VIETNAM NATIONAL UNIVERSITY HO CHI MINH CITY

UNIVERSITY OF SCIENCE

----------*----------

FACULTY OF ELECTRONICS AND TELECOMMUNICATIONS

High Quality Education

PhD. Trong-Tu Bui, Msc. Trung-Khanh Le

Basic Electronics Lab

2018

Contents

LAB 1

ANALOGUE MEASURING INSTRUMENT

I. GOAL

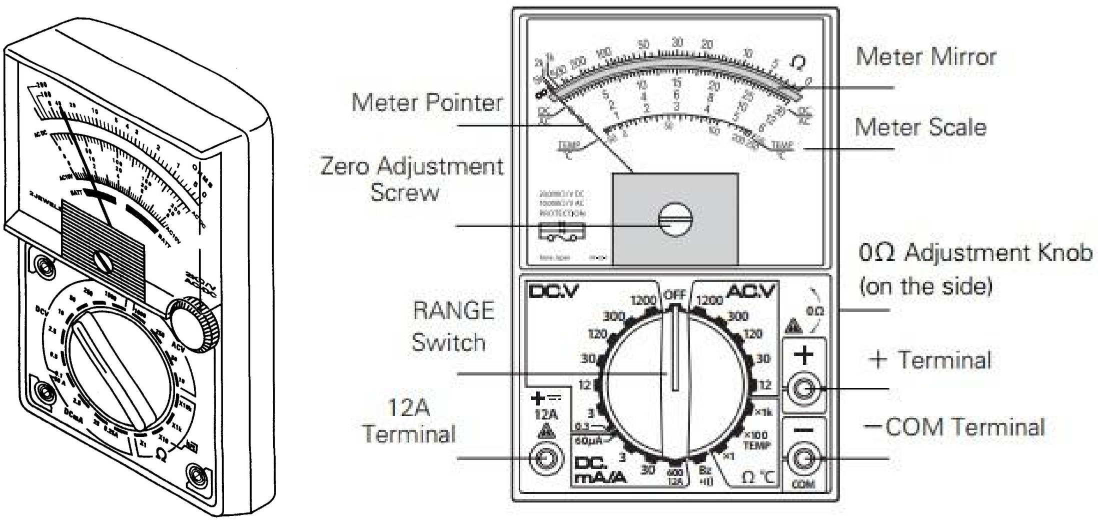
In this lab, student will have skills:

- Using Analog VOM.
- Reading and measuring resistor values, testing electronic components such as capacitor, inductor, transformer, diode and BJT.

II. SUMMARY OF THEORY

a. Analog VOM

Picture 1.1 describes fundamental components of an Analog VOM using galvanometer.



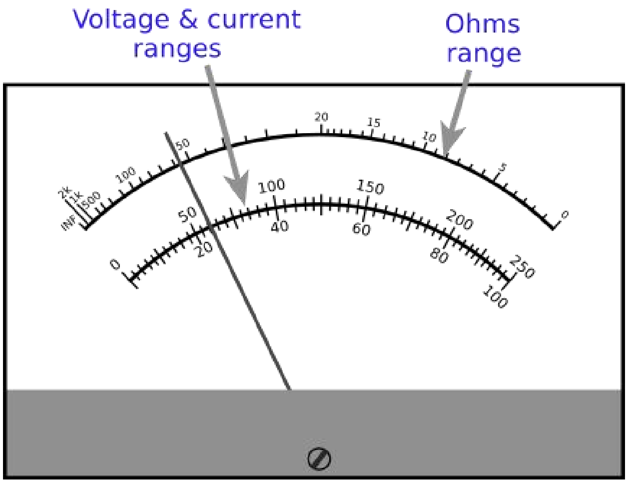
Picture 1.1. Analog VOM using galvanometer.

On this equipment:

- ***-COM*** terminal is connected to Black probe.
- +***Terminal*** is connected to Red probe.
- ***0ΩADJ*** knob is used to calibrate 0 Ohm value. It is required in measuring resistant value.
- ***RANGE Switch*** is used to select which electronic unit will be measured and its scale. Normally, an Analog VOM has DC.V to measure DC voltage, AC.V to

measure AC voltage, DCmA/A to measure DC current and Ω to measure resistance.

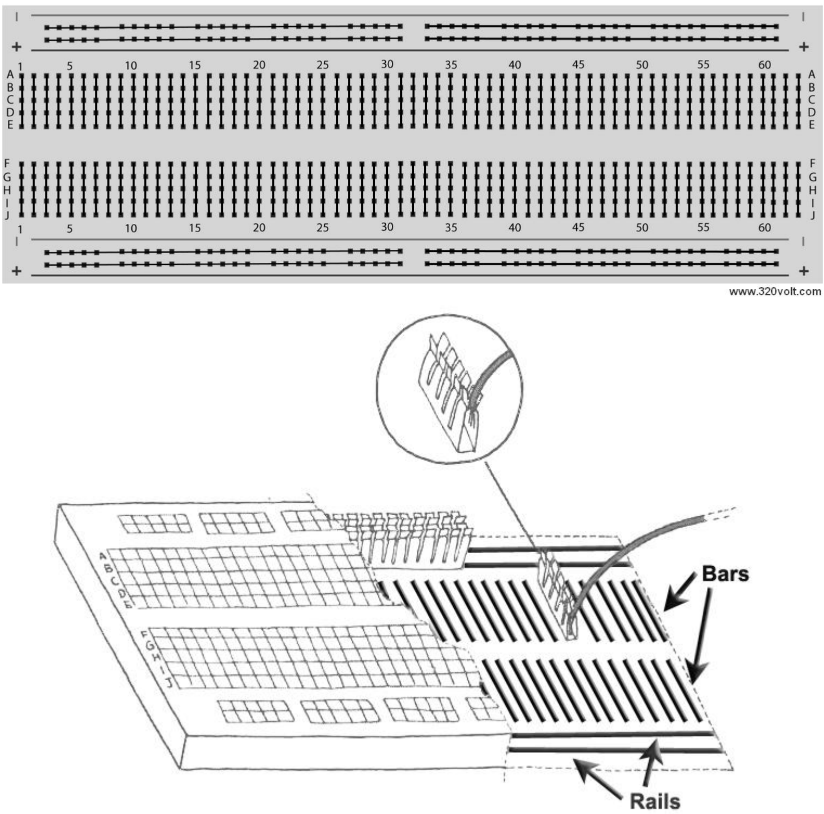
- ***Zero Adjustment Screw*** to calibrate the Zero position of meter pointer (normally, at the left side).
- Picture 1.2 shows ranges and unit on a display of an Analog VOM.



Picture 1.2. Range and Unit.

b. Breadboard

Breadboard is a tool to help building electronic circuits with wires and components. It has many strips of metal (copper usually) which run underneath the board. Picture 1.3 shows the internal structure of a breadboard.



