your circuit ,the circuit becomes as shown in Fig.(2-4).

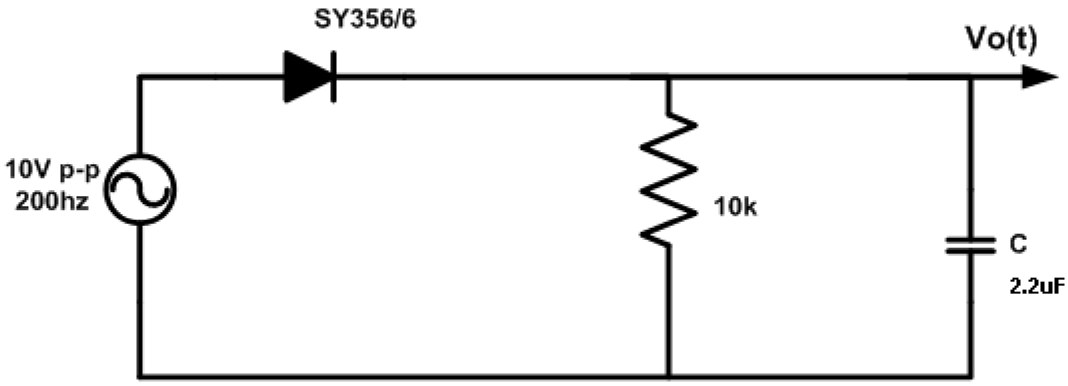


Fig. (2.4)

7. Observe the output waveform on the oscilloscope and measure peak-to-peak ripple and rms ripple voltage **using ac coupling**.
8. Measure the mean value of Vo(t) **using dc coupling**, then calculate the ripple factor.

9. Now replace the 2.2 µF capacitor by a much larger value of 47µF ,making sure to connect the + side of the capacitor to the diode cathode (the capacitor is electrolytic and MUST be connected in the correct polarity).

Questions:

- Is the ripple now less than or more than it was with the lower value of the capacitor?
- Is the mean rectified voltage now greater or less?

B. *FULL-WAVE RECTIFICATION*

Diode bridge circuit as a full wave rectifier:

1. Construct the circuit of Fig.(2-5).

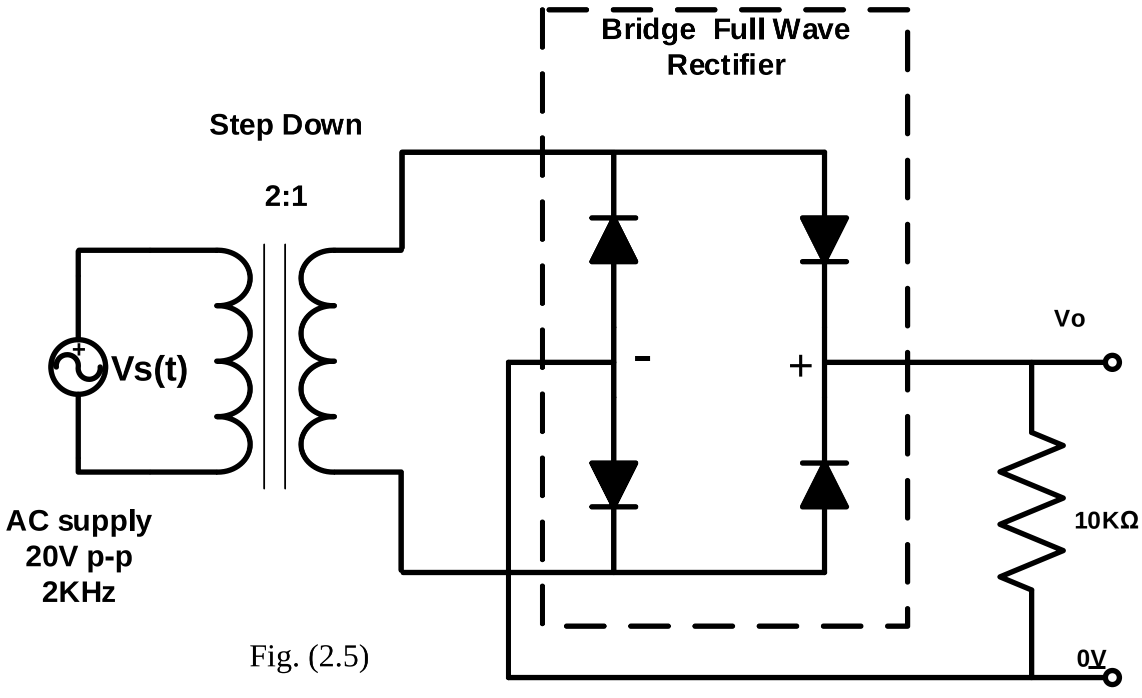


Fig. (2.5)

2. Connect the oscilloscope to the output.
3. Draw the output waveform as seen on the oscilloscope and take a picture showing key quantities.
4. Measure the dc and ac components of the voltage across the load using DVM.
5. Now add a capacitor of 2.2µF to your circuit, and observe the output on the oscilloscope.

Questions:

- When the capacitor connected, what is the change on the waveform, why?
- Does the ripple voltage change with frequency?
- If the input frequency is low do you need a larger or a smaller capacitor to achieve the same smoothing as when the frequency is higher?

III. other applications:

A. clipping:

1. Connect the circuit as shown in Fig.(2-6).

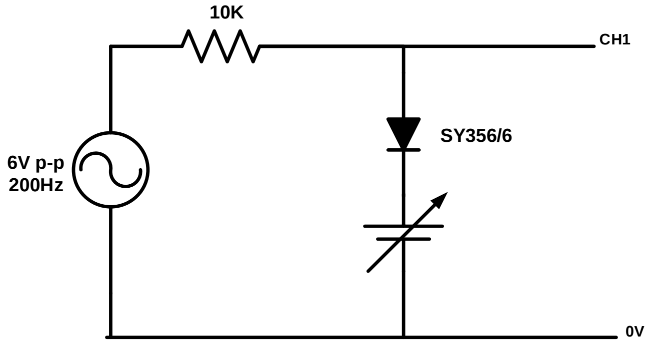


Fig.(2-6)

2. Connect the oscilloscope to the output of the circuit.
3. Set the power supply variable control to zero (fully anti-clockwise) and sketch the output waveform.
4. Increase the dc source slightly and notice what happens to the output waveform.

Questions:

- What difference is there between the input and output wave?
- At what voltage is the output wave form chopped off?
- If the dc is 2V, at what voltage are the positive peaks chopped off?
- If the ac is 10V p-p, does the clipping voltage change?
- What is the relationship between the clipped level and the dc voltage in the two cases?

B. Clamping:

1. Connect the circuit shown in Fig.(2-7).

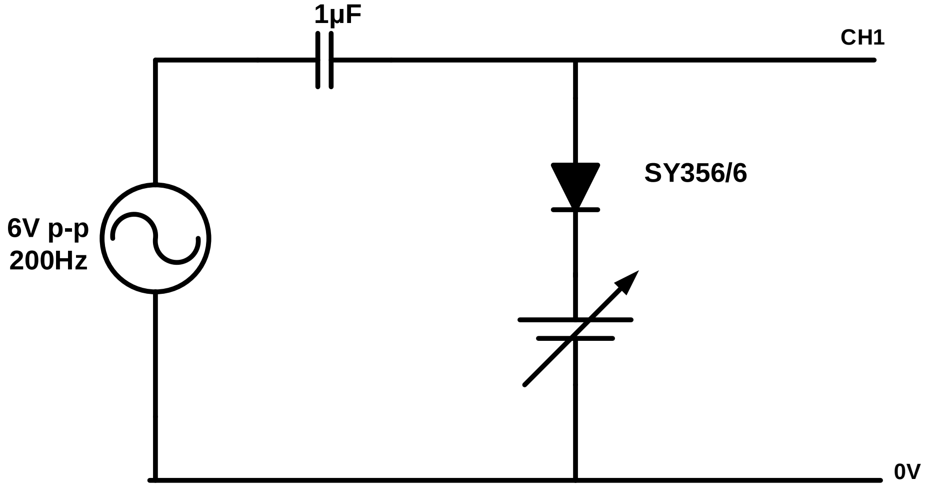


Fig.(2-7)

2. Follow the same steps you had followed in the previous part A (clipping).

Questions:

- Does the output wave form alternate about the same dc level as the input waveform?
- To what value is the positive peak of the output waveform clamped, if the ac input signal is $5V_{PK}$?
- Does the positive peak still stay clamped to the same level?
- Can you see any relation between the reference voltage setting and the clamping level.

C. ***VOLTAGE MULTIPLIER CIRCUITS***

1. Set up the circuit as shown in Fig.(2-8).

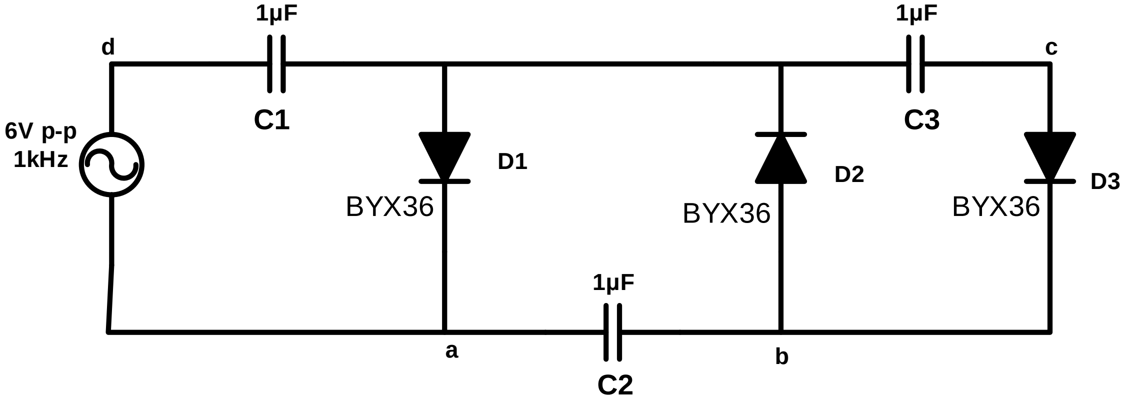


Fig.(2-8)

2. To see how the circuit works as a doubler measure the dc voltage between points a and b using DVM.
3. To see how the circuit works as a Tripler measure the dc voltage between points c and d.
4. Measure the voltage across each capacitor. Are they of the same value?

Questions:

- Is the output voltage between a,b twice the peak of the input voltage.
- Is the output voltage between c,d three times the peak of the input voltage.
- What is the peak inverse voltage across each diode?
- Compare the results of the above questions with the theoretical values.

Experiment#3 ENEE3102

The Transistor Biasing and the DC Parameters

Objectives:

1. To investigate the supplies connections to the transistor.
2. To investigate the characteristics of the transistor for varying dc supply voltage to its three connection models.

Pre-lab Work:

You have to apply the PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need.

Procedure:

<u>I. The Transistor Biasing:</u>

1.Connect the circuit as seen in Fig.(3-1) using the PnP transistor .

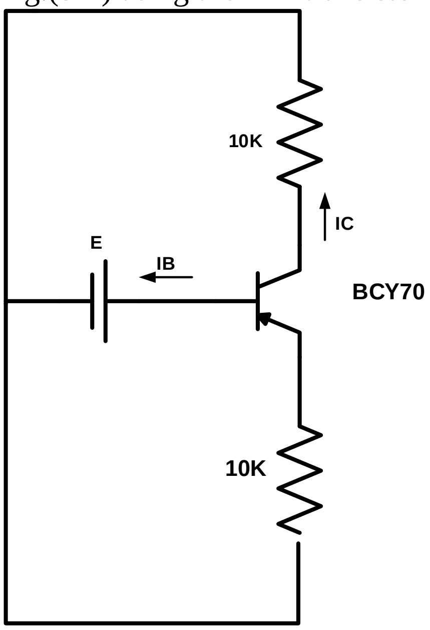


Fig.(3-1)

Switch on the supply and bring the variable dc up to 4V.Record the I_E, I_C and I_B.
2. Switch off the supply and return the variable d.c knob to min., then reverse the connections of the supply such that both junctions are forward biased. Repeat step 4.
3. Let us now examine what happens when one junction is forward biased and one is reverse biased .

Connect the circuit as seen in Fig.(3- 2) .Which junction is forward biased and which is reverse biased?

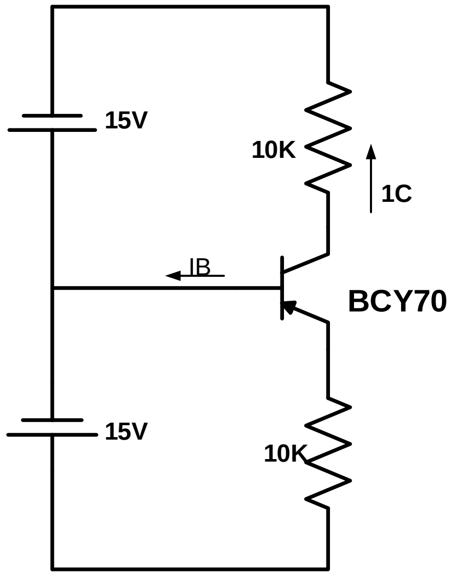


Fig.(3-2)

4. Repeat step 1 to 3using npn transistor, reverse the supplies terminals. Switch on the supply Are the currents now similar to those recorded for pnp transistor?

