Birzeit University
Faculty of Engineering & Technology

Circuits & Electronics Lab

ENEE2103 Circuits and Electronics Laboratory

FEBRUARY 14, 2021

Evaluations and Grading:

1. Prelabs (pspice simulations for the circuits under test) 15%
2. Quizez 15%
3. Reports(Three reports) 30%
4. Final Exam (Practical + Theoretical + Report Writing) 40%

Table of Contents

Lab Safety Instructions and Rules

يجب على الطلبة ارتداء الكمامات طوال فترة المختبر والحفاظ على التباعد اثناء اجراء التجارب وذلك بالتناوب على اجراء الأجزاء المختلفة من التجارب حسب تعليمات مدرس المساق

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 circuits and 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 electric circuits and components including transient, steady state and frequency response.
- 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

Experiment # 1

Basic Measurement Techniques

➢ Objectives:

1. In this experiment, we shall study how to use the digital multi-meter, power supply, function wave generator, and the oscilloscope.
2. To make some measurements in the lab.
3. Measuring Phase between two signals.

➢ Equipment Required:

1. Board, resistors, capacitors and wires.
2. Digital Multi-meter GDM-8135.
3. Oscilloscope TDS 2002B.
4. Power Supply Unit TK286 [0-20V] Variable DC.
5. *Function Wave Generator GFG-8215a.*

➢ Procedure

Part A: Measuring resistance

1. Use digital multi meter (DMM) to measure resistances you have been given, record results.

Part B: Measuring voltage

1. Connect the circuit shown in figure 1.14, set the input power supply to 10V, measure the voltage across each resistor, record and discuss results.

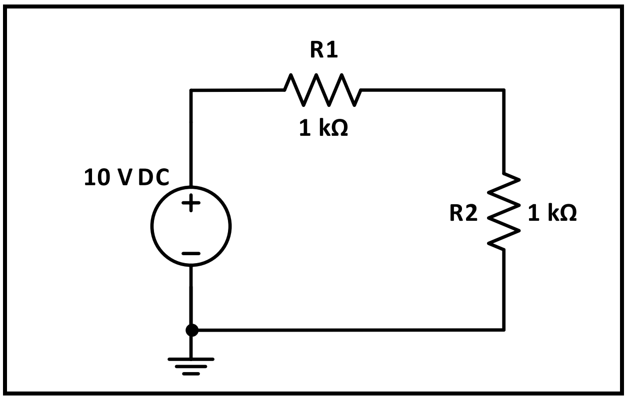


Figure 1.14

Part C: Measuring current

1. For the same circuit of figure.1.14 measure the current through each resistor, record and discuss results.

Part D: Using oscilloscope and function wave generator

1. Connect the circuit shown in Figure 1.15, set the input voltage to sinusoidal voltage at 8 V_{PP} and 1 kHz (you will need to connect channel 1 of the oscilloscope to the input voltage).

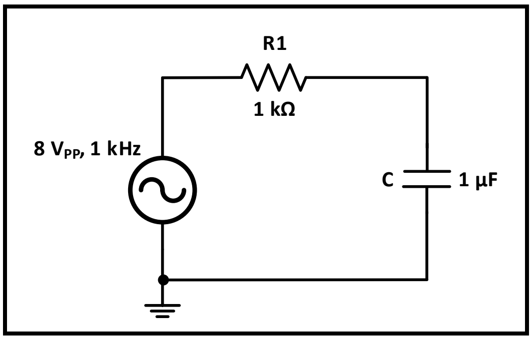


Figure 1.15

2. Connect channel 2 of the oscilloscope to the capacitor, use measure menu to view characteristics of the voltage across capacitor.
3. Use DMM to measure the current in the circuit, and the voltage across the resistor and the capacitor.

Experiment#2 ENEE2103

Circuit Laws and Theorems

Objectives:

1. To use to measure the resistance, the voltage and the current.
2. To test the validity of the KVL and KCL laws.
3. To verify the validity of the voltage and current division rules.
4. To test the validity of the superposition theorem.
5. To determine the Thevinin and Norton equivalent circuits.

Equipment:

1. Digital Multimeter.
2. Feedback Prototype Board
3. DC power supply
4. Discrete Resistors, Resistance Decade Box

Pre-lab:

1. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
2. Verify if Simulation Results match the expected results

Procedure:

A. ***KVL,KCL***

1. Measure the value of the resistances given to you by the lab instructor and make sure they match those in Fig. (2.1)
2. Connect the circuit of Fig (2.1). **Consider Rx=R1=R4=1kΩ , R5=3.3kΩ,R6=4.7kΩ**
3. Set the voltage source to 15 volts
4. Set the resistance decade box R_x to the value 1kΩ .
5. Measure and fill the values required in table 2.1(value and sign according to the used sign convention).
6. Change the value of R_x to the half of the first value.
7. Reapeat step 5
8. From your measurements verify the validity of KCL , KVL for the circuit. **Keep the circuit connected for the following part.**

Vs	Pot	R1		R4		R5		R6		Rx	
		V1	I1	V4	I4	V5	I5	V6	I6	Vx	Ix
15V	Rx										
15V	0.5 Rx										

Table (2.1)

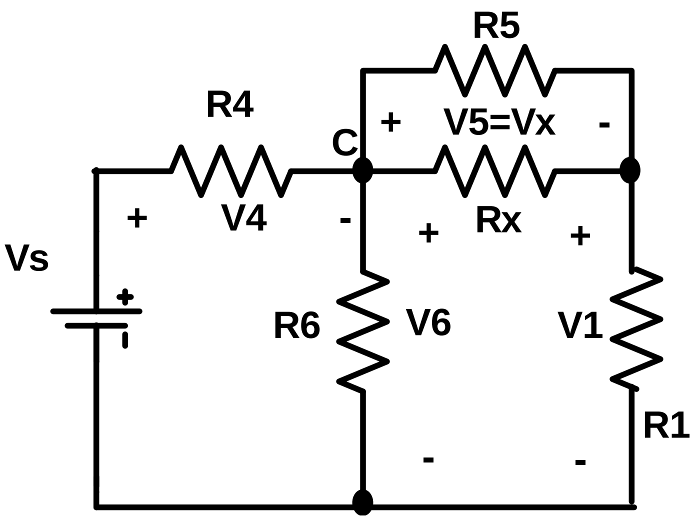


Fig (2.1)

B. Voltage & Current Division:

I. Voltage division

1. In the circuit of Fig (2.1) disconnect the resistance R_5 from the circuit
2. Measure the voltages on all the branches of the circuit.
3. Change R_x as shown in table2.2 and repeat step2 for all the given values.
4. In each case apply the resistors values into the voltage division formula
9. Do the measured values satisfy the voltage division rule? **Keep the circuit connected for the following part.**

Vs (volt)	Pot.	V1	V4	V6	Vx
10	Rx				
10	0.5Rx				

Table (2.2)

II. Current division

1. In the circuit of Fig (2.1) reconnect R_5 and replace R1 by a short circuit and set R_x to its start value.
2. Measure the curents in all the resistive branches of the circuit and fill the values in table2.3.
3. Change R_x as shown in table 3 and repeat step 2 for all the given values.
4. In each case apply the resistors values into the current division formula
5. Do the measured values satisfy the theoretical values of the current division rule?

Vs (volt)	Pot.	I4	I5	I6	Ix
10	Rx				
10	0.5Rx				

Table (2.3)

C. ***Superposition:***

1. Connect the circuit of Fig (2.2).
2. Set the source V_{s1} to 5 volts and V_{s2} to 10 volts.
3. Set the variable resistor R_x to the value of 1K.
4. Measure the current and the voltage on R_6
5. Set V_{S1} to zero and V_{s2} to 10 volts measure the current and the voltage on R_6
6. Set V_{s1} to 5 volts and V_{s2} to zero and measure the current and the voltage on R_6
7. Define the relation between the three current values measured in (4,5,6)
8. Define the relation between the three voltage values measured in (4,5,6)

Vs1(volt)	**Vs2 (volt)**	**V6 (volt)**	**I6 (mA)**
5	**10**		
0	**10**		
5	**0**		

Table (2.4)

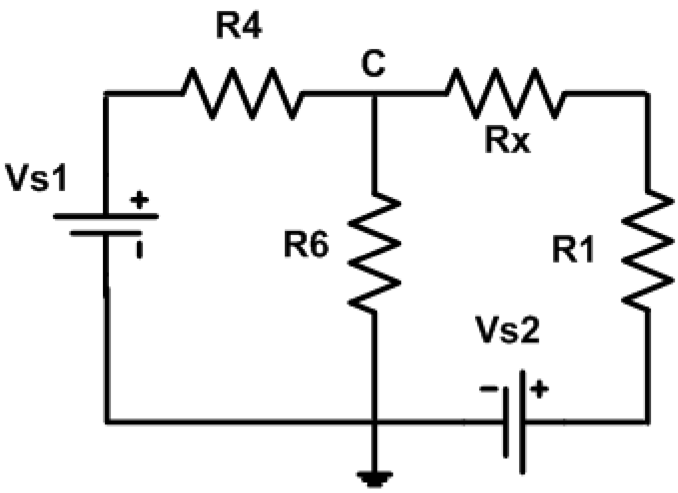


Fig (2.2)

D. ***Thevinin and Norton equivalent circuits:***

1. Reconnect the circuit of Fig (2.2).
2. Set the V_{s1} to 5volts and Vs2 to 10 volts and measue voltage across R1.
3. Disconnect R_1 and measure the voltage on the terminals (a,b) where R_1 was connected as in Fig (2.3).[Voc- open circuit voltage]
4. Short circuit the terminals (a, b) and measure the current in the short circuit (Isc).
5. Disconnect the voltage sources and short circuit the terminals where each source was connected.
6. Measure the resistance from the terminals (a,b) (Rab=Rth).
7. Connect the voltage source in series with the variable resistance from the potentiometer (VR1) as in Fig (2.4) , do not connect R1.
8. Set the voltage source to Voc measured in step (3) and the variable resistance to Rth measured inn step (4).
9. Measure the voltage on the opened terminals of the series connection.

10. Short circuit the terminals of the series connection and measure the current in the short circuit.
11. Define the relation between the voltage values measured in steps (3,9)
12. Define the relation between the current values measured in steps (4, 10)
13. Compute the ratio between the measured voltage Voc and current Isc in steps (3,4)
14. Define the relation between the computed value in step (13) and the resistance measured in step (5)
15. Connect the resistance R1 across terminal a-b of Fig. 2.4 and measure the voltage across it?
16. Compare voltage across R1 from step (15) to its value measured in step (2)

17. What is the relation between the circuit of Fig.2.2 and the circuit of Fig.2.4 constructed in steps (7,8)? Refer to the electric variables on the port (a, b).
18. Compare the short circuit current value with the Norton current source determined by computation.

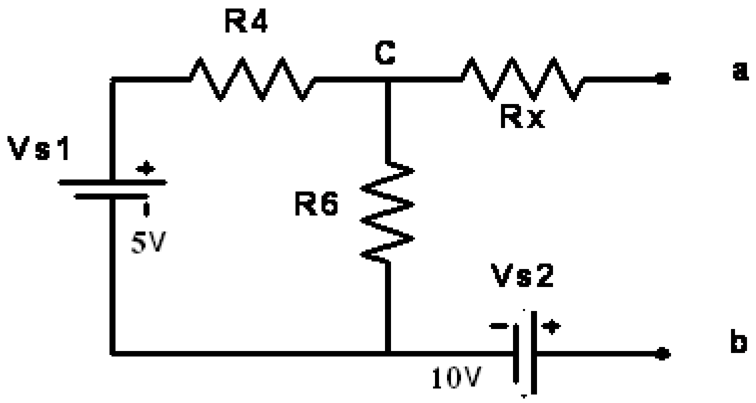


Fig (2.3)

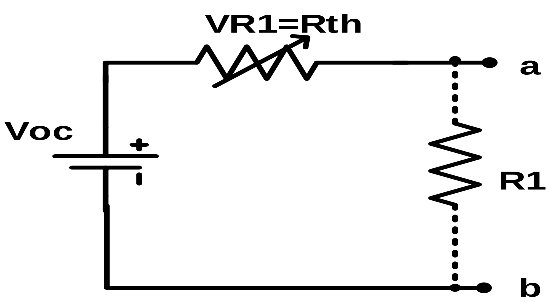


Fig (2.4)

Components List: R1=R4=1 kΩ , R5=3.3 kΩ, R6=4.7kΩ
Resistance Decade Box

Experiment#3 ENEE2103

First and Second Order Circuit

Objectives:

1. To use the Oscilloscope to measure electric values.
2. To test and analyze the time responses of RL and RC circuits.
3. To test and analyze the time response of the second order RLC circuit.
4. To test the effect of the initial state of the dynamic elements on the time response.
5. To determine the first and second order circuits parameters from the circuit response.

Equipment:

1. Digital Multimeter.
2. Oscilloscope (TDS-2002B).
3. Power supply.
4. Signal generator.
5. **Discrete Capacitors and Resistors, Inductance decade box, Resistance decade box**

Pre-lab:

3. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
4. Verify if Simulation Results match the expected results

Procedure:

A. ***RC Circuit:***

1. Set the resistance Decade Box to the value of R3 then measure Value of C1 using the RLC meter.
2. Connect the circuit of Fig (3.1)
3. Set the signal generator to square wave 5Vp-p and 50Hz **with dc offset=2.5V**
4. Connect the Oscilloscope to the Capacitance terminals.
5. From the response displayed on the oscilloscope, determine the value of the system time constant.
6. Determine the steady state voltage value on the capacitor.
7. Use your measurements of the time constants to determine the value of the capacitance.

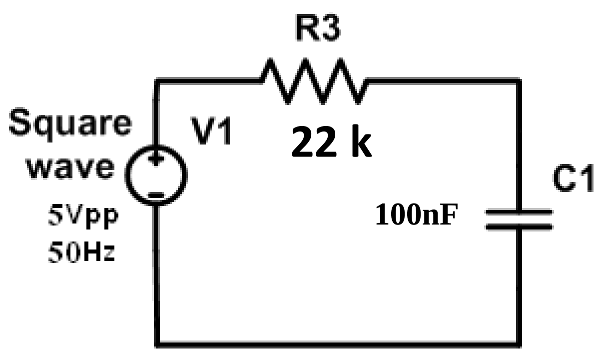


Fig (3.1)

B. ***RL Circuit:***

1. Connect the circuit of Fig (3.2)
2. Set the signal generator to generate a periodic square waveform with 10Vp-p and frequency=500Hz , dc offset=5V.
3. Connect the oscilliscope to display the voltage response of the inductor.
4. Measure the time constant of the circuit and the steady state values of the voltage and current responses.
5. Determine the behavior of the voltage and current responses in relation to the element characteristic equation.
6. Change the period of the periodic square wave to $T=2\tau_L$ and display the result.
7. Write your conclusion about the displayed waveform.

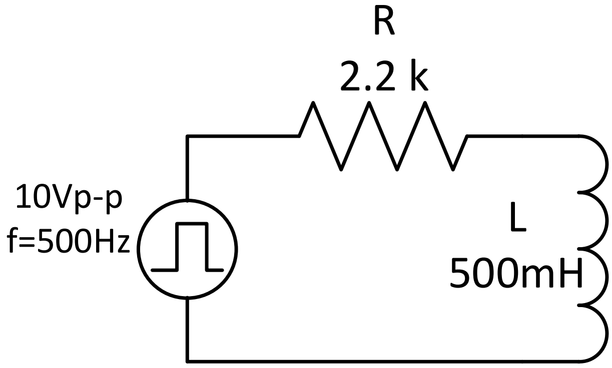


Fig (3.2)

C. ***RLC Circuit:***

I. **Response type:**

1. Connect the circuit of Fig (3.3)

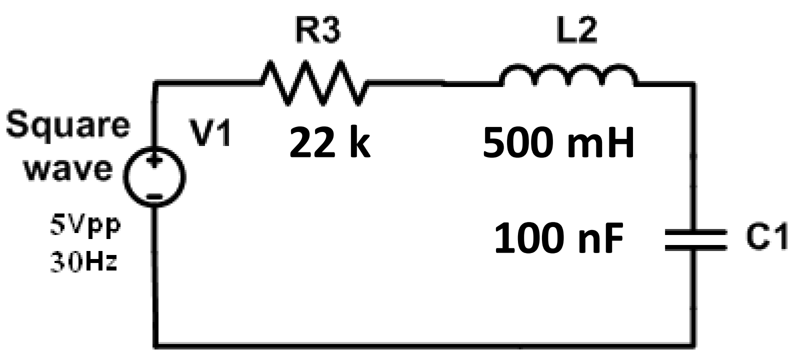


Fig (3.3)

2. Connect the oscilloscope to measure the voltage in the capacitor.
3. Set the signal generator to generate a periodic square waveform with ± 2.5 volts and 30Hz, dc offset=2.5V
4. Determine the type of the response.
5. Replace R3 by resistance decade box
6. Calculate the value of the resistance that satisfied the critical damping and the under damping conditions.
7. Change the variable resistor with steps so that you can detect a change in the type of the response.
8. Refine your steps around the value for which the transition occurs so that to detect the transition point.

9. Determine the type of the response in each case.

II. Response parameters:

1. Set the value of R_x to (define value through test) so that to get an under damped response
2. Use the cursor to measure the decay-envelope time constant (τ), the damping coefficient (α) and the damped frequency (ω_d) as shown in Figure 3.3.
3. Double the value of C1 and Measure the parameters defined in step2 noting the effect.
4. Reset the capacitance to its initial value
5. Reduce value of L2 to half its value and note the effect on previous parameters.

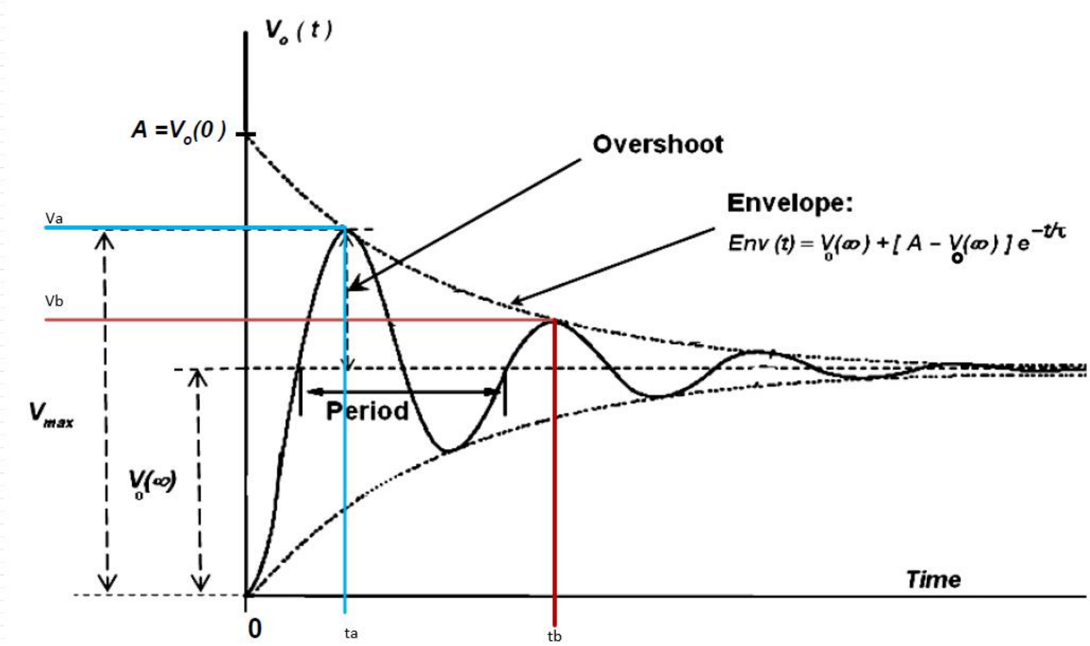


Figure 3.3

Decay time constant $$\tau = \frac{t_b - t_a}{ln\left(\frac{V_a - V_{o(\infty)}}{V_b - V_{o(\infty)}}\right)}$$

Damping Coefficient $$\alpha = \frac{1}{\tau}$$

Damped radian frequency $$\omega_d = \frac{2\pi}{t_b - t_a}$$

Components List:
22kohm , 2.2kohm
100nF (2)
Resistance Decade Box
Inductance Decade Box

Experiment#4 ENEE2103

Sinusoidal Steady State Circuit Analysis

Objectives:

1. To use the Oscilloscope, the DMM, the Wattmeter for AC electric quantities measurement..
2. To measure the circuit elements impedances and voltage and current phasors.
3. To verify the validity of the Circuit theorems in the sinusoidal steady state.
4. To measure the power in sinusoidal steady state circuits.

Equipment :

1. Digital Multimeter.
2. Oscilloscope (TDS-2002B).
3. Power supply.
4. Function generator.

Pre-lab:

1. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
2. Verify if Simulation Results match the expected results

Procedure:

A. *Impedance:*

1. Connect the circuit of Fig (4.1)
2. Value of R_x =2.2 kΩ
3. Set the signal generator to generate a sinusoidal waveform with amplitude 5 volts and frequency 1 kHz.
4. Measure the total impedance of the circuit using DMM by measuring the total voltage and current. Find the phase shift between total voltage and current using the oscilloscope cursor menu.
5. Repeat the step (4) with the signal frequencies: 500 Hz , 1500 Hz. Fill in the results in table 4.1
6. Write your conclusions about the variation of the impedance of the resistor with the frequency.
7. Connect the circuit of Fig (4.2).
8. Repeat the steps (2-5) with the signal frequencies: 500Hz , 1500 Hz. Fill in the results in table 4.2
9. Write your conclusions about the variation of the impedance of capacitor with the frequency.
10. Connect the circuit of Fig (4.3)
11. Repeat the steps (2-5) with the signal frequencies: 500Hz , 1500 Hz. Fill in the results in table 4.3
12. Write your conclusions about the variation of the impedance of the inductor with the frequency.

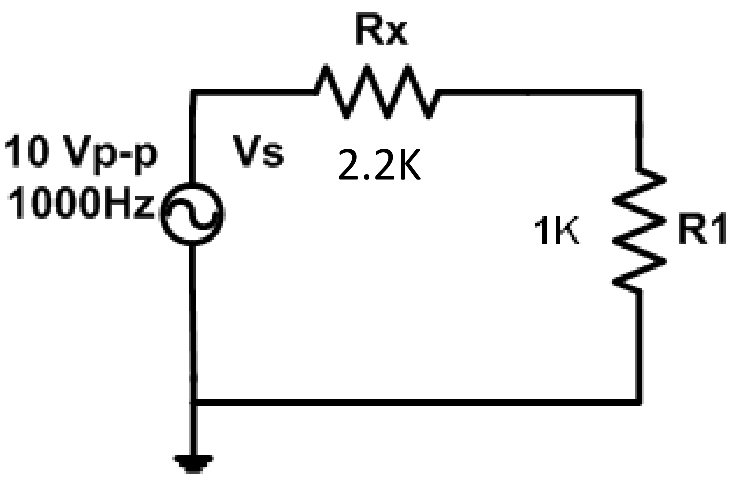


Fig (4.1)

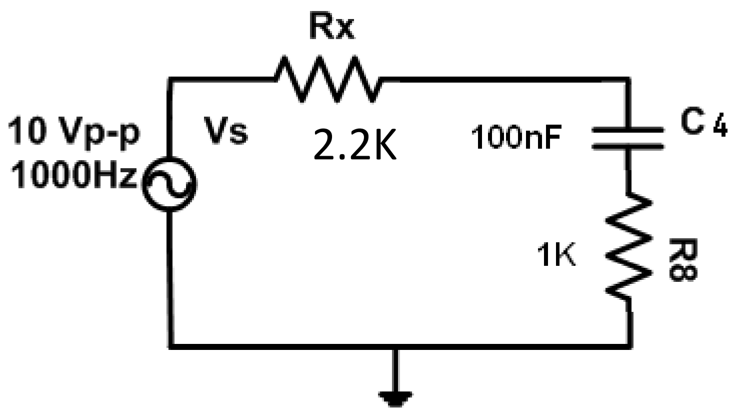


Fig (4.2)

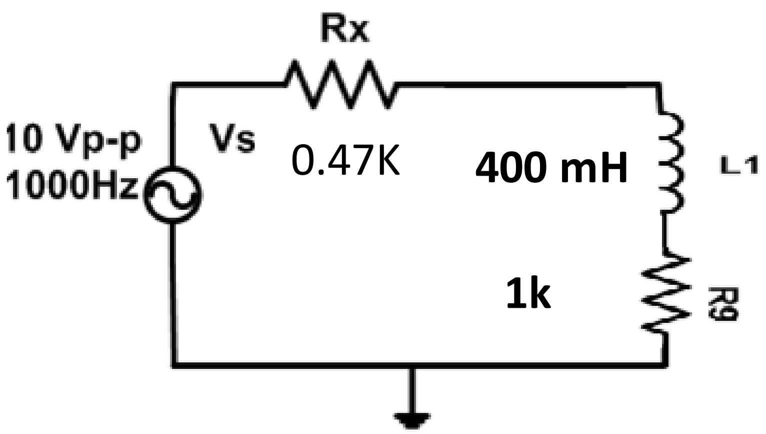


Fig (4.3)

B. *Capacitive and inductive behavior:*

1. Connect the circuit in Fig (4.4) (note : use table 4.5 to fill in the results)
2. Set the generator to generate a sinusoidal waveform with amplitude 5 volts and frequency 1 kHz.
3. Measure the phase shift between the total current and the voltage.
4. Repeat the step (3) incrementing the frequency 2 kHz, 4 kHz,6 kHz,8 kHz .
5. Determine the resonance frequency fo experimentally ***(note that at fo, both voltage and current will be in phase)***

6. Write your conclusions about the circuit behavior in relation to the capacitive and inductive and the resistive behavior.
7. Set the generator frequency to the resonance frequency found in 5.
8. Connect another 100nF capacitor in parallel to C2 and explain the behavior of the circuit according to the circuit response.
9. Disconnect the extra capacitor and double the value of L3 and Explain the behavior of the circuit according to the circuit response.

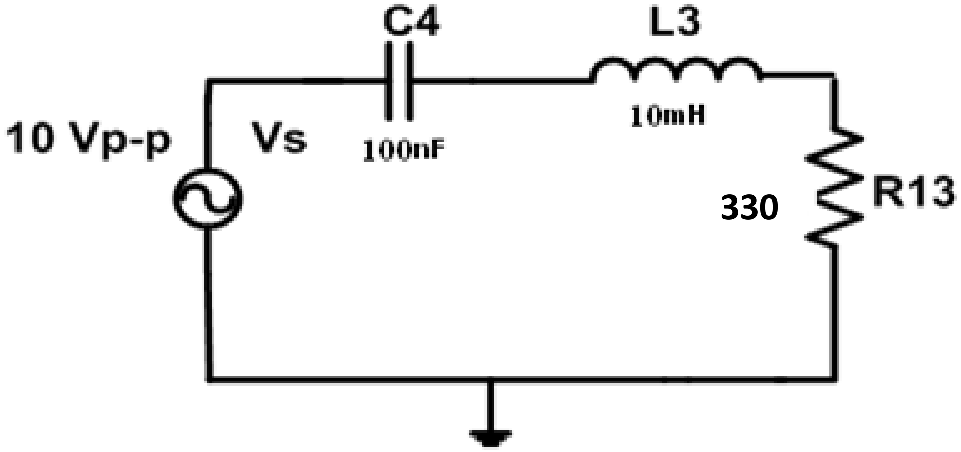


Fig (4.4)

C. *Sinosoidal steady state power:*

1. Set the voltage source to amplitude 2.5 V and frequency 2 kHz
2. **Connect** the circuit in Fig (4.5)
3. **Measure** the rms voltage and the rms current across R6, L1 , C2 and R1
4. **Measure** the phase shift between Vs and Is; Vc and Ic and fill table 4.5.

   Notice that in order to measure phase shift between Vc, Ic , you need to add a 10 Ω resistor in series with C2. Find the phase shift between voltage of C2+ Rx and the voltage of Rx.
5. **Compute** the active power (average power), the reactive power and the power factor in each element.
6. **Verify** the validity of the conversation of energy law (∑input Power=∑Output Power)

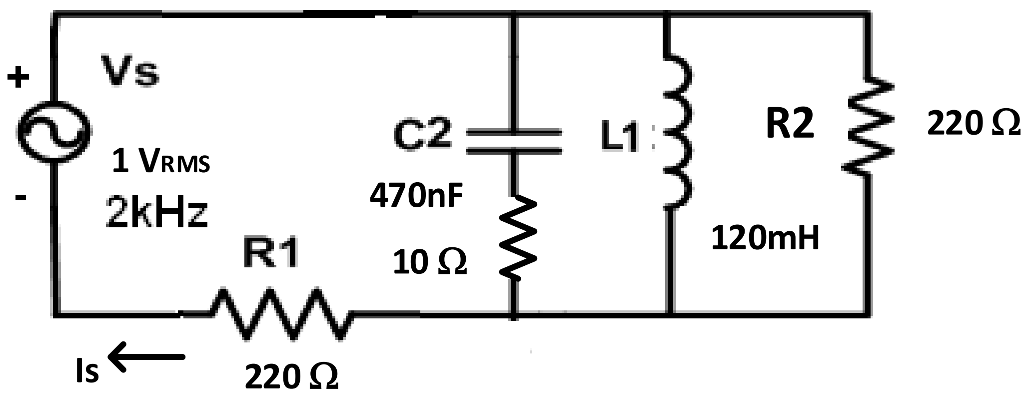


Fig (4.5)

Table 4.1

f [Hz]	Vrms	Irms	Δt	Phase shift
500				
1k				
1.5k				

Table 4.2

f [Hz]	Vrms	Irms	Δt	Phase shift
500				
1k				
1.5k				

Table 4.3

f [Hz]	Vrms	Irms	Δt	Phase shift
500				
1k				
1.5k				

Table 4.4

f	1k	2k	4k	6k	8k	fo
Δt						
(Θ_{Vs}- Θ_{Is})						

Table 4.5

$V_{(R1)}$	**V_L=$V_{(R6)}$**	**I_L**	**I_{R6}**	**V_s**	**I_s**	**(Θ_{Vs}- Θ_{Is})**	**V_c**	**I_c**	**(Θ_{Vc}- Θ_{Ic})**

Experiment #5 **ENEE2103**

Filters

Objectives:

1. To use the Oscilloscope, the DMM, for AC electric quantities measurement..
2. To measure the Amplitude and phase of frequency variable sinusoidal signal.
3. To implement and measure the characteristics of circuits with Op-Amp.
4. To verify the Characteristic behavior of the different types of filters.
5. To verify the differences between the active and the passive filters.