Picture 1.3. Breadboard and inside

III. PRACTICE

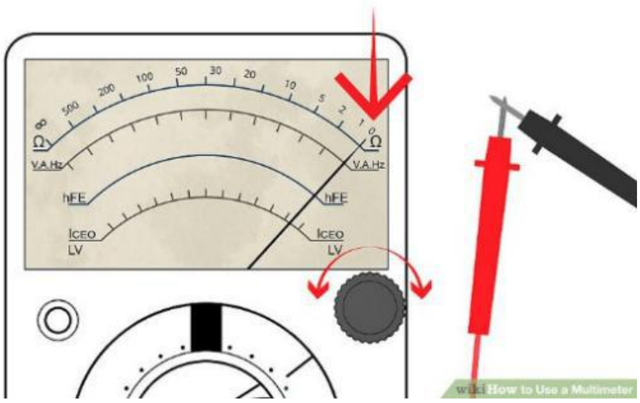
a. Equipment

- Analog VOM.
- Breadboard, resistors, capacitors, inductors, transformer, diodes and BJT.

b. Measure OHM with Analog VOM

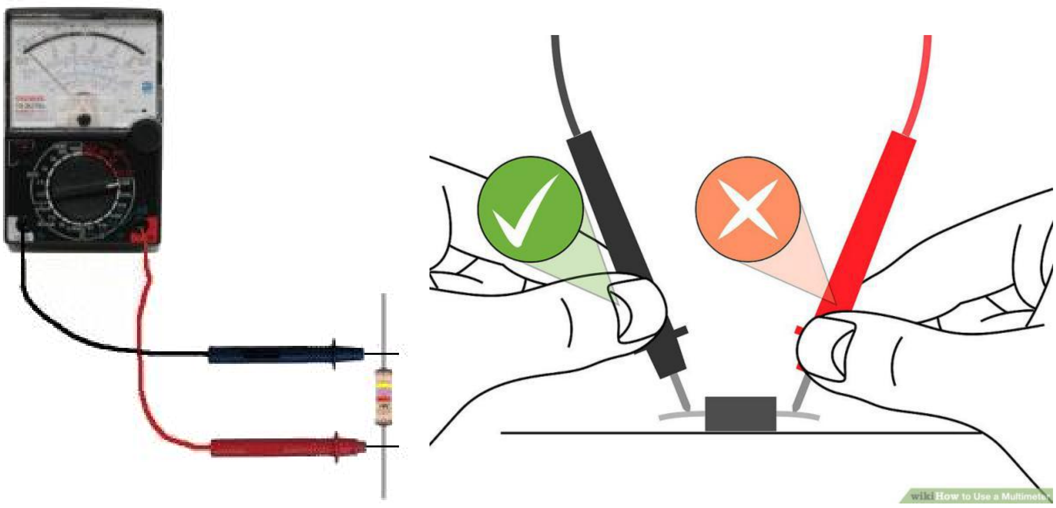
**Note: if you are planning to measure Ohm on a circuit, its power supply must be turned off before using Ohmmeter.*

- Step 1: Select a suitable OHM scale.
- Step 2: Touch two probes (Red and Black) of VOM to each other.
- Step 3: Adjust the ***0ΩADJ*** knob to move needle to Zero OHM position.



Picture 1.4. Adjusting Zero point

- Step 4: Place the two probes at the two terminals of a resistor to measure as in Picture 1.5.

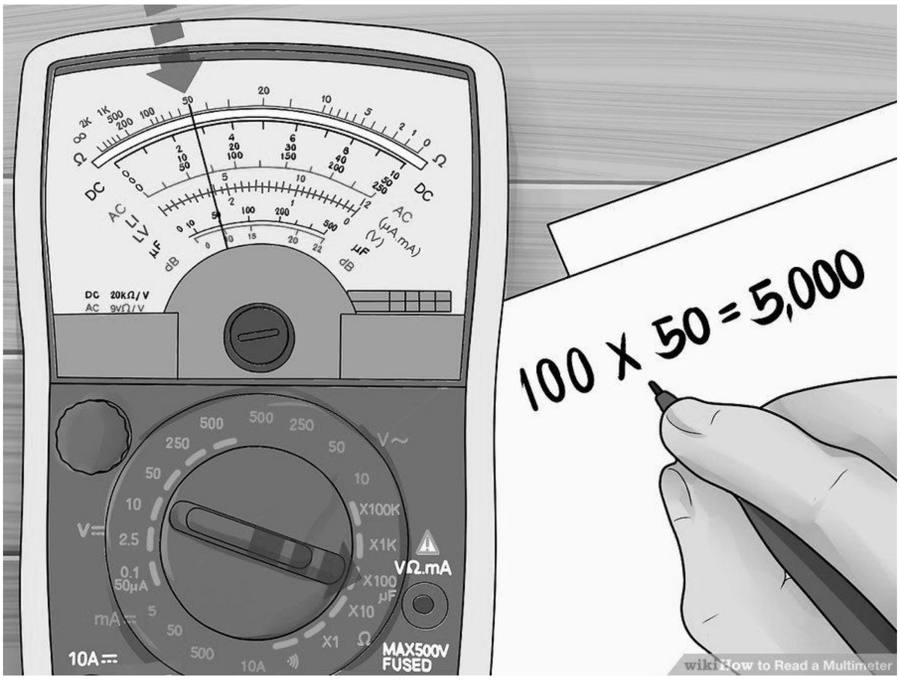


Picture 1.5. Measuring resistor with Analog VOM

- Step 5: Read the value on display and compare to value calculated from color code of the resistor.

*** HOW TO READ OHM VALUE ON DISPLAY*

- *X1 scale:*
  Value = Needle position (ex.: 20 Ω X 1 = 20 Ω)
- *X10 scale:*
  Value = Needle position X 10 (ex.: 20 Ω X 10 = 200 Ω)
- *X100 scale:*
  Value = Needle position X 100 (ex.: 20 Ω X 100 = 2000 Ω)
- *X1k scale:*
  Value = Needle position X 1 k Ω (ex.: 20 Ω X 1 k = 20 kΩ)
- *X10k scale:*
  Value = Needle position X 10 k Ω (ex.: 20 Ω X 10 k = 20 kΩ)



Picture 1.6. Reading value on Analog VOM display

c. Testing capacitor with Analog VOM

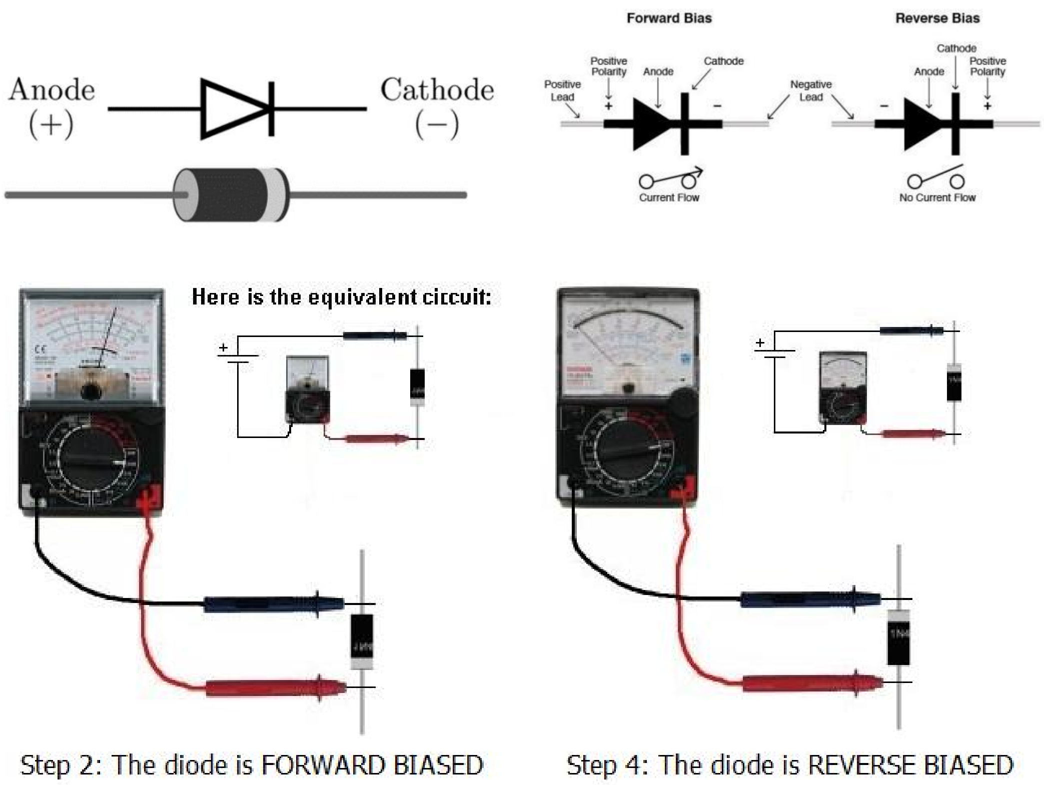
- Step 1: Select a suitable OHM scale.
- Step 2: Place the two probes onto two terminals of a capacitor.
- Step 3: Tracking movement of needle:
  - If needle goes up then goes down, the capacitor is good.
  - If needle goes up without goes down, the capacitor is shorted.
  - If needle does not move, the capacitor is opened or the current OHM scale is not suitable (*large scale should be used for small capacitance and otherwise*).