<u>II. The Transistor DC Parameters:</u>

<u>A : Input Characteristic:</u>

1. Connect up the circuit as shown in Fig.(3-3). Turn the variable dc knob to min, and the 10k potentiometer to zero.

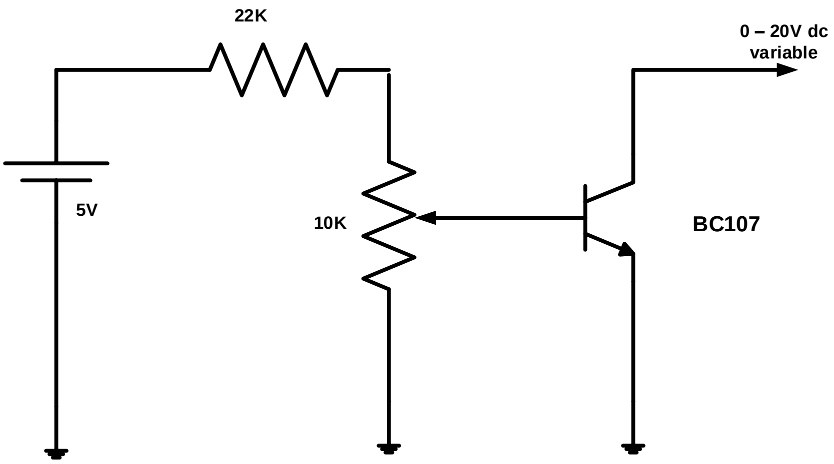


Fig.(3-3)

2. Switch on the power supply. For V_{CE} = 0V and I_B = 0 µA, measure VBE.
3. Rotate the 10K potentiometer slowly clock wise till base current is 5 µA .Measure V_{BE} for V_{CE} is still 0 V. Repeat for I_B as shown in Table 3.1 , and continue for other values of V_{CE}.
4. Plot a graph of V_{BE} against I_B .V_{BE} on the y.-axis.

Table 3.1

V_{CE}	I_B [μA]	V_{BE} [V] Measure
0	0	
	5	
	10	
	15	
0.2	0	
	5	
	10	
	15	
0.4	0	
	5	
	10	
	15	

B:Forward Current Transfer Characteristic:

1. For the same circuit of Fig.(3-3), return the pot. to zero and reset V_{CE} to 2.5V. Measure I_C for I_B as in table 3.2
2. Repeat step (1) for V_{CE} as in table 3.2
3. Plot graph of I_C against I_B with I_C on the X -axis.

Table 3.2

V_{CE}	I_B [μA]	I_C [mA]←
2.5	0	
	5	
	10	
	15	
5	0	
	5	
	10	
	15	
15	0	
	5	
	10	
	15	

C: Reverse Voltage Characteristic:

1. For the same circuit of Fig.(3-3), switch off the supply. Ensure that the 10K pot. is set to zero and variable knob is at minimum
2. Switch on the supply and let I_B= 2.5μA .Record V_{BE} for V_{CE} settings as in Table 3.3
3. Plot graphs of V_{BE} against V_{CE}, V_{BE} on the Y-axis..

Table 3.3

I_B [μA]	V_{CE} [V]	V_{BE} [V] Measure
2.5	0	
	0.5	
	2	
	5	
	15	
5	0	
	0.5	
	2	
	5	
	15	
15	0	
	0.5	
	2	
	5	
	15	

D: The output Characteristics:

1. For the same circuit of Fig.(3-3) .Ensure that dc knob is set to min. and the pot. is set to zero. Switch on the supply.
2. Set the pot. so that I_B is 2.5µA . Record I_C for V_{CE} settings as in Table 3.4
3. Plot graph of V_{CE} against I_C with V_{CE} on X- axis..

Table 3.4

I_B [µA]	V_{CE} [V]	I_C [mA] <= Measure
2.5	0	
	0.5	
	2	
	5	
	15	
5	0	
	0.5	
	2	
	5	
	15	
15	0	
	0.5	
	2	
	5	
	15	

Question:

:From your graphs, calculate the four h-parameters such that :

.

- $H_{IE} = (\Delta V_{BE}/\Delta I_B)|_{vce=0}$.----------------------> input characteristic
- $H_{RE} = (\Delta V_{BE}/\Delta V_{CE})|_{ib=0}$.--------------------> reverse characteristic
- $H_{OE} = (\Delta I_C/\Delta V_{CE})|_{ib=0}$.----------------------> output characteristic
- $H_{FE} = (\Delta I_B/I_C)|_{vce=0}$.------------------------> forward current transfer characteristic.

Experiment #4 ENEE3102

BJT Transistor As An Amplifier, CE, CC, CB Connection

Objectives:

3. To investigate the effect of applying sinusoidal signal to a transistor connected in common emitter.
4. To investigate the properties of the transistor amplifier in common emitter, common collector, and common base connection.

Pre-lab Work

You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab

Procedure:

I. COMMON EMITTER TRANSISTOR AMPLIFIER.

1. Connect the circuit of Fig.(4-1).

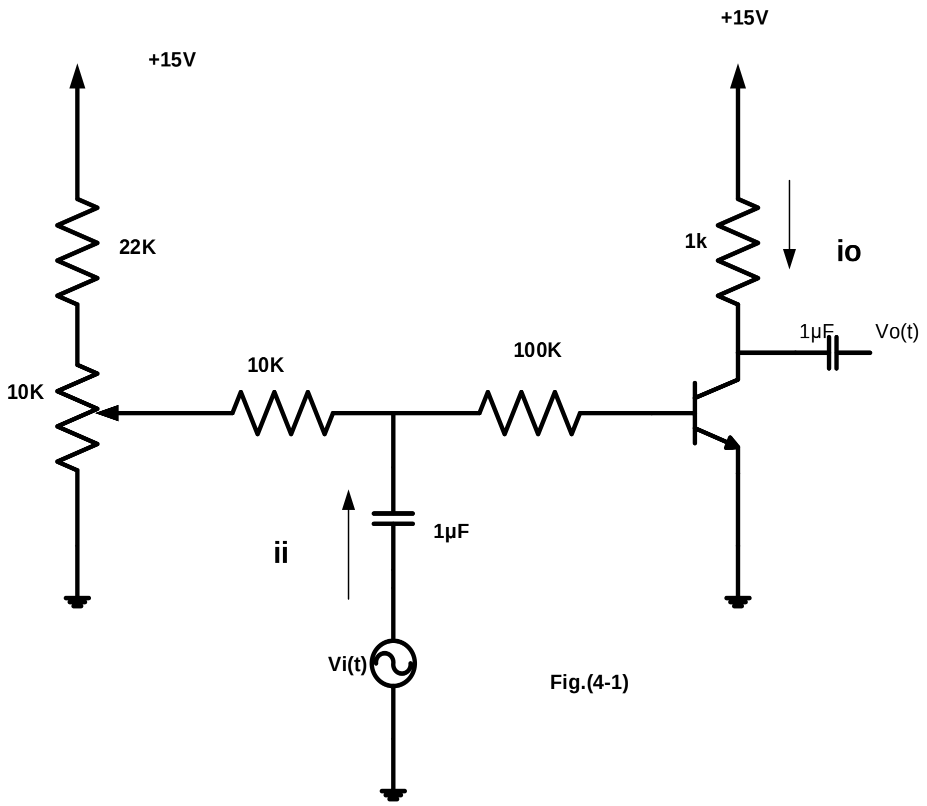


Fig.(4-1)

2. switch on the power supply and the function generator.
3. Set the function generator frequency to 1KHz sine wave and amplitude to zero.
4. Adjust the base bias potentiometer for a DC collector voltage(V_C) of 8 volts.
5. Switch on the oscilloscope and connect its channels to the base and the output of the circuit.
6. Turn up the function generator output until the output of the circuit is 8 volts peak-to –peak.

7. Calculate the voltage gain of the transistor ,Vo/Vbe.

8. Calculate the voltage gain of the amplifier , Vo/Vi

9. Using DMM measure the AC currents for both the base and the collector of the transistor.

10. Calculate the current gain of the transistor Ic /Ib.

11. Using DMM measure the AC currents for both io and ii.

12. Calculate the current gain of the amplifier , Io /Ii.

13. Calculate the input impedance of the amplifier , Zi = Vi /Ii .

II. COMMON COLLECTER TRANSISTOR AMPLIFIER.

1. Connect the circuit as shown in Fig.(4-2).

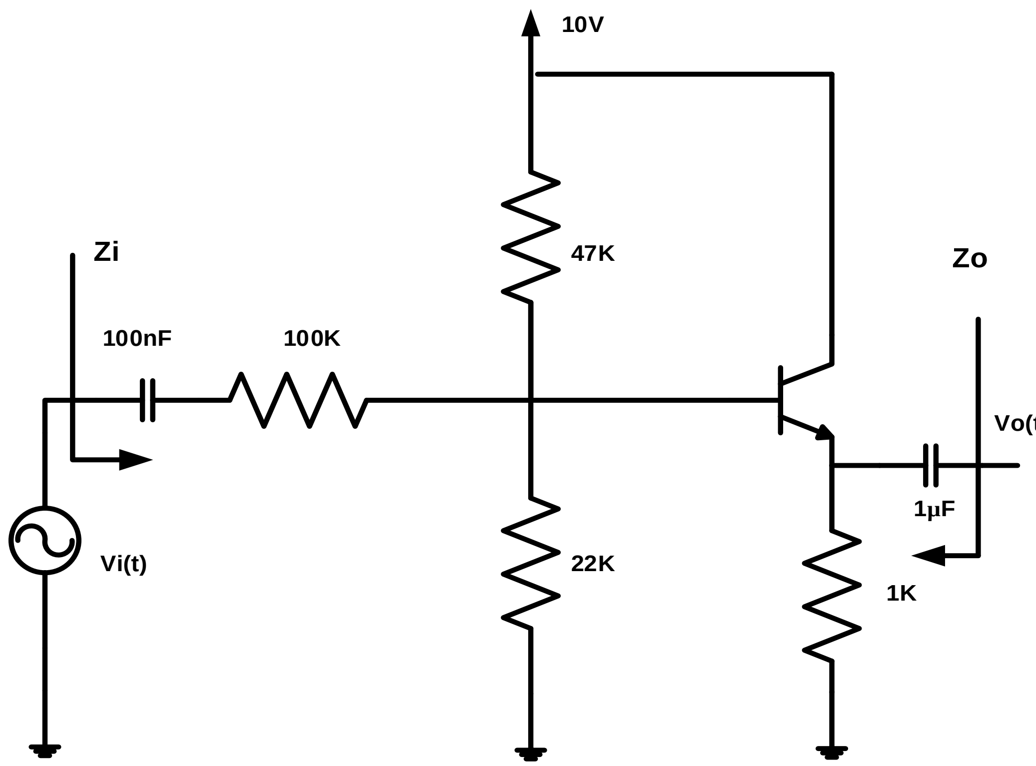


Fig.(4-2)

2. Ensure that the variable dc control is at min.

3. Switch on the power supply and adjust the variable dc voltage to give a Vcc of +10 volts.
4. Set the sine wave generator to a frequency of 1KHz ,but either disconnect its output ,or turn its output amplitude to zero, so there is no signal input to the circuit.

5. Measure the quiescent bias voltages of the circuit,V_E and V_B ,using DVM.

6. Increase the output amplitude of the sine wave generator until an output amplitude from the amplifier is about 2volts peak-to-peak.

7. Measure the ac input voltage needed to achieve this output .

8. Calculate the voltage gain.

9. Measure the ac voltage across the 100KΩ input resistor .

10. Calculate the input current using your measured value of voltage across the input resistor.

11. From the output voltage and the load resistor value calculate the ac output current.

12. Calculate the current gain.

13. From your measured values you can calculate the input impedance Z_{in}.

14. To find the output impedance of the amplifier, you should take off the input sine wave generator and replace it with a short circuit, then you have to connect the generator to the output (emitter) via a capacitor, and measure its output voltage and current.

15. Enter your results in table 4.1.

Quantity	**Measured values**
V_{in}	
V_{out}	
i_{in}	
i_{out}	
	Calculated values
$A_V = V_{out}/V_{in}$	
$A_i = i_{out}/i_{in}$	
$Z_{in} = V_{in}/i_{in}$	
Z_{out}	

Table 4.1

Questions:

- How is the output quiescent voltage related to the input?
- How do the parameters compare with those of the common emitter stage?

III. COMMON BASE TRANSISTOR AMPLIFIER.

1. Connect the circuit of Fig. (4.3).

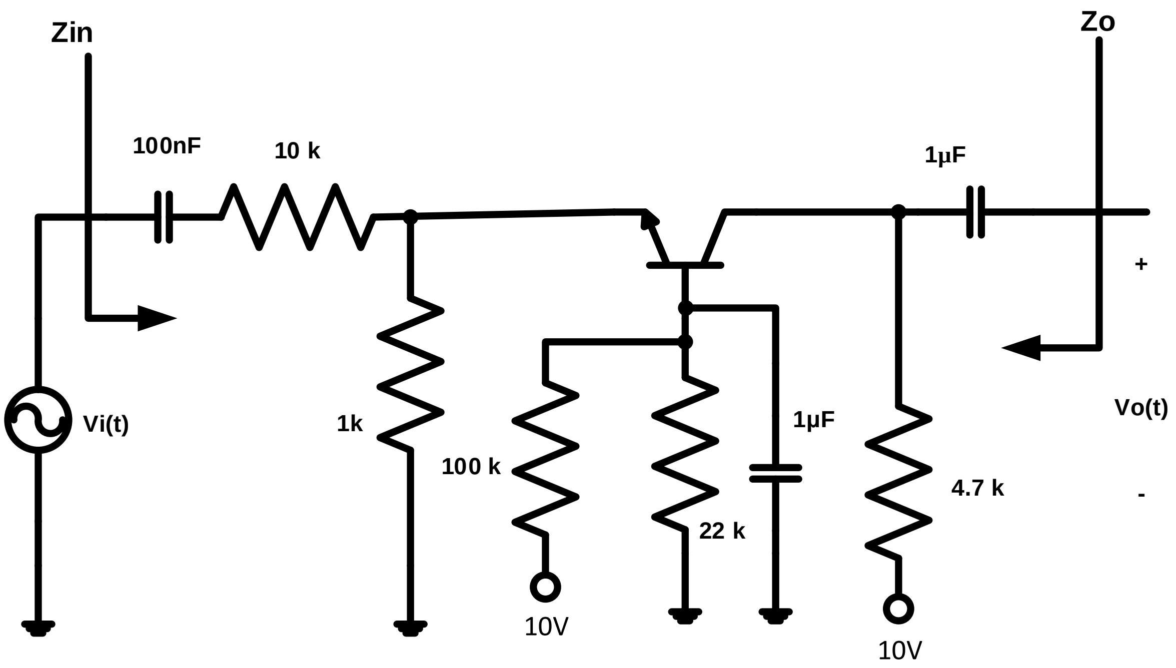


Fig.(4-3)

2. Ensure that the variable dc control is at min.
3. Switch on the power supply and adjust the variable dc voltage to give a Vcc of +10 volts.
4. Set the sine wave generator to a frequency of 1 kHz ,but either disconnect its output ,or turn its output amplitude to zero, so there is no ac signal input to the circuit.
5. Measure and record the quiescent bias voltages and currents I_B, I_C, V_{BE} , V_{BC} and V_{CE}. ,using DVM.
6. Increase the output amplitude of the sine wave generator until an output amplitude from the amplifier is about 2volts peak-to-peak.
7. Measure the ac input voltage needed to achieve this output. What happens if the ac input is increased further?
8. Calculate the voltage gain Av.