Equipment :

1. Digital Multimeter.
2. Oscilloscope (TDS-2002B).
3. Power supply.
4. Function Generator
5. Discrete components:
   Opamp: uA741
   Resistors: 1 kΩ , 2.2 kΩ ;
   Capacitors: 220nF, 470nF
6. Inductance decade box

Pre-lab Work:

1. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
2. Verify if Simulation Results match the expected results

Procedure:

A. *Passive filters:*

I. First order circuits:

1. Connect the circuit of Fig (5.1) (make sure to measure R1 and C1 provided to you)

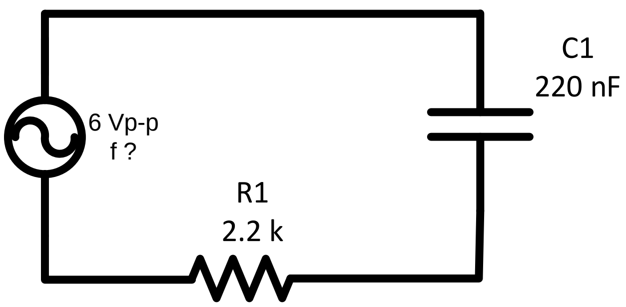


Fig (5.1)

2. Set the signal generator to generate a sinusoidal waveform 3 volt amplitude and **f =20 Hz** and measure the output voltage on the capacitor using the DMM.
3. Determine the phase shift between the **circuit current and the input** voltage using the oscilloscope, Put ch1 on ac source and ch2 on R1 to represent the current.
4. Measure Vi, V_R and Vc using DMM **(fill results in table 5.1)**

5. Measure the value of cut-off frequency (fc) by finding the max value of Voltage across the capacitor Vc_max (use DMM) first, then fc is the frequency for which

Vc=0.707 Vc_max. **(note: value of fc=1/(2πReq.C)).**

6. Repeat steps 3, 4 for frequencies ranging from 0.1fc - 20fc and fill results in table 5.1 . Make sure to take enough points to be able to draw the magnitude frequency plots [dB/decade]

7. From the magnitude-frequency plots of (V_R / Vi) and (V_C / Vi) determine the filter type in each case.

8. From the plots of step 7. Determine approximately the 3-db cut-off frequency in each of the two cases.

9. Analyze theoretically the phase behavior with the frequency and determine if the filter introduces a phase shift.

II. Second Order Fiters:

1. Connect the circuit of Fig (5.2)

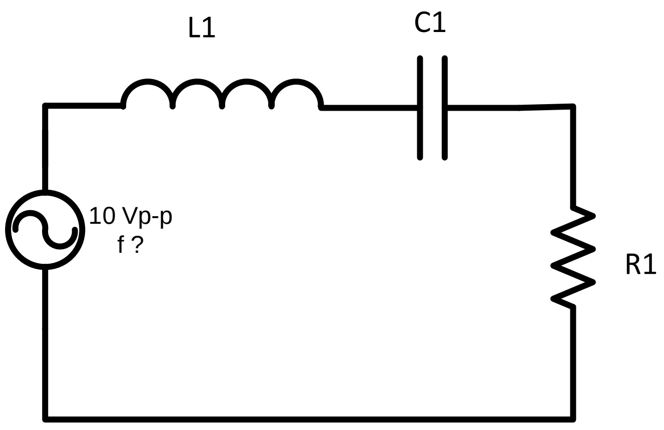


Fig (5.2)

2. Set the component values **R1=1 kΩ , L1=100 mH and C1=470nF.**

3. Set the signal generator to generate a sinusoidal waveform with amplitude 5 volts and a frequency value that equals the resonance frequency (fo) measured experimentally and verified theoretically.

4. Measure the voltages across each component: V_R, V_L, V_C using the DMM. (fill in the results in in table 5.2)

5. Determine the values of fc1 and fc2 experimentally and compare to theoretical values.

6. Repeat step 4 for frequencies ranging from 0.1fc1 - 20fc2, take enough points to be able to draw the magnitude frequency plot [dB/decade] as in table 5.2

7. From the magnitude-frequency plots of (V_R / Vi) and ((V_C + V_L)/ Vi) determine the filter type in each case.

8. From the plots of step 6. Determine approximately the 3db cut-off frequency in each of the two cases. **Does it match the theoretical values?**

Active filters:

1. Connect the circuit of Fig (5.3). Don't forget to connect the bias voltage sources

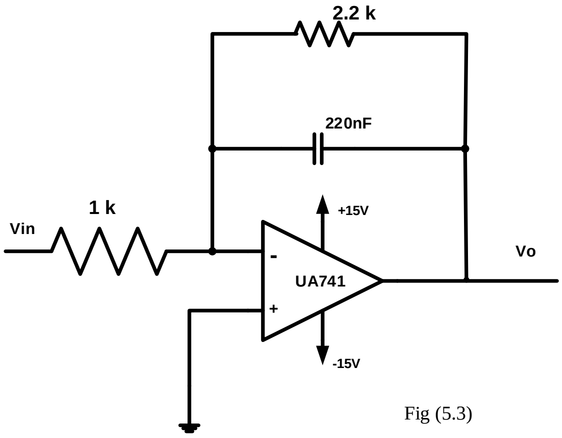


Fig (5.3)

2. Calculate the expected value of cut-off frequency f_c? Does it match the value obtained from the measurement and simulation?

4. Set the signal generator to generate a sinusoidal waveform with 3 volts amplitude and **f** =**f**$_c$ and measure the input and output voltage using the DMM.

5. Repeat step (4) for frequencies within the following range ($0.1f_c < f < 40f_c$), make sure to take enough number of points in table 5.3 to construct the magnitude bode plot.

7. Draw the magnitude bode plot (20 log(Vo/Vi)) based on measurements of step (5), determine the filter type, and approximately determine the 3db cut-off frequency.

8. In your report compare the simulation results for the magnitude and phase frequency response for the passive and active filter of part A-I and part B (from 10Hz to 10 MHz)

Frequency	Vin(Vrms)	V_R(Vrms)	V_C(Vrms)	$\theta_V-\theta_I$	log f	20 log (V_R/V_{in})	20 log (V_C/V_{in})
20 Hz	1Vrms						
0.1 fc	1Vrms						
0.5 fc	1Vrms						
fc	1Vrms						
	1Vrms						
4fc	1Vrms						
	1Vrms						
10fc	1Vrms						
	1Vrms						
14fc	1Vrms						
	1Vrms						
20fc	1Vrms						

Table 5.1➔ ac sweep

Frequency	Vin(Vrms)	V_R(Vrms)	V_C(Arms)	V_L(Vrms)	(VC+VL)(Vrms)	log f	20 log (V_R/V_{in})	20 log (V_c+V_L/V_{in})
0.1 fc1								
fc1								
0.5fo								
fo								
2fo								
fc2								
10fc2								
14fc2								
20fc2								

Table 5.2

Frequency	Vin(Vrms)	V_o(Vrms)	log f	20 log (V_o/V_{in})
0.1 fc				
0.4fc				
0.8fc				
Fc				
1.5fc				
4fc				
10fc				
20fc				

Table 5.3

Experiment # 6 ENEE2103

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:

1. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
2. Verify if Simulation Results match the expected results

Procedure:

I. DIODE CHARACTERISTICS

1. Connect the Circuit of Fig. (6.1).

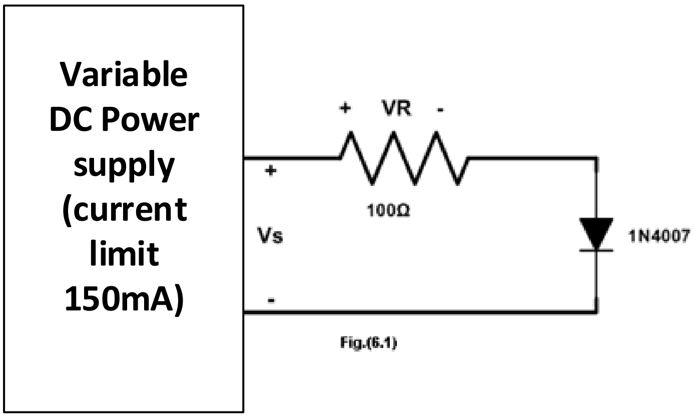


Fig.(6.1)

2. Set the current limit of the dc power supply to 150mA.
3. Switch on the power supply and adjust it from zero to 1 volt in 0.1V steps and in 0.5 steps from 1V to 3V.
4. For each setting measure the value of VR.

Table 6.1

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			

5. Calculate V_D and I_D and enter them in the table 6.1 .
6. 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.(6-3).

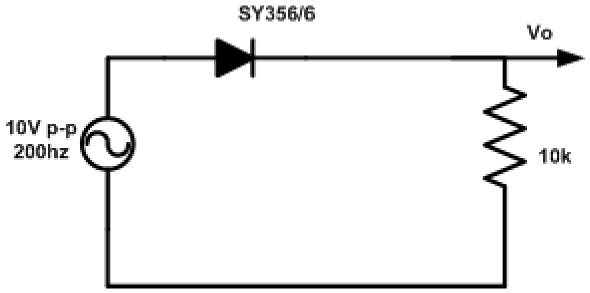


Fig.(6-3)

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.(6-4).

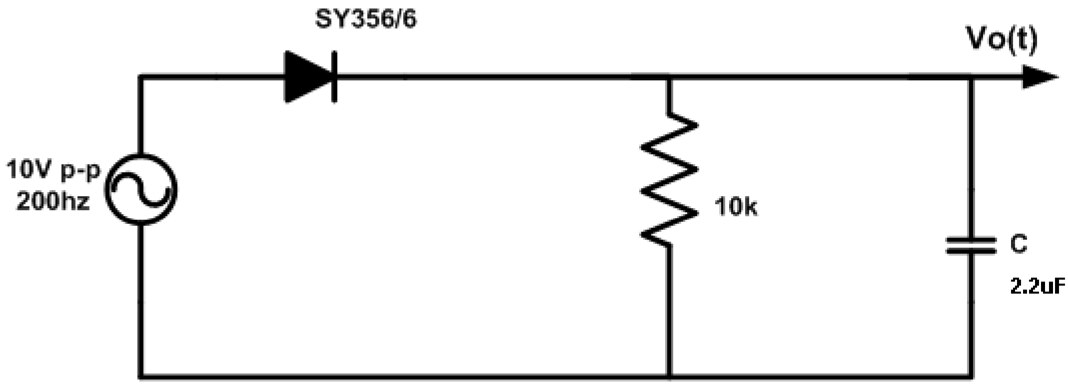


Fig.(6-4)

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.(6-5).

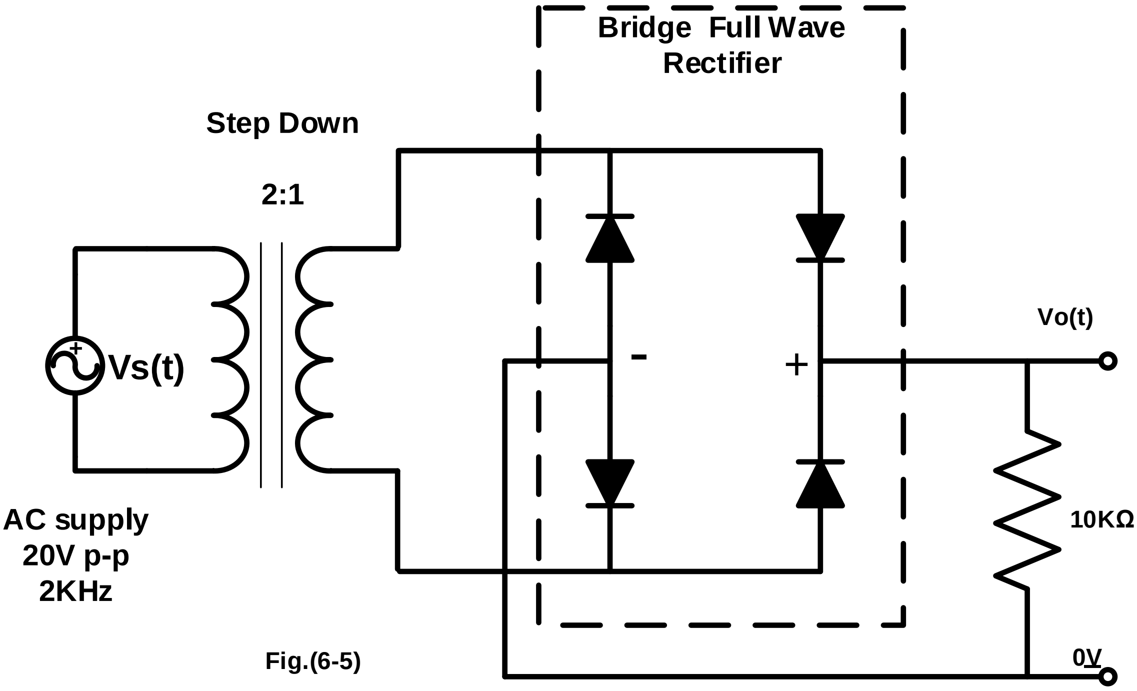


Fig.(6-5)

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?
- What is the effect of frequency on the ripple? When the input frequency is reduced, do you need a larger or a smaller capacitor to achieve the same ripple?

III. other applications:

A. clipping:

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

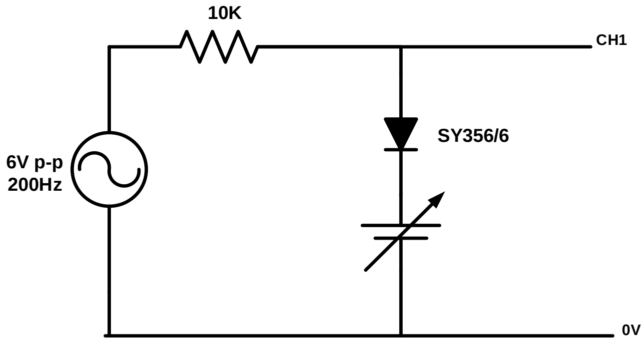


Fig.(6-6)

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 (take photos of input and output for three different values of dc voltage: 0V, 1.5V and 4V)

Note: make sure to have dc coupling for oscilloscope channels

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.(6-7).

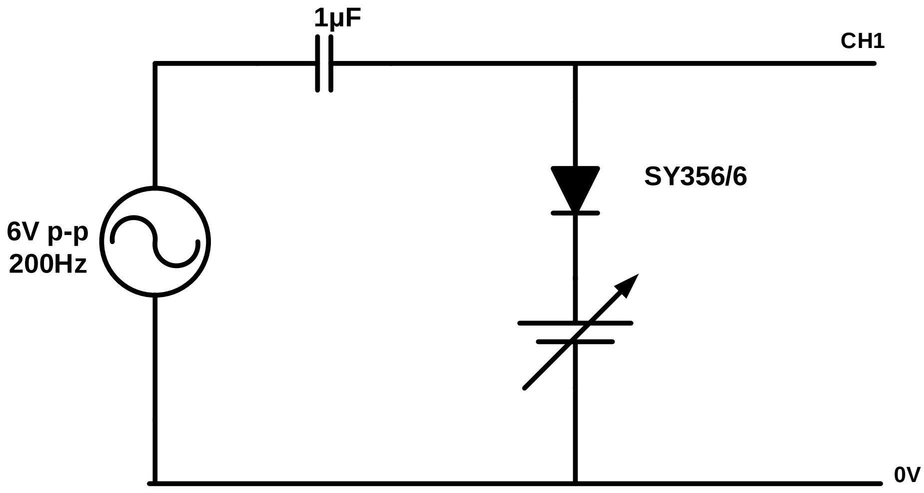


Fig.(6-7)

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

3. Take photos of both input and output for three different values of dc voltage: 0V, 1.5V and 4V

Note: make sure to have dc coupling for oscilloscope channels

<u>Questions:</u>

- Does the output wave form alternate about the same dc level as the input waveform?
- To what value the positive peak of the output waveform is 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. <u>VOLTAGE MULTIPLIER CIRCUITS</u>

1. Set up the circuit as shown in Fig.(6-8).

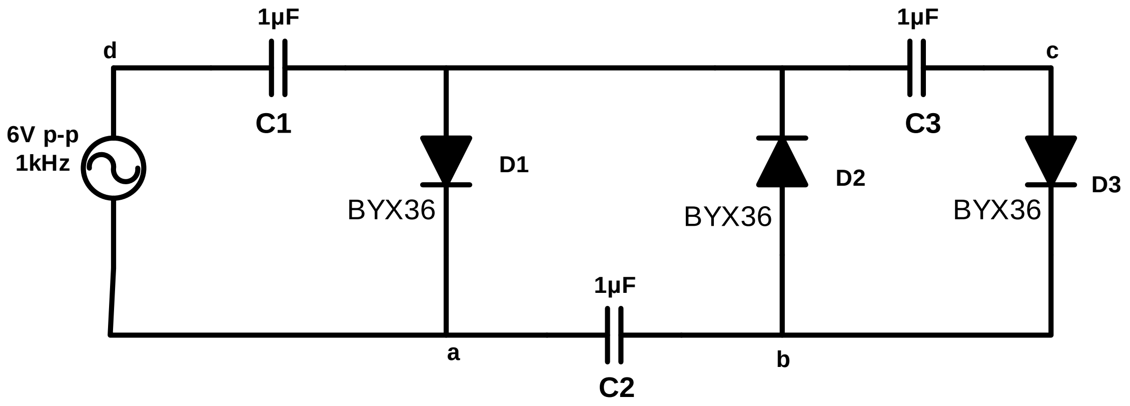


Fig.(6-8)

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

<u>Questions:</u>

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

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

Objectives:

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

Pre-lab Work

1. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
2. Verify if Simulation Results match the expected results

Procedure:

I. ***COMMON EMITTER TRANSISTOR AMPLIFIER.***

A.CE amplifier with voltage divider - bias

1. Connect the circuit of Fig. (7.1).

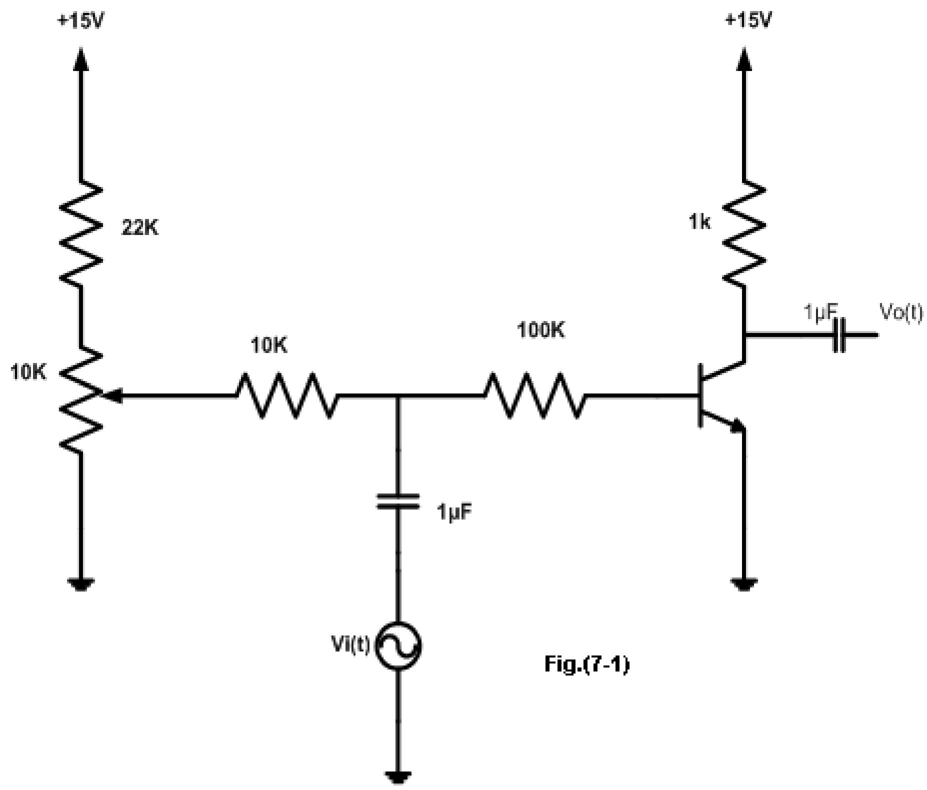


Fig.(7-1)

2. Switch on the power supply and the function generator.
3. Set the function generator frequency to 1 kHz sine wave and amplitude to zero.
4. Adjust the base bias potentiometer for a DC collector voltage (V_C) of 8 volts or as close as possible to it. Measure and record I_C,I_B, V_{CE}, V_{BE} and V_{BC}
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.(make sure there is no distortion due to saturation or cut-off)
7. Use oscilloscope to measure and record the input signal $v_i(t)$, the base voltage $v_B(t)$ and the output signal $v_o(t)$.
8. Calculate the voltage gain of the transistor $Av = Vo\ (t) / Vi\ (t)$ and $Av_1 = Vo\ (t) / V_B\ (t)$
9. Using DMM measure the AC currents for both the base and the collector of the transistor.
10. Calculate the current gain of the amplifier and the input impedance of the transistor amplifier.
11. What is the effect of the 100 kΩ resistor on the voltage gain?

II. <u>COMMON COLLECTER TRANSISTOR AMPLIFIER.</u>

1. Connect the circuit of Fig. (7.2).

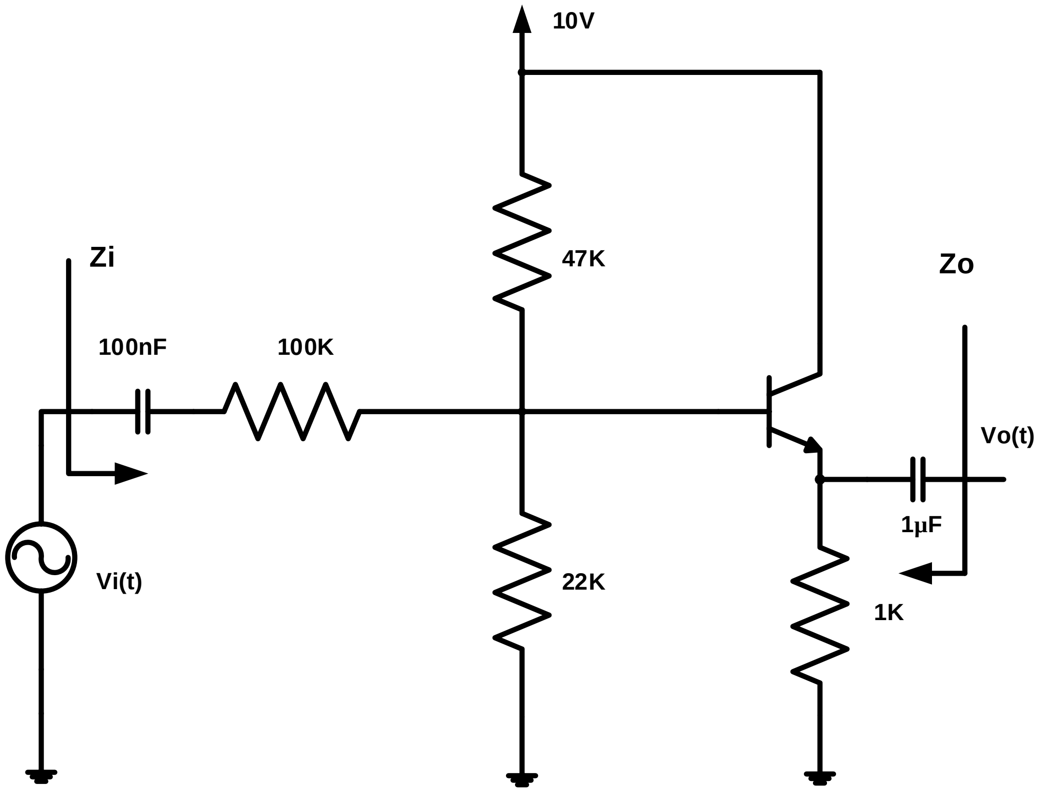


Fig.(7-2)

2. Ensure that the variable dc control knob is at minimum.
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 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. (make sure the waveform is undistorted).

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

8. Calculate the voltage gain Av.

9. Measure the ac voltage across the 100 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.

12. Calculate the current gain Ai.

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 table 7.1.

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

Table 7.1

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. (7.3).

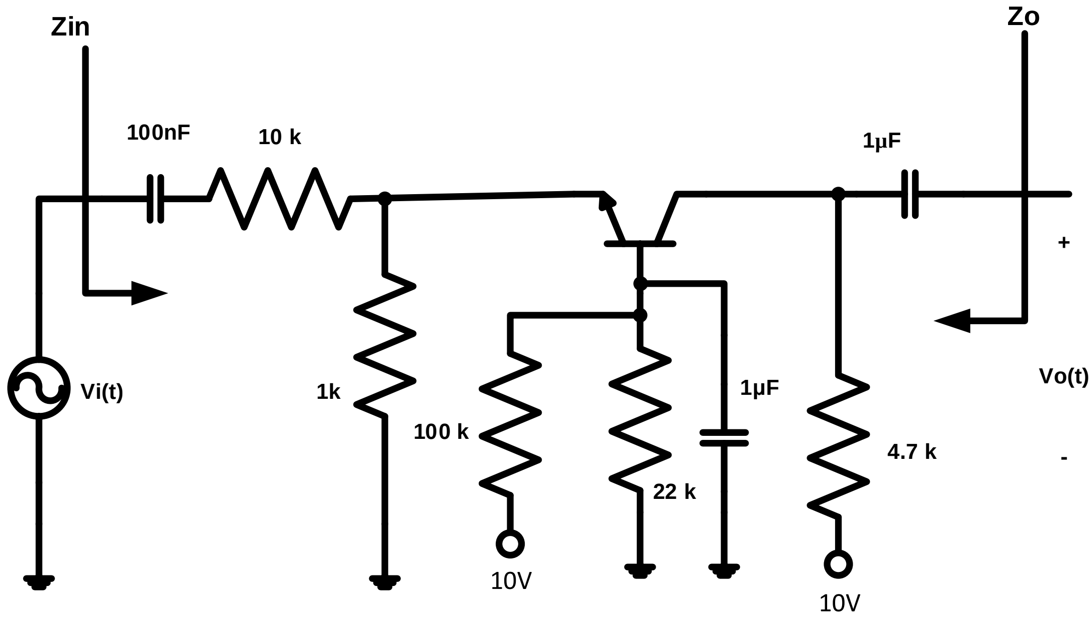


Fig.(7-3)

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.

16. Calculate the current gain.

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

18. 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.

19. Enter your results in a table like table 7.1.

<u>Questions:</u>

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

Experiment #8 ENEE2103

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:

1. Simulate the circuits in the procedure section and determine the required values (set the parameters that must be assigned by the instructor in the procedure to proper values).
2. Verify if Simulation Results match the expected results

Procedure:

I. CHARACTERESTICS OF AN N-CHANNEL JFET.

1. Connect the circuit of Fig.(8-1) taking into account the polarities.

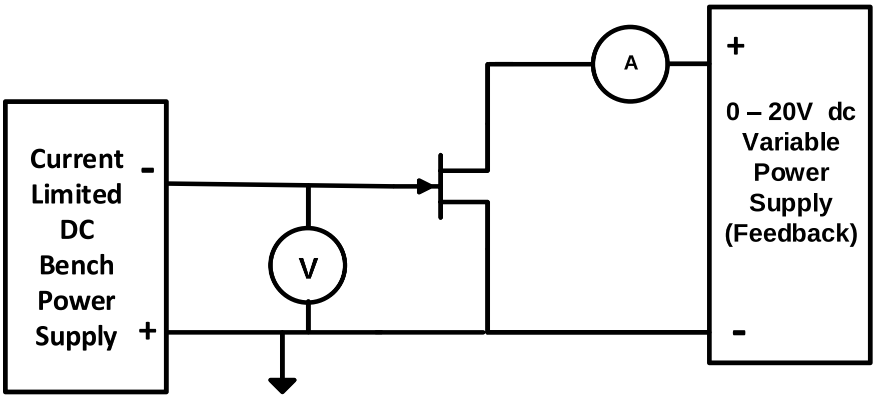


Fig. (8.1)

2. Set the current limit on the bench power supply to its minimum value .
3. Set the voltage to zero , then switch on the power supply.
4. Set the V_{DS} to the first value in table 8.1 , and then read I_D for each value of V_{GS}.
5. Repeat for all the values of V_{DS} in the table, recording the corresponding I_D values.
6. Plot the results from your table onto your graph, drawing one curve of I_D against V_{DS} for each value of V_{GS}.

Table 8.1

	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							

7. 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.

Note: When preparing the prelab in Pspice use dc and parametric sweep to get the curves measured in Table 8.1

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: trans-conductance g_m =(change in I_D)/(change in V_{GS}).