d. Testing inductor and transformer with Analog VOM

- Step 1: Select X1 on OHM scale.
- Step 2: Measure resistance of an inductor.
- Step 3: Measure resistances of primary coil and secondary coil.

e. Testing diode with Analog VOM

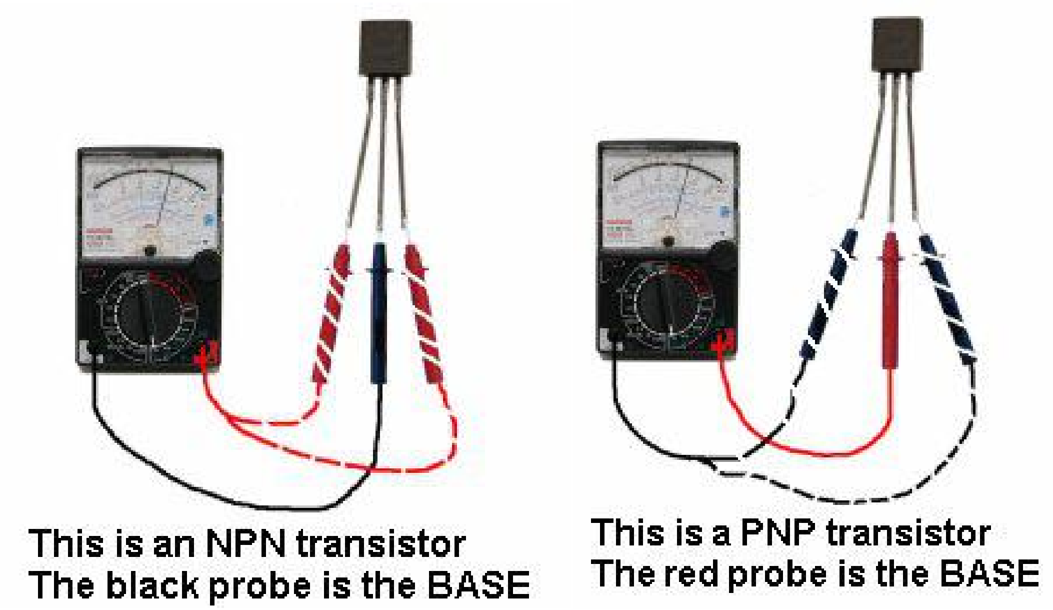
- Step 1: Select X10 or X100 on OHM scale
- Step 2: Place Red probe on Cathode terminal, Black probe on Anode terminal of a diode.
- Step 3: Tracing movement of needle:
    - If needle goes up, the diode may be good.
    - If needle does not move, the diode is broken.
- Step 4: Place Black probe on Cathode terminal, Red probe on Anode terminal of a diode.
- Step 5: Tracing movement of needle:
    - If needle does not move, the diode is good.
    - If needle goes up, the diode is shorted.



Picture 1.7. Testing diode with Analog VOM

f. Testing BJT with Analog VOM

- Step 1: Select X10 or X100 on OHM scale
- Step 2: Try the 6 combinations and when you have the black probe on a pin and the red probe touches the other pins and the meter swings nearly full scale, you have an NPN transistor. The black probe is BASE.
    - If the red probe touches a pin and the black probe produces a swing on the other two pins, you have a PNP transistor. The red probe is BASE
    - If the needle swings FULL SCALE or if it swings for more than 2 readings, the transistor is FAULTY.



Picture 1.8. Testing BJT with Analog VOM

g. Measure DC Voltage with Analog VOM

- Step 1: Select the maximum DCV scale.
- Step 2: Place Black probe on the lower voltage point (usually GND), Red probe on higher voltage point.
- Step 3: Read value from display.
- Step 4: If the value is too small to read, select lower DCV scale.

h. Measure AC Voltage with Analog VOM

- Step 1: Select the maximum ACV scale.
- Step 2: Place Black probe on the lower voltage point (usually GND), Red probe on higher voltage point.

- Step 3: Read value from display.
- Step 4: If the value is too small to read, select lower ACV scale.

IV. PREPARATION AT HOME

Equations for converting between voltage source and current source?

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 1: ANALOGUE MEASURING INSTRUMENT

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ………………………………………………, **student ID:** ………………..

- name: ………………………………………………, **student ID:** ………………..

TABLE OF RESULTS		
Home question	Equation	
b	Resistor values	Read: …………………… Measured:………………………… Read: …………………… Measured:………………………… Read: …………………… Measured:…………………………
c	Capacitor test	Scale: ………………Minimum value of needle: ……………
d	Inductor	Resistance value: ………………………………………………
	Transformer	Primary resistance: …………………………………………… Secondary resistance: …………………………………………
e	Diode test	Scale: …………………Minimum value of needle: …………..
f	BJT test	BJT type:………………………………………………… Pin positions:
g	DC voltage	Measured value:……………………………………………
h	AC voltage	Measured value:……………………………………………

------- END OF REPORT -------

LAB 2

DIGITAL MEASURING INSTRUMENT

I. GOAL

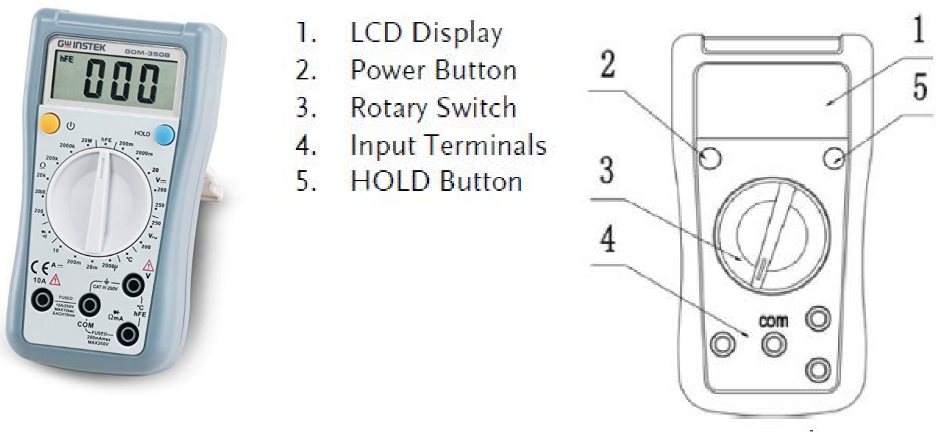
In this lab, student will have skills:

- Using Digital VOM, Oscilloscope and Function Generator.
- Reading and measuring resistor values, testing electronic components such as capacitor, inductor, transformer, diode and BJT.

II. SUMMARY OF THEORY

a. Digital VOM (DMM)

Picture 2.1 describes fundamental components of a Digital VOM (DMM)



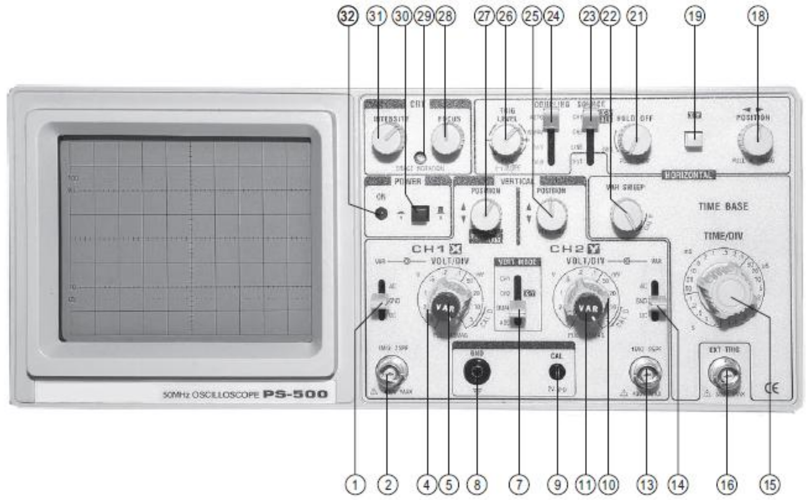
Picture 2.1. Digital VOM.