9. Measure the ac voltage across the 10 kΩ input resistor.

10. Calculate the input current using your measured value of voltage across the input resistor.

11. From the output voltage and the load resistor value calculate the ac output current.

1. Calculate the current gain.

2. From your measured values you can calculate the input impedance Z_{in}.

3. To find the output impedance of the amplifier, you should take off the input sine wave generator and replace it with a short circuit, then you have to connect the generator to the output (collector) via a capacitor, and measure its output voltage and current.

4. Enter your results in table 4.2.

Quantity	**Measured values**
V_{in}	
V_{out}	
i_{in}	
i_{out}	
	Calculated values
$A_V = V_{out}/V_{in}$	
$A_i = i_{out}/i_{in}$	
$Z_{in} = V_{in}/i_{in}$	
Z_{out}	

Table 4.2

Questions:

- How is the output quiescent voltage related to the input?
- How do the parameters compare with those of the common emitter stage?

Experiment #5 ENEE3102

The Field-Effect Transistor

Objectives:

1. To understand the difference between the bipolar and the field effect transistor.
2. To examine the characteristics of N-channel JFET when using as a common source and common drain.

Pre-lab Work:

You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab.

Procedure:

I. CHARACTERESTICS OF AN N-CHANNEL JFET.

14. Connect the circuit of Fig.(5-1).

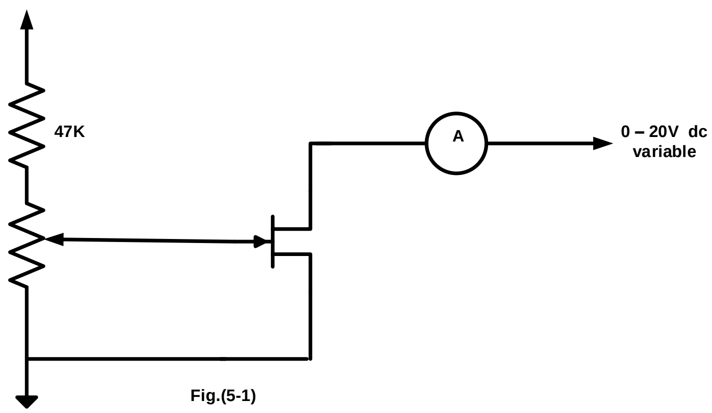


Fig.(5-1)

15. Set the potentiometer anti-clockwise and the variable dc voltage to zero.

16. Switch on the power supply.

17. Set the V_{DS} to the first value in table 5.1 , and then read I_D for each value of V_{GS}.

18. Repeat for all the values of V_{DS} in the table ,recording the corresponding I_D values.

19. Plot the results from your table onto your graph ,drawing one curve of I_D against V_{DS} for each value of V_{GS}.

	I_D(mA) for V_{DS}=(V)						
Vds / Vgs	**0**	**0.5**	**1**	**2**	**5**	**10**	**15**
0							
-0.5							
-1.0							
-1.5							
-2.0							
-2.5							

Table 5.1

20. Now go back to your circuit and set V_{DS} to 10 V and V_{GS} to -1.0 V ,then try to measure I_G.

Questions:

- From your graph ,above which values of V_{DS} is I_D almost unaffected by V_{DS} when V_{GS}=0?
- For a given value of V_{DS} , (say 10 V),do equal changes of V_{GS} cause equal changes of I_D?
- Can you measure I_G or is it too small?
- From your graph ,estimate the change in I_D for 0.5 change in V_{GS} when V_{DS} =10 V , and V_{GS} -1.0 V ,then find the transconductance of the transistor(g_m).

Note: transconductance = g_m =(change in I_D)/(change in V_{GS}).

II. A JFET AMPLIFIER.

16. Connect the circuit as shown in Fig. (5-2).

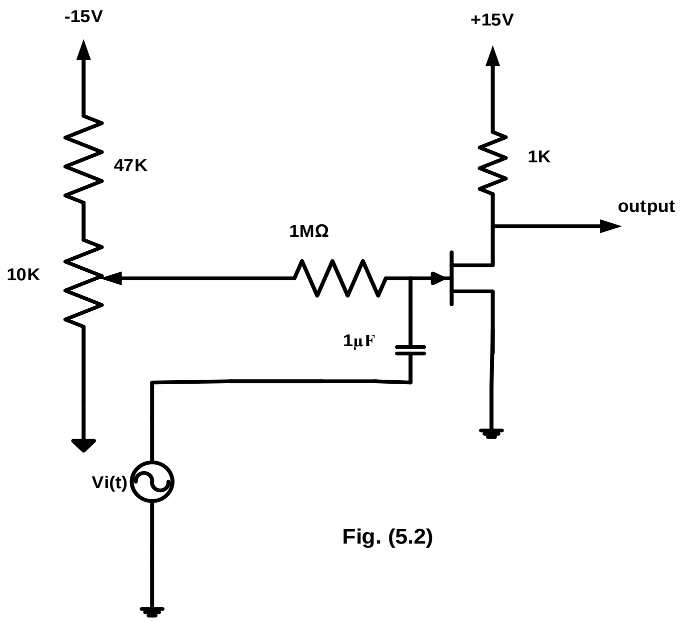


Fig. (5.2)

17. Set the sine wave generator to a frequency of 1 kHz ,but either disconnect its output ,or turn its output amplitude to zero, so there is no signal input to the circuit.

18. Set the potentiometer to give a value of +10 V for V_{DS}.

19. Now apply an input of 2volts peak-to-peak from the generator and observe the output on the oscilloscope.

20. Measure the peak-to-peak output voltage and calculate the voltage gain .

21. Now Measure the ac input current and voltage using DMM and calculate the input impedance Zin seen by the source Vi(t)

III. COMMON DRAIN AMPLIFIER.

1. Connect the circuit as shown in fig. (5-3).

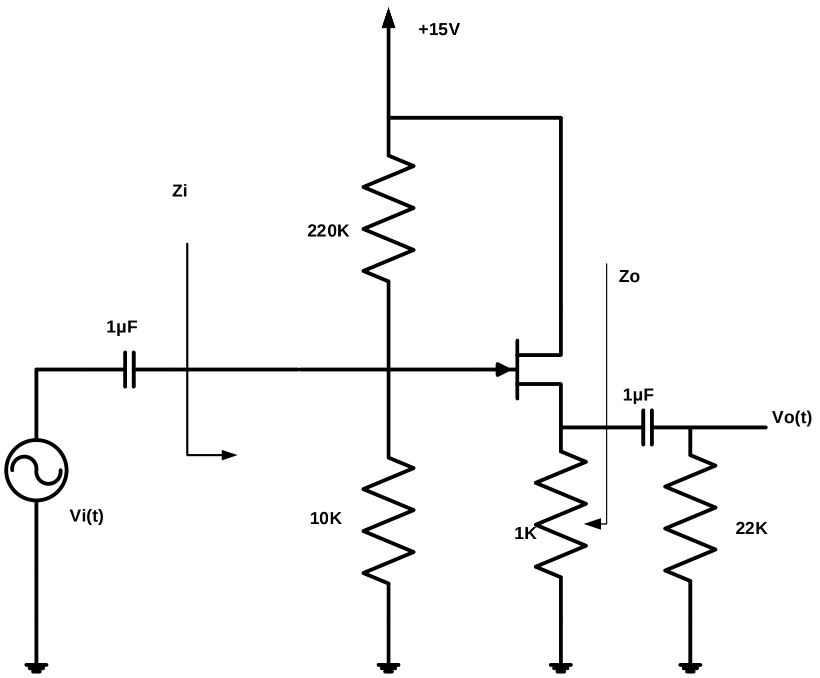


Fig.(5-3)

2. Set the sine wave generator to a frequency of 1 kHz ,but either disconnect its output ,or turn its output amplitude to zero, so there is no signal input to the circuit.

3. Measure the DC voltages of V_G, and V_S.

4. Now apply an input of 0.4 volts peak-to-peak from the generator and observe the output on the oscilloscope.

5. Calculate the voltage gain and the phase shift between the input and output voltage.

6. Measure the values of Z_{in} and Z_{out} using the appropriate voltages and currents at the places shown in the previous figure.

Question:

- Compare the voltage gain of step 5 with the theoretical gain value.

III. CONSTANT CURRENT SOURCE.

- Connect the circuit as shown in fig.(5-4).

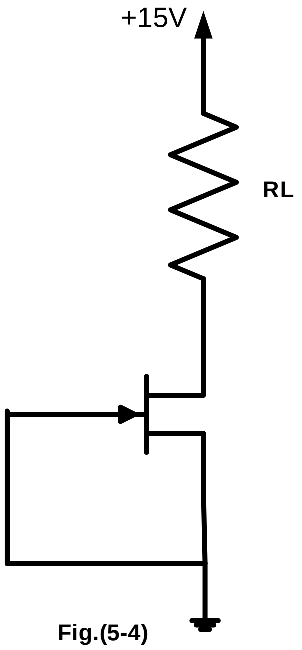


Fig.(5-4)

- Measure the values of the I_D ,and calculate V_R for each value of the resistor ,then record your result in a table like table 5.2

R_L(KΩ)	I_D(mA)	V_L(V)
0.1		
0.22		
0.33		
0.47		
0.56		
1		
1.5		
2		
3		

Table 5.2

Question:

- Calculate the maximum theoretical value of load resistance for which the JFET- current source can provide constant current.
- Comment on the measured range of load resistance in comparison to the theoretical range of load resistance.

Experiment #6 ENEE3102

Multistage Amplifiers and Frequency Response

Objectives:

1. To investigate what happens when transistor amplifier stages are connected one after another.
2. To investigate the effect of frequency changes on the gain of the amplifier

Pre-lab Work:

You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab.

Procedure:

A. ***MULTISTAGE AMLIFIER***

I. MULTISTAGE AMPLIFIER DESIGN.

1. Let us design a two stage amplifier with a voltage gain of 50 to give a peak to peak output of 5V , As shown in Fig.(6-1) : $Av_1 = 25$, $Av_2 = 2$, Vi = 100 mVp-p .

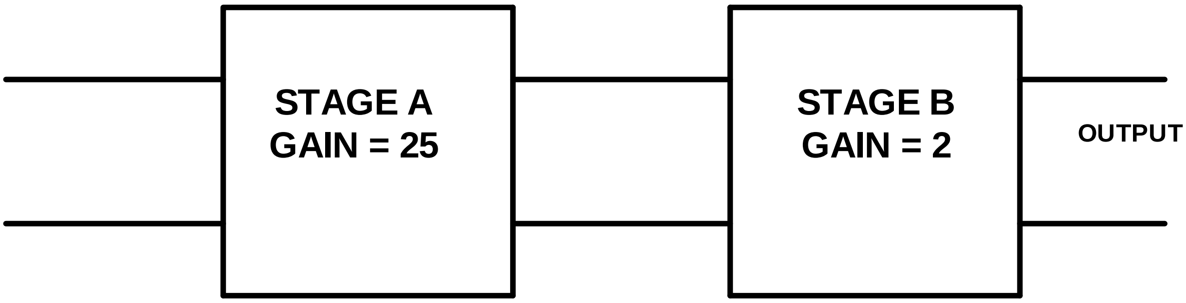


Fig.(6-1)

Design the first stage of the amplifier for the h- parameters of a BC107 transistor are : hie = $4x10^3$ Ω , hoe = 10^{-4} ℧ , hfe = 300 , hre = 10^{-3} ,V_{CC} =10V.

2. Use a bypass capacitor of 100 μF in parallel with R_E of first stage , the second stage emitter resistor is left un-bypassed.

3. Connect up the first stage of the circuit you have designed.

4. Ensure that the variable dc control knob is at min. Switch on the PSU and adjust the variable dc voltage to give Vcc of +10V.

5. Measure the quiescent voltage of the collector, the emitter and base with respect to ground. (Measure V_{B1}, V_{E1}, V_{CE1})

6. Connect the sine wave generator via a 100 μF capacitor to the base of the transistor Q1, and adjust the generator output to 100mVp-p at f = 1 kHz.
7. Measure the output from the first stage, and compare this with the required output.