II. A JFET AMPLIFIER.

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

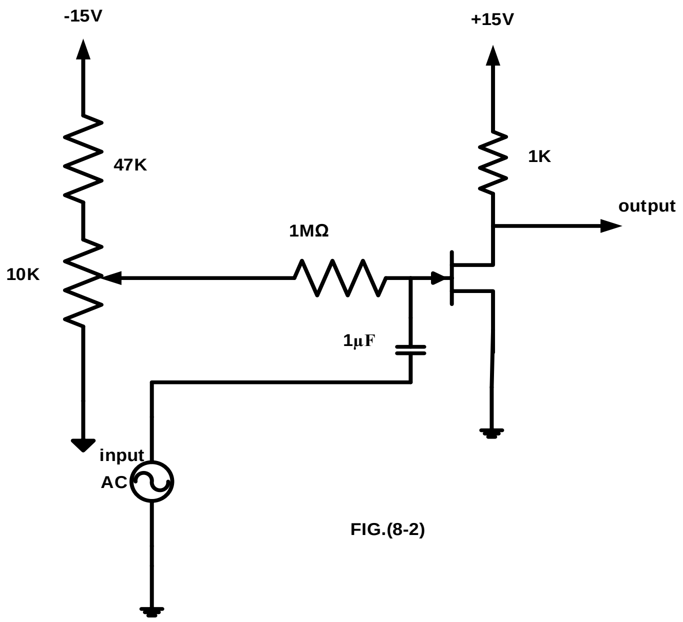


FIG.(8-2)

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

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

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

6. Measure the ac input current and voltage using DMM and calculate the input impedance Zin seen by the source.

III. COMMON DRAIN AMPLIFIER.

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

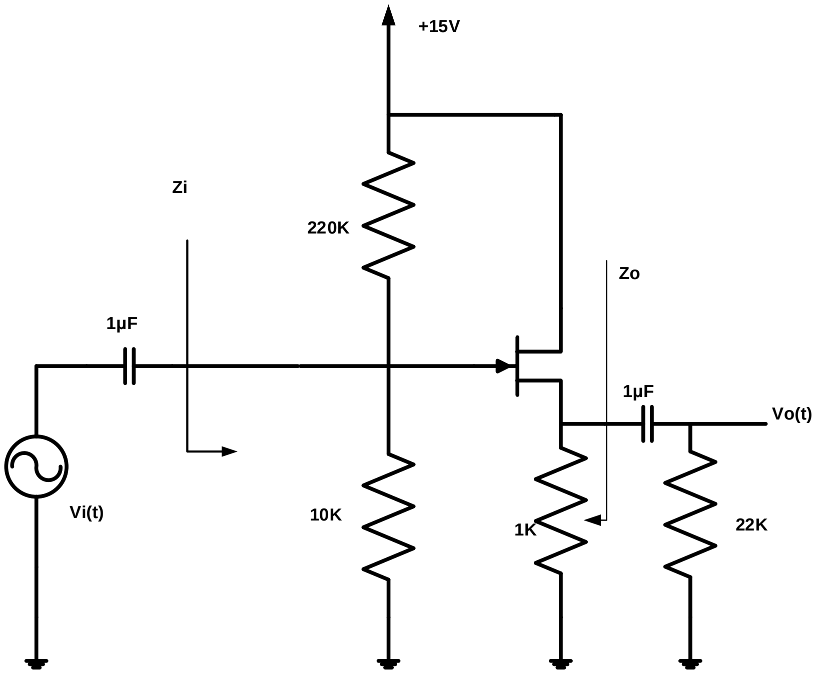


Fig.(8-3)

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.(8-4).

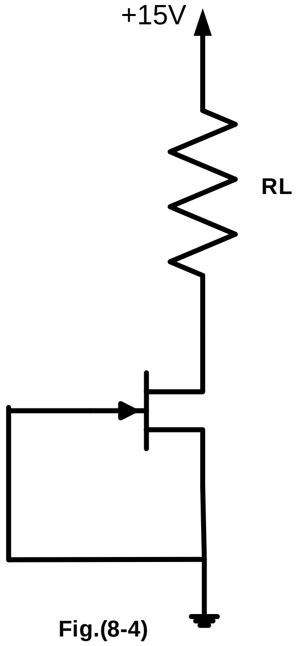


Fig.(8-4)

- Measure the values of the V_R ,and calculate the current I_D for each value of the resistor, then record your result in a table like table 8.2

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

Table 8.2

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 #9 ENEE2103

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 30 to give a peak to peak output of 2.5V , As shown in Fig.(9-1) : $Av_1 = 15$, $Av_2 = 2$, Vi = 100 mVp-p .
   Note: use common emitter amplifier configuration with voltage divider bias and emitter stabilization resistor for each stage,

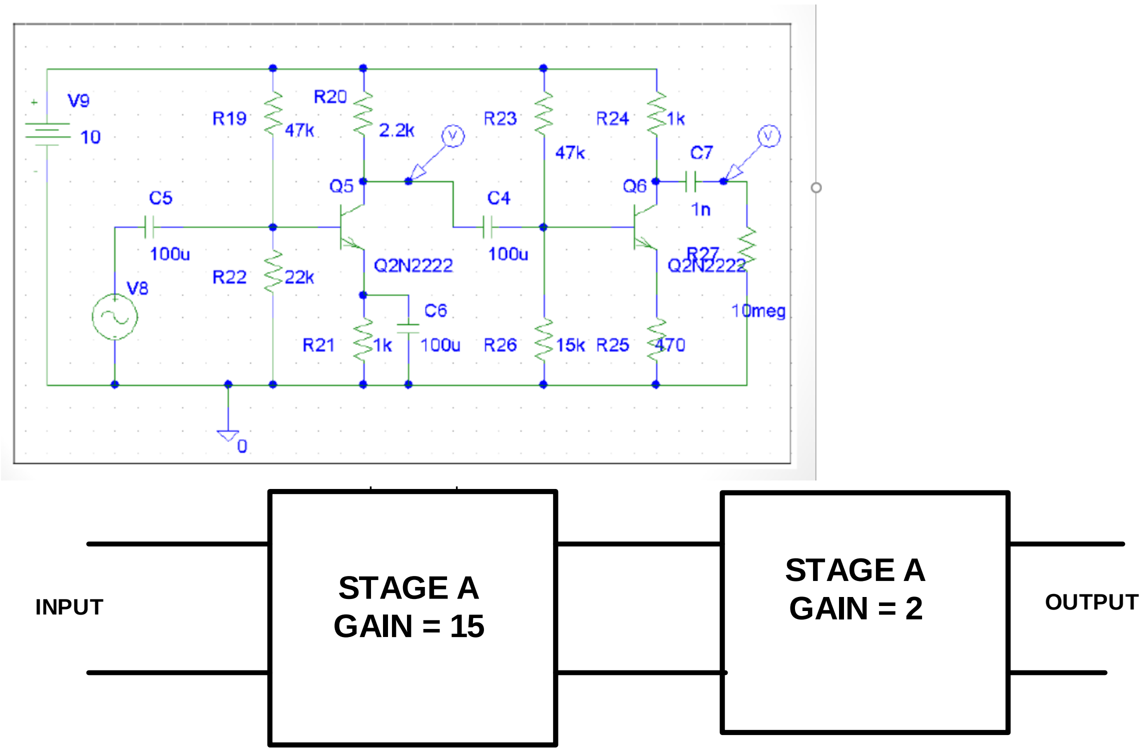


Fig.(9-1)

Design the first stage of the amplifier for the h- parameters of a BC107 transistor are : $hie = 2x10^3\ \Omega$, $hfe = 200$, $V_{CC} = 10V$.

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 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 1.5Vp-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 100 mVp-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 10Hz , 100 Hz ,200,300,400,500Hz, 1 kHz , 10 , 30, 100 ,300 , 500, 700, 1000 and 2000 kHz .

2. Calculate Vout/Vin for each frequency.

3. Record your results as in table 9.1.

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

Table 9.1

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.

Experiment #10 ENEE2103

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.(10-1), V1 is controlled by the potentiometer and V2, is obtained from the variable dc source on the trainer.

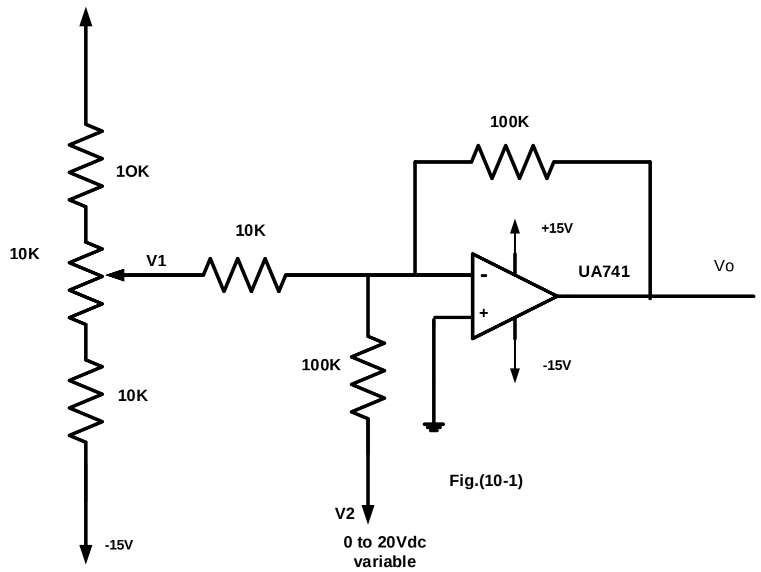


Fig.(10-1)

2. Measure the output voltage for V1, V2 as shown in table 10.1.

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 10.1

3. Calculate the expected output voltage for each step using the formula : Vo $=XV_1 + YV_2$ where X, Y is the resistors ratios.

II. *Voltage Follower Application*

1. Set up the circuit of Fig.(10-2).

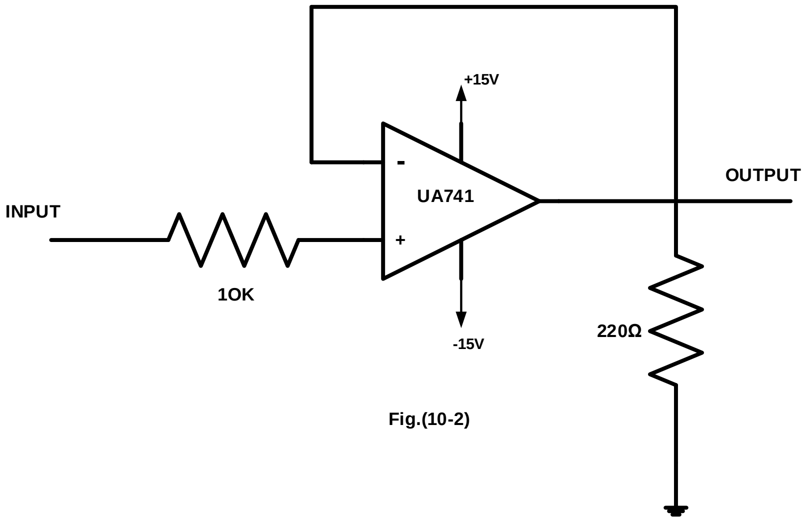


Fig.(10-2)

2. Draw the output Vo(t) for Vi(t) is 2V p-p sinusoidal with 100Hz.
3. Measure and records Vo for Vi (1V,2V,3V,4V,5V,6V,and 7V).
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 ?
- For what applications is this circuit used?
- What is the relation between your Vi, Vo?
- What the approximate value of maximum output current of the op-amp?

III. *Comparator Application*

1. Set up the circuit of Fig.(10-3).

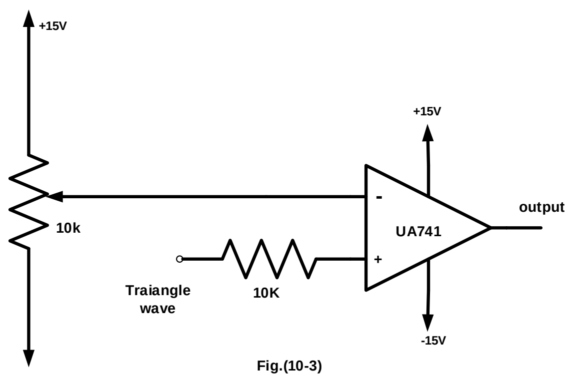


Fig.(10-3)

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

1. Set up the circuit of Fig.(10-4).

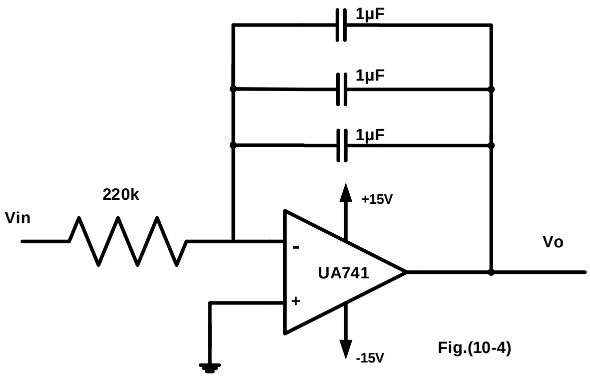


Fig.(10-4)

2. Put an input signal with $2V_{P-P}$ and a frequency of 30 Hz, and draw the output signal from the oscilloscope.

3. Repeat for all types of signals you have.

4. Set up the circuit of Fig. (10-5).

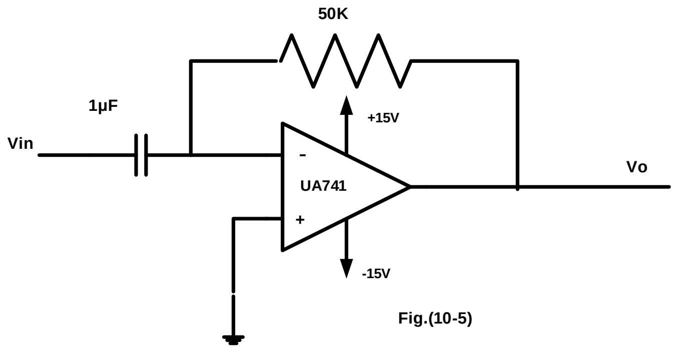


Fig.(10-5)

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.(10-6).

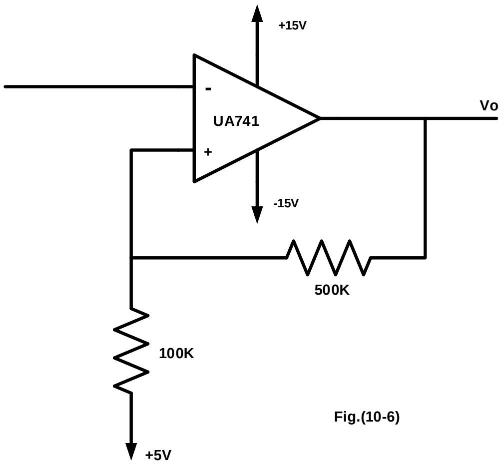


Fig.(10-6)

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.(10-7).

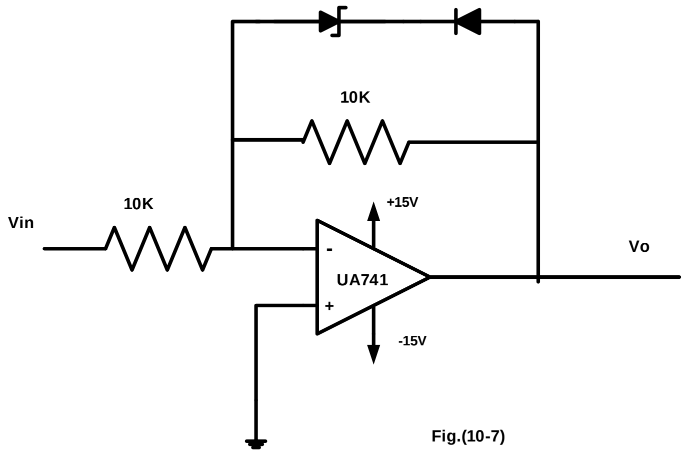


Fig.(10-7)

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 #11 ENEE2103

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.

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 (11-1).*

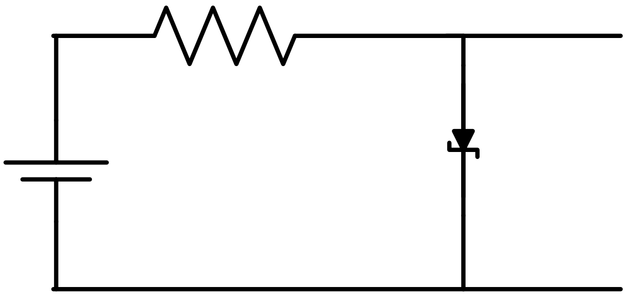


Fig.(11-1)

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 11.1.

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 11.1

4. Connect the circuit shown in Fig(11-2).

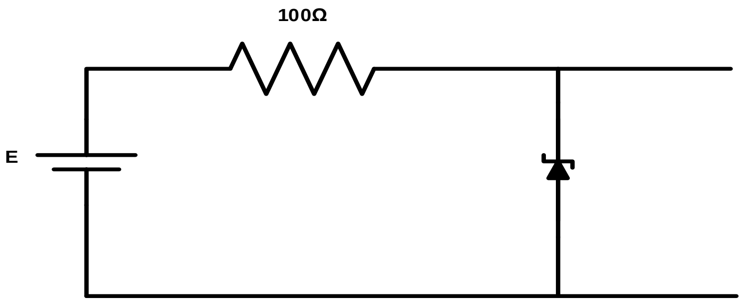


Fig.(11-2)

5. Set the applied voltage E to (0.1, 0.5, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10,11,12,13,14,15)V.

6. For each value of E, measure the voltage across the zener diode and calculate the current through the zener diode. (Fill in Table 11.2)

Table 11.2

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			

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(11-3).

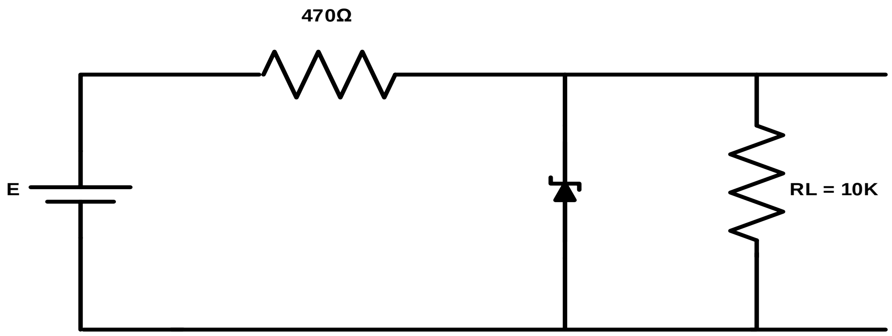


Fig.(11-3)

9. Set E to (10,11,12,13,14)V and measure the load voltage V_L. (Fill Table 11.3)

Table 11.3

E	10	11	12	13	14
V_L					

10. With E set to 10V measure the load voltage V_L for R_L=(8.2K,6.8K,4.7K,2.2K) and Fill in Table 11.4

Table 11.4

R_L	8.2k	6.8k	4.7 k	2.2k
V_L				

II. THE VOLTAGE REGULATED POWER SUPPLY.

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

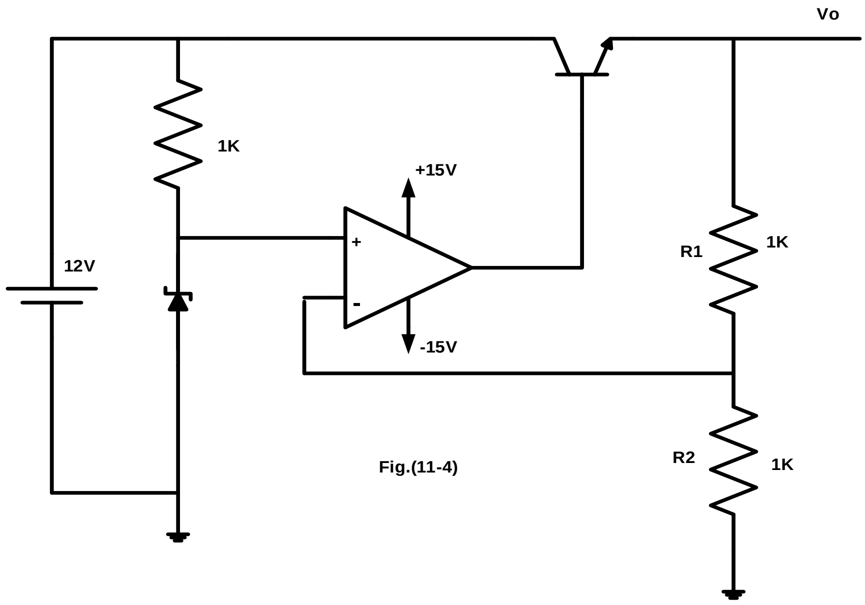


Fig.(11-4)

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.

Table 11.5

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

Table 11.6

RL	R2	Vo
1kΩ	470 Ω	
1kΩ	2.2kΩ	

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

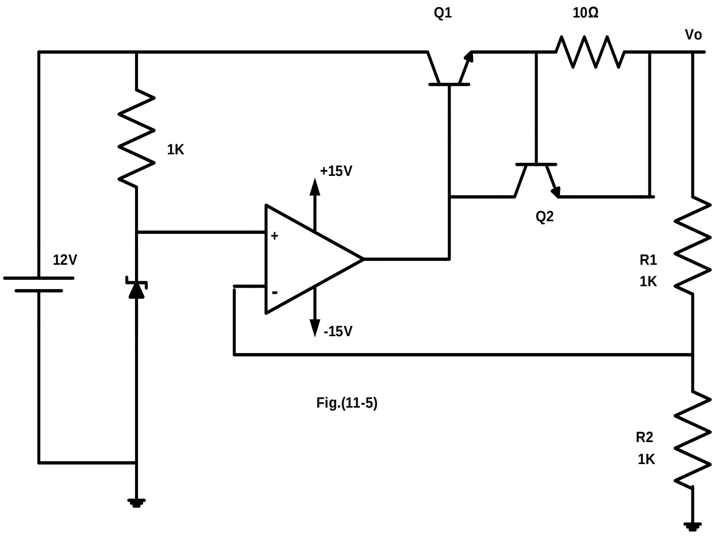


Fig.(11-5)

8. Measure Vo.

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

Table 11.7

R_L	Open	1kΩ	680 Ω	470 Ω	220 Ω	100 Ω	50Ω	40Ω
Vo								
Io								

Table 11.8

RL	R2	Vo
1kΩ	470 Ω	

II. THREE TERMINAL FIXED VOLTAGE REGULATOR 7805.

1. Connect the circuit of Fig (11.6).

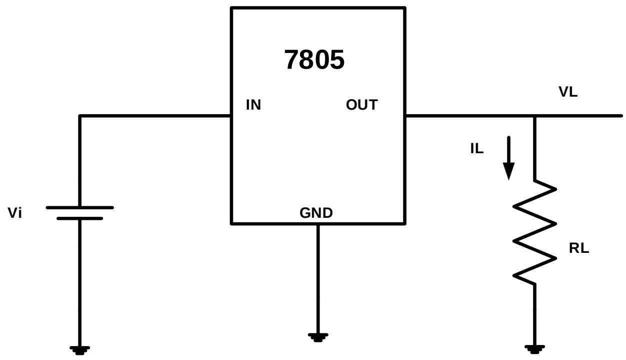


Fig.(11-6)

2. With Vi=10V measure IL and VL for the load resistances listed in the table 11.9.

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

Table 11.9.

3. Using the results of table 9.2 , 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 11.10. 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 11.10

Using the results of table 11.10 , determine the line regulation of the 7805 .

line regulation $= \Delta V_L/\Delta Vi$

III. THE LM317 ADJUSTABLE VOLTAGE REGULATOR.

1. Connect the circuit of Fig.(11.7).

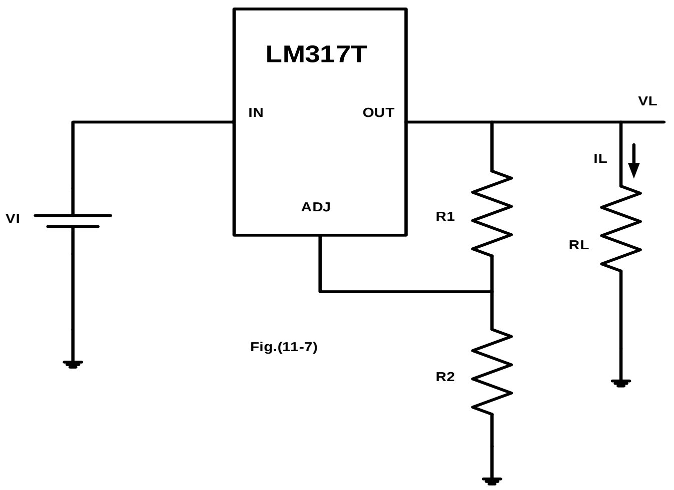


Fig.(11-7)

2. With Vi=10V, R1=100Ω, RL=1k , adjust R2 as shown in table 11.11.

R_2(Ω)	V_L(V)	I_L(m A)
0		
100		
200		
300		
500		
700		

Table 11.11

3. Measure and record VL,IL for each R value.

4. With RL =1k , R1=100 ohm , R2=220 , adjust Vi as listed in table 11.12.

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

Table 11.12

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 11.13.

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

Table 11.13

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

Appendix A

Introduction to Simulation and Report Writing

➢ Objective:

This experiment provides an introduction to the basics of circuit's simulation using "PSPICE 9.1 schematic" which will be used in pre-labs through this course. Also, an introduction to technical report writing is provided.

➢ Equipment Required:

1. Personal computer with PSPICE 9.1 and Microsoft word.

➢ Introduction:

Through the course this lab students have to prepare for experiments by studying theoretical background related to experiment, and by doing pre-labs which includes simulation of circuits.

Computer simulation programs, such as PSPICE, simulates the behavior of electric circuits on a digital computer and tries to emulate both the signal generators and measurement equipment such as multimeters, oscilloscopes, and curve tracers. Though, by using PSPICE it is like you have a virtual circuit lab on your personal computer.

Also, by using PSPICE students can easily subject the circuit to various stimuli (such as input signals and power supply variations) and to see the results plotted out graphically using PSPICE's post processor called Probe. Therefore, students can obtain results before they come to lab, and the laboratory experiments become reinforcement to the subject matter at hand.

We will be using PSPICE 9.1 student version for simulation, and you can use newer versions provided by OrCAD. The first part of this experiment is dedicated to introduce students to the different applications of PSPICE that are related to the lab.

The second part of this experiment is dedicated to introduce students to the guidelines of report writing. The Bachelor of Science degree in Electrical Engineering involves numerous courses that require written reports. These courses also include laboratories, which require reports too. The fact is, once you graduate, industry will require you to write well. In some cases, you will be involved in writing proposals or possibly final design reports. Certainly, you will always be required to write short reports and memos detailing your activities. The second part of this experiment is dedicated to introduce you to the basic guidelines of report writing.