- ***LCD*** displays measured values.
- ***Power Button*** turns on or off the equipment.
- ***Rotary Switch*** selects which electronic unit will be measured and its scale.
- ***Input Terminals*** connects to probes, the ***COM*** hole connects to Black probe.
- ***HOLD Button*** pauses the equipment and keep the last value on LCD. To measure continuously, release this button from pressed state.

Digital VOM is easier to use than Analog VOM, its display is clear and easy to read out small values.

Operating controls: reference to Digital VOM Manual in Appendix A.

b. OSCILLOSCOPE

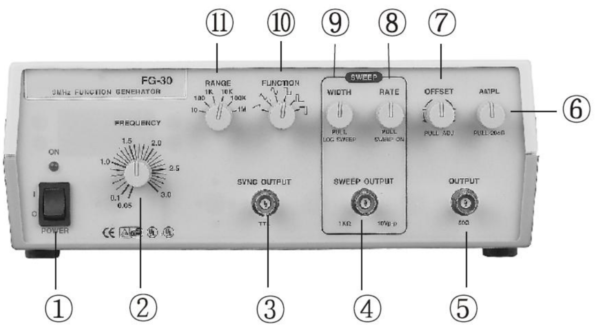


Picture 2.2. Oscilloscope

Operating Controls, Indicators and Signal input connectors: reference to Oscilloscope Manual in Appendix B.

c. Function Generator

Function Generator is a device generating waves for testing. Picture 2.3 shows a picture of a Function Generator.



Picture 2.3. A Function Generator.

Operating Controls, Indicators and Signal input connectors: reference to Function Generator Manual in Appendix C.

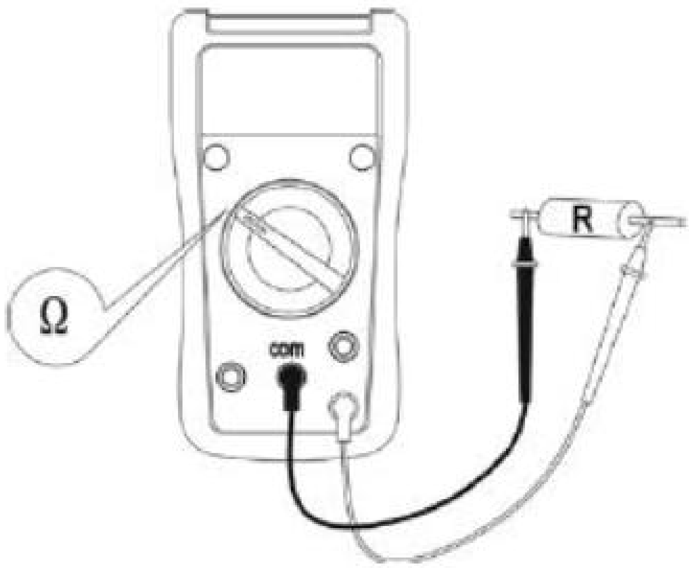
III. PRACTICE

d. Equipment

- Digital VOM.
- Breadboard, resistors, capacitors, inductors, transformer, diodes and BJT.
- Oscilloscope
- Function Generator.

e. Measure OHM with Digital VOM

- Step 1: Turn on Digital VOM
- Step 2: Connect Black probe to COM hole, Red probe to ΩmA hole.
- Step 3: Select suitable Ω scale.
- Step 4: Place the two probes onto two terminals of a resistor.
- Step 5: Read value on LCD, unit of value is the unit of the selected Ω scale.

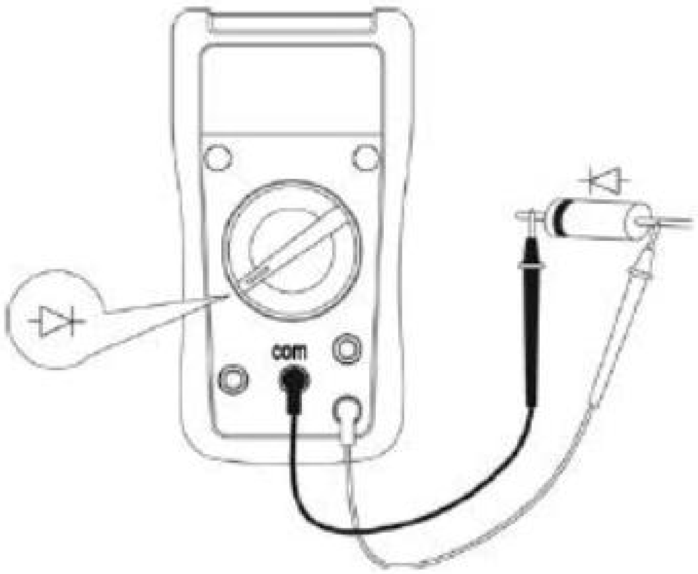


Picture 2.4. Measure resistance with Digital VOM

f. Testing diode with Digital VOM

- Step 1: Turn on Digital VOM
- Step 2: Connect Black probe to COM hole, Red probe to ΩmA hole.
- Step 3: Select diode symbol ⊳|.
- Step 4: Place Red probe on Anode terminal, Black probe on Cathode terminal of a diode.
- Step 5: If LCD value is different from "1", the diode may be good.

- Step 6: Place Black probe on Anode terminal, Red probe on Cathode terminal of a diode.
- Step 7: If LCD value is "1", the diode is good.



Picture 2.5. Testing diode with Digital VOM

g. Measure DC Voltage with Digital VOM

- Step 1: Select the maximum V⎓ scale.
- Step 2: Place Black probe on the lower voltage point (usually GND), Red probe on higher voltage point.
- Step 3: Read value from display.
- Step 4: If the value is too small to read, select lower scale.

h. Measure AC Voltage with Digital VOM

- Step 1: Select the maximum V~ scale.
- Step 2: Place Black probe on the lower voltage point (usually GND), Red probe on higher voltage point.
- Step 3: Read value from display.
- Step 4: If the value is too small to read, select lower ACV scale.

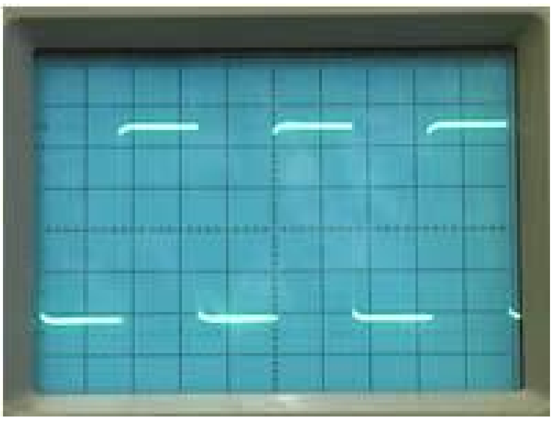
i. Oscilloscope and Function Generator

Oscilloscope

- Step 1: Select scale on probe of Oscilloscope to X1

Picture 2.6. Select scale X1 on Oscilloscope probe

- Step 2: Turn ON POWER (30), LED (32) will light when Oscilloscope is powered on.
- Step 3: Vary INTENSITY (31) to change brightness.
- Step 4: Vary FOCUS (28) to select focus of beam on display.
- Step 5: Select input channel using VERT MODE (7) to channel 1 (CH1).
- Step 6: Select SOURCE (23) to CH1.
- Step 7: Make sure that X-Y (19) button is not pressed.
- Step 8: Rotate VAR (5) clockwise until hearing a "click" sound.
- Step 9: Rotate VAR SWEEP (22) clockwise to the most right position.
- Step 10: Connect probe to CAL (9) to test Oscilloscope and the probe.
- Step 11: Select AC-GND-DC Switch (1) to GND.
- Step 12: Vary POSITION (27) until seeing a line of beam in the middle of screen.
- Step 13: Select AC-GND-DC Switch (1) to AC.
- Step 14: Rotate TIME / DIV (15) to position .5 mS
- Step 15: Vary VOL / DIV (4) until seeing a square wave on screen.