Question:

- What is the gain of your stage

8. Connect up the second stage of the circuit you have designed.

9. Ensure that the variable dc control knob is at min. Switch on the PSU and adjust the variable dc voltage to give Vcc of +10V.

10. Measure the quiescent voltage of the collector, the emitter and base with respect to ground. (Measure V_{B2}, V_{E2}, V_{CE2})

11. Connect the sine wave generator via a 100µF capacitor to the base of the transistor Q2, and adjust the generator output to 2.5Vp-p at 1 kHz frequency.

12. Measure the output from the second stage.

13. The two stages must be now connected together to form the complete amplifier.

14. Ensure that the variable dc control knob is at min. Switch on the PSU and adjust the variable dc voltage to give Vcc of +10V. Make sure that both transistors are still biased in the linear region.

15. Adjust the generator connected to the input of first stage to 100mVp-p at f = 1 kHz and measure the output from the second stage.

Questions:

- What is the voltage gain of the two stage amplifier?
- What would happen if the coupling capacitor used did not have negligible impedance at 1 kHz?

Note:
Keep the circuit connected for the next part.

II. FREQUENCY RESPONSE.

1. For the same circuit of part I , adjust the output amplitude of the sine wave generator to a suitable value and measure the output amplitude from the amplifier for the frequency 50Hz , 100 Hz ,200,300,400,500Hz, 1 kHz , 50 , 30, 100 ,300 , 500, 700, 1000 and 2000 kHz
2. Calculate Vout/Vin for each frequency.

3. Record your results as in table 6.1.

Frequency(Hz)	Vin(pk-pk)	Vout(pk-pk)	Av	log f	20 log (Av)
50					
60					
80					
100					
200					
300					
400					
500					
1k					
10k					
30k					
100k					
300k					
400k					
500k					
700k					
1000k					
2000k					

Table 6.1

1. Plot a graph of magnitude frequency response (20 log (Av) **vs** log f) i.e. dB/Decade scale.

Questions:

- Does the output amplitude vary with frequency?
- What happens to the output amplitude at low frequency?
- What causes this effect at low frequencies?
- What would you expect happen to the gain of the circuit at high frequencies?
- Between which frequency limits does the amplifier have a constant gain?
- At which frequencies is the gain 0.707 times the maximum gain?
- What is the difference between these two frequencies? Mark these two points on your graph.

III. DIFFERENTIAL AMPLIFIER.

1. Set up the circuit as shown in Fig. (6.2).

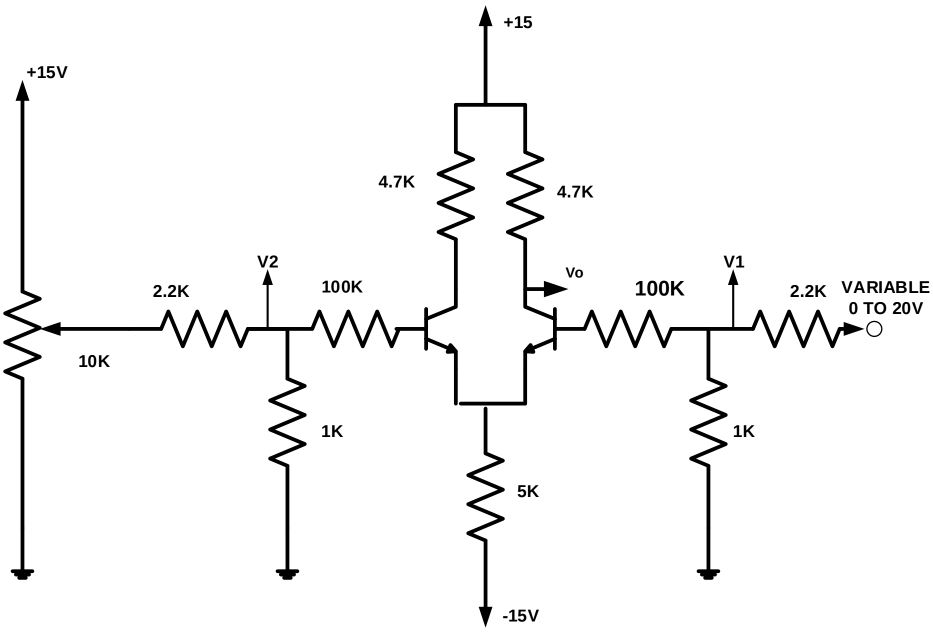


Fig.(6-2)

2. Switch on the power supply .
3. Measure the output voltage Vout, for input voltage V1 and V2 of 0, 1, 2, 3, 4 and 5V.

4. Tabulate your results as in table 6.2.

V1=V2(V)	Vout(V)
0	
1	
2	
3	
4	
5	

Table 6.2

5. Plot a graph of Vout against V1 = V2 .

Questions:

- What is the slope of the graph ?
- As Vout = Ac Vc, what does the slope of the graph represent ?
- What is your experimental value of Ac ?
- If Vc is made zero instead of Vd, what will the expression of Vout become?

6. Set up the circuit as shown in Fig. (6.3).

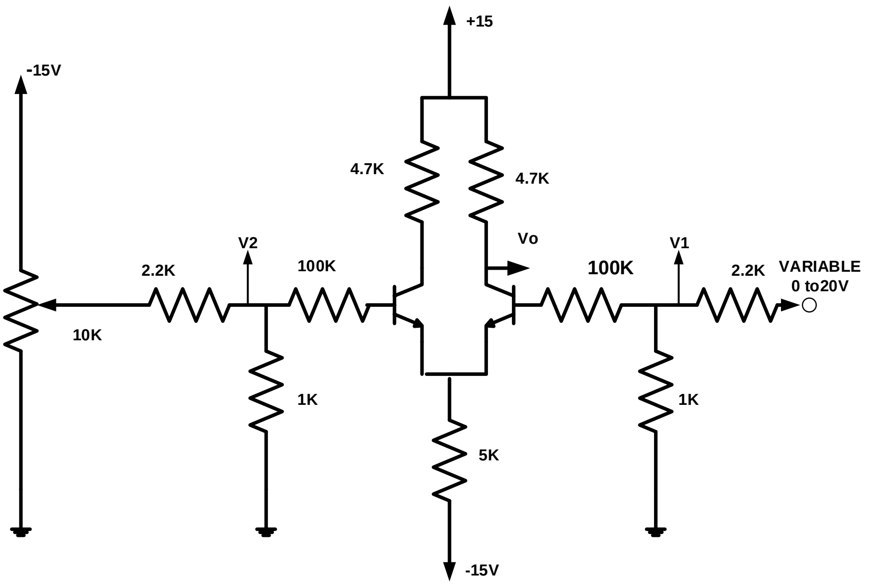


Fig.(6-3)

7. Connect a voltmeter between each of the amplifier inputs(V1 and V2) and 0 volt, and set both inputs at 0 volts .

8. Record the resultant Vout .

9. Measure and record Vout for V1, V2 as indicated in table 5.3.

V1(V)	*V2(V)*	*Vout(V)*
0	***0***	
0.05	***-0.05***	
0.1	***-0.1***	
0.15	***-0.15***	
0.2	***-0.2***	

Table 6.3

10. Plot a graph of these results, and calculate the slope of the graph .

Questions:

- What does the slop represent ?
- What is your experimental value of Ad ?
- From the Ac and Ad values found, calculate the common mode rejection ratio of the amplifier .

Experiment #7 ENEE3102

Power Amplifiers

Objectives:

1. To understand the classes of power amplifier.
2. A knowledge of the design of a power amplifier using push-pull techniques.

Prelab:

You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab.

PROCEDURE:

I. THE CLASSES OF POWER AMPLIFIER.

1. Connect the circuit of fig.(7-1)

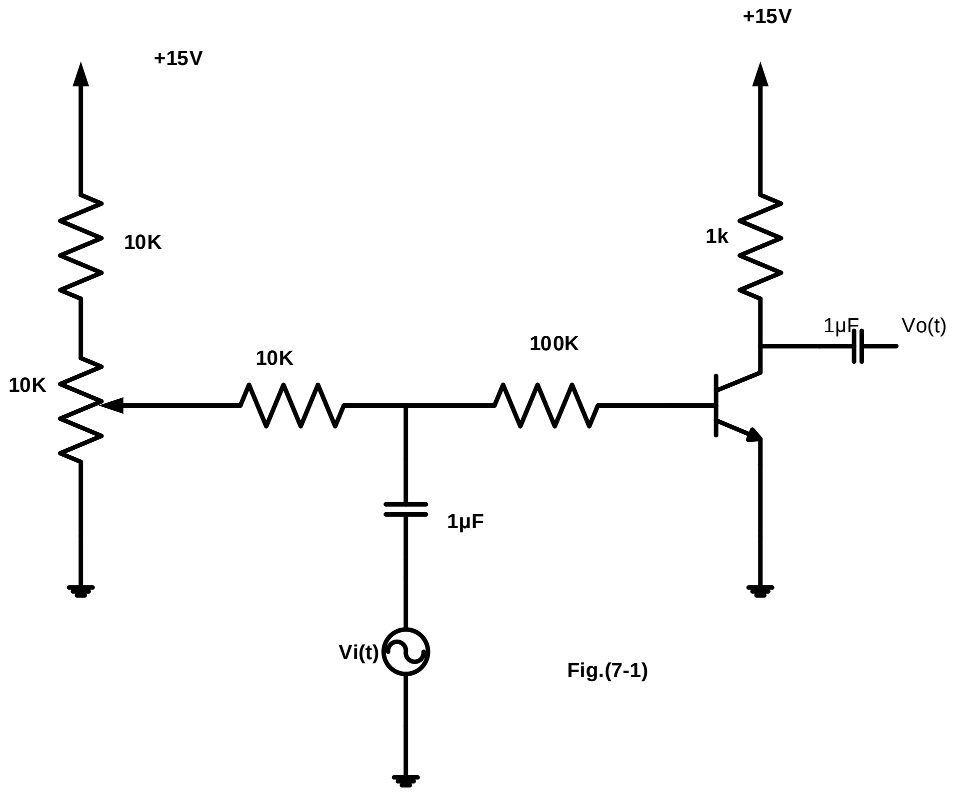


Fig.(7-1)

2. Switch on the power supply, function generator, and oscilloscope.

3. Start with the bias control at zero (fully clockwise).

4. Connect the oscilloscope channels to the input and output of the circuit.

5. Turn up the output of the function generator until you see a waveform that shown in Fig.(7-2.a),comparison between the input waveform and the output waveform shows distortion due to insufficient bias.

6. Determine the class of the amplifier for this case.

7. Turn down the signal amplitude a little , and turn up the bias potentiometer, the waveform should look like Fig.(7-2.b).

8. Again, determine the class of amplifier.

9. Turn the bias up further, and the output waveform should look like Fig.(7-2.c),the transistor is now saturating.

10. Set the function generator output to zero, then reset the bias to give a collector voltage of about 7.5V on the meter , and an output like Fig.(7-2.b), then turn up the signal amplitude until a waveform like fig.7.2(d) is displayed.

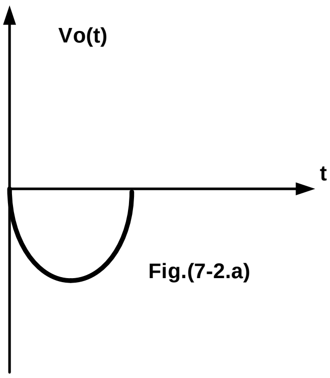


Fig.(7-2.a)

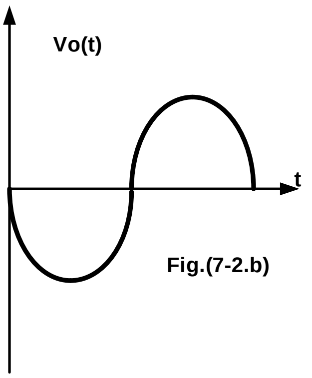


Fig.(7-2.b)

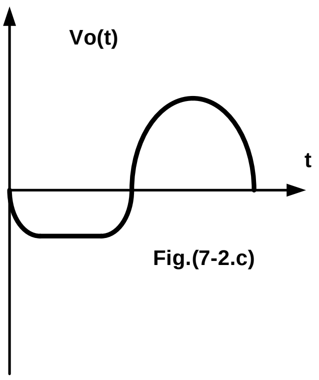


Fig.(7-2.c)

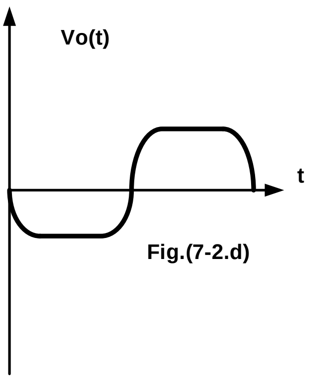


Fig.(7-2.d)

II. PUSH-PULL AMPLIFIER.

1.Connect the circuit as shown in Fig.(7-3).