➢ Installing PSPICE 9.1

- To download PSPICE 9.1 student version, visit the following link:
  http://www.electronics-lab.com/downloads/circutedesignsimulation/?page=5

You will find a list of programs, scroll down until you see **PSPICE 9.1 Student Version**, then click download. The setup is straight forward, however, in case you needed help check the following video and follow steps.
https://www.youtube.com/watch?v=tCFjjHY94Ro

➢ Building and simulating circuits

- To open PSPICE 9.1, search your computer for **"Schematic"** and open the program. This is shown in figure.1 for windows 10 users. **Note: zoom in to see figure details if not clear.**

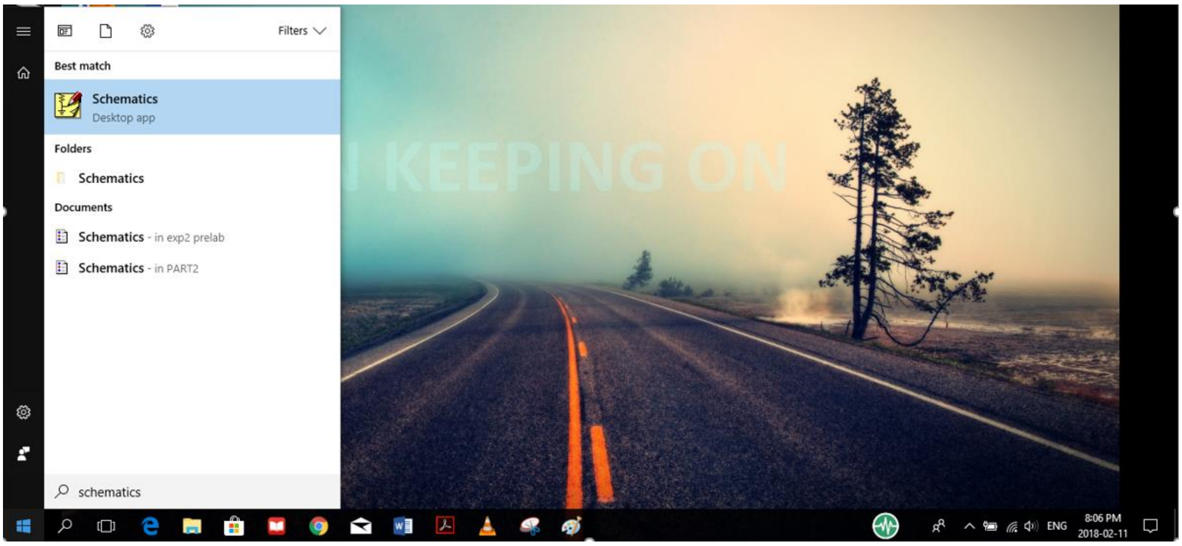


Figure.1

- Click on Schematics icon, then you will see the window shown on figure.2

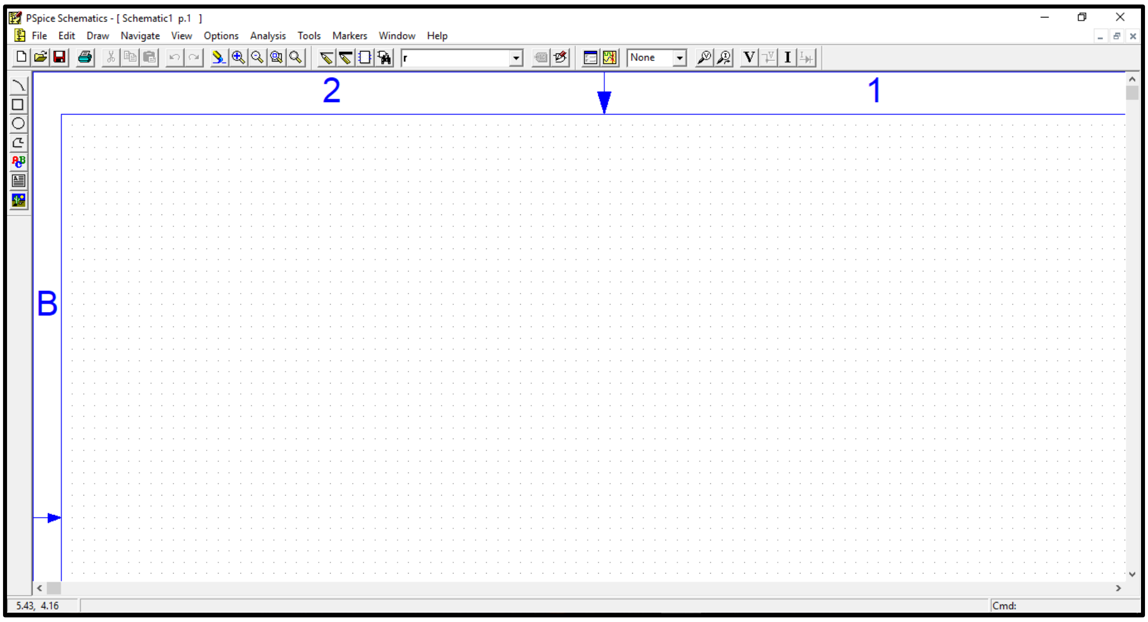


Figure.2

- To build the circuit of figure.3 on PSPICE, click on **"get new part"** icon shown in figure.4:

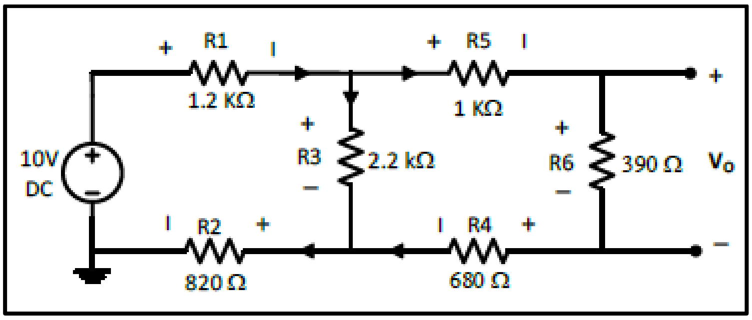


Figure.3

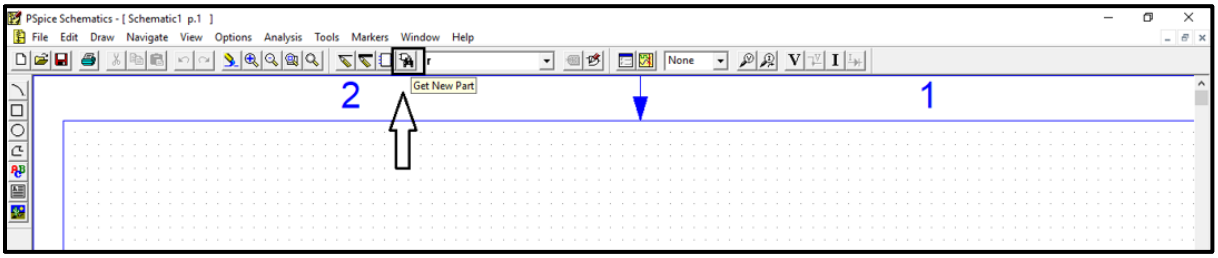


Figure.4

- When you click on get new part icon, you will see the window shown on the left side in figure.5. This is a list of all components available in the program. Click on "advanced" so you can see the picture of part you want to add as shown on the right side in figure.5.

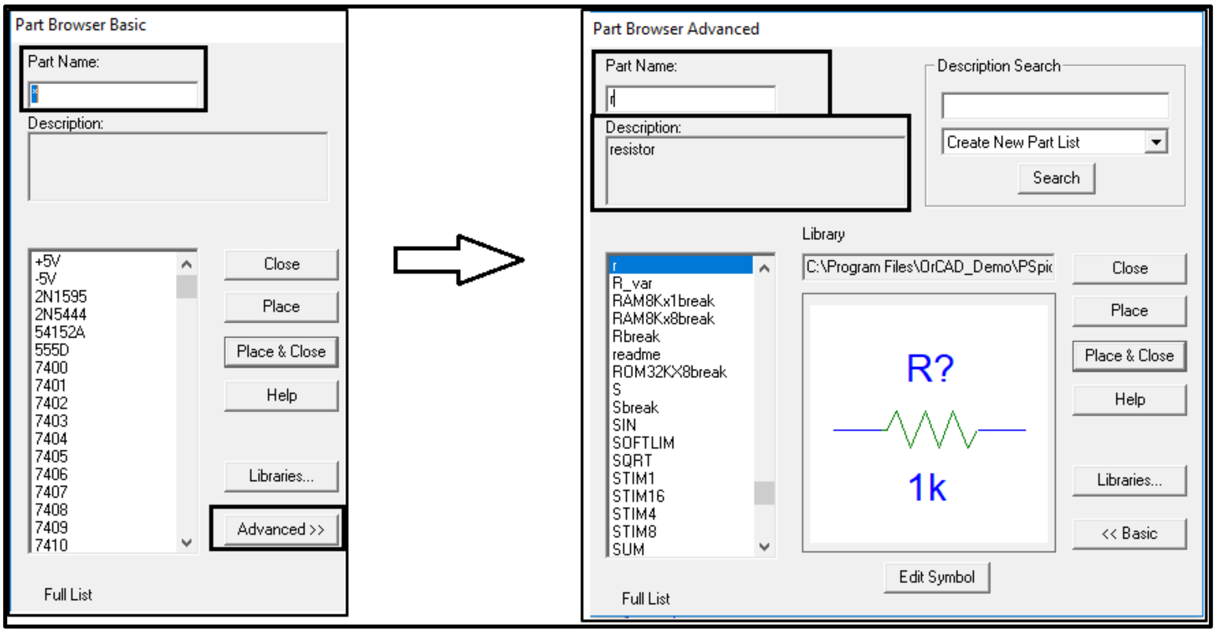


Figure.5

- To add a circuit part, you can search by typing part name in the specified field. After finding the part you want, click on "place and close", then click wherever you want to add a part. After you finish adding components, click on "Esc" on your keyboard to end the mode.
- Note that each circuit element in PSPICE have a specific name that might be different from its name in circuit analysis. Table.1 provides the names of mostly used circuit elements in the circuit lab.

Table.1

Circuit element	Part name in PSPICE
Resistor	R
Capacitor	C
Inductor	L
DC Voltage source	Vdc
DC Current source	Idc
Periodic square voltage source	Vpulse
AC Voltage source: two types	vsin (sinusoidal voltage source used in transient analysis)
	vac (variable frequency source used in ac sweep analysis)
Ground	gnd_analog
Operational amplifier	ua741

- Note that when you place parts, each circuit part in PSPICE has a name and a value as shown in figure.6 Always make sure to place circuit parts in PSPICE in a similar way to the circuit given in manual. For example, check the way the parts are placed in figure.6.

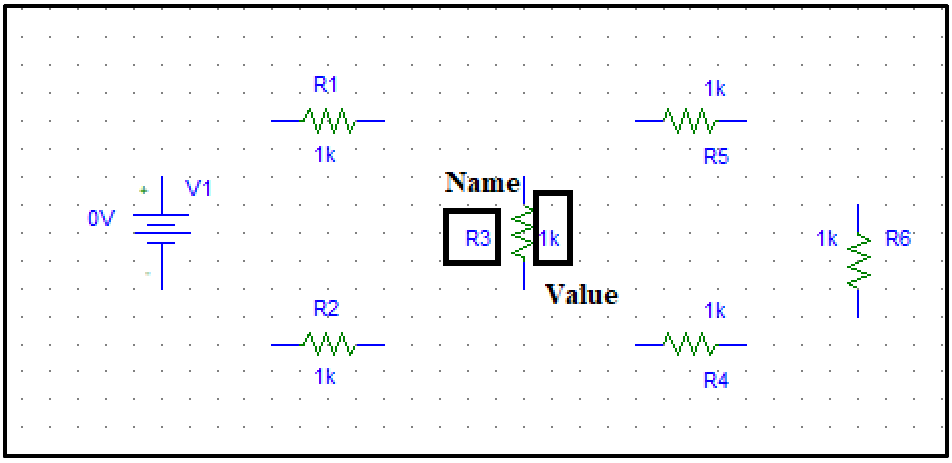


Figure.6

- To connect the components, click on "draw wire" icon shown in figure.7, **a common mistake** that students do is clicking on "draw bus" icon which is next to "draw wire". Avoid doing this mistake!
- Also avoid drawing wire over components (shorting parts). Click where you want each vertex of the wire. Each click ends a wire segment and starts a new one

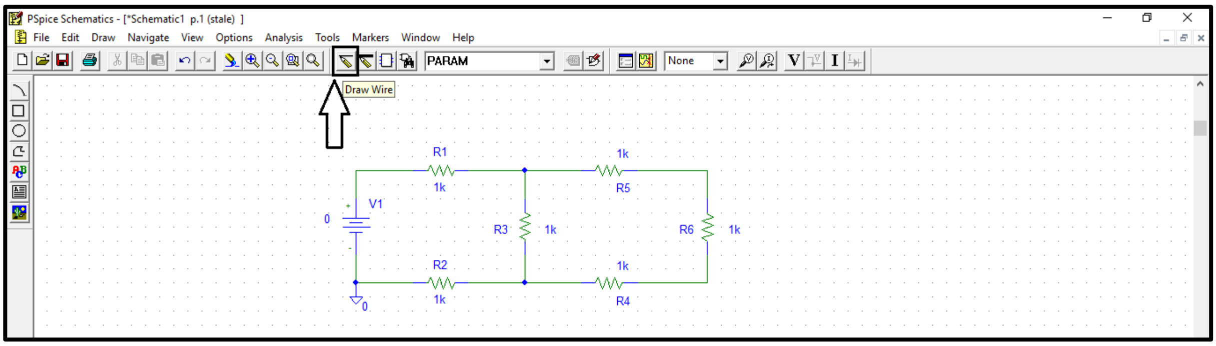


Figure.7

- To change a part value e.g. resistance, voltage source … etc. double click on its value and the window in figure.8 will show up, type the value you want inside the box then click ok.

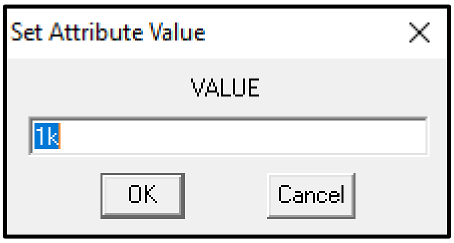


Figure.8

- When elements such as resistors and voltage sources are given values, it is convenient to use unit prefixes. PSPICE supports the prefixes listed in Table.2. Note that the letter must immediately follow the value – no spaces. Also, PSPICE is case insensitive so, there is no difference between 1M and 1m in PSPICE.

Table.2

PSPICE Unit Prefixes		
K - kilo - 10^3	MEG - mega - 10^6	G - giga - 10^9
M - Millie - 10^{-3}	U - micro - 10^{-6}	N - Nano - 10^{-9}

- **PSPICE requires that all schematics have a ground**, the voltage there will be zero and all other node voltages are referenced to it. **If you do not place a ground, you will get an error and will not be able to simulate your circuit**. The part you need is either the analog ground (GND_analog) or the earth ground (GND_earth) which are equivalent, you can get them from "get new part option". In this example, we used GND_analog.
- Your schematic is finished now and ready for saving, click on "save" then on "simulate" button as shown in figure.9.

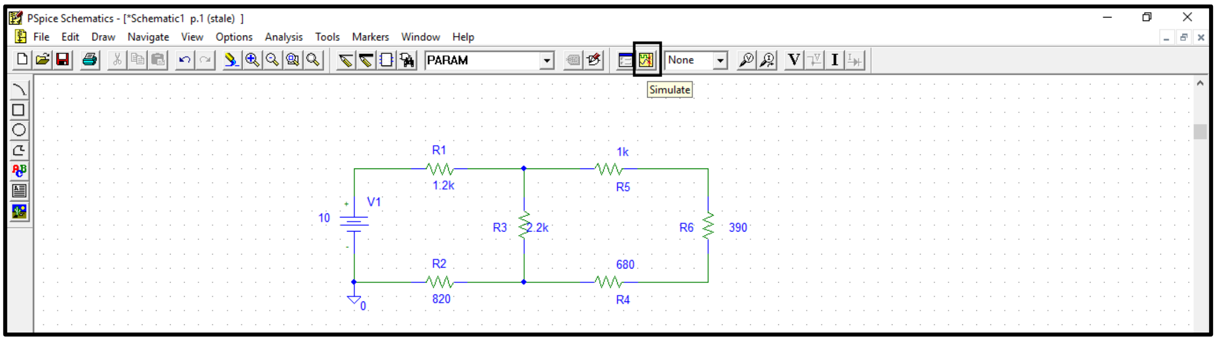


Figure.9

- When you click on "simulate" button the window shown in figure.10 will show up (simulation output window), if your circuit doesn't have any errors you will see the message in the box in figure.10.

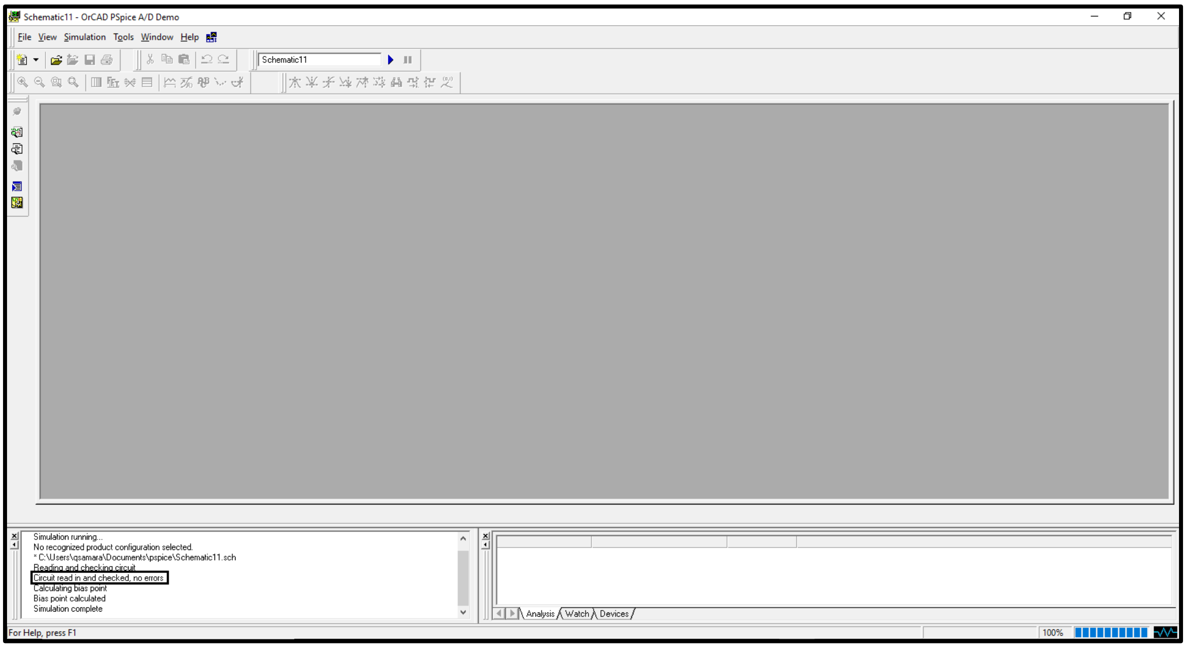


Figure.10

- Return to the schematics window and click on **“V”** and **“I”** buttons shown in figure.11. By clicking on these buttons, PSPICE displays the voltage on each node “**with respect to ground”** and the current on each branch.

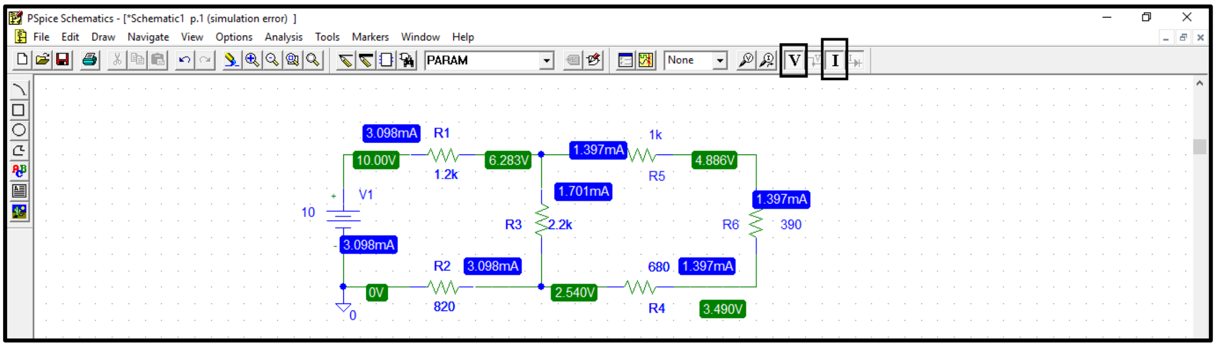


Figure.11

- **The Passive Sign Convention and PSPICE:**

All currents and voltages in PSPICE and Schematics obey the passive sign convention shown in Figure.12. The voltage across the element is defined positive at node 1 with respect to node 2, and current is entering a device from its “1” end and leaving its “2” end.

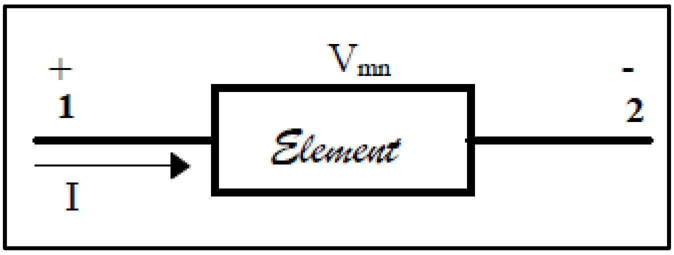
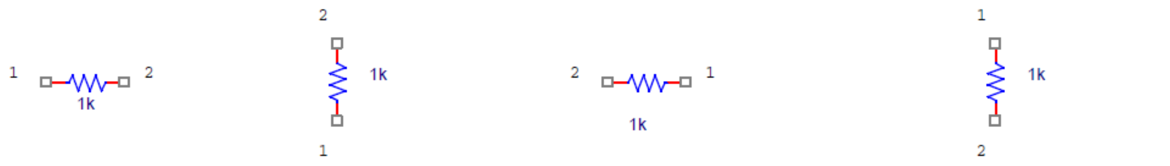


Figure.12

Version Second Semester 2020-2021

All two leaded passive components have an **implied** “1” end and a “2” end (not visible on schematic). Whenever you place a component, it takes a default position, for example, a resistor, capacitor, or inductor will take a default position with its “1” end to the left as shown in figure13-(a). A component may be rotated by activating it, then right-clicking and selecting Rotate, or by typing the letter “r” (see b). Each rotation moves the component counterclockwise by 90º. To get the “1” end facing up, you must rotate the component 3 times from its default position as indicated in (c).



(a) Default (b) First rotation (90 deg) (c) Second rotation (90 deg) (d) Third rotation (90 deg)

Figure.13

- Knowing about component layout is important when you are viewing your results in Probe to be discussed later.

➢ Types of Analysis Performed by PSPICE

PSPICE is capable of performing four main types of analysis: Bias Point, DC Sweep, AC Sweep/Noise, and Time Domain (transient).

- **Bias Point:**

The Bias Point analysis is the starting point for all analysis. In this mode, the simulator calculates the DC operating point of the circuit. **This is the type which was explained in the previous section.**

- **DC Sweep**

The DC Sweep analysis varies a circuit part e.g. voltage source, global parameter (resistor) over a specified range in an assigned number of increments in a linear or logarithmic fashion.

- **Time Domain (transient)**

The Time Domain (transient) analysis is probably the most popular analysis. In this mode, you can plot the various outputs as a function of time.

- **AC Sweep/Noise**

The AC Sweep/Noise analysis varies the operating frequency in a linear or logarithmic manner. It linearizes the circuit around the DC operating point and then calculates the network variables as functions of frequency.

➢ Example on DC Sweep

- To perform a DC sweep for the same circuit of the previous section, click on the "setup analysis" icon shown in figure 14.

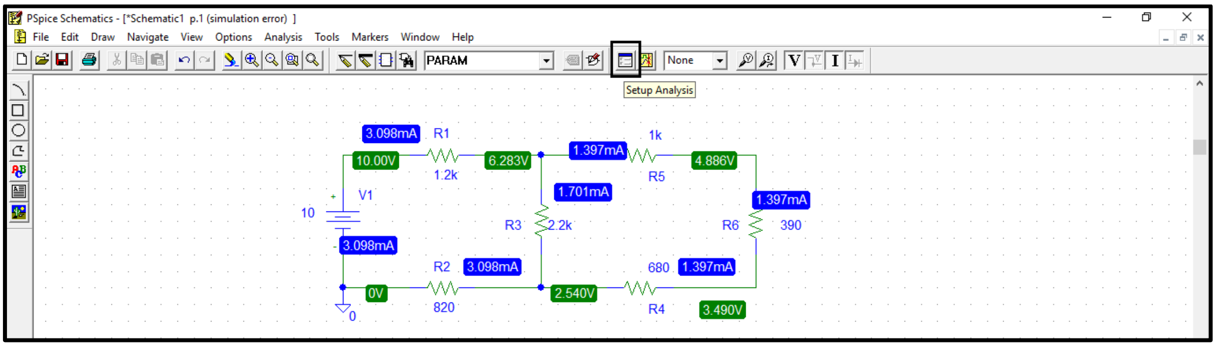


Figure.14

- The window shown in figure.15 will show up:

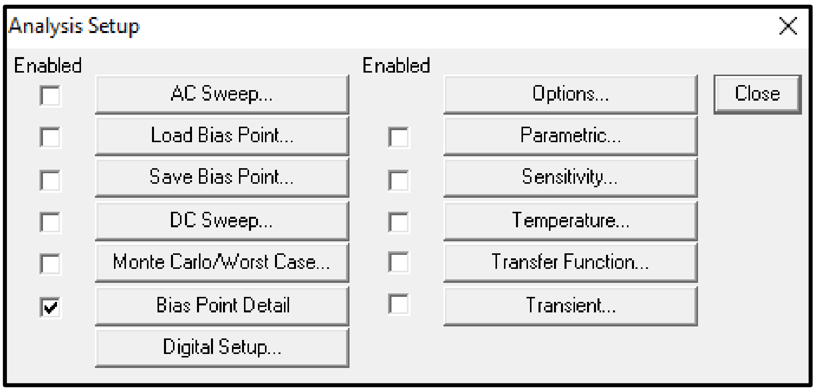


Figure.15

- Click on "DC sweep" and the window shown in figure 16 will show up, you have to fill the spaces indicated inside boxes in figure.16.

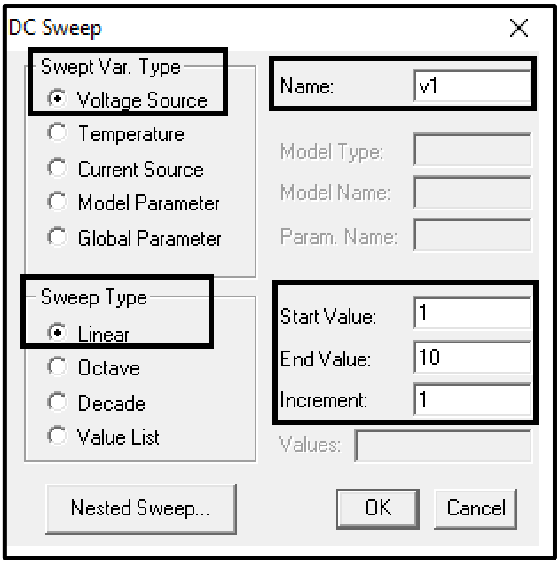


Figure.16

- Click on simulate, then the simulation output window will be as shown in figure.17:

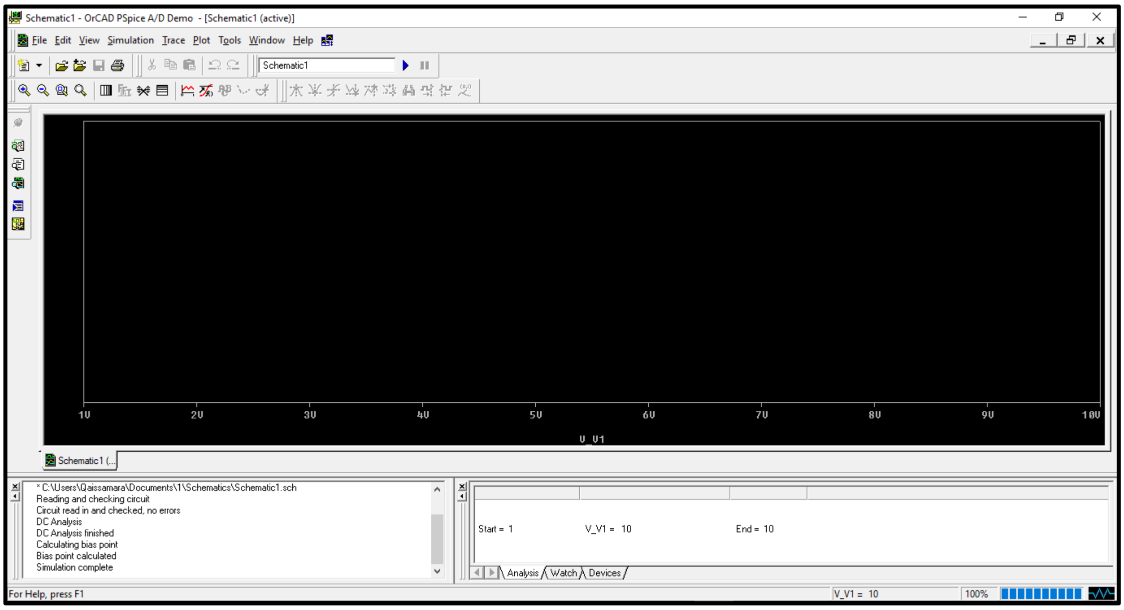


Figure.17

- The output now is displayed as curve rather than a point, to display the voltage across R6 for example click on "Voltage Marker" and place it where you want as shown in figure.18.