Picture 2.7. Square wave on Oscilloscope screen

- Step 16: Using equations in TIME MEASUREMENTS, FREQUENCY MEASUREMENTS and MEASUREMENT OF VOLTAGE BETWEEN TWO POINT ON A WAVEFORM in Oscilloscope Manual (Appendix C) to calculate period, frequency and peak-peak voltage of the captured wave.

Function Generator

- Step 17: Turn on POWER (1) on Function Generator, LED ON will light when Function Generator is powered on.
- Step 18: Rotate RANGE (11) to 1K.
- Step 19: Rotate FUNCTION (10) to Sine wave.
- Step 20: Rotate FREQUENCY (2) to 1.5 position.
- Step 21: Rotate OFFSET (7) counter-clockwise to the most left position.
- Step 22: Connector cable to OUTPUT (5).
- Step 23: Connect the cable in step 22 to Oscilloscope probe.
- Step 24: Check waveform on Oscilloscope screen and calculate period , frequency and peak-peak voltage of this wave.

IV. PREPARATION AT HOME

Equation to calculate voltage between two points on Oscilloscope?

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 2: DIGITAL MEASURING INSTRUMENT

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ……………………………………………, **student ID:** ………………..

- name: ……………………………………………, **student ID:** ………………..

TABLE OF RESULTS		
Home question	Equation	
b	Resistor values	Read: …………………… Measured:………………………… Read: …………………… Measured:………………………… Read: …………………… Measured:…………………………
c	Diode test	Forward baised value:………………………………………..
d	DC voltage	Measured value:……………………………………………
e	AC voltage	Measured value:……………………………………………
f	Oscilloscope	Period:……………… Frequency:…………… Vpp:………….. Waveform:
	Function Generator	Period:……………… Frequency:…………… Vpp:………….. Waveform:

------- END OF REPORT -------

LAB 3
ELECTRONIC CAD SOFTWARE

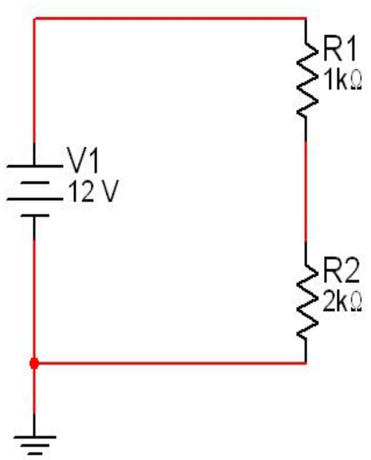
I. GOAL

In this lab, student will have skills:

- Understanding netlist of schematic.
- Using Electronic CAD software in schematic design and simulation.

II. SUMMARY OF THEORY

Electronic CAD software is computer program helping engineers in designing schematic, PCB and running circuit simulation. To simulate a circuit, SPICE is the most popular tool. The following picture and code shows an example of SPICE program:



** Any text after the asterisk '*' is ignored by SPICE*

** Voltage Divider*

vV1 1 0 12

rR1 1 2 1000

rR2 2 0 2000

*.OP * perform a DC operating point analysis*

.END

Picture 3.1. Voltage Divider circuit and Netlist.

There are many Electronic CAD softwares over the world. In this lab, OrCAD PCB Designer Lite is used in practice because of rich features and free. OrCAD Lite is fully functional and offers every feature of OrCAD, limited only by the size and complexity of the design. There is no time limit for OrCAD Lite, you can use it as long as you want.

- OrCAD Capture: it is one of the most widely used schematic design solutions for the creation and documentation of electrical circuits.

- OrCAD CIS: CIS (component information system) is product for component data management, along with highly integrated flows supporting the engineering process
- OrCAD PSpice® A/D and Advanced Analysis: OrCAD® PSpice® and Advanced Analysis technology combine industry-leading, native analog, mixed-signal, and analysis engines to deliver a complete circuit simulation and verification solution.
- OrCAD PCB Editor: it is a tool to design a PCB (Printed Circuit Board) from a schematic.