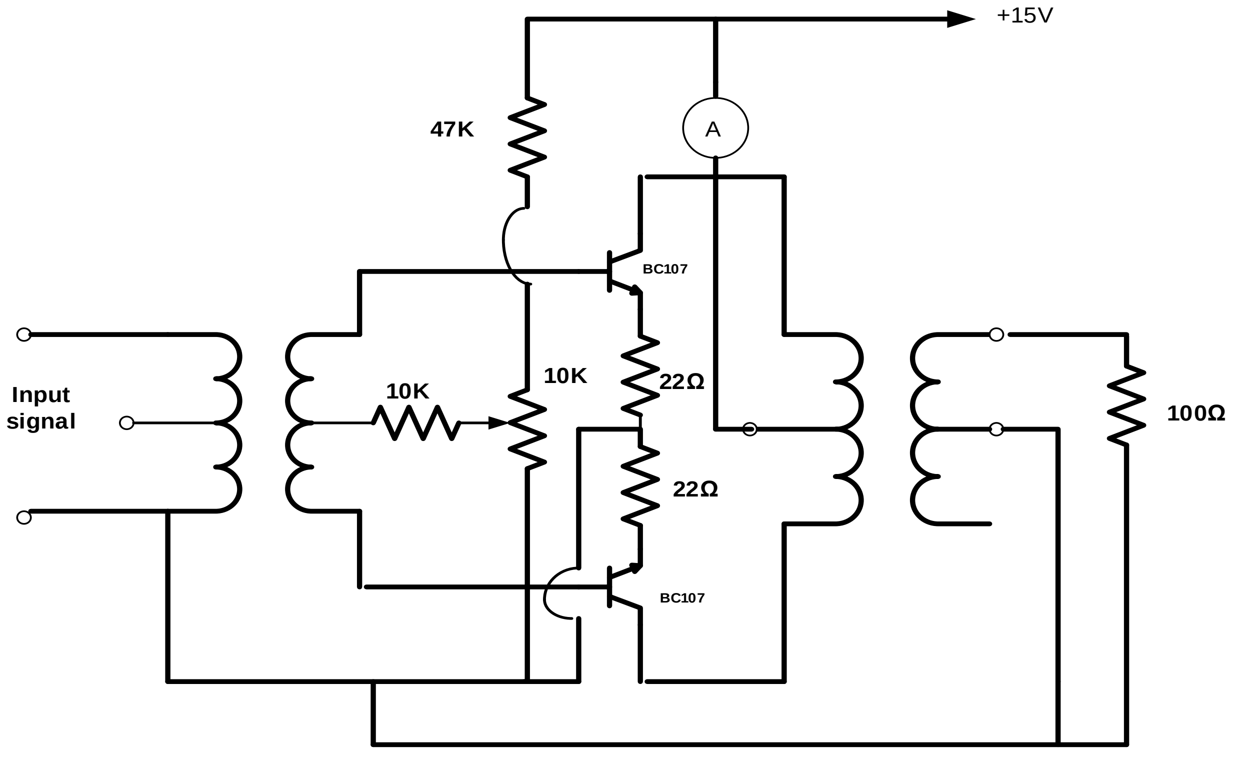


Fig.(7-3)

2. Switch on the equipment and set the DC supply current to zero by setting the bias potentiometer fully anticlockwise to zero.
3. Set the function generator output to zero.
4. Turn up the function generator output to give a peak-to-peak voltage of 4 volts at 1KHz.
5. Switch on the oscilloscope and connect its channels to the input and output of the circuit .
6. The output waveform should shows a severe crossover distortion.
7. Reduce the input to 0.5 volt peak-to-peak, notice that the output disappears.
8. Slowly apply bias by turning the potentiometer clockwise, observing the output waveform, but do not exceed 10mA on the meter.
9. Now set the function generator so that the output of the amplifier is 4V peak-to-peak and adjust the bias potentiometer to eliminate crossover distortion.
10. Measure the output peak-to-peak voltage and record it in a table like table 7.1.

Load Resistor (Ω)	Output Voltage (volts)		Output Power (mW)
	Peak-to-peak	RMS	
320			
220			
150			
100			
69			
50			
41			

Table 7.1

11. Change the load resistor for the other values in the table, entering the output voltage each time.

12. Convert each output voltage reading to its root mean square equivalent by using the formula:

$$Vrms= \frac{VPK-PK}{2\sqrt{2}}$$

From this the output power can be calculated using the formula:

$$Power=(V_{rms})^2/R_{load}.$$

13. Enter the power output in the result table.

14. Select the resistor that matches the output impedance most closely, and replace the load resistor with it.

15. Turn up the function generator output for maximum undistorted output waveform .

16. Measure the output power of the amplifier by measuring the peak-to-peak output voltage .

17. Measure the supply current and using the formula :

$$Pin=V_{supply}*I_{supply}.$$

Calculate the input power to the amplifier.

18. Calculate the efficiency of the amplifier.

Questions:

- Which value of load resistor gives maximum power?
- What does this imply about the output impedance of the amplifier?

III. COMPLEMENTARY PUSH-PULL AMPLIFIER.

1. Connect the circuit as shown in fig. (7-4).

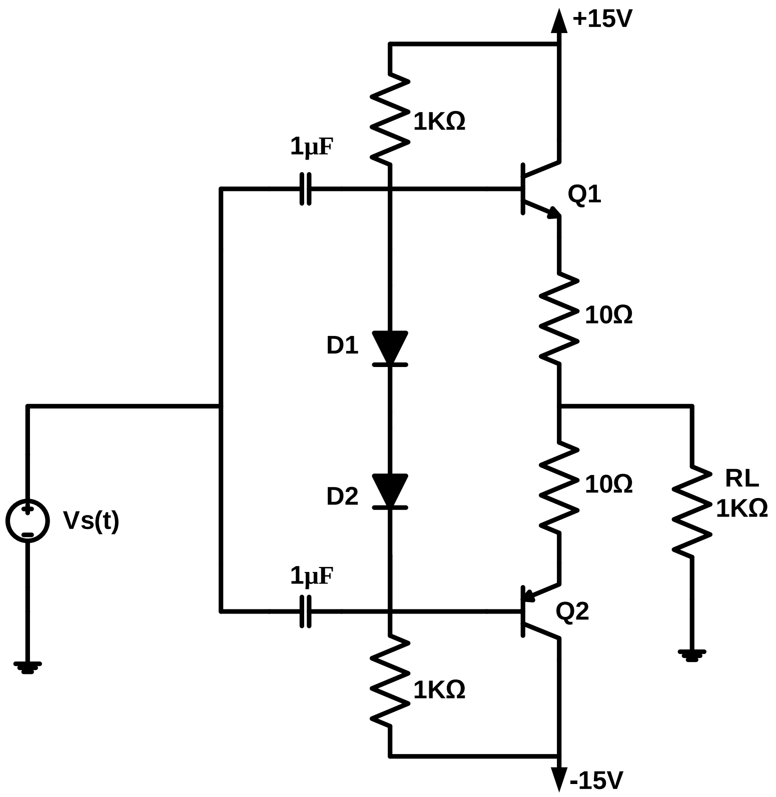


Fig .(7-4)

2. Turn up the function generator output to give 1 volt peak - to - peak at 2KHz.
3. Calculate the power efficiency of the amplifier.
4. Repeat steps (1 – 3) for RL = 100Ω .

Experiment #8 ENEE3102

The SCR,DIAC ,and UJT

Objectives:

1. To investigate the SCR triggering, breakdown voltage, and holding current.

2. To investigate the DIAC and UJT.

Prelab:

You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab.

PROCEDURE:

I. THE SCR.

1. Connect the circuit of Fig. (8-1).

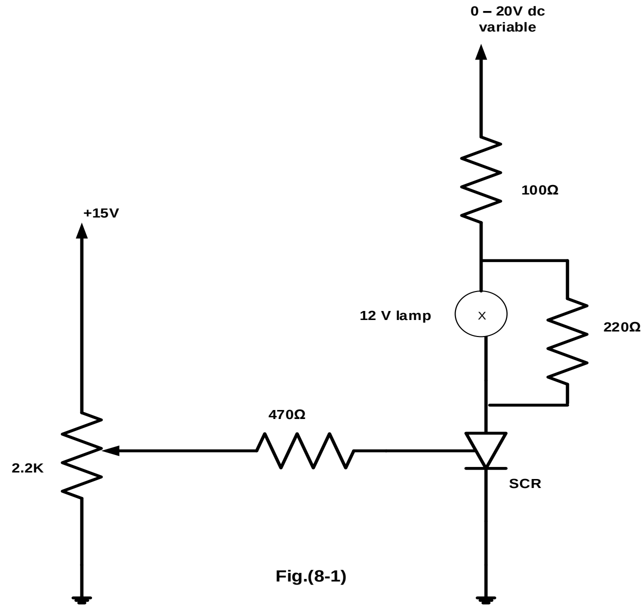


Fig.(8-1)

2. Set the variable dc voltage to 12V,turn the potentiometer to zero(clockwise), and switch on the supplies.

3. Slowly rotate the potentiometer, observing the gate current meter continuously, until the lamp suddenly lights, record the gate current at which this occurs.

4. Switch off the supplies, and return the potentiometer to zero.

5. Repeat the measurement several times, to ensure that you have the correct value of the trigger current I_{GT}.

6. Now trigger the SCR on again and measure the voltage from anode to cathode, this is the saturation voltage V_{AK}(sat).

Question:

- Do you expect the saturation voltage to be greater than 0.6V?explain.

7. Finally connect the DMM in series with the lamp ,trigger the SCR on .

8. temporary disconnect the gate connection ,slowly reduce the supply voltage until the lamp has no light, note the value of current at which this occurs.

9. Repeat this procedure several times to ensure that you have the correct value, what you have found is the holding current (I_H).

Question:

- What do you think will happen ,if you trigger the SCR on and then reduce the gate current to zero again? confirm your answer by experiment.

10. Replace the 12Vdc source with 12V ac rms.

11. Repeatedly increase and decrease the gate current, notice and explain what you think is happening.

Question:

- Why does the lamp burn less brightly than it did with 12 V dc supply?

12. Using the oscilloscope monitor and draw, the anode voltage with respect to the input voltage.

II. THE DIAC.

1. Connect the circuit as shown in Fig. (8-2).

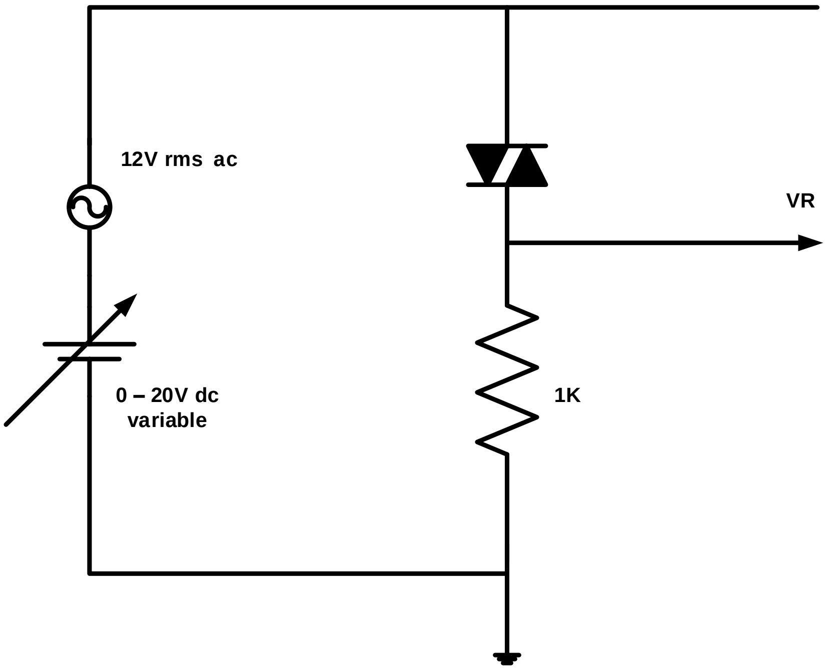


Fig.(8-2)

2. Set the variable dc supply to zero and switch on the supplies.

3. Switch on the oscilloscope and connect its channels to the DIAC , and resistor terminals.

4. Measure V_{BR} and V_R from the oscilloscope, note that V_{BR} is the DIAC break over voltage and V_R is the load voltage.

5. Calculate the DIAC current immediately after switch on.

6. After determining the DIAC current and voltage ,try to construct an approximate characteristics of the DIAC.

III. THE UJT.

1. Connect the circuit as shown in Fig. (8-3).

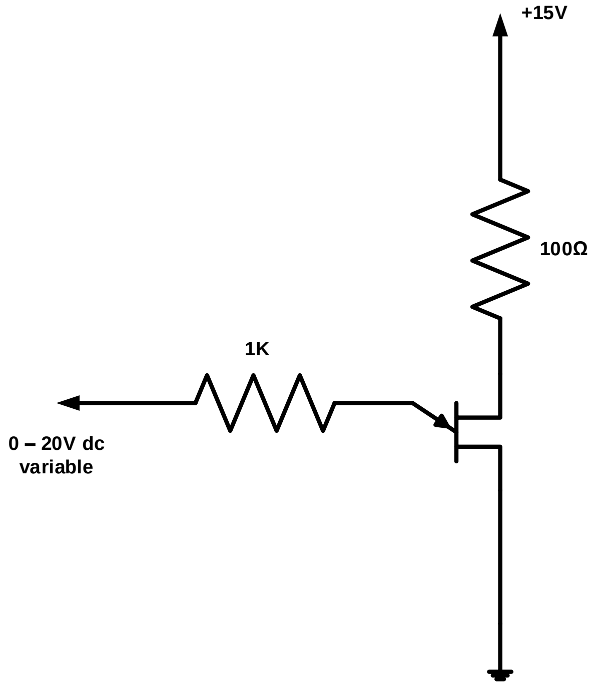


Fig.(8-3)

2. Set the variable dc voltage to zero and switch on the power supply.

3. Slowly increase the variable dc until the emitter current suddenly increases. That is the UJT switches on.

4. Record the value of V_{EB1} just before switch on,also the values of V_{EB1} and I_E just after switch on

5. Slowly reduce the variable dc voltage until the emitter current suddenly switches off again.

6. Record, in a table like table 8.1 ,the values of I_E and V_{EB1} just before switch off, and the value of V_{EB1} just after switch off .

Condition	I_E [mA]	V_{BE} [V]
Just before switch on		
Just after switch on		
Just before switch off		
Just after switch off		

Table 8.1

7. Now you can construct the characteristic for your UJT .

Questions:

- What do you think will happen if V_{EB1} is made negative?
- Study your graphs for the DIAC and the UJT , and determine what are the main differences between the characteristics of the two trigger devices?