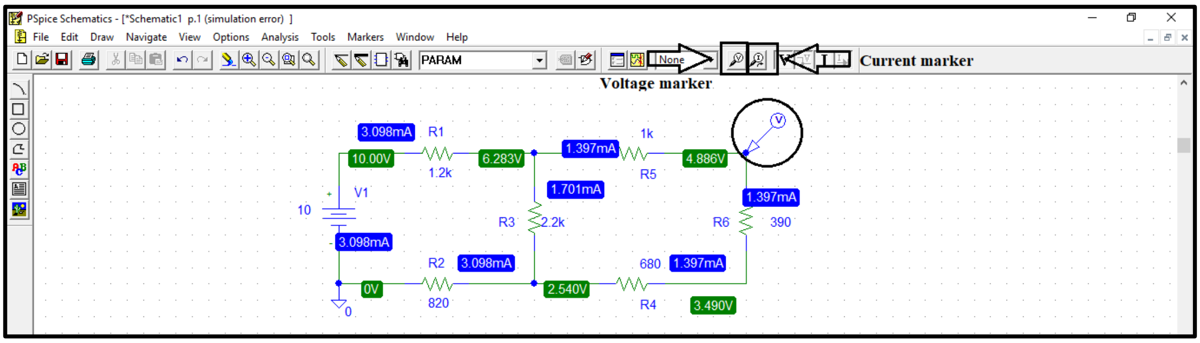


Figure.18

- When you place the marker, the output simulation window will be as shown in figure.19. this is a curve of the voltage at the node you placed the marker **with respect to ground.**

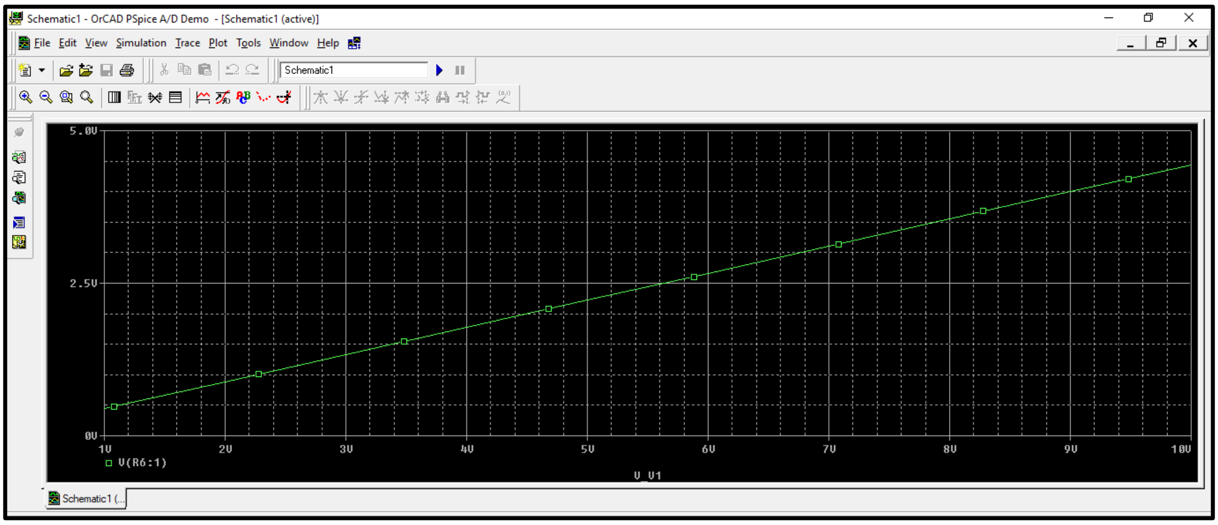


Figure.19

- If you want to display the voltage on R6 only, then go to schematic window, click on marker, and select "mark voltage differential" as shown in figure.20. (you will place two markers)

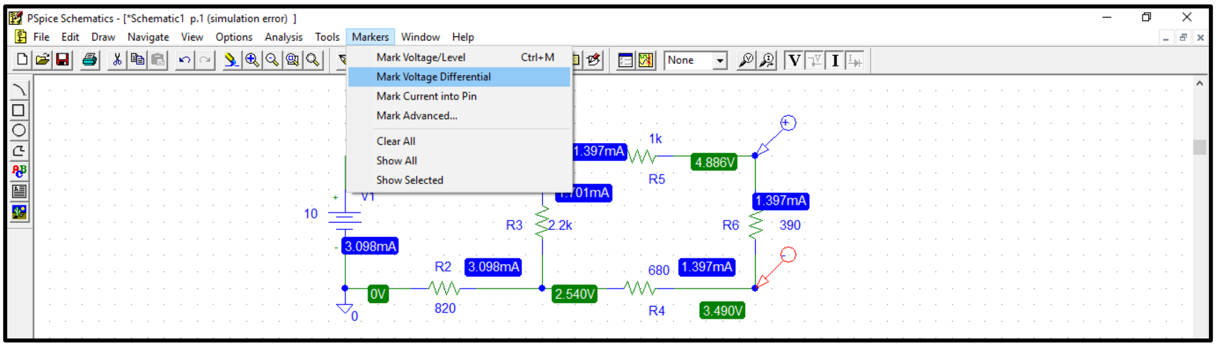


Figure.20

- The output now will be as shown in figure.21. **Note the difference on y-axis range!**

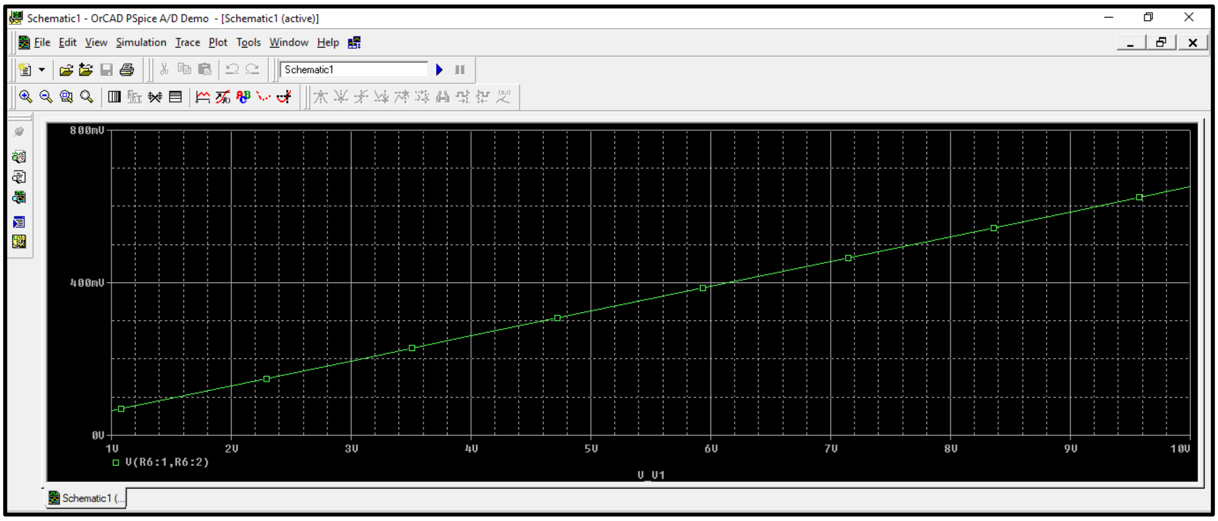


Figure.21

- You also can add curves, (traces) from simulation output window by clicking on traces and select add trace as shown in figure 22. You show current trace by clicking upon Current Maker (note that current marker must be placed on the terminals of a part to display current).

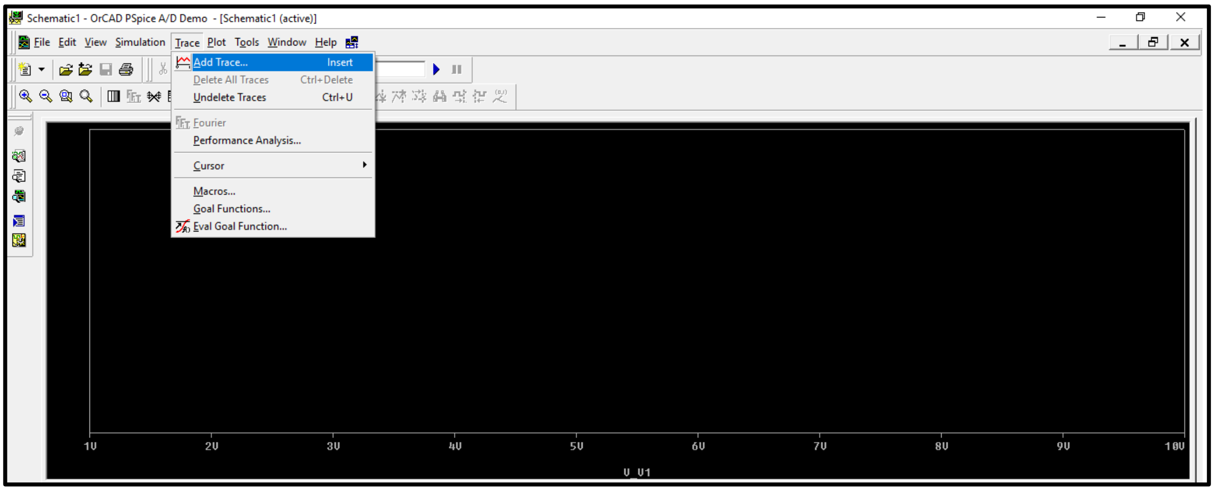


Figure.22

- Then the window shown in figure 23 will show up, in this window you will find a list of all variables in the circuit: (voltages and currents on all parts) to add a trace just click on it and it will show up in the field indicated in the box in figure.23.

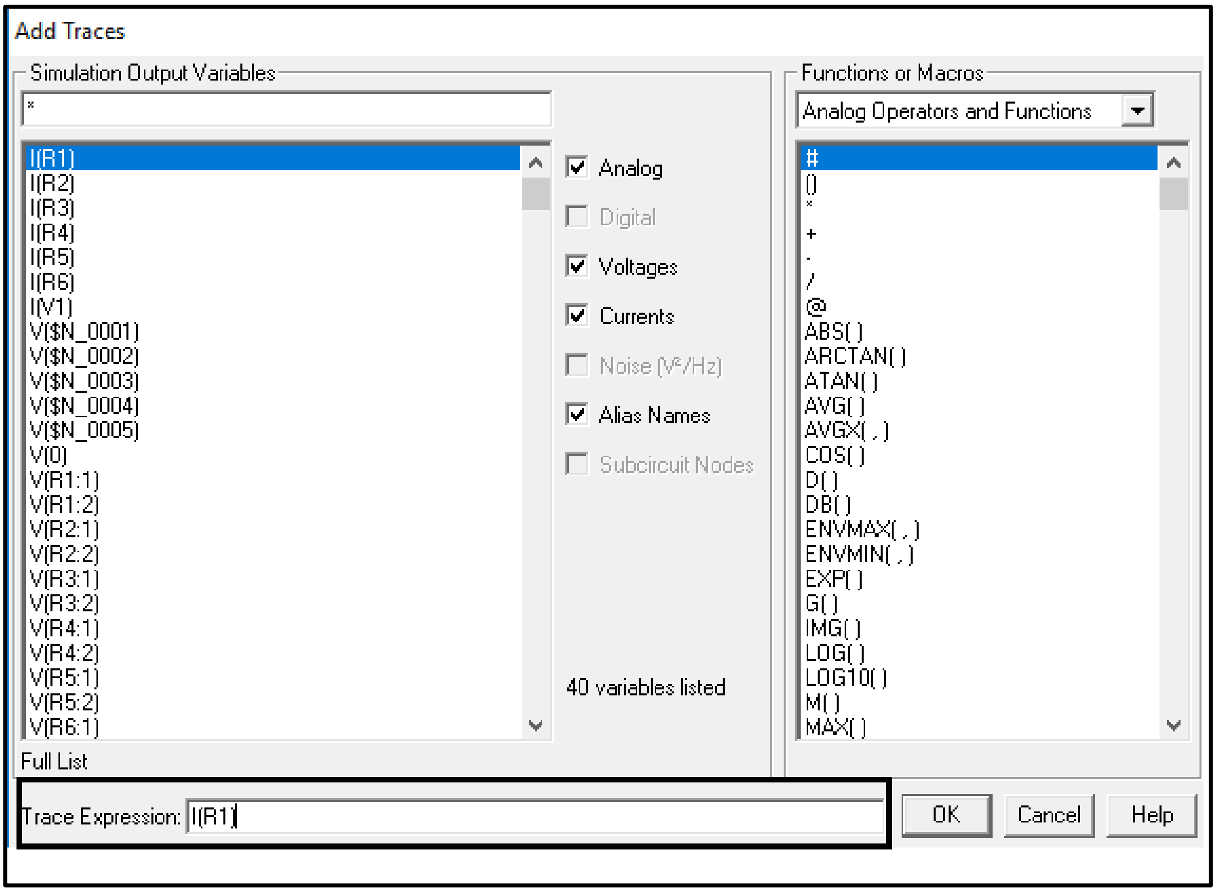


Figure.23

- When you finish selecting variables click on ok, and the trace you selected will be shown in simulation output window as shown in figure.24.

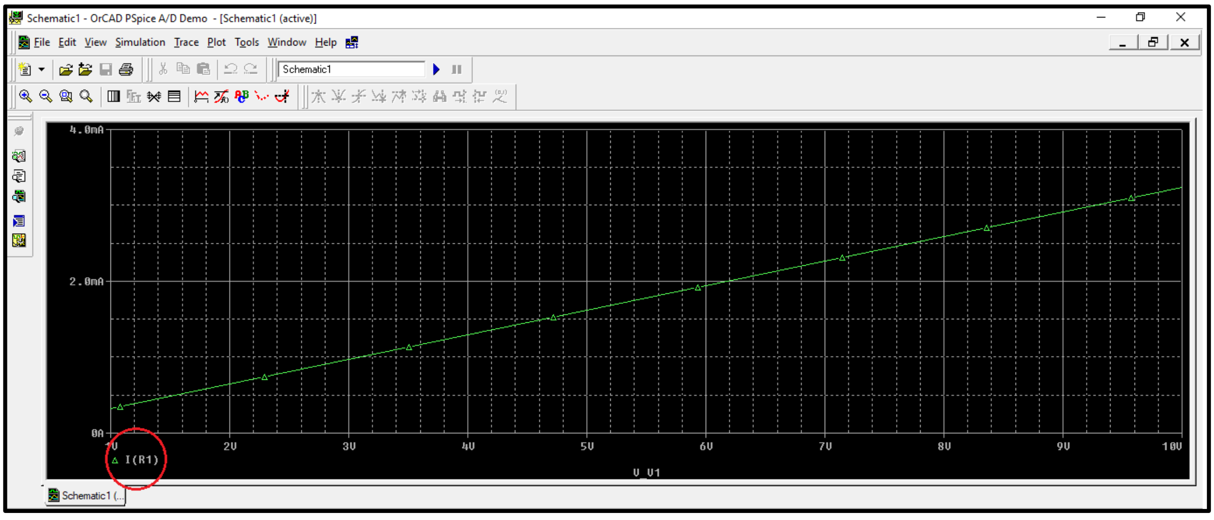


Figure.24

- In the “add trace window you can also add a trace that is a combination of more than one circuit variable e.g. to add a trace of the power dissipated in R6, note the equation inserted in figure 25.

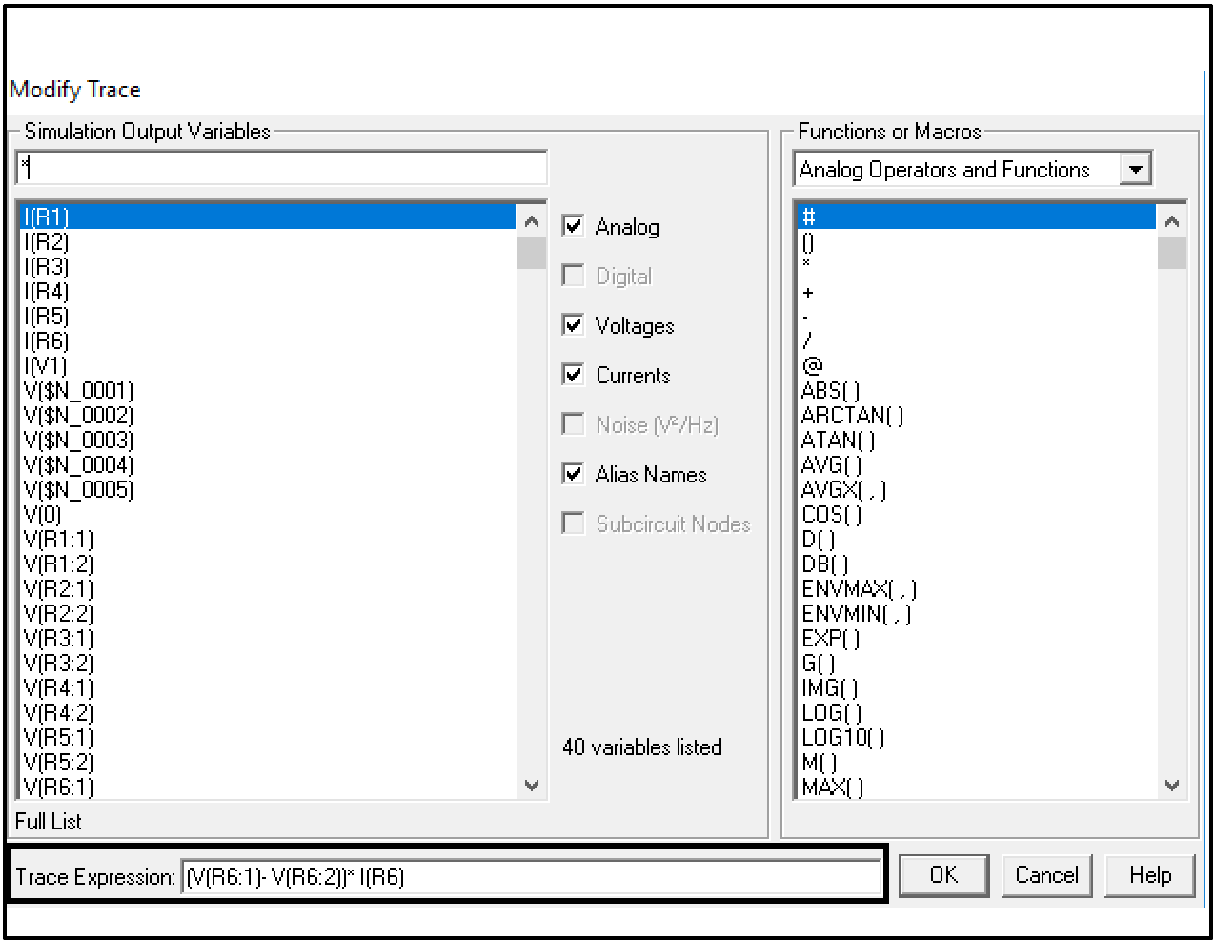


Figure.25

- The output is displayed in figure.26, note that the y-axis unit now is Watt.

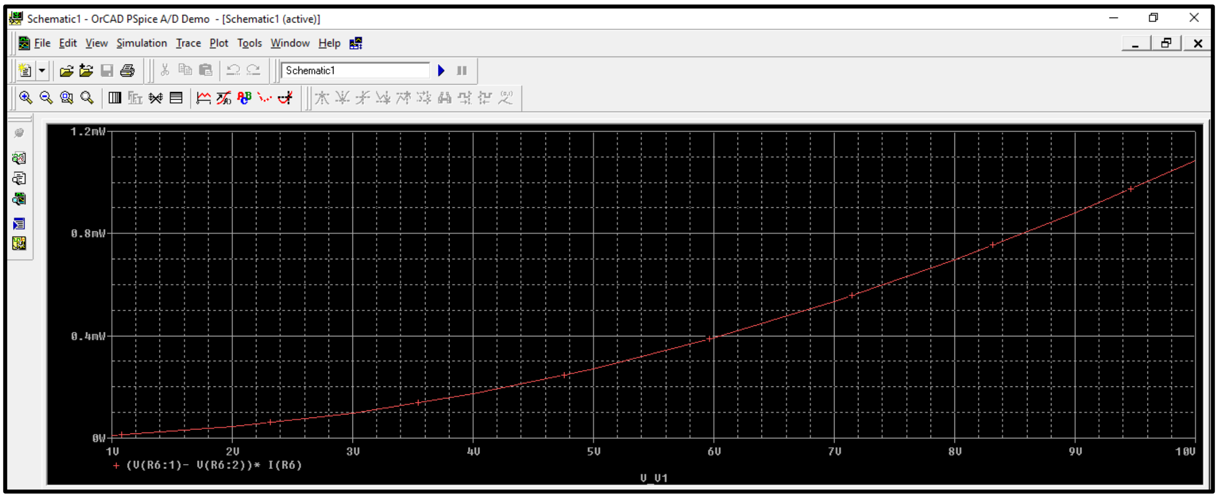


Figure.26

- **To perform a DC sweep for a resistor in the same circuit**, you need to add "param" part, go to "get new part" and search for "param", as shown in figure.27.

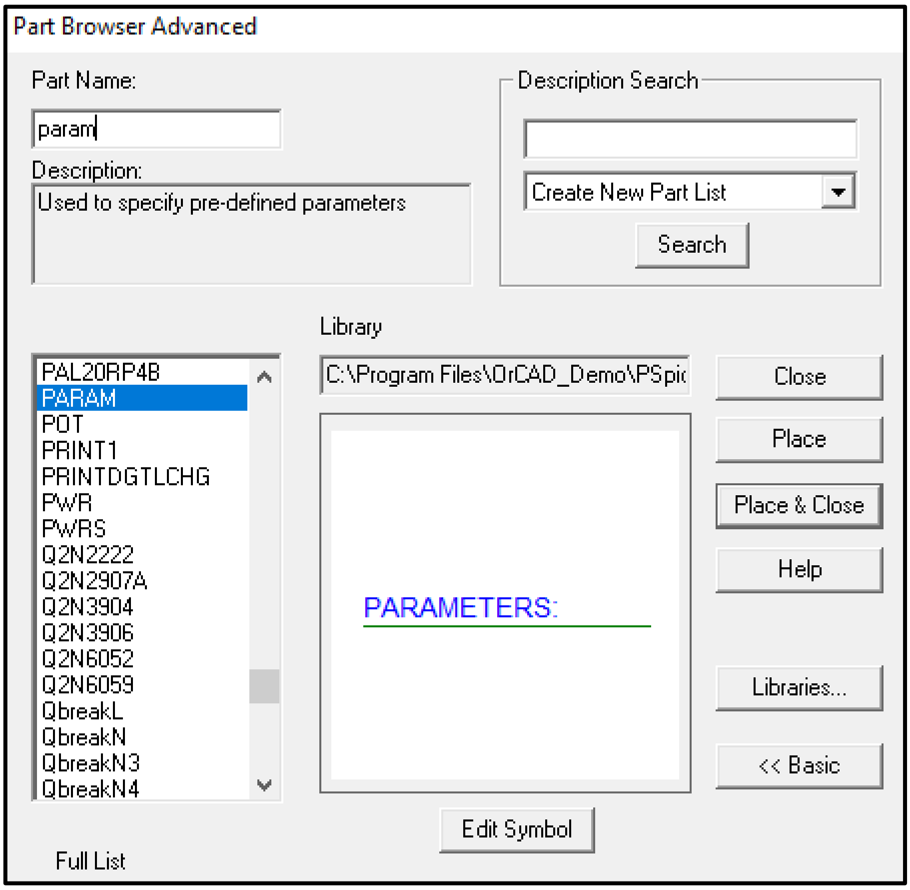


Figure.27

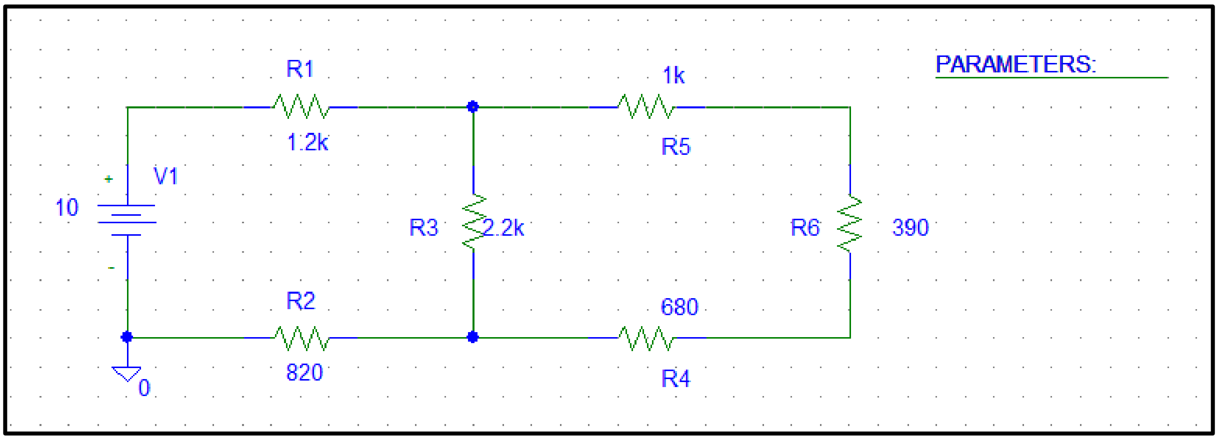


Figure.28

- Double click on "param", and the window shown in figure.29 will show up, insert parameters as shown in figure.

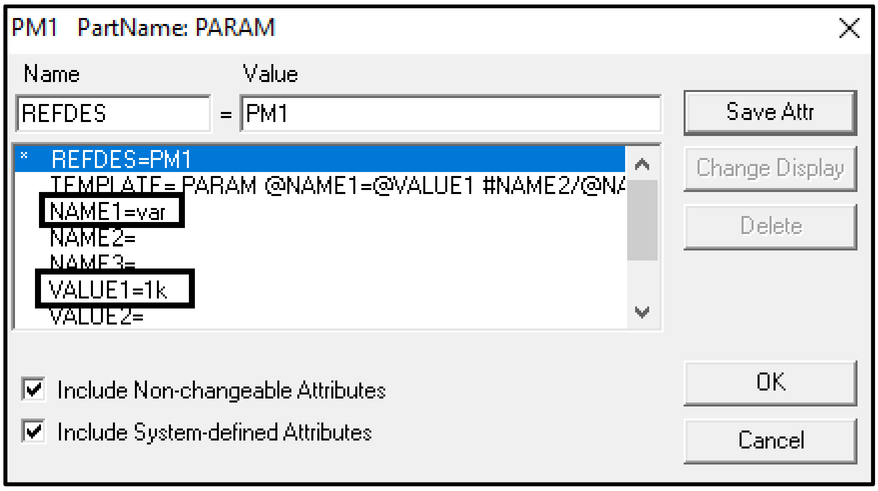


Figure.29

- Click on "setup analysis" and select DC sweep, fill in parameters as shown in figure 30.

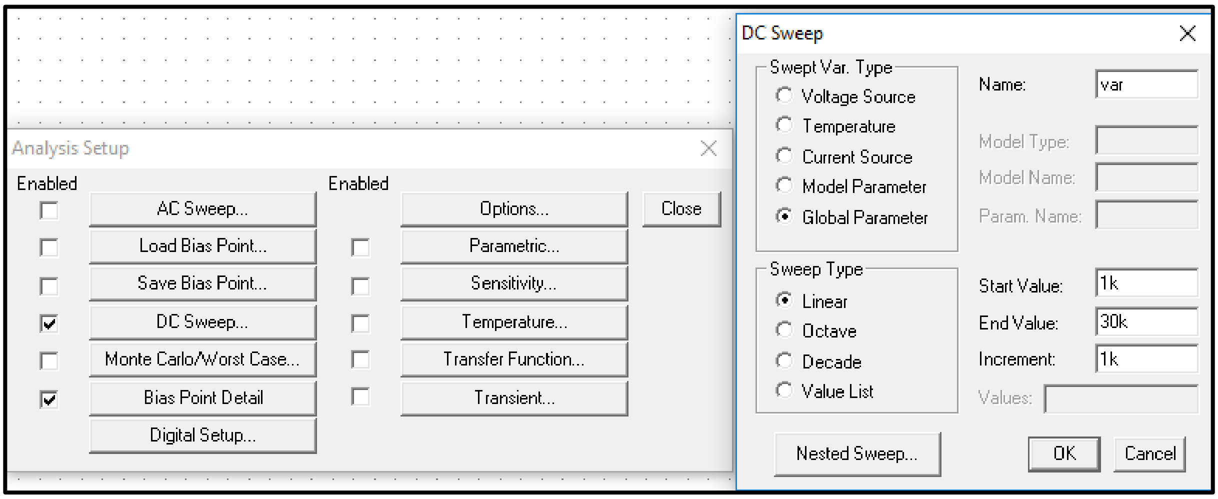


Figure.30

- Type the name of the global parameter inside curly brackets {}, as shown in figure 31.

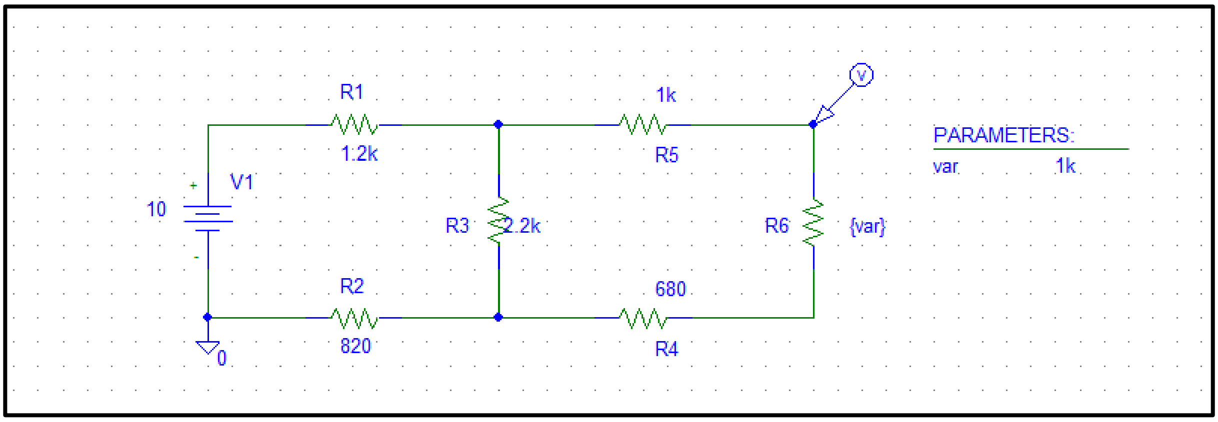


Figure.31

- Click simulate and check the output as shown in figure.32.

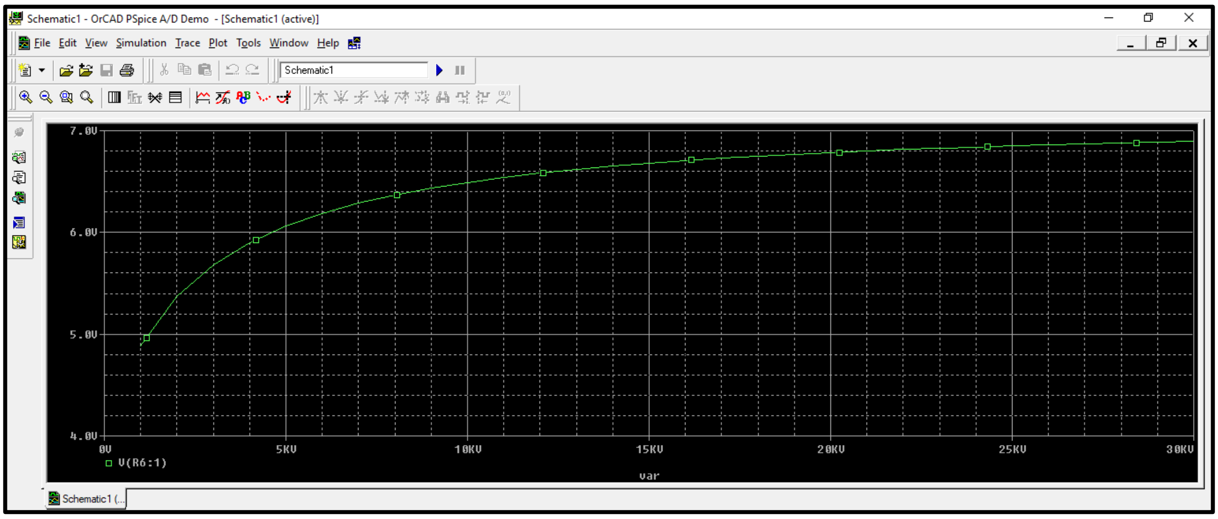


Figure.32

- **To change trace properties such as width and color, right click on the trace and select properties as shown in figure.33.**

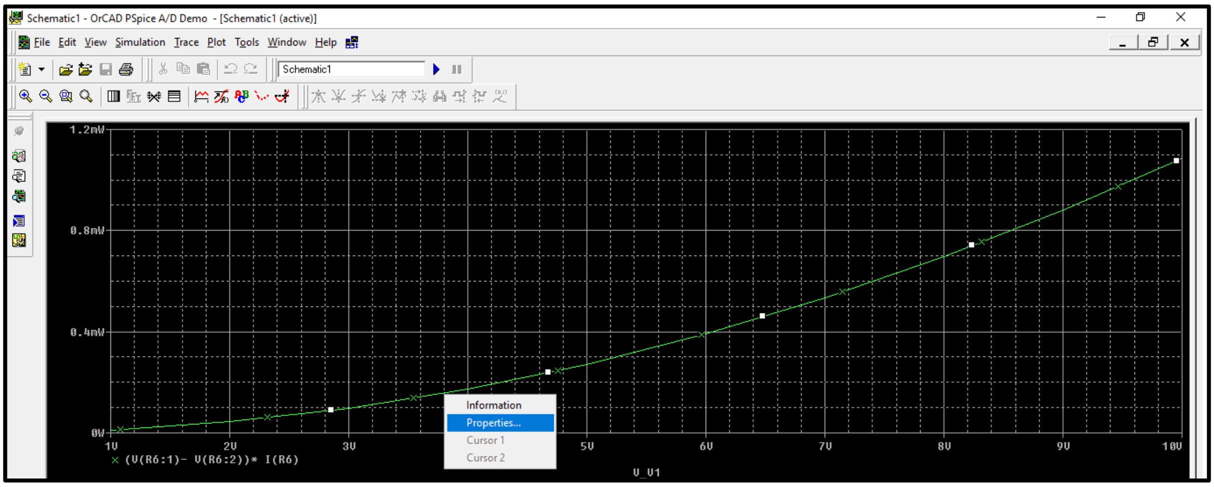


Figure.33

- The window shown in figure.34 will show up, you can change trace properties.