III. PRACTICE

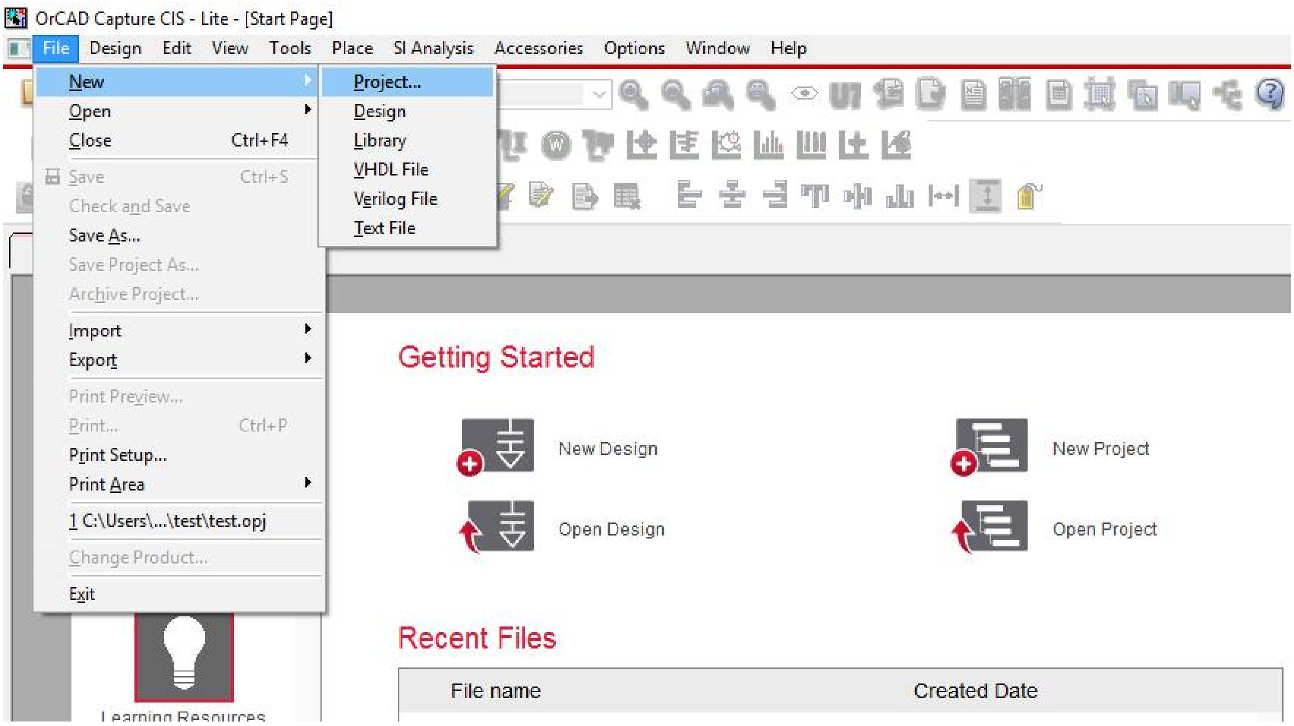
a. Equipment

- OrCAD PCB Designer Lite

b. Schematic

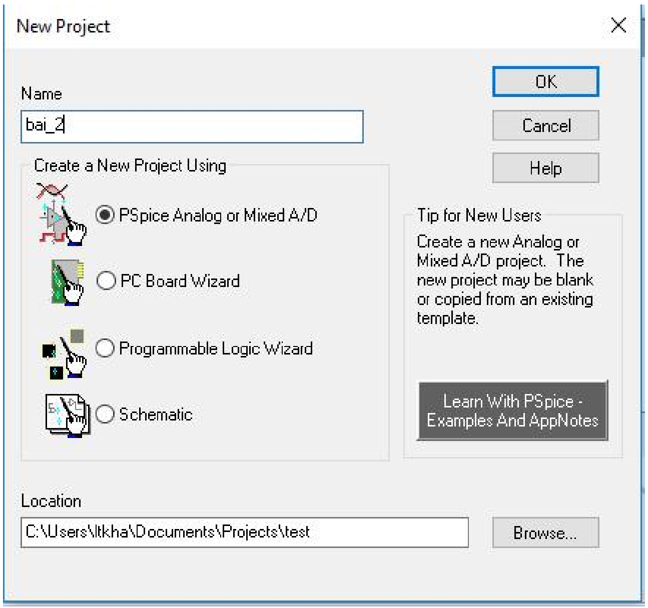
Step 1: open Capture CIS Lite from Start Menu or Shortcut on Desktop.

Step 2: select File->New->Project



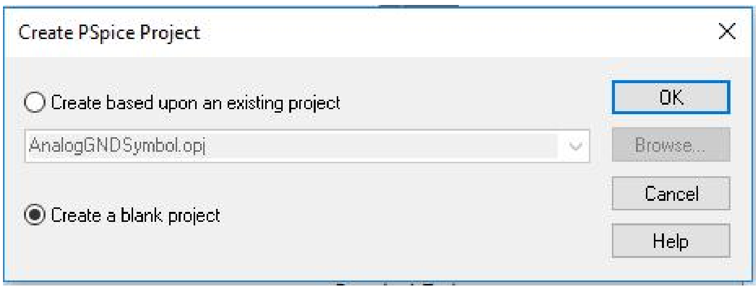
Picture 3.2. File->New->Project

Step 3: in New Project box, put a name for project in Name, select design type as in the picture below. After that, select OK to continue.



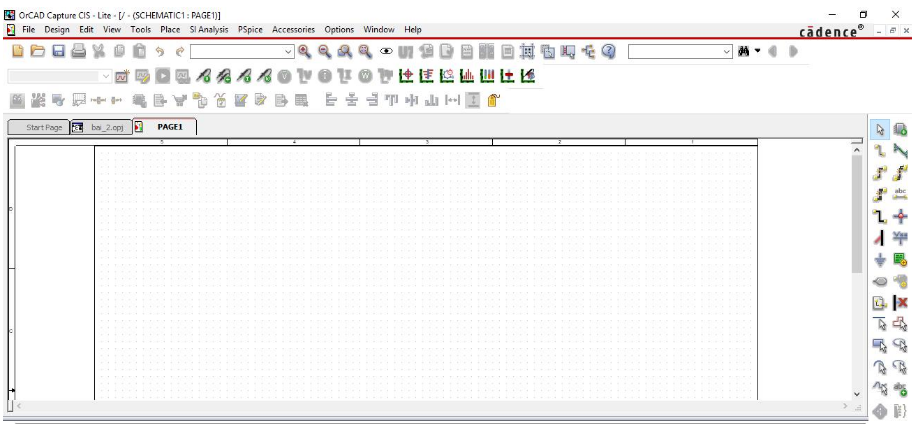
Picture 3.3. Set name and type for project

Step 4: OrCAD will ask you to create a blank project or using existing project, select "Create a blank project" as in Picture 3.4.



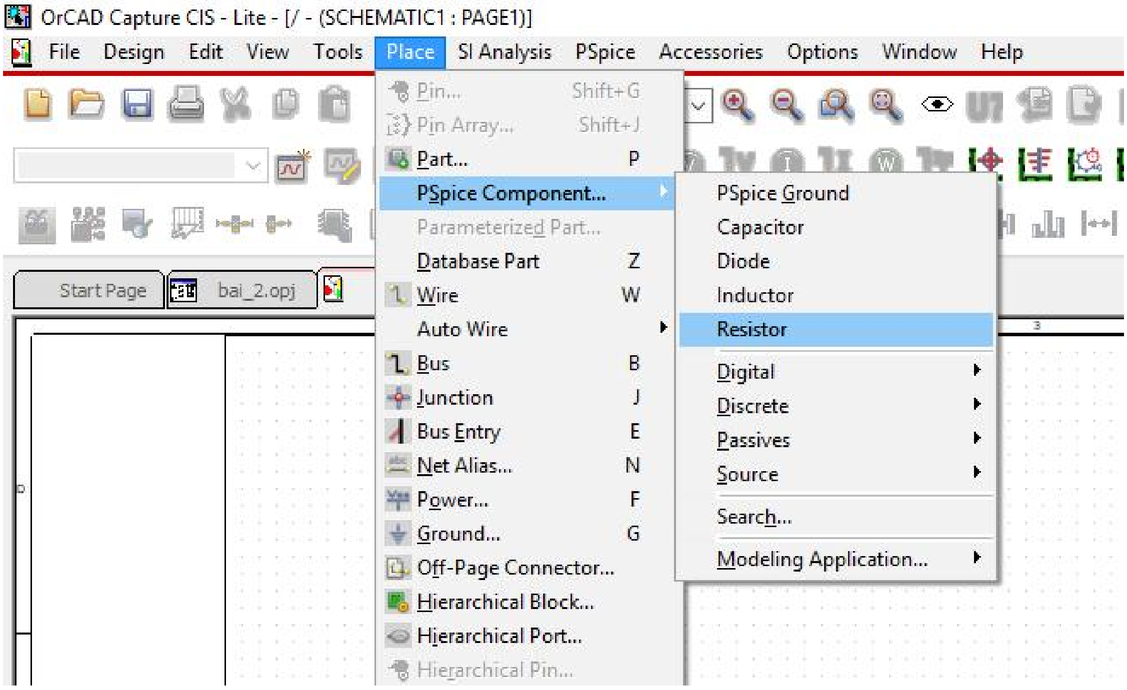
Picture 3.4. Create a blank project

Step 5: a blank page will open as in Picture 3.5



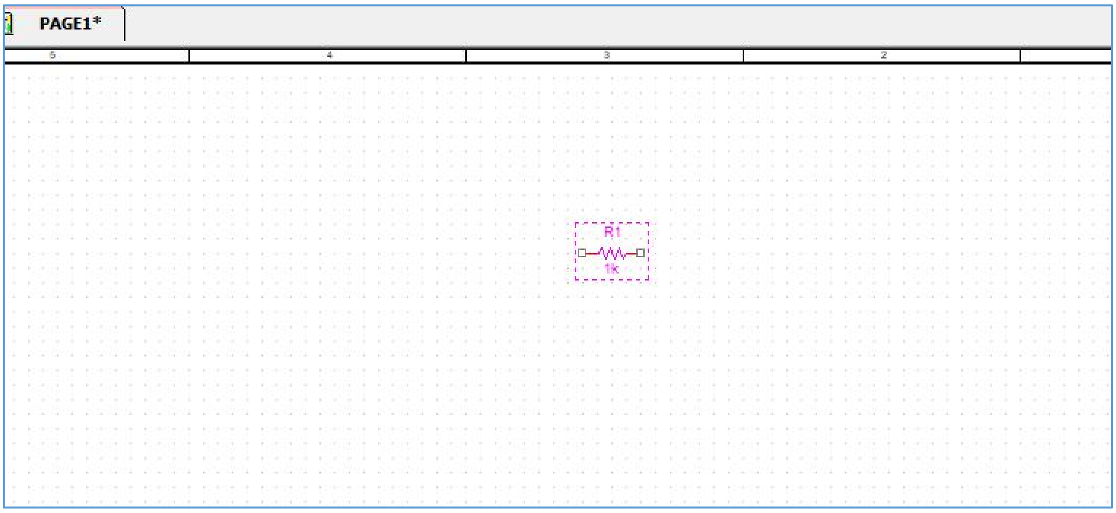
Picture 3.5. Blank page for design

Step 6: go to Place->PSpice Component…->Resistor to pick up a resistor. After that, a resistor symbol will occur under your cursor.