IV. USAGE OF A UJT IN A RELAXATION OSCILLATOR.

1. Connect the circuit as shown in Fig. (8-4).

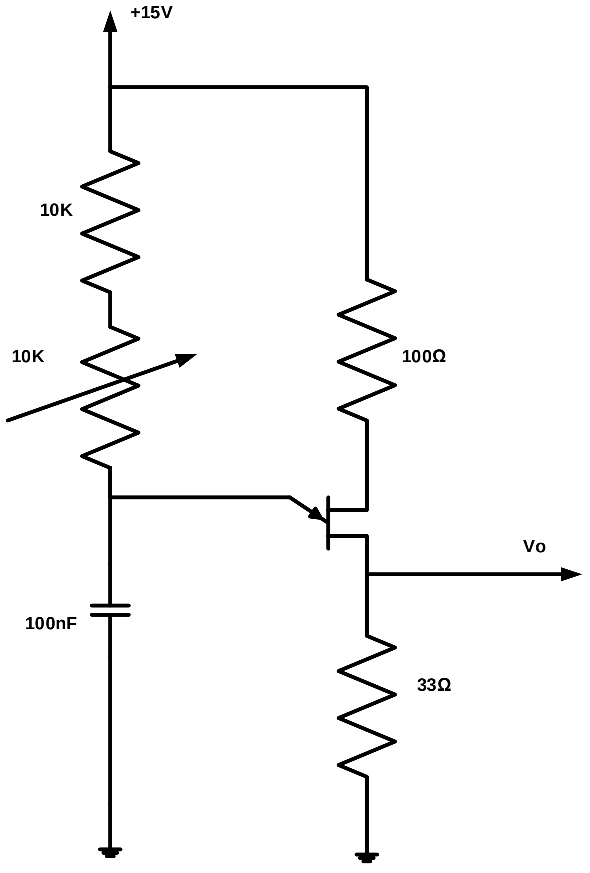


Fig.(8-4)

2. Switch on the power supply .

3. Switch on the oscilloscope and connect its channels to $V_o(t)$,and $V_C(t)$.

4. For variable resistor values(1k,2k,3k,4k,5k),sketch the waveforms of $V_o(t)$,and $V_C(t)$.

5. Measure and record the frequency of $V_o(t)$ in table 8.1.

R(KΩ)	Frequency(KHz)
1	
2	
3	
4	
5	

Table 8-1

Experiment #9 ENEE3102

The Operational Amplifier

Objectives:
To investigate the application of the op. amp circuits such as adding, Voltage follower, Comparator, Integrator and Differentiator.

Prelab:
You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab.

PROCEDURE:
I. Adding Application
1. Set up the circuit of Fig.(9-1), V1 is controlled by the potentiometer and V2, is obtained from the variable dc source on the trainer.

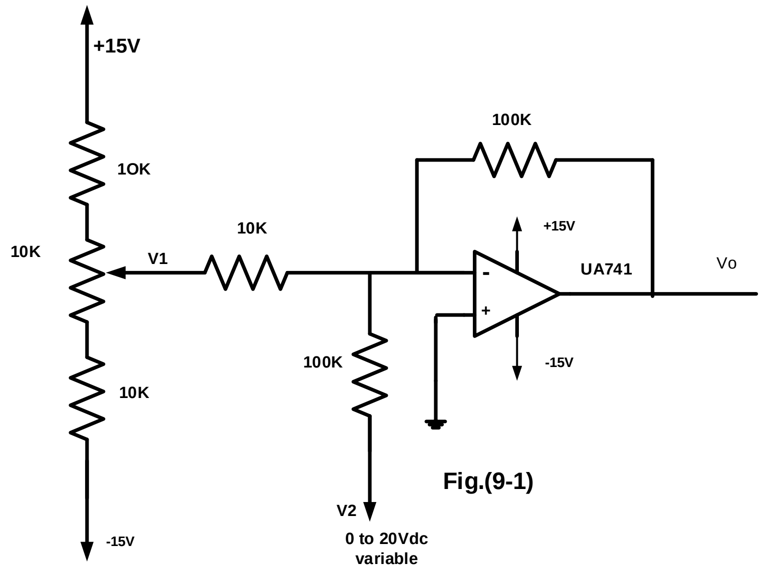


Fig.(9-1)

2. Measure the output voltage for V1, V2 as shown in table 9.1.

Input voltage		Output voltage	
V_1	V_2	V_O	Calculated voltage
0.5	2		
0.1	6		
0.3	4		
-0.9	2		
-1.1	4		
-1.5	6		

Table 9.1

3. Calculate the expected output voltage for each step using the formula :

   $V_o = XV_1 + YV_2$ where X, Y is the resistors ratios.

II. ***Voltage Follower Application***

1. Set up the circuit of Fig.(9-2).

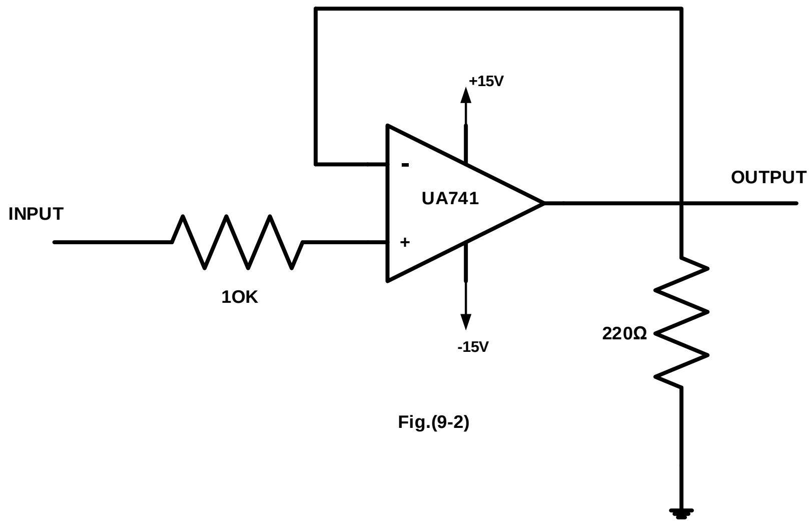


Fig.(9-2)

2. Measure and records Vo for Vi (1V,2V,3V,4V,5V,6V,and 7V).

3. Draw the output Vo(t) for Vi(t) is 2V p-p sinusoidal with 100Hz.

4. Change RL (220Ω) to 1KΩ , then measure and records Vo

   for Vi (6V,8V,10V,12V,,and 15V).

Question:

- Is this circuit has similar properties as the emitter follower. Explain your answer.
- For what applications is this circuit used?
- What is the relation between your Vi, Vo?

III. Comparator Application

1. Set up the circuit of Fig.(9-3).

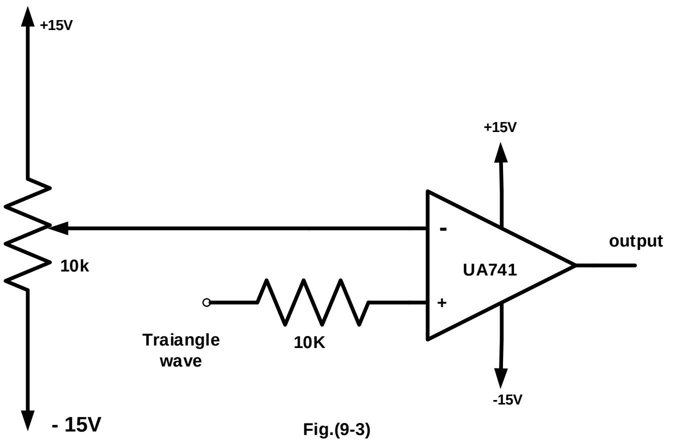


Fig.(9-3)

2. Use 1 kHz triangular input signal from the function generator.
3. Set the triangle input signal to 2 Vp-p and change the dc reference voltage so that you obtain an output of positive Vsat then negative Vsat and a square wave output.
4. For each of these cases draw the output voltage and record the value of the dc reference voltage.

Question:

- What happens to the amplifier output?. For what application is this circuit used?
- Is there any similarity between this circuit and the diff amplifier, what is the shape of the output?

IV. Integrator and Differentiator

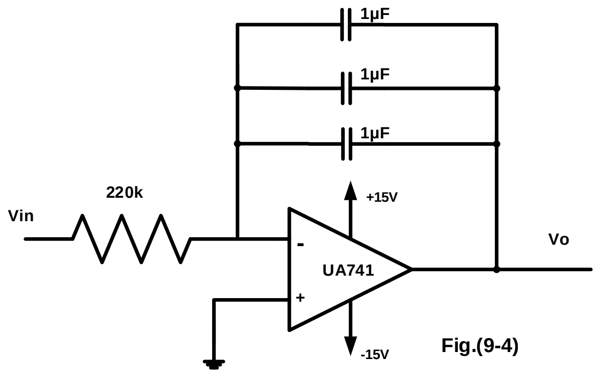


Fig.(9-4)

1. Set up the circuit of Fig.(9-4).
2. Put an input signal with $10V_{P-P}$ and a frequency of 100Hz ,and draw the output signal from the oscilloscope.
3. Repeat for all types of signals you have.

4. Set up the circuit of Fig. (9-5).

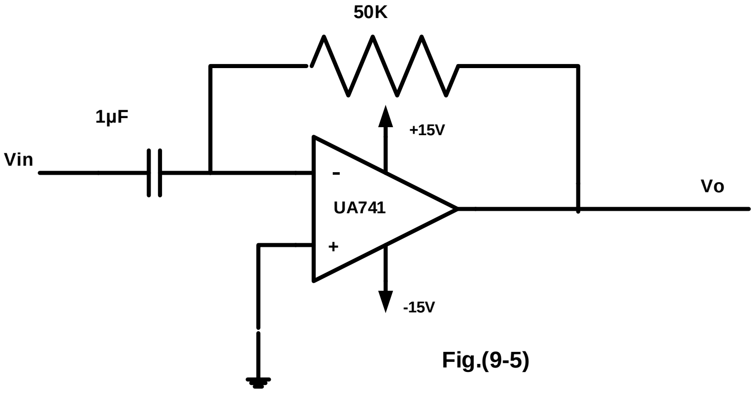


Fig.(9-5)

5. Repeat steps 2 and 3, and record your results.

Question:

- Is the output realize the differentiation and integration theory?

V. To investigate the effect of adding hysteresis:

1. Connect the Schmitt trigger circuit shown in Fig.(9-6).

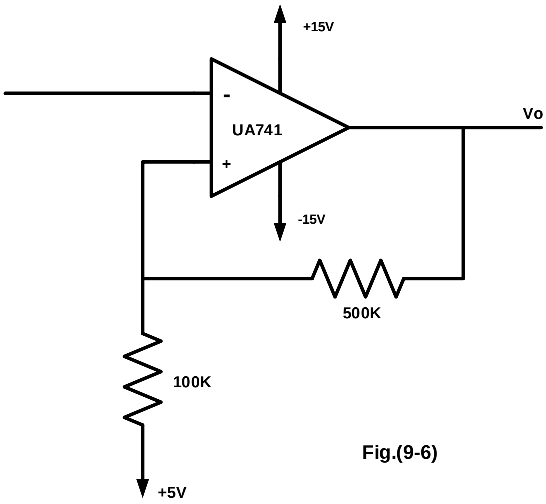


Fig.(9-6)

2. Put Vi(t) = 15Vp-p sine wave of frequency 1 kHz.

3. Sketch the output voltage with respect to Vi(t).

4. Indicate the levels of Vi(t) where Vo(t) changes its level.

5. Calculate the theoretical lower and upper trigger levels for the circuit above and compare them with those of measured values.

VI. Active Clipping Circuit:

1. Connect the circuit shown in Fig.(9-7).

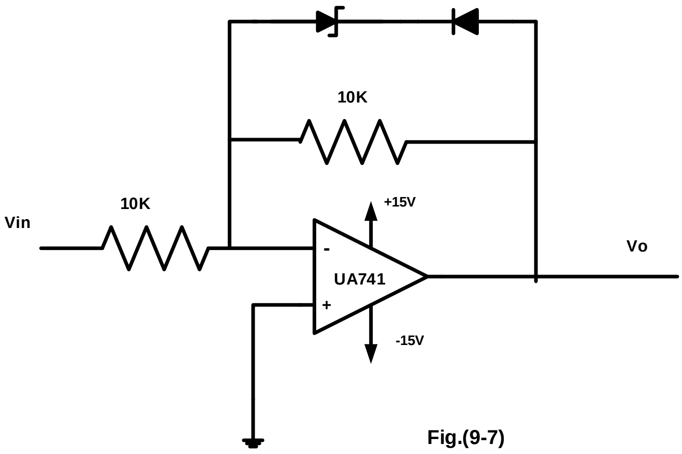


Fig.(9-7)

2. With Vi(t) of 1 kHz , vary the amplitude until you have a clipped output voltage .

3. Sketch the output voltage with respect to input voltage and record the levels of Vo(t) .

4. Reveres both diode connections and repeat steps 2,3 above.

Experiment #10 ENEE3102

Zener Diodes and Voltage Regulators

Objectives:

1. To construct the I.V characteristic of a zener diode.
2. To demonstrate the use of zener diode as voltage regulator.
3. To examine the operation of the voltage regulator.
4. To demonstrate the circuit and operation of astable multivibrator

Pre-lab Work:

You have to apply PSPICE simulation to all practical circuits shown in the procedure below, and you have to do all necessary calculation you will need in the lab.

Procedure:

•

I.ZENER DIODE.