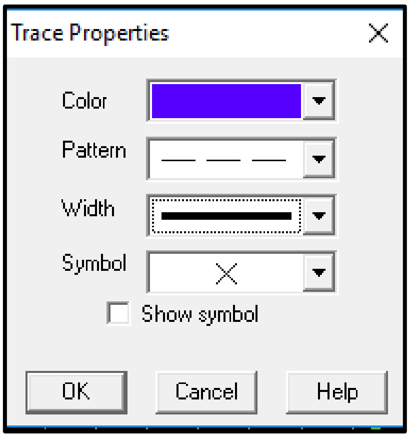


Figure.34

- After changing trace properties, the simulation output window will be shown in figure.35, **always make sure to increase the trace width in your prelab.**

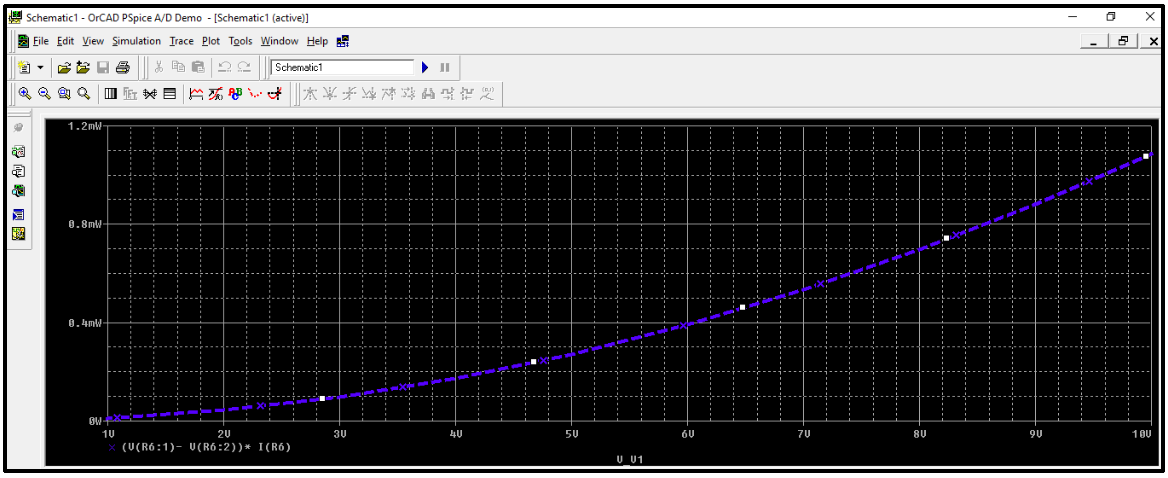


Figure.35

- **To copy the output curve to a word document**, go to the simulation output window and click on window then select "copy to clipboard", then select ok as shown in figure.36.

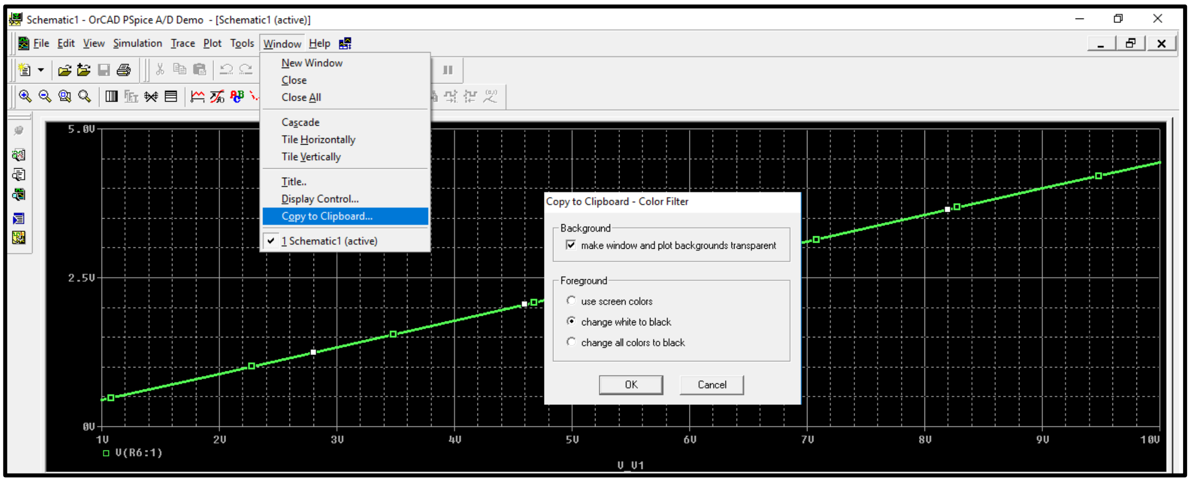
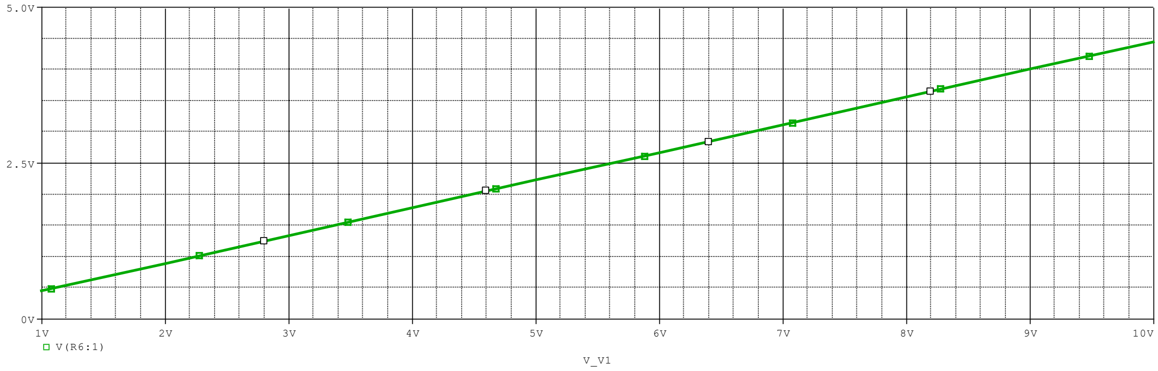


Figure.36

- Then click "paste" or "ctrl+v" inside word document and the output curve is copied a shown.



➢ Example on Transient analysis

- The circuit shown in figure.37 contains a square wave source which is a function of time, so we will use transient analysis.

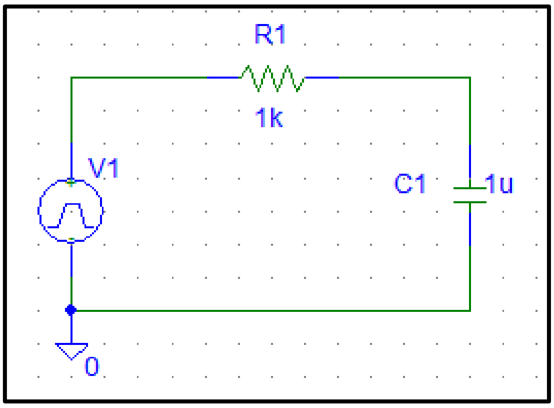


Figure.37

- The source name is "vpulse" and can be found on the components list as shown in figure.38.

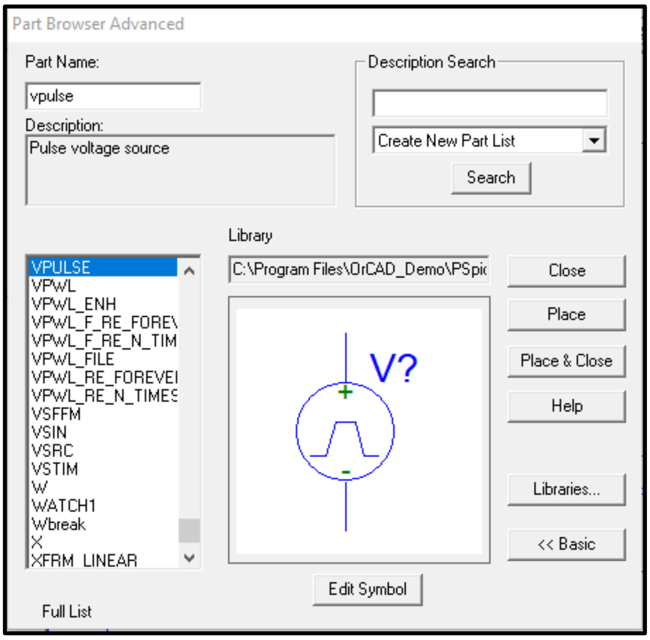


Figure.38

- Double click on the source to insert its parameters, the window in figure.39 will show up.

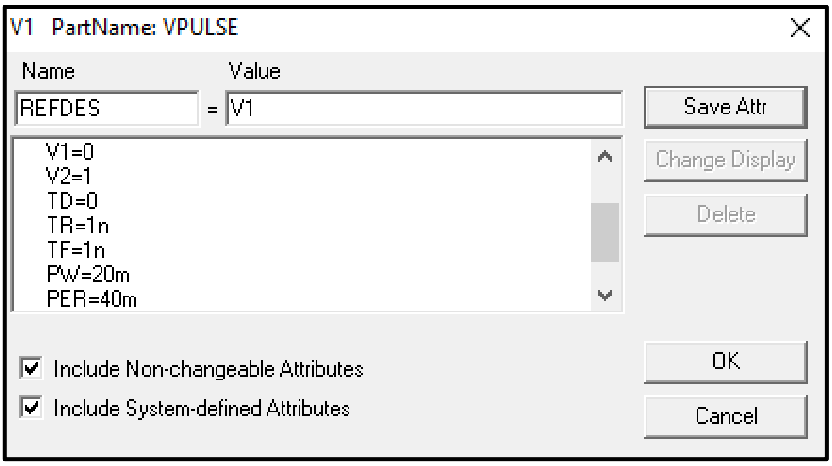


Figure.39

- Table.3 provides the details of this parameters.

Table.3

Parameter	Description	Value
V1	Lowest value	Usually 0
V2	Highest value	Depending on the question e.g. 1
TD	Delay Time	Usually 0
TR	Rise Time	1n
TF	Fall Time	1n
PW	Pulse Width	(1/ 2xFrequency) e.g. 20m
PER	Period	(1/ Frequency) e.g. 40m

- To perform a transient analysis for the circuit of figure.37, click on the “setup analysis” icon and select “Transient”, the window in figure.40 will show up.
- There are **two important parameters** that must be determined carefully in transient analysis so you can picture the output properly, these are: **Final time, and step ceiling.**
- **Final time** is the time till which you want to see the output signal, it is usually selected three times the period of the source.
- **Step ceiling** controls the accuracy **(smoothness)** of the output plots, it is usually obtained by dividing the final time on 10^4.

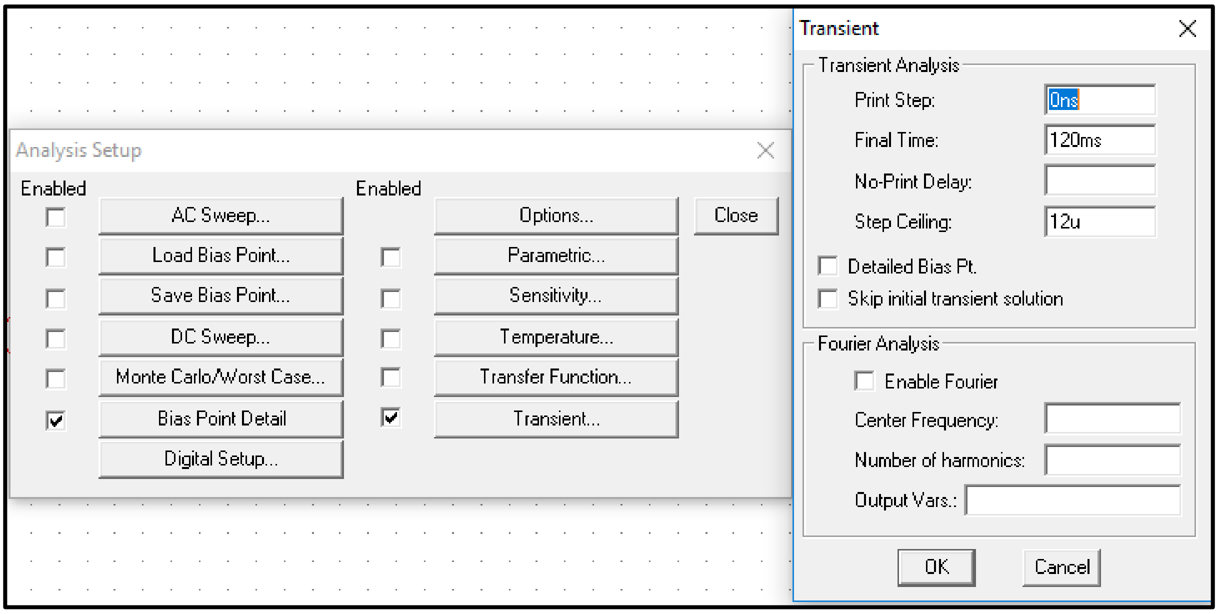


Figure.40

- The output is now displayed in figure.41.

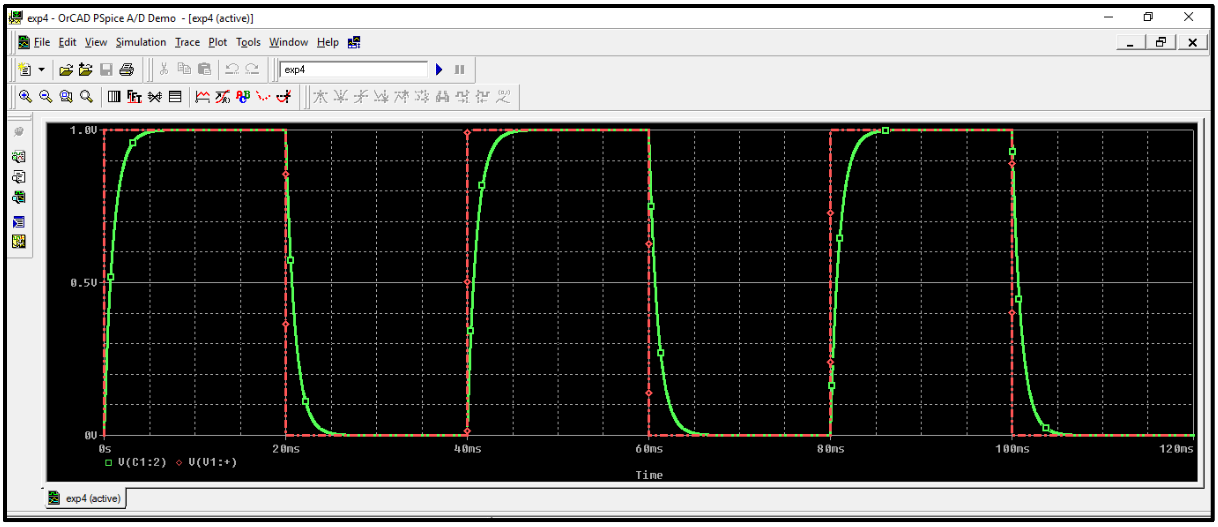


Figure.41

- Cursors are used often used to determine certain points on traces to use cursors click on “toggle cursors” icon shown in figure.42.
- There are two cursors in PSPICE (a1), and (a2), you can move between them by clicking the right and the left buttons of the mouse.
- Each cursor has a reading of x-axis and a reading of y-axis, look at figure.42.

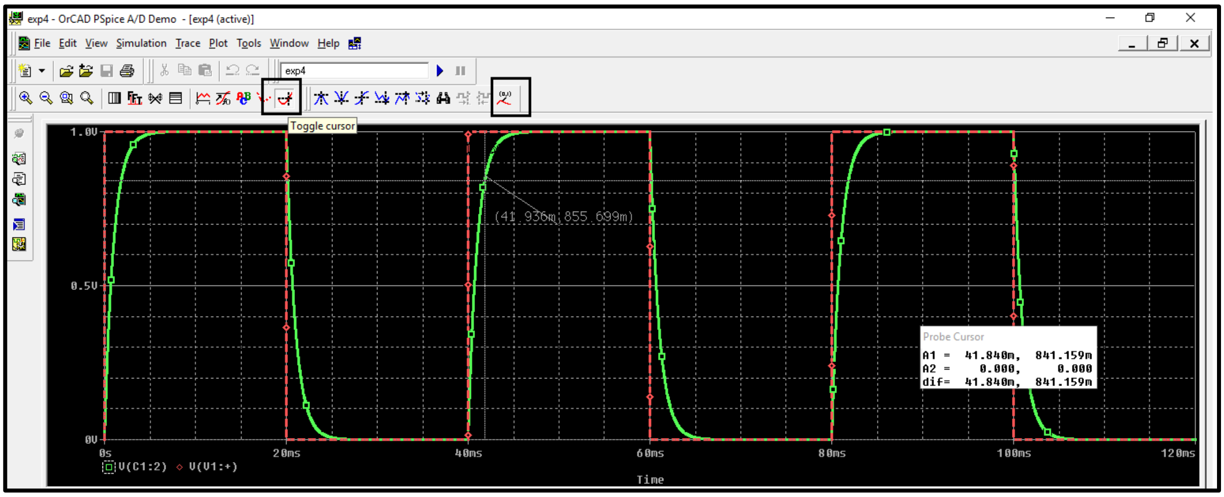


Figure.42

- Another example on transient analysis, sinusoidal voltage source (vsin).

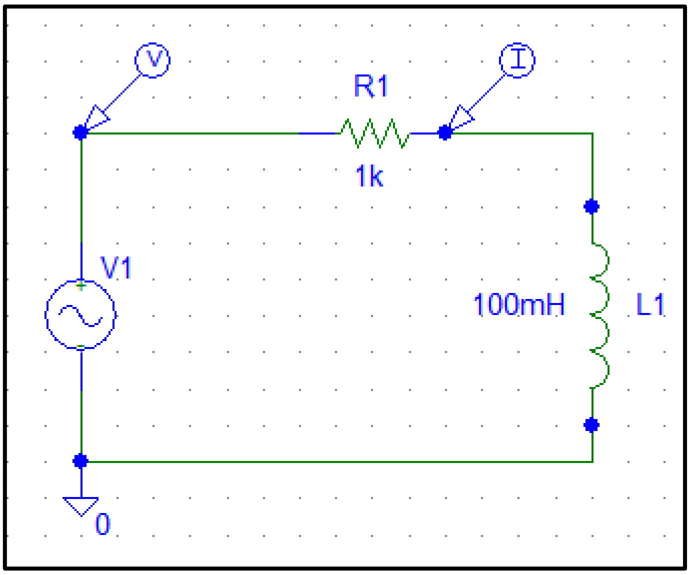


Figure.43

- Source settings.

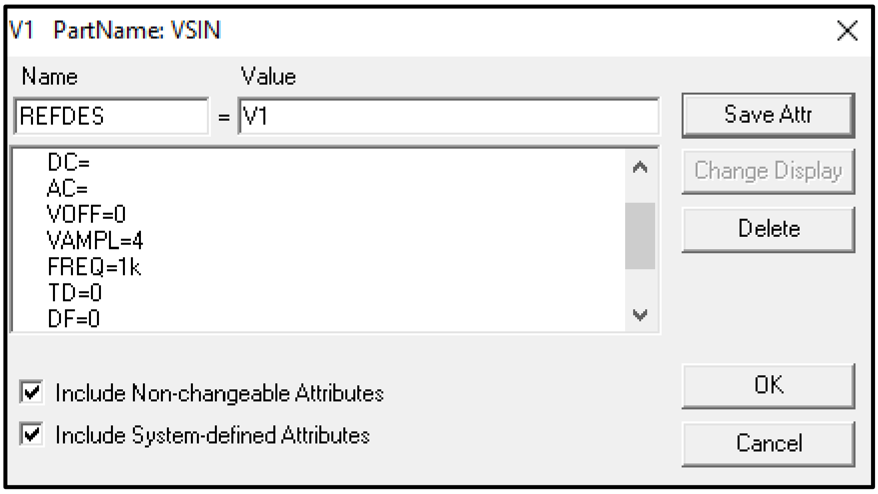


Figure.44

- In the simulation output window, we are viewing the current and voltage, but the voltage is in the range of (-4 to 4) and the current is in the range of (-4m to 4m) so we need **to add a separate y-axis to the output window.**
- Click on "plot", then select "add y-axis", note that you have different y-axes as shown in figure.45, you can move between axes by clicking on the axis you want to add a trace to.

Figure.45

- Voltage and current traces are shown in figure.46, each trace is on a different y-axis range.

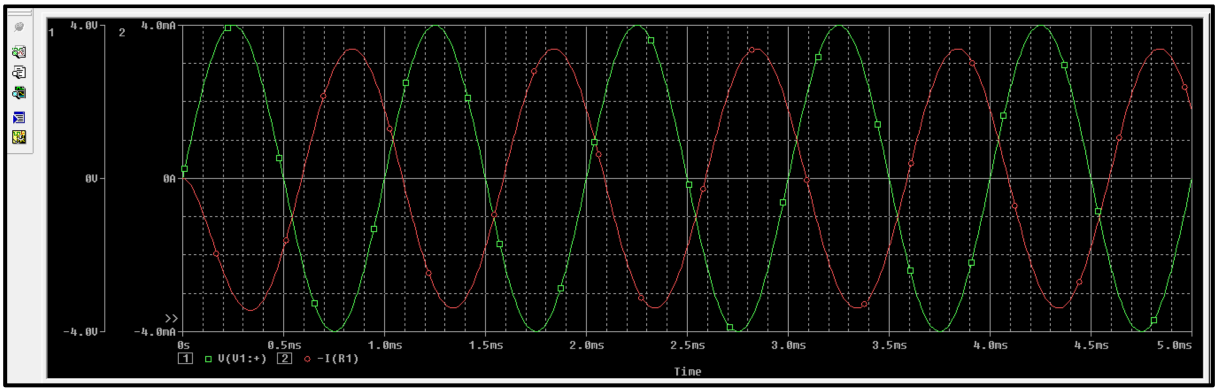


Figure.46

➢ Example on AC sweep analysis

- The circuit shown in figure.47 contains an (vac) source which is used for AC sweep analysis.
- Double click on the ac source to insert setting: only insert voltage magnitude to 1.

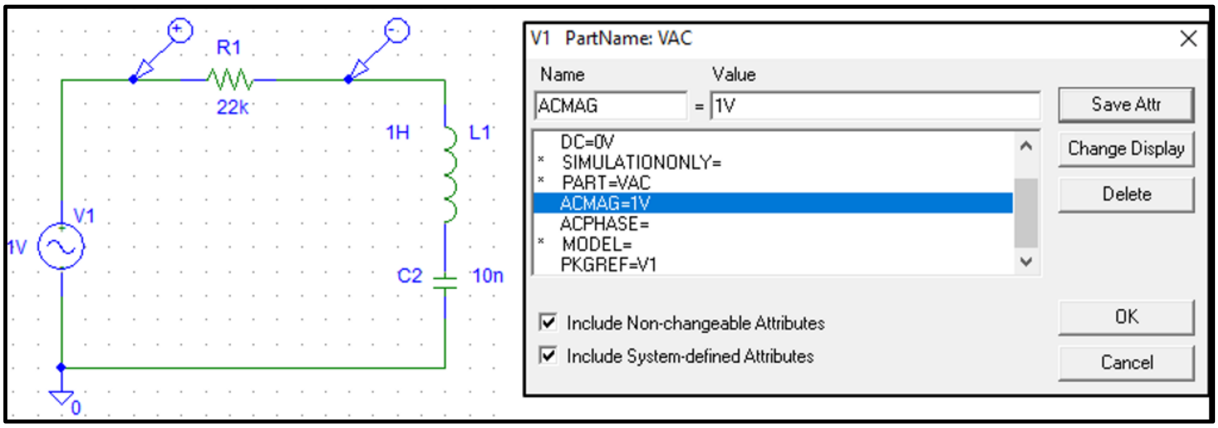


Figure.47

- Setup analysis setting: usually select "AC sweep type" to decade.

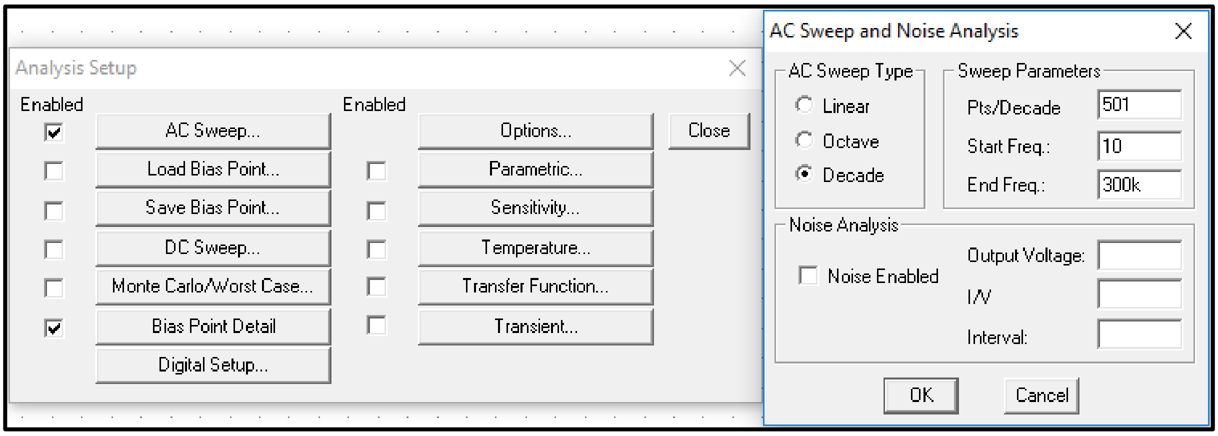


Figure.48

- Output window:

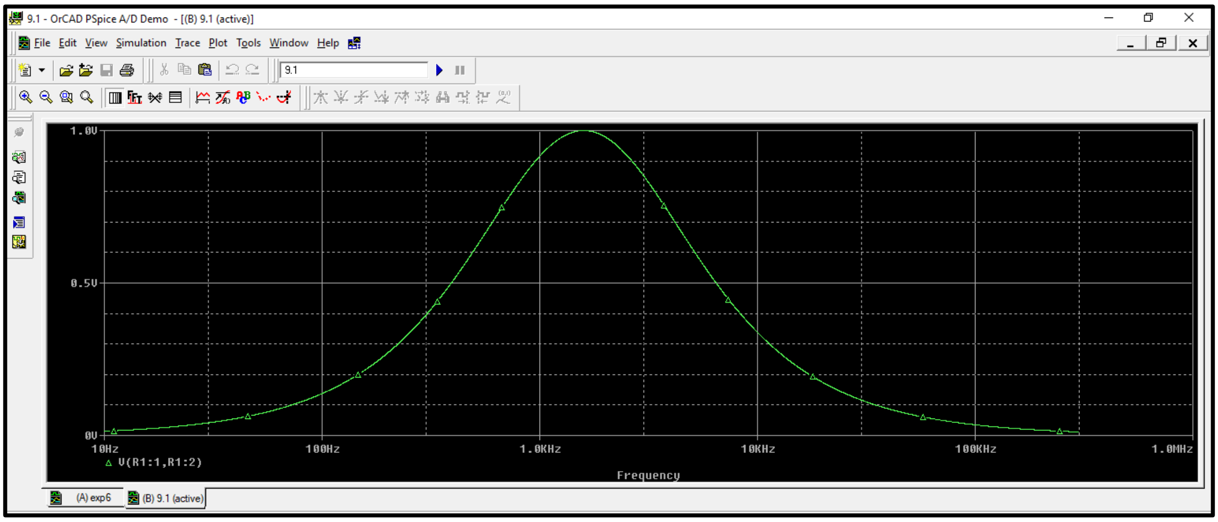


Figure.49

- To get the magnitude in dB:

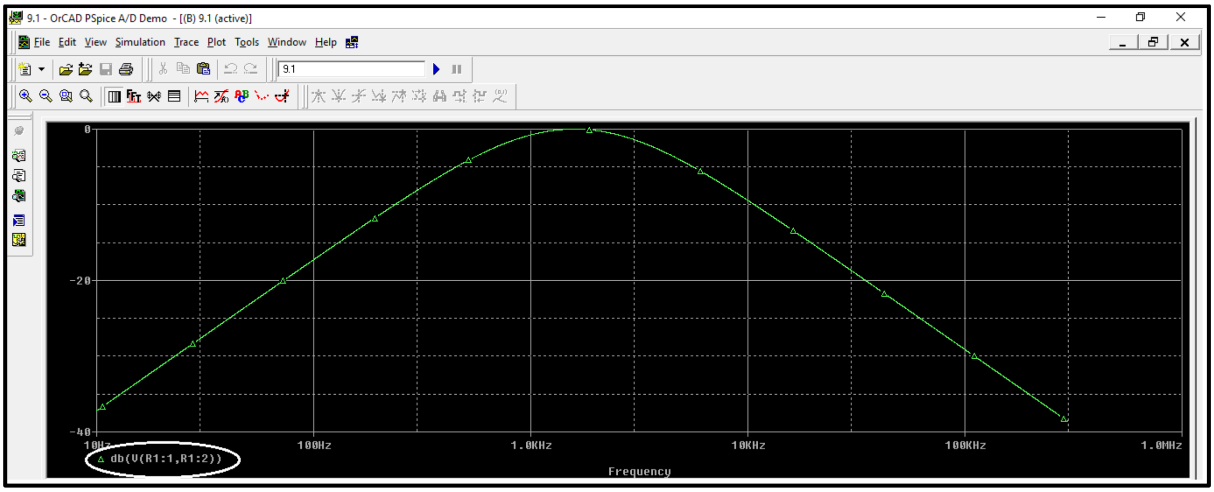


Figure.50

- To get the phase:

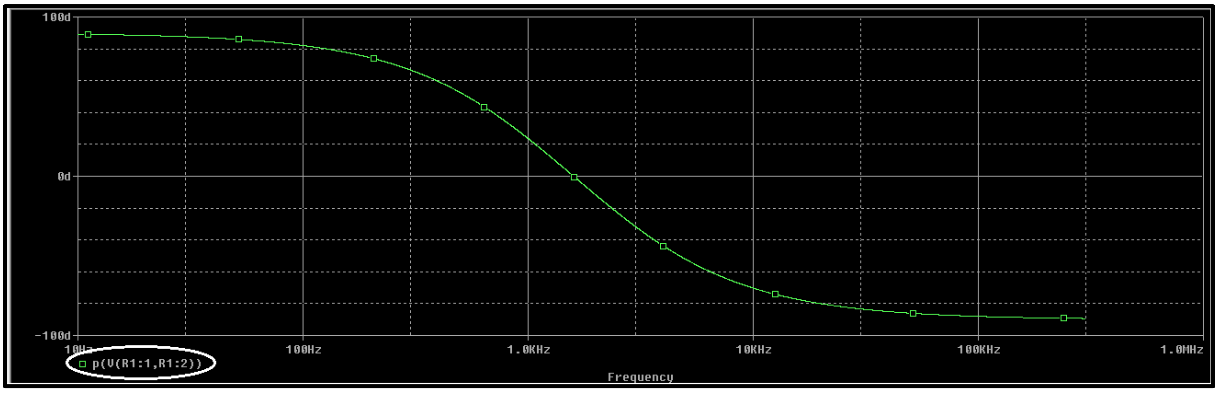


Figure.51

- Another example:

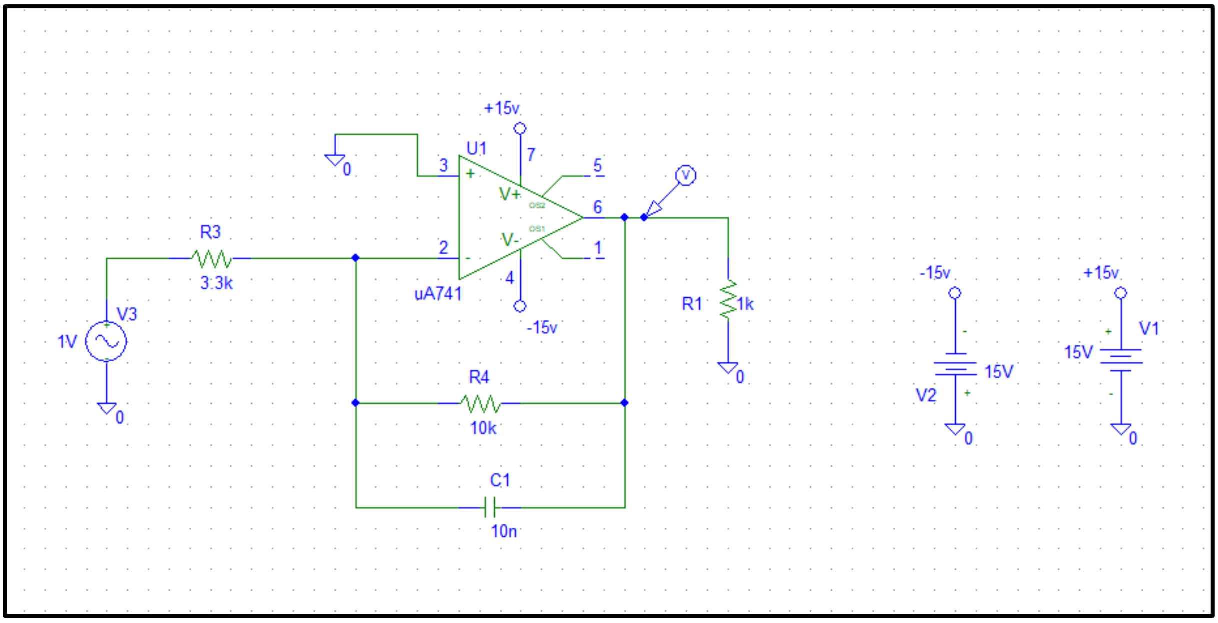


Figure.52

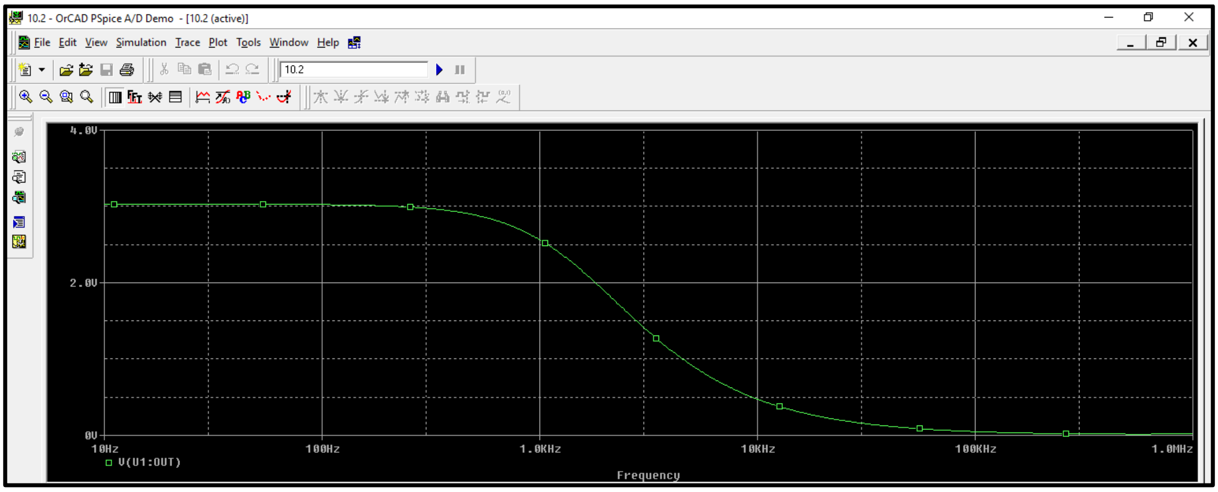
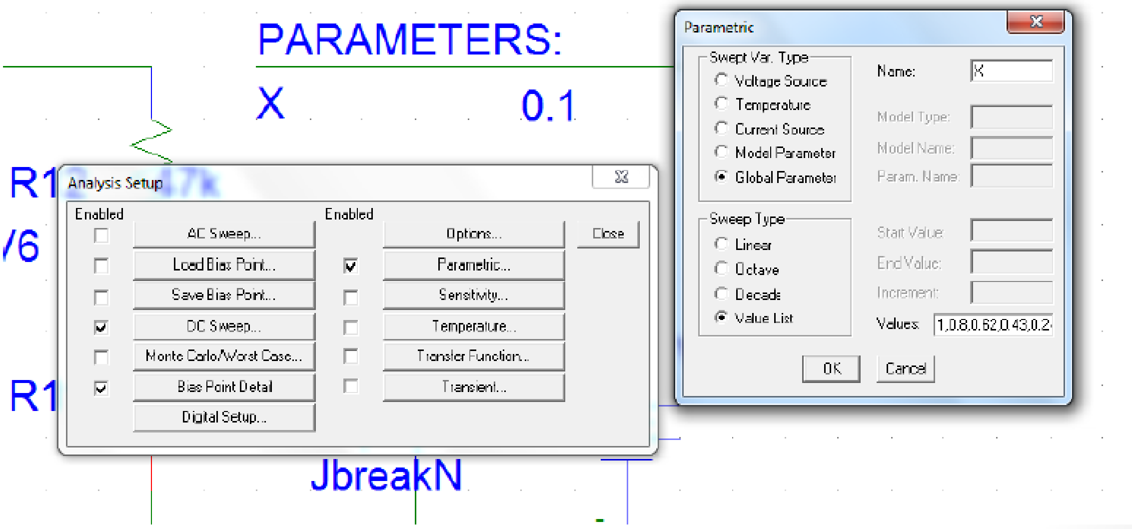
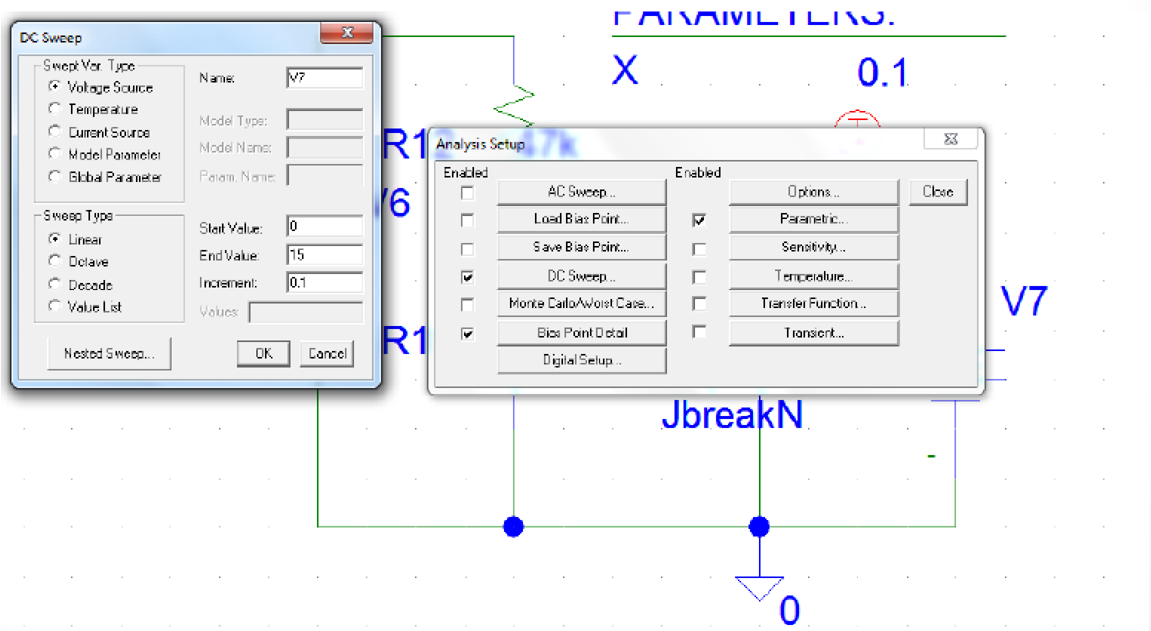


Figure.53

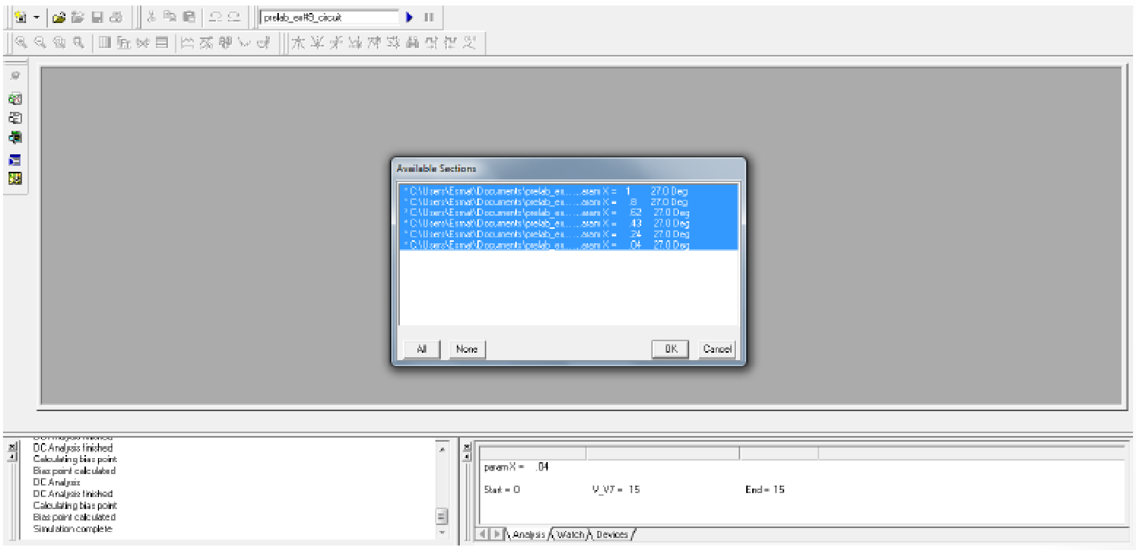
Parametric Sweep With DC sweep to plot the characteristic curves of the JFET

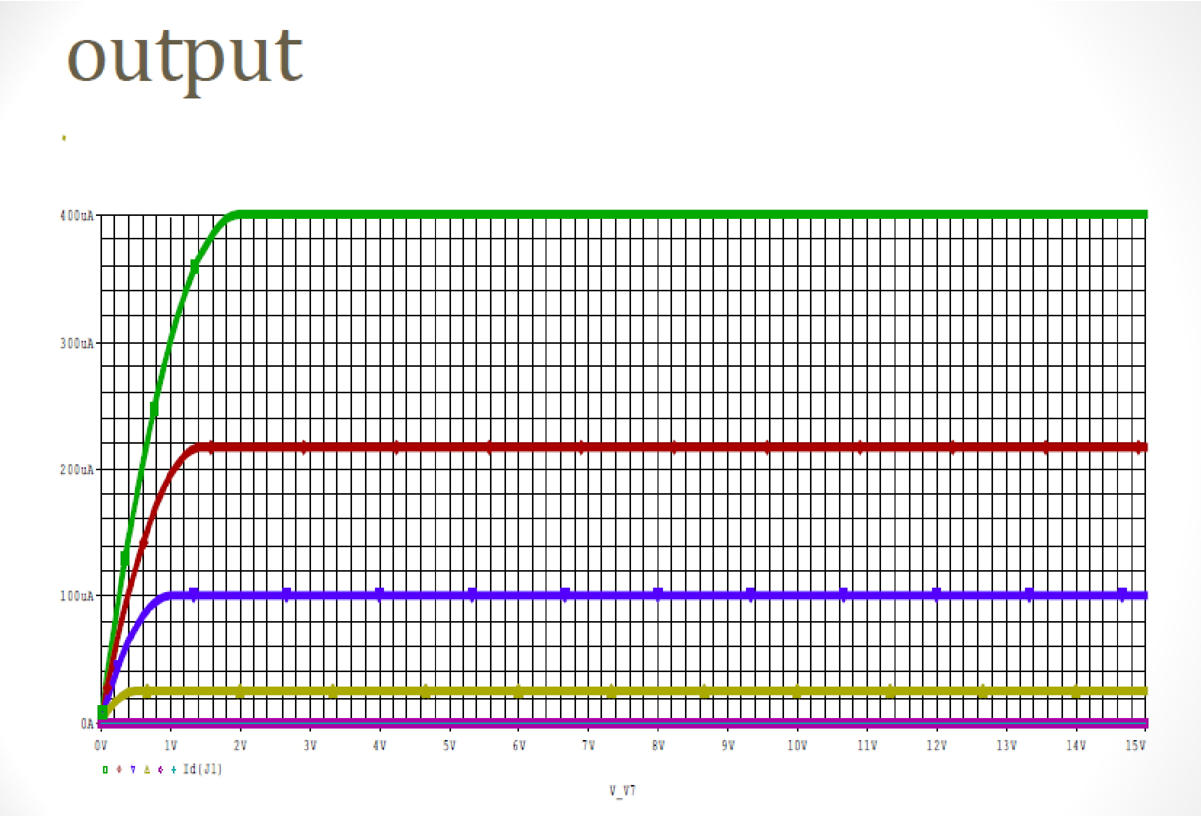


Parametric Sweep With DC sweep to plot the characteristic curves of the JFET



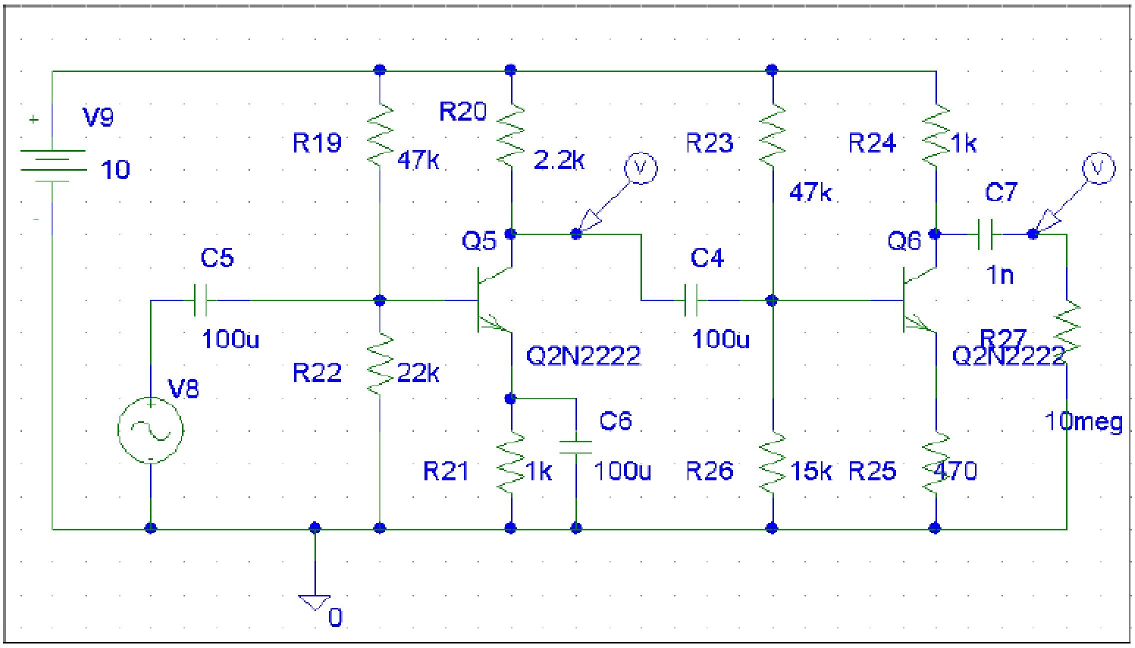
Output



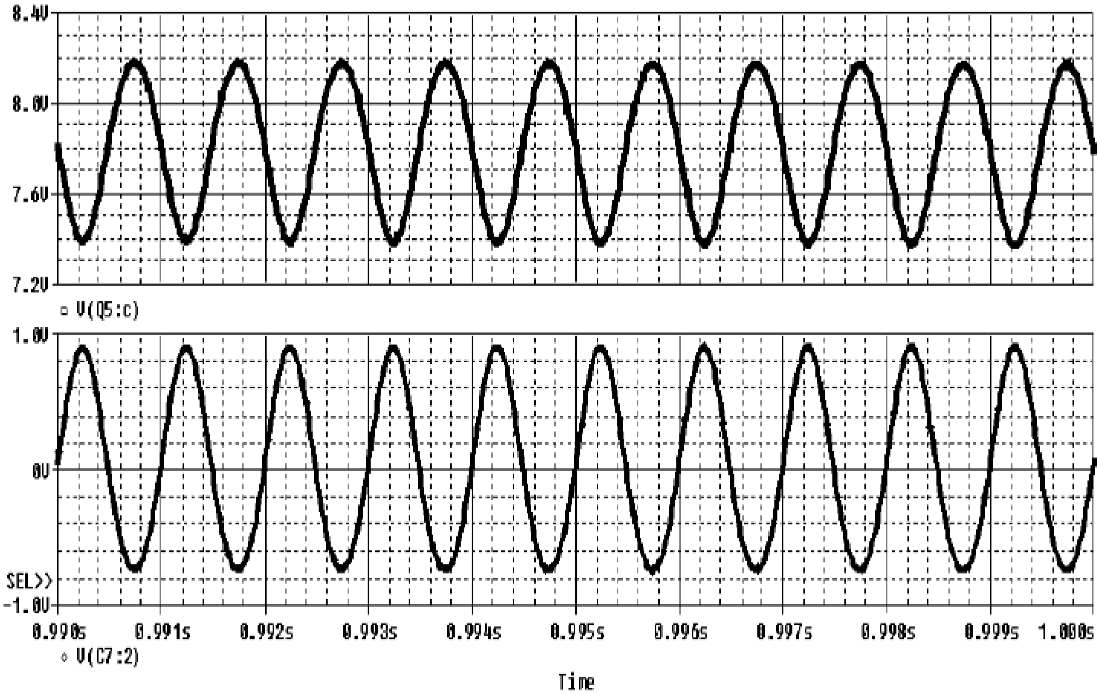

output
400uA
300uA
200uA
100uA
0A
0V
1V
2V
3V
4V
5V
6V
7V
8V
9V
10V
11V
12V
13V
14V
15V
Id(J1)
V_V7

1

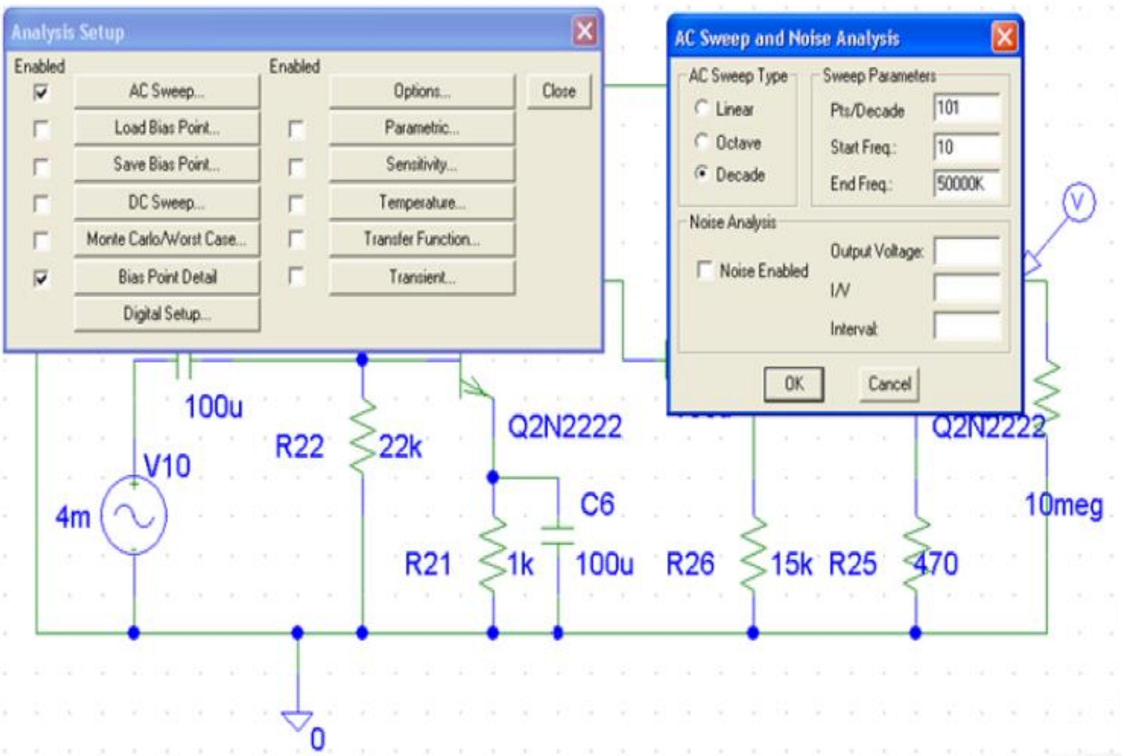
Multi stage amplifier



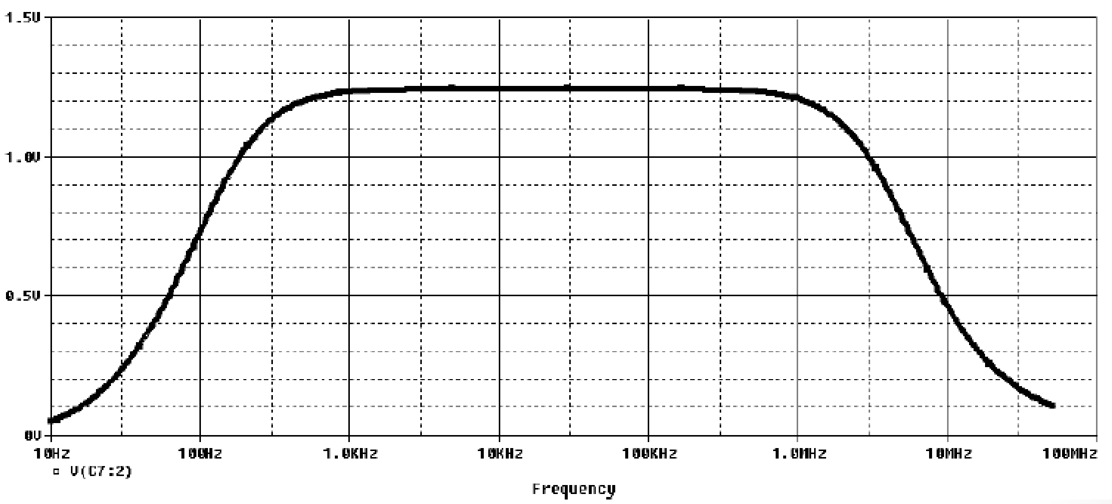
The output voltage when the input voltage is sinosiodal with 12mVp-p



Frequency Response



Frequency response



Report writing guidelines

An experiment report is an important tool to communicate the experiment results and findings to others, it should be organized and written in clear a way. **Reports should be original and contain the basic required elements detailed below, copying from any source will result in a zero grade and proper academic punishment.**

The report must contain the following sections:

1. Cover page

Cover page must contain: course number and name, section number, experiment name and number, author name, date performed, lab partner(s) name(s), and the instructor's name.

2. Table of contents:

A list sections headings and page numbers.

3. Abstract

This section provides a brief summary explaining the aim of the experiment, and the methods used. Use your own words, do not copy objectives from manual.

4. 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.
- If you used information from any resources, state them in references.
- Explain symbols in the mathematical formulas, and use graphical representation of formulas (curves) where applicable.

5. 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.

6. Data, Calculations, and Analysis of results:

- Your report must include a section for data aside from the sheet you used to record data in the lab, which must be attached at the end of your report. The data should be recorded in clear and readable fashion, be provided in tables where possible, and should have units.
- Calculations should be performed to get the required quantities from measured ones, you must show **a detailed sample calculation showing any equations used**.

- You have to discuss all results comparing with what was theoretically expected from prelab exercise and explaining any differences. Discuss results **qualitatively** i.e. no need to state numeric results for each result as the experiments usually contains measurements of many quantities, it is enough to state the general behavior of results. The discussion 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?
- Any questions in the procedure section must be answered in this section when discussing results.

7. Conclusions

Restate the main objectives and how or to what degree they were achieved. What principles were validated by the experiment? Were there any major experimental complications? How the result can be improved in the future if the experiment is repeated (optional)?

8. References:

List any references you used in writing your report, examples on IEEE references formatting are given below:

example of textbook:

[1] J.W. Nilsson and S.A. Riedel, Electric Circuits. Reading, MA: Addison-Wesley, 5th ed., 1996, pp. 111-113.

example of Internet web page:

[2] Approximate material properties in isotropic materials. Milpitas, CA: Specialty Engineering Associates, Inc. web site: www.ultrasonic.com, downloaded Aug. 20, 2001.

General Format Guidelines

1. Use bold font with size (14 point) for titles (Times Roman or Arial), and 12 points elsewhere. Also, use 1.5 line spacing, and justify all paragraphs.
2. Place page numbers on all pages, bottom (though title page is page 1, don't display the number 1 on the title page).
3. Equations are centered and the equation numbers are right justified. The equation number is placed in ( ). See example.

$$F(s) = \int_{-\infty}^{\infty} e^{-st} f(t)dt \tag{1}$$

4. Center all tables and include **a heading and caption with the appropriate table number above each table**. For example, "Table 1: Voltage measurements for Part 3"
5. Figures must be centered, and the **figure number and caption are centered beneath** the figure. For example, "Figure 1: Circuit schematic of Butterworth filter".
6. All graphs must be done with a computer (e.g. Microsoft Excel or even Matlab).
7. All graphs require labels and units on the axes, and require a legend for more than one set of y-axis data.
8. Include a leading zero when a number's magnitude is less than 1 (use 0.83 instead of writing .83). Include a space between any number and an associated unit (i.e., 3.4 mA, not 3.4mA).
9. Use your word processor for Greek symbols for common engineering quantities as β, π, φ, ω, and Ω. Also use Microsoft word to make any necessary superscripts and subscripts. (Use $V = 10R^2$ instead of V = 10R^2).
10. Verb Tense:
    - Use past tense when describing a procedure that was implemented in order to produce your results. For example, "After constructing the circuit of Fig. 1, power was applied."
    - Use present tense when analyzing the results and making conclusions. For example, "The data shows that the efficiency of the process is 92%." Also, when making reference to a figure or data within the report, use present tense. For example, "The test setup is shown in Fig. 1."