Picture 3.6. Place->PSpice Component…->Resistor

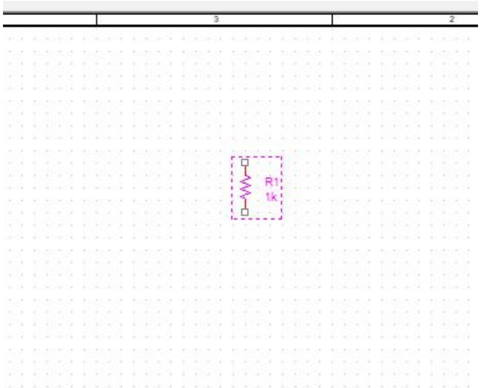
Step 7: left-click on white page to place the resistor above.



Picture 3.7: Placing a resistor

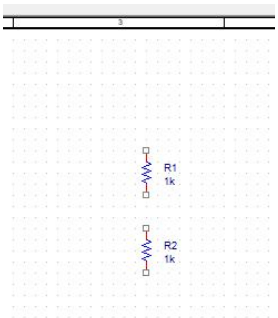
Step 8: select the resistor above, hold left button on your mouse and move your mouse to move the component to anywhere on the design page.

Step 9: select the resistor above, press R to rotate it 90 degree.



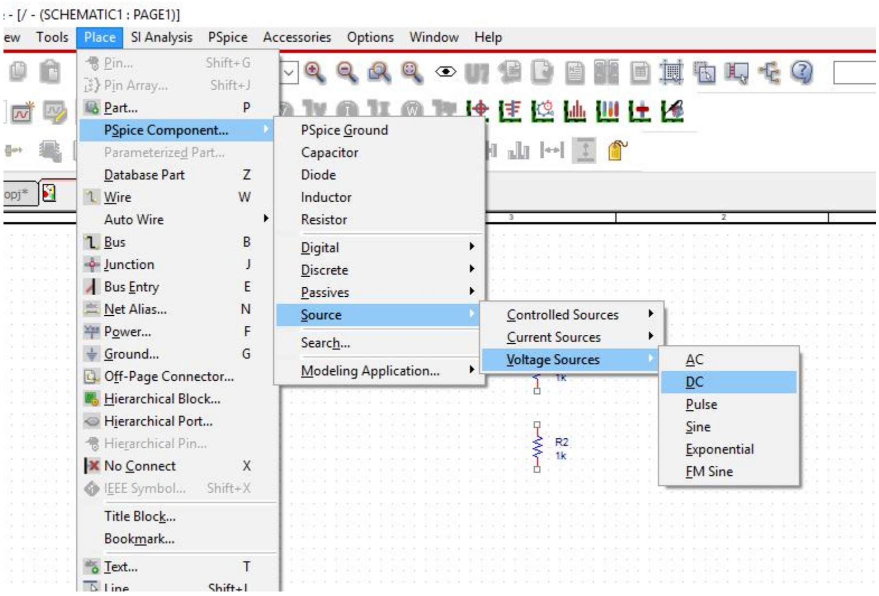
Picture 3.8. Select then press R to rotate 90 degree.

Step 10: Place another resistor under the resistor above (follow the step 6 to step 9)



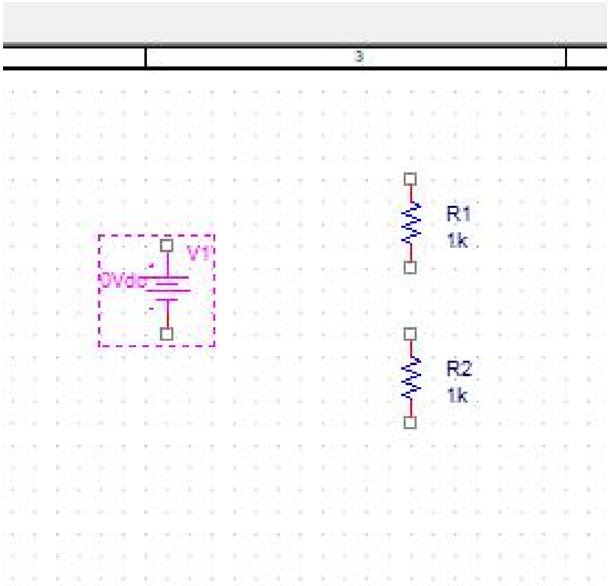
Picture 3.9. Place R2

Step 11: go to Place->PSpice Component…->Source->Voltage Sources->DC to place a DC voltage source



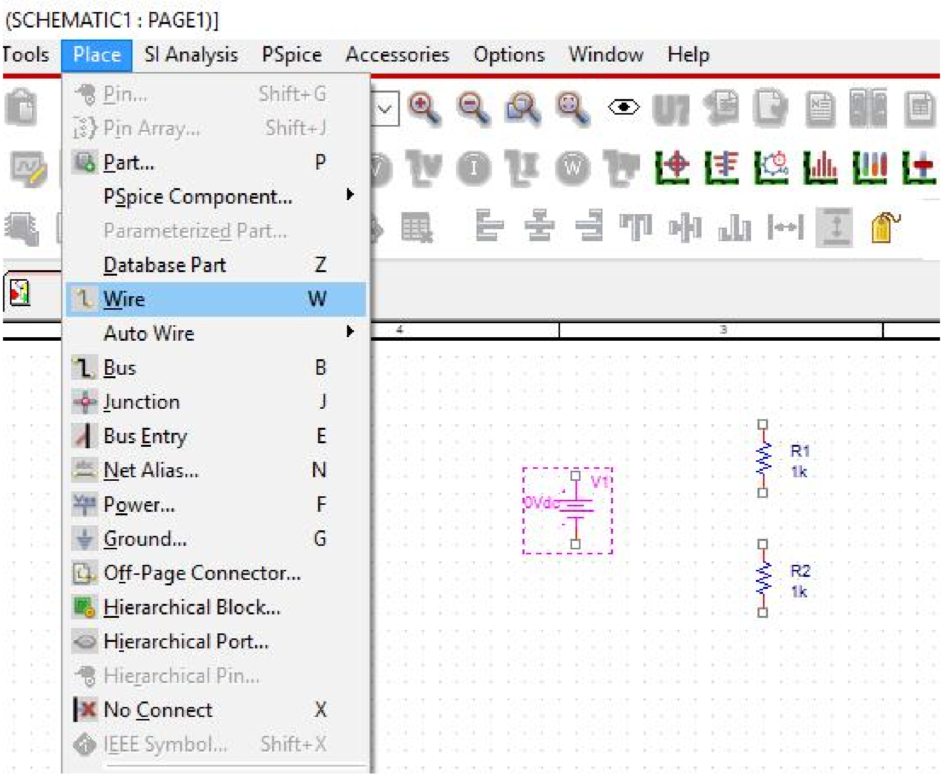
Picture 3.10. Place->PSpice Component…->Source->Voltage Sources->DC

Step 12: click on design page to place V1



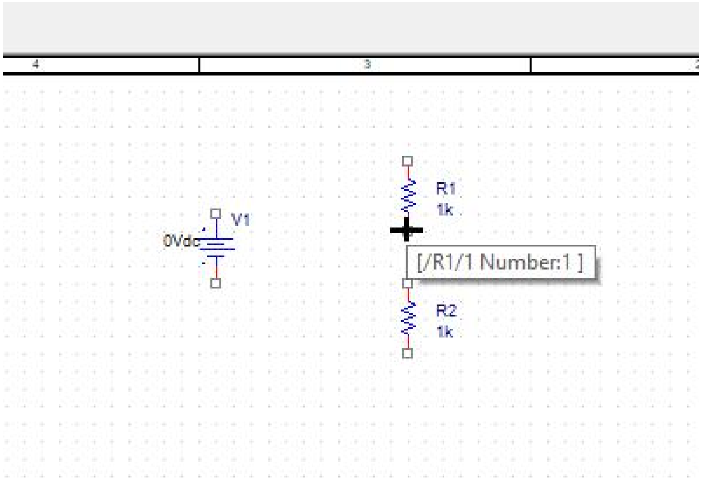
Picture 3.11. Place a DC voltage source

Step 13: go to Place->Wire (or press W) to go to wiring mode. If you want to quit from this mode, press Esc on your keyboard.