1. *connect the circuit shown in Fig (10-1).*

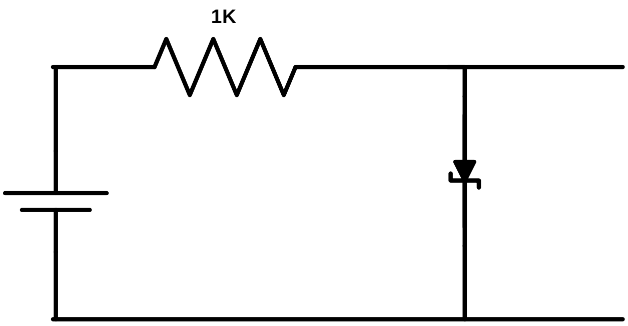


Fig.(10-1)

2. Set the applied voltage E to (0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9, 1,2,3,4)V.
3. For each value of E, measure the voltage a cross the resistor, the forward current through the zener diode, and the voltage a cross the zener diode and fill as in table 10.1.

E(V)	V_R(V)	V_Z(V)	I(m A)
0.1			
0.2			
0.3			
0.4			
0.5			
0.6			
0.7			
0.8			
0.9			
1			
2			
3			
4			

Table 10.1

4. Connect the circuit shown in Fig(10-2).

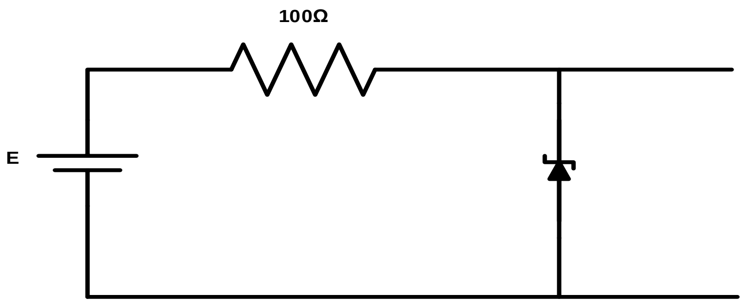


Fig.(10-2)

5. Set the applied voltage E to values shown in table 10.2.

6. For each value of E, measure the voltage across the zener diode and the current through the zener diode.

E(V)	Vz(V)	I(m A)
0.1		
0.5		
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		

Table 10.2

7. Using the results obtained in steps 3 and 6 constitute a graph of the characteristic of the zener diode.

8. Connect the circuit shown in Fig (10-3).

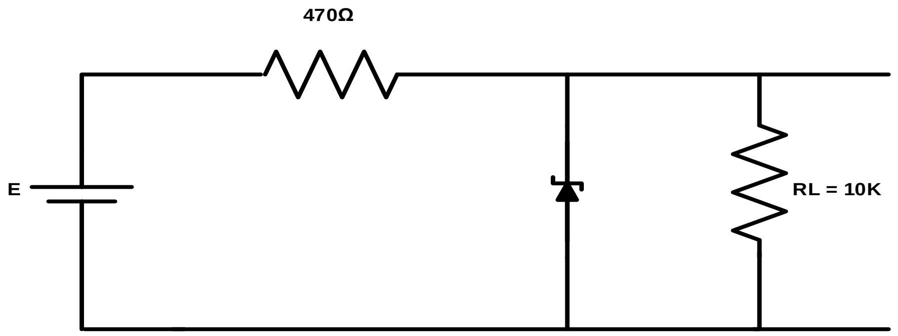


Fig.(10-3)

9. Set E to (10,11,12,13,14)V.
10. Measure the load voltage V_L and Fill table 10.3

E	10	11	12	13	14
V_L					

Table 10.3

11. With E set to 10V measure the load voltage V_L for R_L=(8.2K,6.8K,4.7K,2.2K).

R_L	8.2k	6.8k	4.7 k	2.2k
V_L				

Table 10.4

<u>II. THE VOLTAGE REGULATED POWER SUPPLY.</u>

1. connect the circuit of Fig.(10-4).

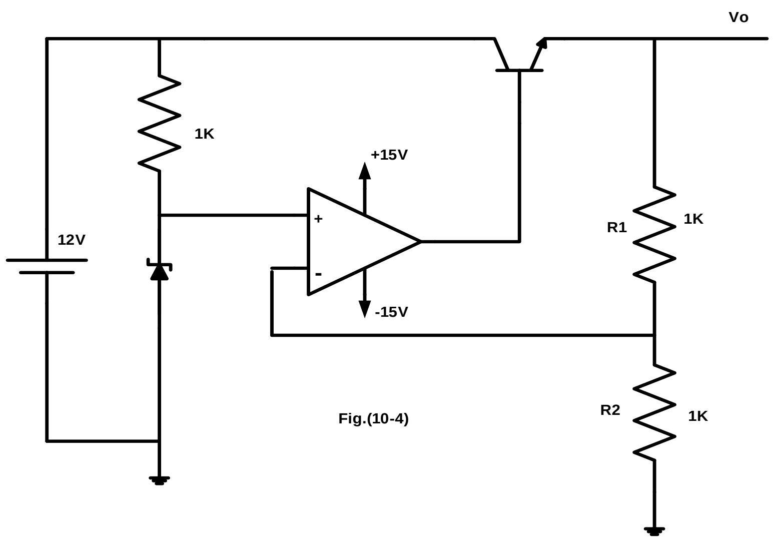


Fig.(10-4)

2. Measure Vo.

3. Attach a 1k load resistor to the output. Measure Io and Vo.

4. Repeat step 3 for load resistance RL = (680 , 470 , 220 , 100) ohm and fill Table

R_L	open	1kΩ	680 Ω	470 Ω	220 Ω	100 Ω
Vo						
Io						

5. Set RL back to 1K .Change the value of R2 to 470 ohm . What is the new output voltage.

6. Change R2 to 2.2k. What is the output voltage now?

7. Connect the circuit shown in Fig.(10-5).

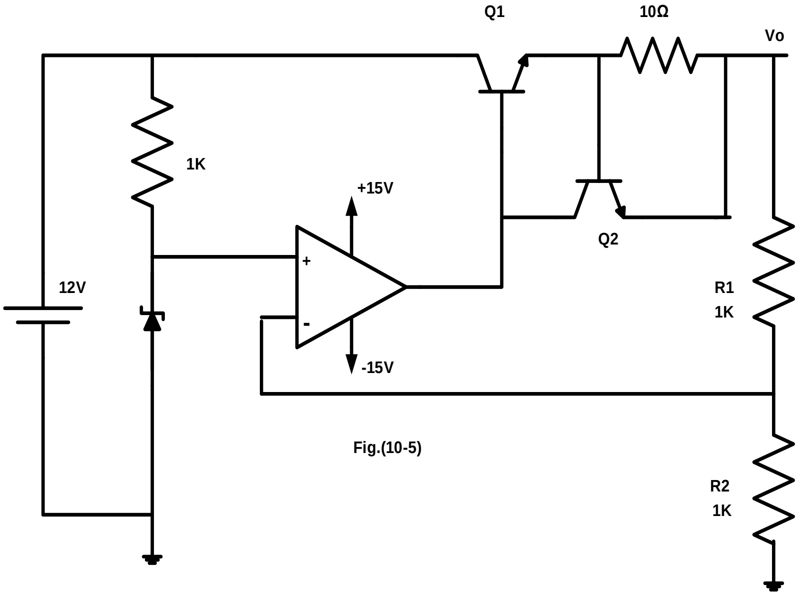


Fig.(10-5)

8. Measure Vo.

9. Repeat steps 3 and 4 . and record your results .

R_L	open	1kΩ	680 Ω	470 Ω	220 Ω	100 Ω	50 Ω
Vo							
Io							

II. THREE TERMINAL FIXED VOLTAGE REGULATOR 7805.

1. Connect the circuit of Fig (10.6).

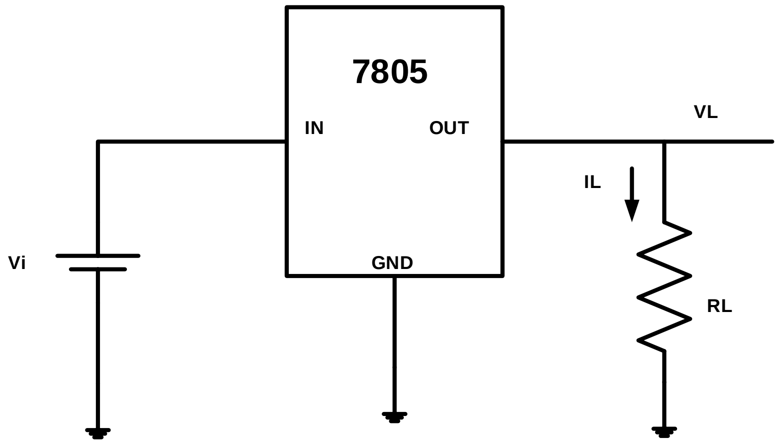


Fig.(10-6)

2. With Vi=10V measure I_L and V_L for the load resistances listed in the table 10.2.

$R_L(\Omega)$	$V_L(V)$	$I_L(m\ A)$
25		
50		
100		
200		
400		
600		
800		
1000		

Table 10.2.

3. Using the results of table 10.2 , determine the load regulation of the 7805.knowing that load regulation = $\Delta V_L / \Delta I_L$.

4. Set RL=100 ohm , adjust the input voltage Vi as listed in table 10.3. Measure V_L and I_L for each input voltage in the table.

Vi(V)	$V_L(V)$	$I_L(m\ A)$
8		
9		
10		
11		
12		
13		
14		
15		

Table 10.3

Using the results of table 11.3 , determine the line regulation of the 7805 .knowing that line regulation $= \Delta V_L / \Delta Vi$

III. THE LM317 ADJUSTABLE VOLTAGE REGULATOR.

1. Connect the circuit of Fig.(10.7).

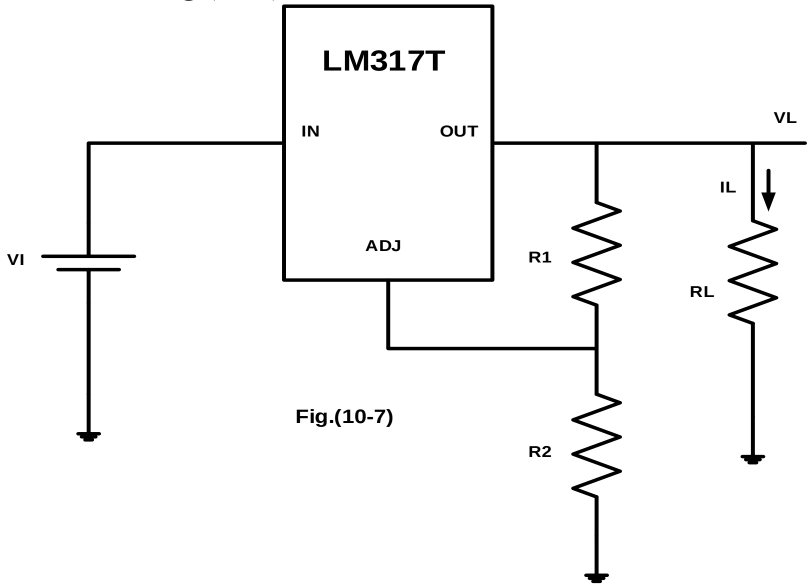


Fig.(10-7)

2. With Vi=10V, R1=100Ω, RL=1k , adjust R2 as shown in table 10.4.

$R_2(\Omega)$	$V_L(V)$	I_L(m A)
0		
100		
200		
300		
500		
700		

Table 10.4

3. Measure and record V_L,I_L for each R value.

4. With R_L =1k , R1=100 ohm , R2=220 , adjust Vi as listed in table 10.5.

Vi(V)	$V_L(V)$	I_L(m A)
10		
12		
14		
15		
16		
17		

Table 10.5

5. Measure and record the load voltage and current for each input voltage value.

6. Using your results, calculate the line regulation for the LM317T voltage regulator.

7. With Vi=10V, R1=100 0hm , R2=220 , adjust RL as shown in table 10.6.

R_L(Ω)	**V_L(V)**	**I_L(m A)**
100		
200		
400		
500		
600		
700		
1000		

Table 10.6

8. Measure and record V_L,I_L for each R_L value.

Experiment #11 ENEE3102

Oscillators

Objectives:

a. To demonstrate the wien bridge oscillator.

b. To demonstrate the RC phase-shift oscillator.

c. To demonstrate the colpitts oscillator.

d. To demonstrate the RC a stable multivibrator.

Prelab:

Perform the PSPICE Simulation on all the practical circuits shown under procedure below, and you have to do all necessary calculation you will need in the lab.

PROCEDURE:

I. THE WEIN BRIDGE OSCILLATOR.

1. Connect the circuit of Fig. (11-1).

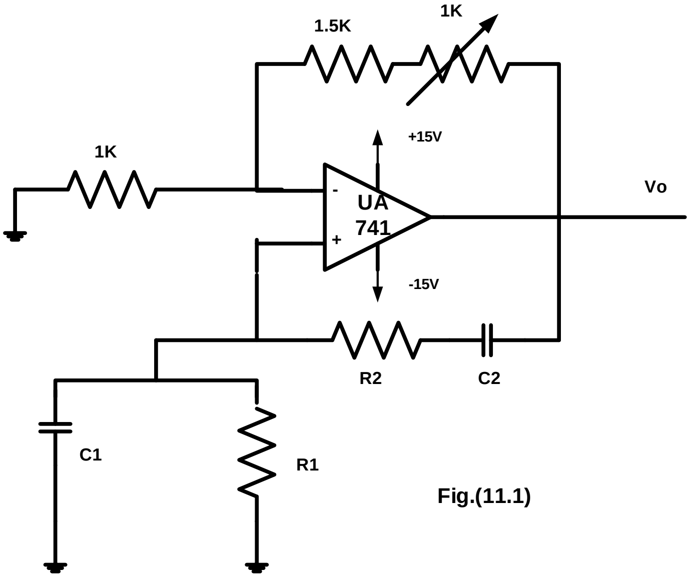


Fig.(11.1)

2. With R1 = R2 = 10k and C1 = C2 = 0.1µF carefully adjust the 1k pot. Until the output wave form has the least amount of distortion. Measure and record the amplitude and frequency of the waveform.