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)?
- Are all elements of the report included?
- Is the report submitted on time?

Appendix B

Introduction to Lab Equipment and Measurement

➢ Objectives:

4. In this experiment, we shall study how to use the digital multi-meter, power supply, function wave generator, and the oscilloscope.
5. To make some measurements in the lab.
6. Measuring Phase between two signals.

➢ Equipment Required:

6. Board, resistors, capacitors and wires.
7. Digital Multi-meter GDM-8135.
8. Oscilloscope TDS 2002B.
9. Power Supply Unit TK286 [0-20V] Variable DC.
10. *Function Wave Generator GFG-8215a.*

➢ Introduction:

- Board:

Used to build circuits on it. Each adjacent four holes marked with a white line forms one node (short circuit), circuit elements are connected between nodes, the board is shown in Figure 1.1.

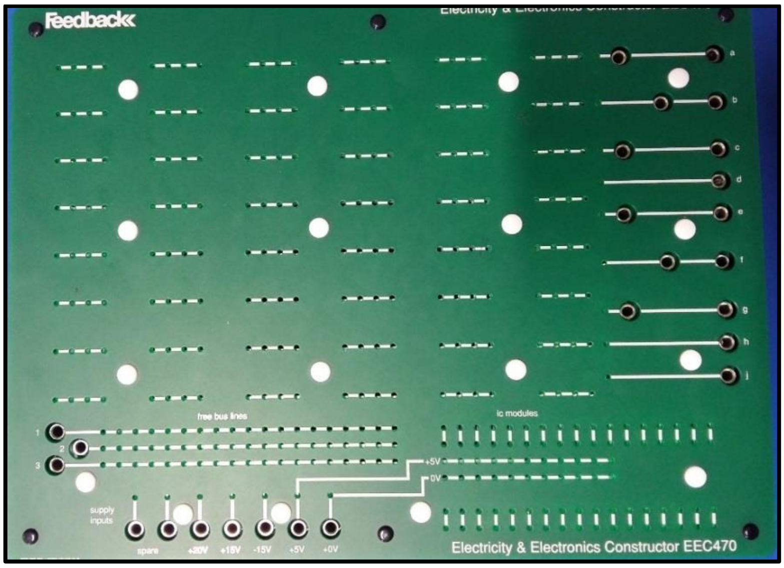


Figure 1.1: board.

- Power Supply Unit:

Power supply unit provide AC and DC voltage output. We will use the Power Supply Unit (Feedbak 92-445) shown in Figure 1.2.

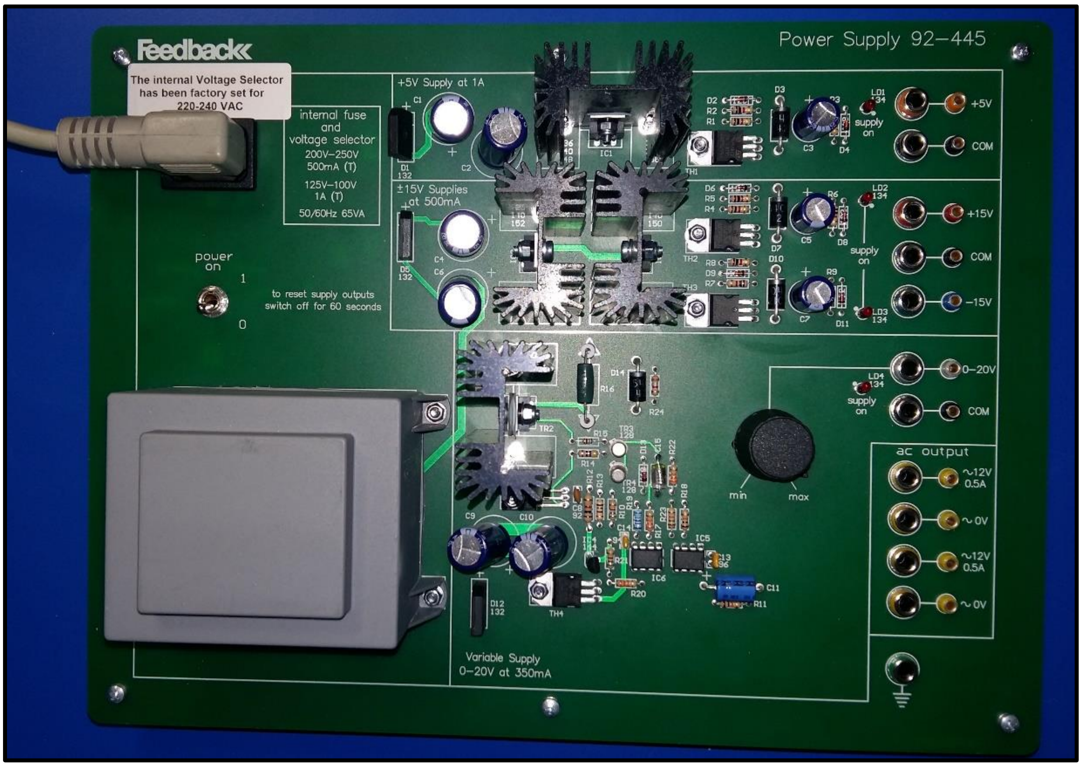


Figure 1.2: Power supply unit.

- Digital Multimeter:

Digital Multimeters are used to measure resistance, voltage and current. This makes the multimeter one of the most important instruments. We use the GDM-8135 in the lab as shown in Figure 1.3. This is typical of such instruments. The power switch is located on the lower right of the front panel, with green color.

Figure 1.3: Digital Multimeter GDM-8135, connected as ammeter.

- The multimeter has three buttons indicated above that determine which type of measurement is being made. For example, if you want to measure resistance, you have to select the resistance mode button. Likewise, current and voltage modes can easily be set.
- Next, you need to attach cables to the multimeter ports. You may use cable with a connector as shown in Figure 1.4.

Figure 1.4: Cable with a connector.

- A black cable inserted in the black port called 'common' or COM. Another red cable is inserted in one of the three red ports depending on the quantity to be measured.
- **To measure voltage or resistance**, the red cable is inserted in the upper red port which is marked (V-Ω).

Upper red Port: Voltage / resistance

Black Port: common

Lower red Ports: Current

- **To measure current**, the red cable is inserted in one of the two lower red ports which are marked 2A max, 20A max. The First port is fused and can measure currents up to 20 A. The second port is used to measure small currents of 2A or less and is fused.
- In circuit lab you will have two multimeters, one is configured to measure currents, and the other is configured to measure voltages and resistances. So, cables are connected to the appropriate ports, and you do not need to reconnect them each time you want to measure a different quantity.

- Measuring resistance:

- Use the multimeter configured to measure voltage and resistance (the one with the red cable attached to the (V-Ω) port). To measure resistance, you need to set the multimeter to the resistance mode by pushing the **'KΩ'** button.

- You must connect the Ohmmeter cables **in parallel** with the element in the circuit for which you want to measure the resistance, as shown in figure 1.5.

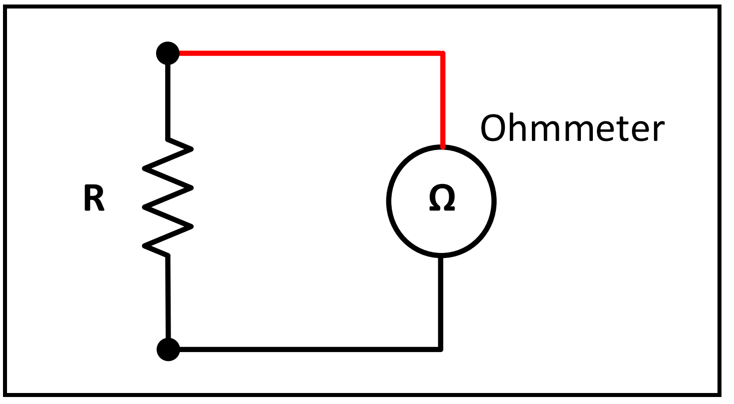


Figure 1.5: Multimeter connection to measure resistance.

- An Ohmmeter usually employs an internal battery across the resistance you are trying to measure. The battery drives a current into the resistor, which is measured by a current sensor. The value of the resistance is the calculated by dividing the battery voltage by the current. **Hence, be careful not to connect any external power supply to the resistor you are trying to measure because an extra current can damage the ohmmeter.** Also isolate the resistor you want to measure from the circuit, since the multimeter calculates the overall Thevenin resistance connected to its terminals**.**
- **Selecting suitable range:** You will see numbers appear on the multimeter display.
    - If these numbers are all zero, then the range setting on the multimeter is too high and you should lower it. The range of measurement is selected by the buttons below the multimeter display.
    - If only the number 1 appears in the left most position, then the range setting is too low and the resistance value is 'out of range'. In this case, increase the range setting.
    - If the display shows actual values, then this is the value of the resistor. Try out different range settings to see which one gives you the most precision.
    - Remember to check your range setting before reading the display e.g. if the range setting is 20 KΩ the display unit is kΩ, if it is 200 Ω the display unit is Ω.

- Measuring voltage:

- Use the multimeter configured to measure voltage and resistance (the one with the red cable attached to the (V-Ω) port). To measure voltage, set the multimeter to the voltage mode by pushing the **'V'** button.
- A voltmeter is connected **in parallel** with the circuit element where the voltage measurement is desired as shown in figure 1.6. Since the voltage across parallel elements

is the same, the voltage measured by the meter will be the same as the voltage across the element to which the meter is connected.

- The internal impedance of the voltmeter is very large, to avoid drawing extra current, thus disturbing the voltage it is trying to measure.

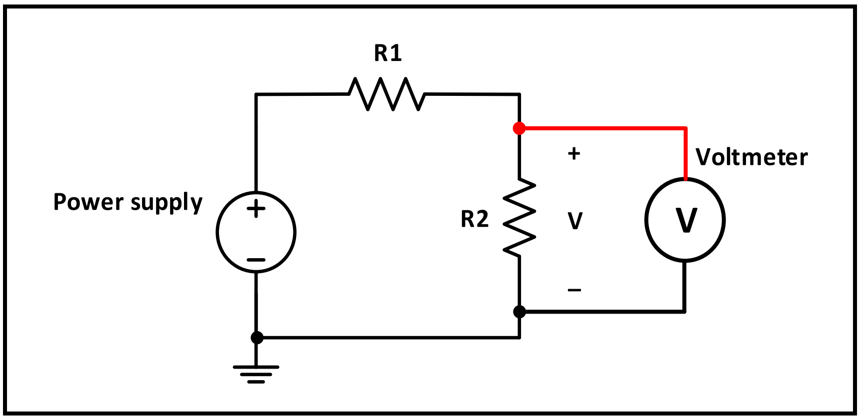


Figure 1.6: Multimeter connection to measure voltage.

- There are two different types of voltage measurements: AC and DC. The button to the left of the voltage mode button toggles the FigFmultimeter from AC to DC mode.
- When measuring voltage in DC mode, the multimeter measures the time average of the signal. In AC mode, the multimeter measures the R.M.S (root mean square) of the signal.
- **A common mistake** is to measure AC voltage in DC mode, the reading will be zero since the average value of AC voltage is zero.
- The red cable is connected to the higher voltage, while the black cable is connected to the lower voltage to produce positive measurement in DC mode.

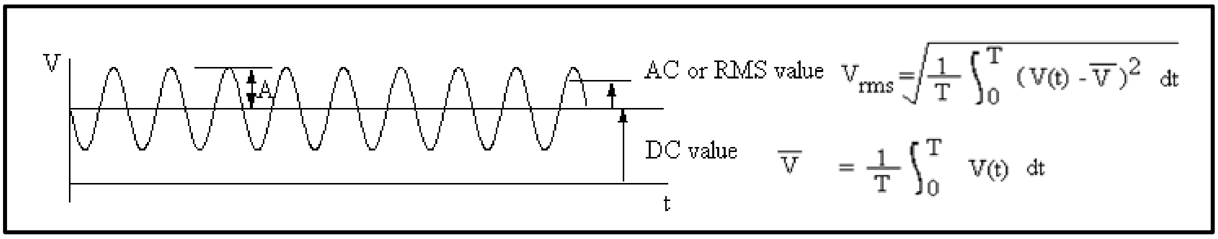


Figure 1.7: Sinusoidal signal

- **Selecting suitable range** for voltage measurement is done in the same way for resistance measurement. Note that the unit of measurement is (mV) or (V) depending on the selected range.
- **To protect the instrument**, you should always select the highest range first when making measurements of unknown voltages, then lower it till you find the suitable range.

- Measuring current:

- Use the multimeter configured to measure current (the one with the red cable attached to the (2A) port). To measure current, set the multimeter to the current mode by pushing the **'mA'** button.
- An ammeter is connected **in series** with the element in the circuit through which you want to measure the current. This means, for example, if you want to measure the current through a cable between two devices, you will have to disconnect that cable, and reconnect it through the meter, as shown in figure 1.8.
- Since the current in each element of a series circuit is the same, the current flow through the meter will be the same as the current flow to the element of interest.
- The ammeter has a very small internal resistance so it does not disturb the current it is trying to measure. Due to this fact, however**, if it is connected by accident in parallel with the circuit element, a large amount of current will flow through it, thus damaging it. That is why ammeters are usually protected by a current limiting fuse.**

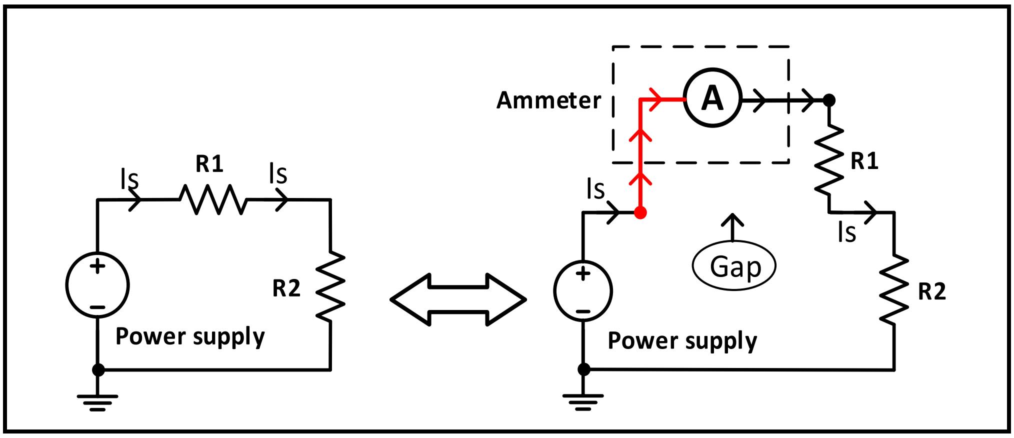


Figure 1.8: Multimeter connection to measure current.

- Current measurement has AC and DC modes similar to voltage measurement.
- The red cable is connected such that the current flows into it and leaves out the black cable to produce positive measurement in DC mode.
- **Selecting suitable range** for current measurement is done in the same way for resistance measurement. Note that the unit of measurement is (mA) or (µA) depending on the selected range.
- **To protect the instrument**, you should always select the highest range first when making measurements of unknown currents, then lower it till you find the suitable range.

- **Function Wave Generator:**

The function generator can be used to produce three different kinds of signals as a function of time, square wave, triangular wave and sine wave.

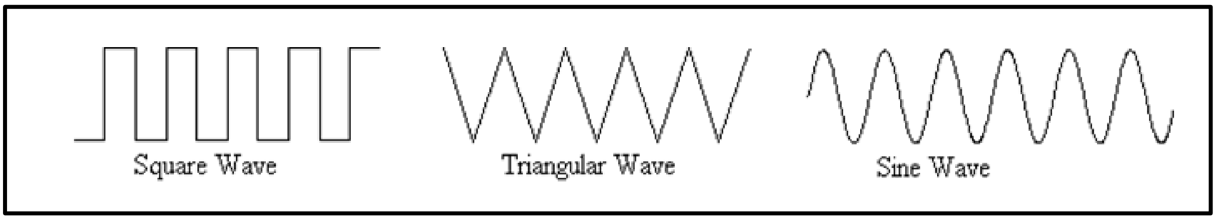


Figure 1.9: Waveforms

The function buttons determine the type of signal the instrument produces. With the sine-wave button pushed in, a sine wave will be produced, for example. The function generator also has range settings to determine the frequency range of the signal. The amplitude of the output signal is controlled using "AMPL" knob. The Function Wave Generator used in lab is the GFG-8215a as shown in Figure 1.10

Figure 1.10: Function Wave Generator GFG-8215a

- **Oscilloscope:**

The oscilloscope is a device designed to display a voltage signal in visual form. It can used for some quantitative measurements, such as voltage amplitude and frequency, it also can be used to compare two separate signals and estimate their relative characteristics. The scope used in lab is the Tektronix TDS 2002B as shown in Figure 1.11.

- **Basic Set Up:**

The oscilloscope is turned on using the power button indicated in figure 1.11. After a few seconds a horizontal line should appear on the screen. The line, called a 'trace' is a plot of voltage (on the vertical scale) against time (on the horizontal scale).

- **Note:** Autoset button used to get the signal direct without any setting on oscilloscope.

Figure 1.11: Oscilloscope Tektronix TDS 2002B

The TDS 2002B is a digital oscilloscope that functions very much like a computer. Most of the settings can be changed through various menus accessible from the top control panel:

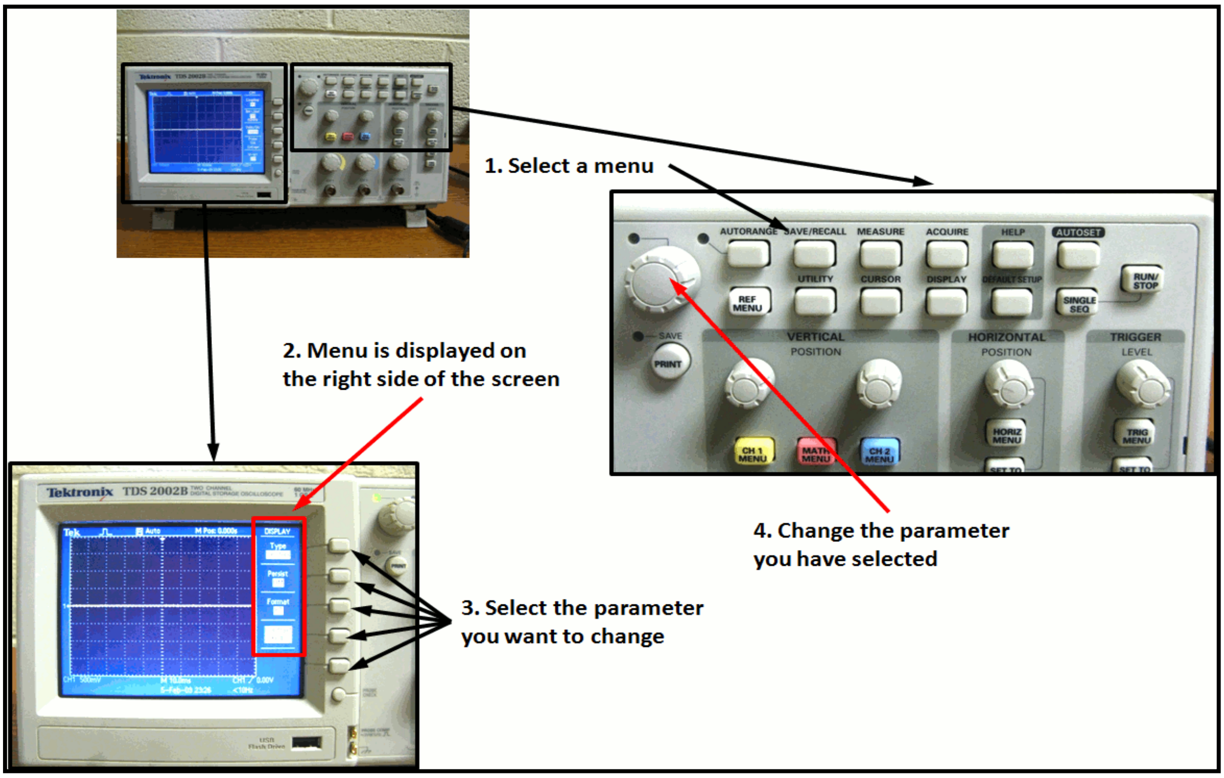


Figure 1.12: Oscilloscope menu

Note that the oscilloscope screen has graduations (or 'divisions'). This is to enable you to make quantitative measurements of signals displayed. You can also try the ***MEASURE*** and ***CURSOR*** menus. The ***MEASURE*** menu allows you to measure various quantities (frequency, mean, peak-to-peak, etc...) for one or two channels. The ***CURSOR*** menu uses two lines to measure any feature on the vertical or horizontal axes. Try these out for yourself. In the section of the front panel labeled "VERTICAL" set the **CH 1 MENU** and **CH 2 MENU** In the HORIZONTAL section set the SEC/DIV knob to 5ms/div (as you turn the knob, you should see the value at the bottom of the screen (just above the date) change). You should see a trace on the screen. Adjust the intensity and focus if needed. Center the line using the position knobs in the HORIZONTAL and VERTICAL sections.

- **Connecting Signals:**

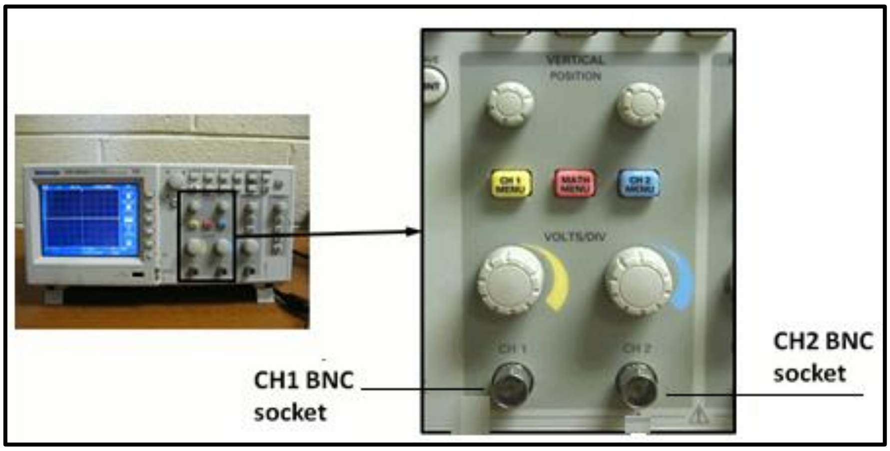


Figure 1.13: connecting signals to oscilloscope

Signals are connected to the scope through one of two BNC sockets, located as shown above. These accept cables with connectors of the type shown below.

BNC connector

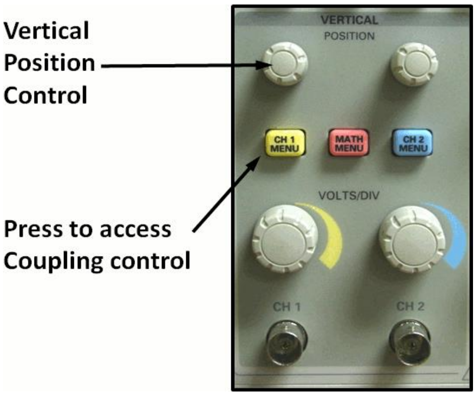
There are two sockets, because the scope can simultaneously display up to two signals. The signals and connectors are identified as CH 1 and CH 2, for channels 1 and 2. Note that the signal is actually provided to the scope through the central pin of the BNC sockets. The outer shield is a ground connection (not just a common) **so you must be a little careful what you connect to it. (E.g. if you set the power supply to provide 5V relative to ground and then connect the 5V to the BNC shield the resulting mismatch might cause problems the scope.)**

When you have made your connections, you need to tell the oscilloscope what to display using the **CH 1 MENU** and **CH 2 MENU** buttons. Pressing these buttons will toggle the display of their respective channel. Channel 1 signal appears as a yellow trace, while Channel 2 signal is blue. It is therefore possible to display the signals from both channels at the same time.

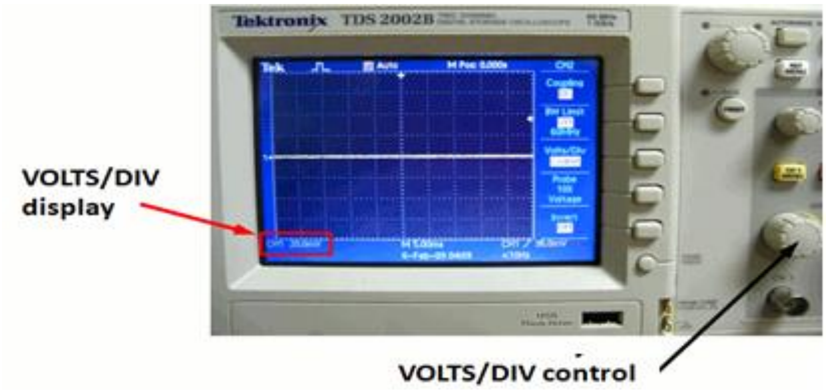
The **MATH MENU** button can be used to display an arithmetic combination of two signals. You can select from addition, subtraction, multiplication, addition or Fourier Transform (FFT). The resulting signal is plotted in red.

- **Setting the Vertical Scale:**

Scopes give you the same control over displaying a signal as you would have over plotting it on a piece of graph paper. You can independently control the origin and the scale of the voltage displayed on the vertical axis. There are duplicate controls for the two channels.



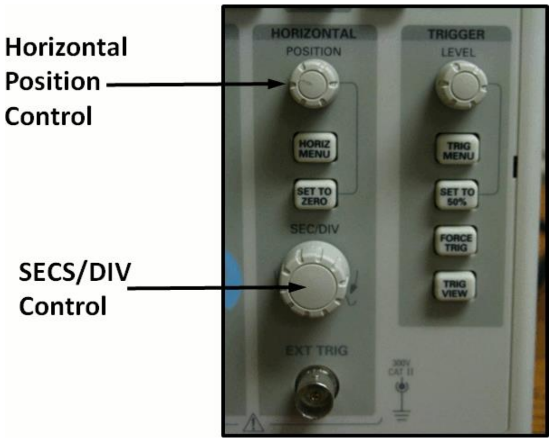
The vertical 'POSITION' controls are used to set the origin. Turning the POSITION control for CH1 for examples simply moves the signal up and down on the screen (i.e. the origin is moving up and down). If your objective is to make an absolute voltage measurement (rather than just look at the form of the signal) you will want to know quantitatively where the origin you've set is. You can do this by selecting GND coupling to plot zero volts on the screen.



For controlling the scale of the voltage axis there is a VOLTS/DIV control. This sets the voltage represented by each of the large vertical divisions on the screen. The actual value set is indicated by the number in their lower left corner of the screen and may be varied from 5 mV per division to 5 V per division.

If you need finer resolution in the increments of VOLTS/DIV you can press the **CH 1 MENU** and change *Volts/Div* from Coarse to Fine. Note that it is perfectly possible to set the POSITION and/or VOLTS/DIV control so that the signal you want to look at, or even the voltage origin, is off the screen so that you see no trace. If you think this has happened the best way to get your signal back is to increase the VOLTS/DIV setting, and then to rotate POSITION knob until the trace appears.

- Setting the Horizontal Scale:



The controls for the horizontal (time) axis are located as shown in the picture. The same horizontal scale is used for both channels, so there is only one set of controls. For controlling the time origin (i.e. for moving the signal horizontally across the screen) there is a POSITION control. For controlling the scale of the time axis there is a knob labeled 'SEC/DIV'. This sets the time represented by each of the large horizontal divisions on the screen. The actual value set is indicated at the bottom of the screen (just above the date) and may be varied from 5 nanoseconds per division to 50 seconds per division.

➢ Procedure

Part A: Measuring resistance

2. Use digital multi meter (DMM) to measure resistances you have been given, record results.

Part B: Measuring voltage

2. Connect the circuit shown in figure 1.14, set the input power supply to 10V, measure the voltage across each resistor, record and discuss results.

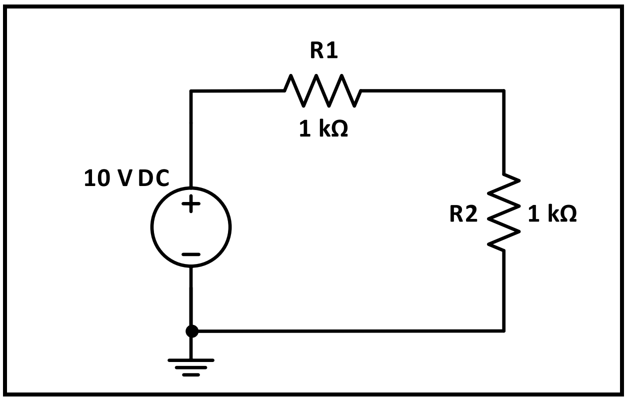


Figure 1.14

Part C: Measuring current

2. For the same circuit of figure.1.14 measure the current through each resistor, record and discuss results.

Part D: Using oscilloscope and function wave generator

4. Connect the circuit shown in Figure 1.15, set the input voltage to sinusoidal voltage at 8 V_{PP} and 1 kHz (you will need to connect channel 1 of the oscilloscope to the input voltage).

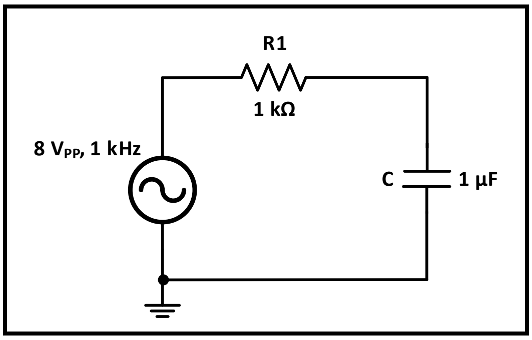


Figure 1.15

5. Connect channel 2 of the oscilloscope to the capacitor, use measure menu to view characteristics of the voltage across capacitor.
6. Use DMM to measure the current in the circuit, and the voltage across the resistor and the capacitor.