3. Replace C1 and C2 with 0.33 µF capacitors and repeat steps 2 and 3.

4. Replace R1 and R2 with 1k resistors and repeat steps 2 and 3.

II. THE RC PHASE SHIFT OSCILLATOR.

1. Connect the circuit of Fig(11-2).

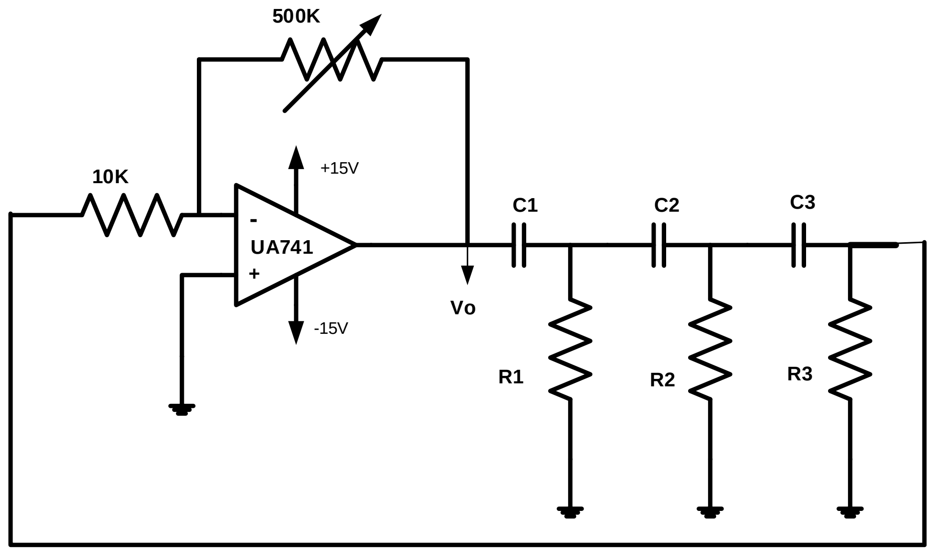


Fig. (11.2)

2. With R1 = R2 = R3 = 1k and C1 = C2 = C3 = 0.33uF , carefully adjust the 500k pot. Until the output waveform has the least amount of distortion.
3. Measure and record the amplitude and the frequency of the waveform obtained in step 2.
4. Replace C1, C2, and C3 with 0.47µF capacitors and repeat steps 2 and 3.

5. Replace R1, R2, and R3 with 470 ohm resistors and repeat steps 2 and 3.

III. THE COLPITTS OSCILLATOR.

1. Connect the circuit of Fig.(11-3) .

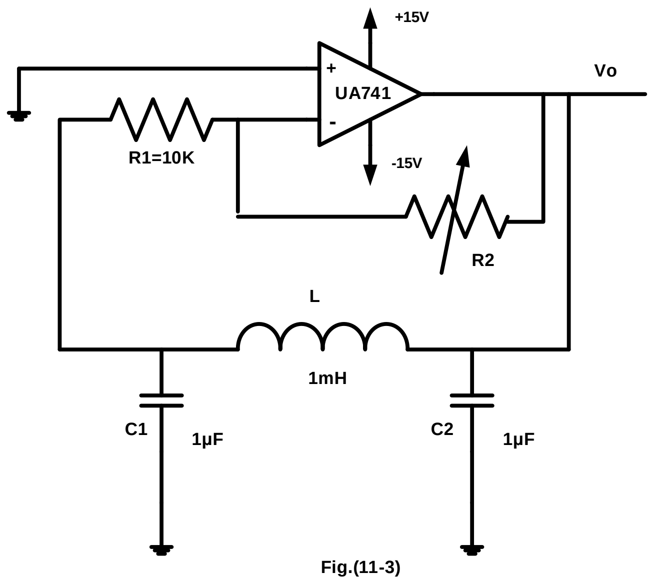


Fig.(11-3)

2. Set the R2 pot. to 500k. Measure and record the amplitude and the frequency of the wave form obtained .

3. Adjust the pot . until the output signal disappear (no oscillations). Record the value of the pot.

4. Replace L with 10mh inductor and repeat steps 2 and 3.

IV. THE RC ASTABLE OSCILLATOR.

1. Connect the circuit of Fig.(11-4).

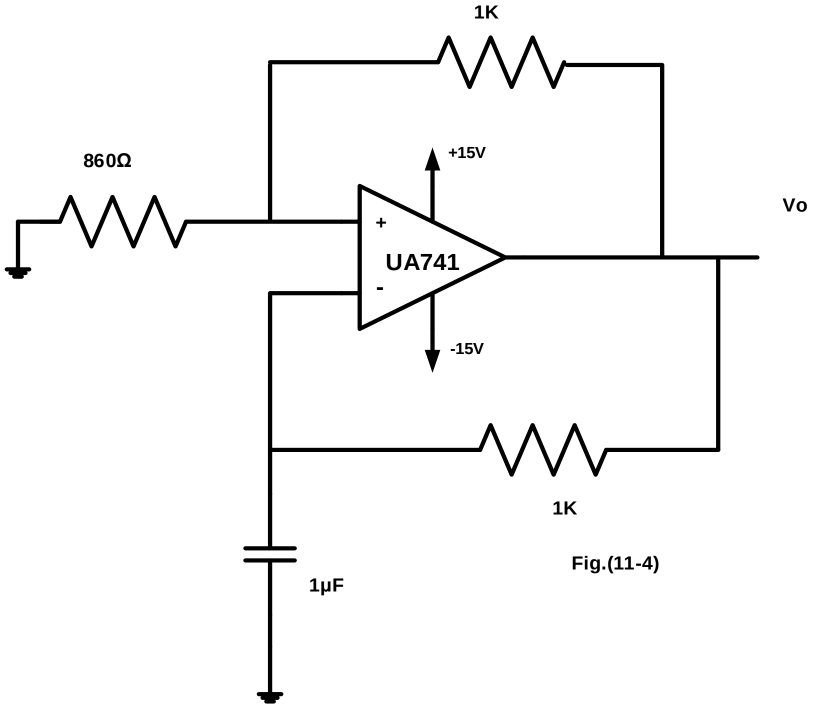


Fig.(11-4)

2.Record the amplitude and the frequency of the waveform.

3.Replace C with 0.1μF capacitor and repeat step 2.

V. THE 555 TIMER CHIP AS AN ASTABLE MULTIVIBRATOR.

1. Connect the circuit shown in Fig.(11-5).

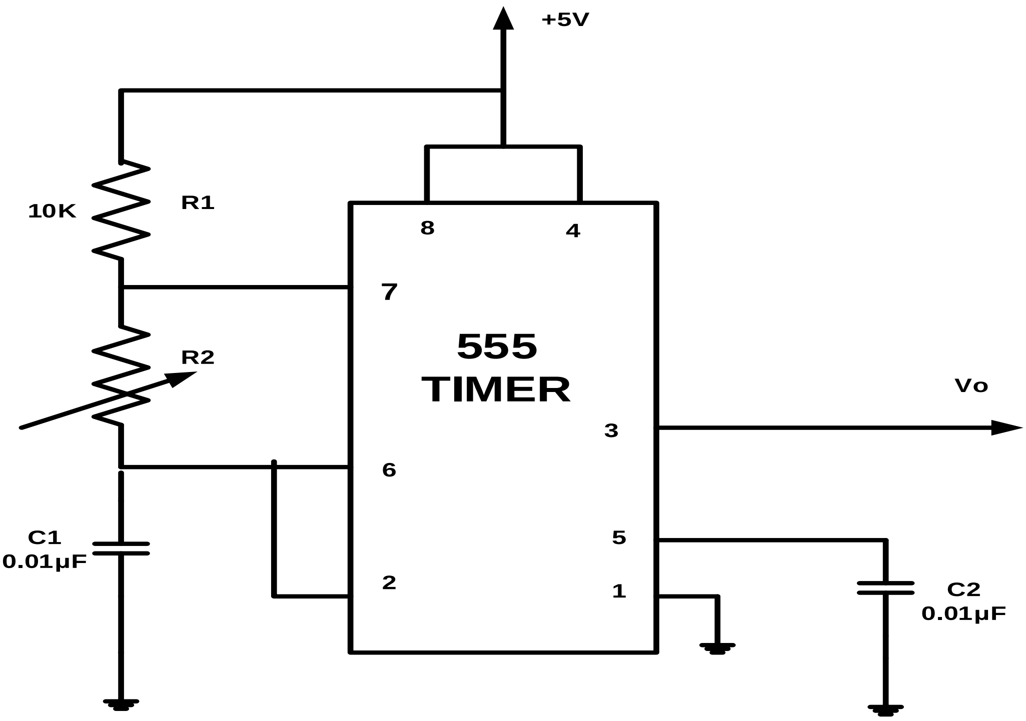


Fig.(11-5)

2. Using the oscilloscope measure the frequency and the duty cycle of the output voltage for different values of variable R2.(10,20,30,40,50k) Table 11.5

3. Sketch Vc(t) connected to terminal 6 for different values of R2.

4. Compare the results of step 2 with the theoretical values.

R2 [kΩ]	Frequency		Duty Cycle	
	Measured	Calculated	Measured	Calculated
10				
20				
30				
40				
50				

Table 11.5

Experiment #12 ENEE3102

Project

Objectives:

To enable students to implement a circuit of a design project given to them, the project assignment will be given during the semester and students should perform calculations, component value choice and simulations of the project before demonstrating a working hardware

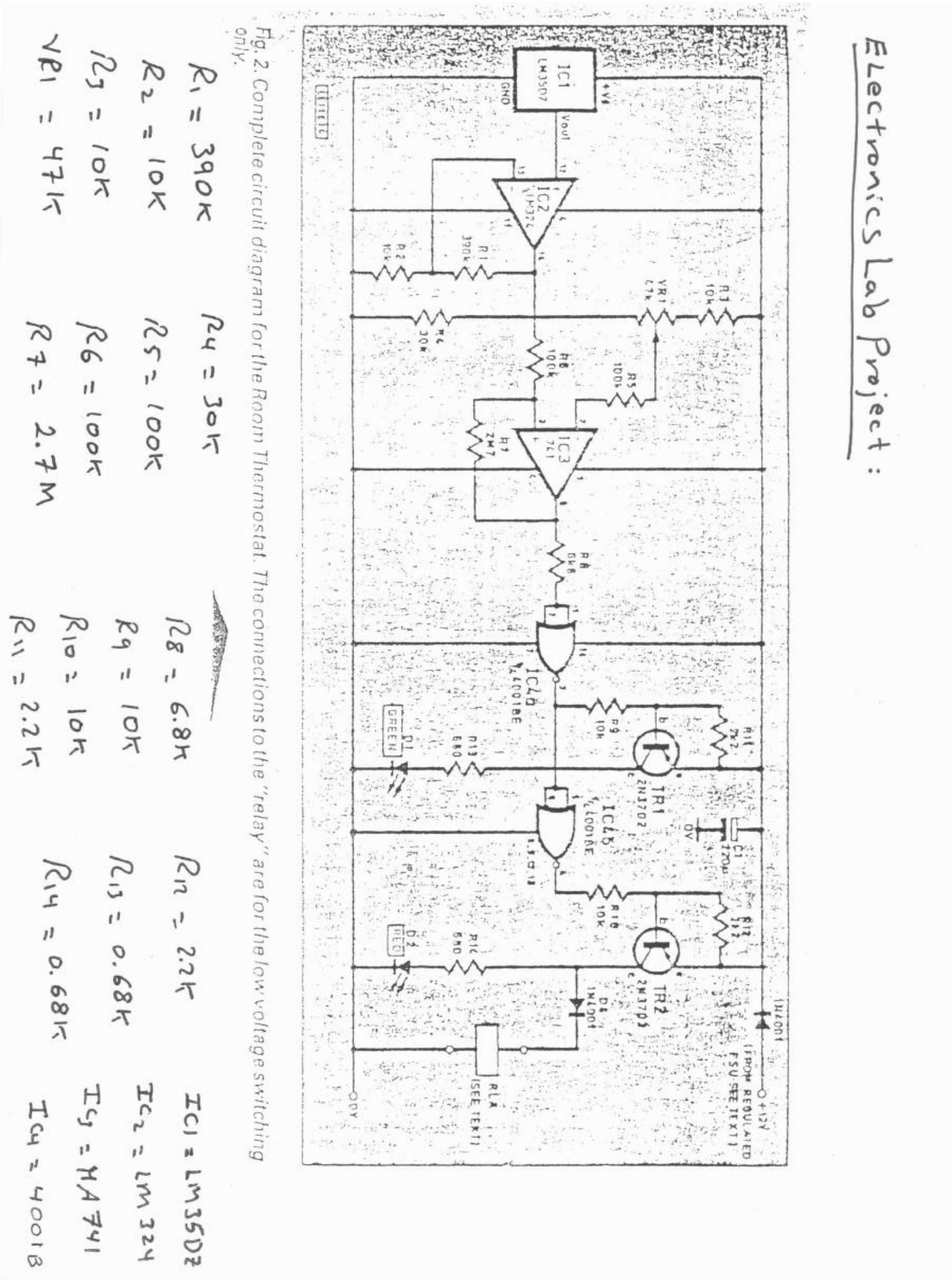


Fig. 2. Complete circuit diagram for the Room Thermostat. The connections to the "relay" are for the low voltage switching only.