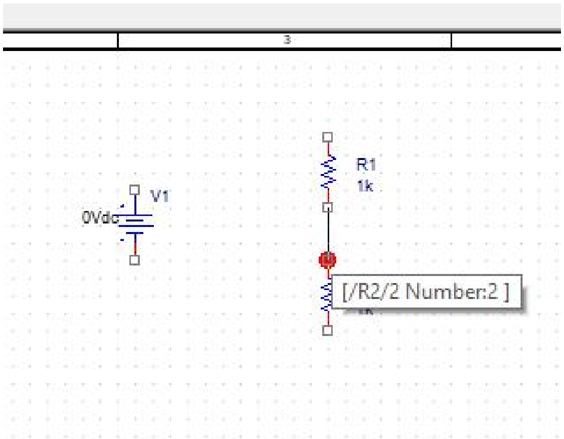
Picture 3.12. Place->Wire

Step 14: left-click on one terminal of R1



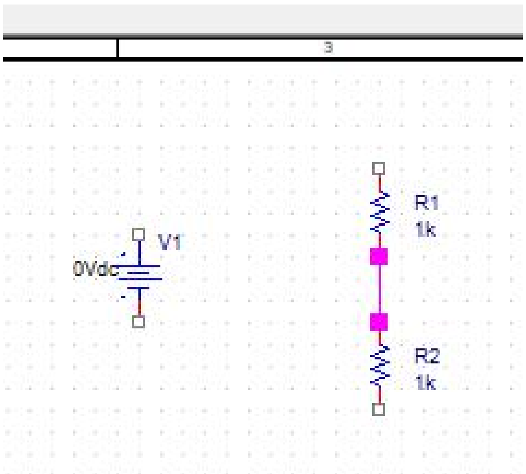
Picture 3.13. Left-click on a terminal of R1.

Step 15: move cursor to a terminal of R2 until see a Red circle.



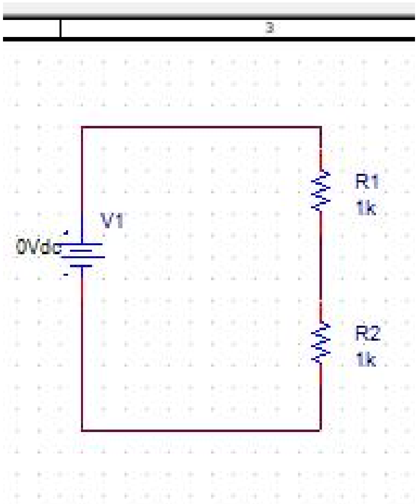
Picture 3.14. Move cursor to a terminal of R2.

Step 16: left-click on the terminal of R2 to finish the routing between R1 and R2.



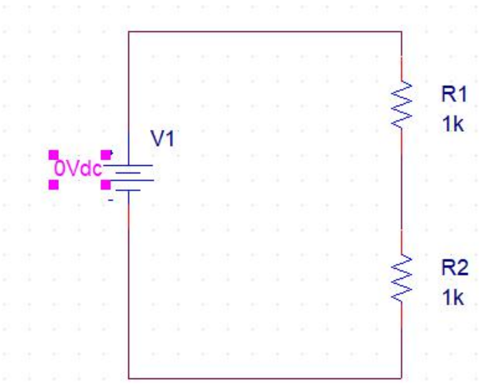
Picture 3.15. Complete routing between R1 and R2.

Step 17: follow step 13 to step 16 to complete the wiring of the circuit as in the picture below.



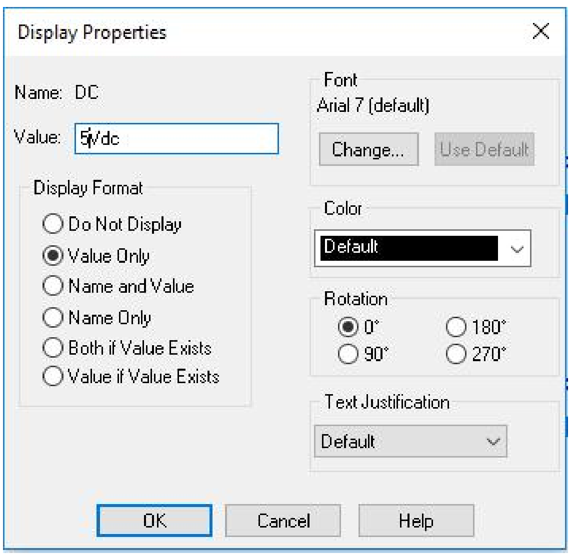
Picture 3.16. Completed circuit

Step 18: double click on “0Vdc”



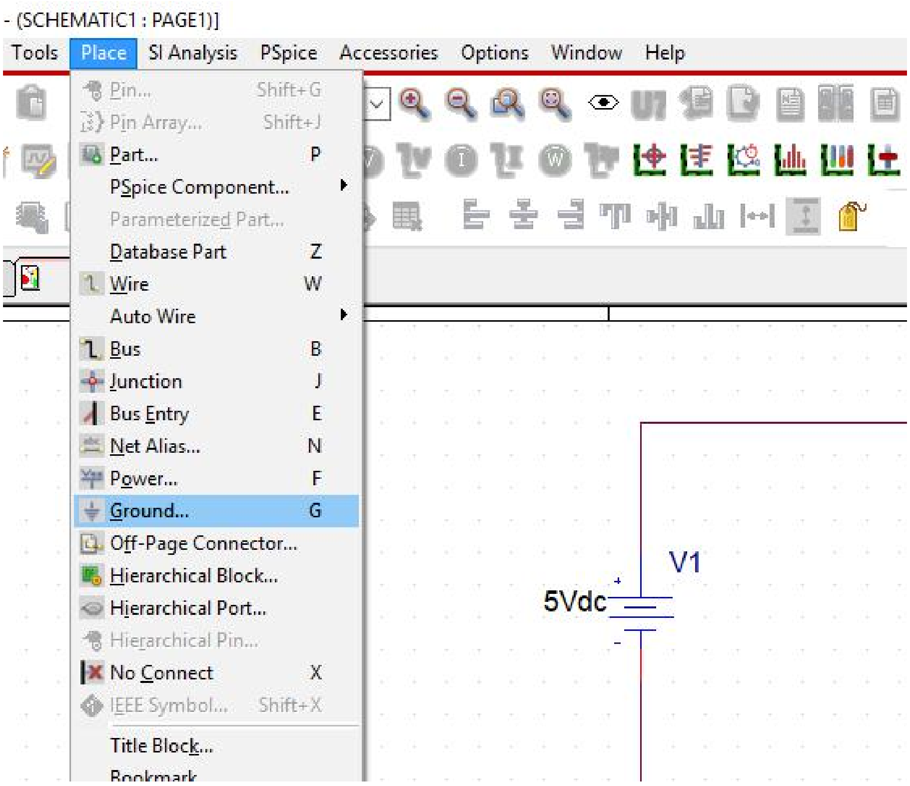
Picture 3.17. Double click on “0Vdc”

Step 19: change Value to 5Vdc then select OK.



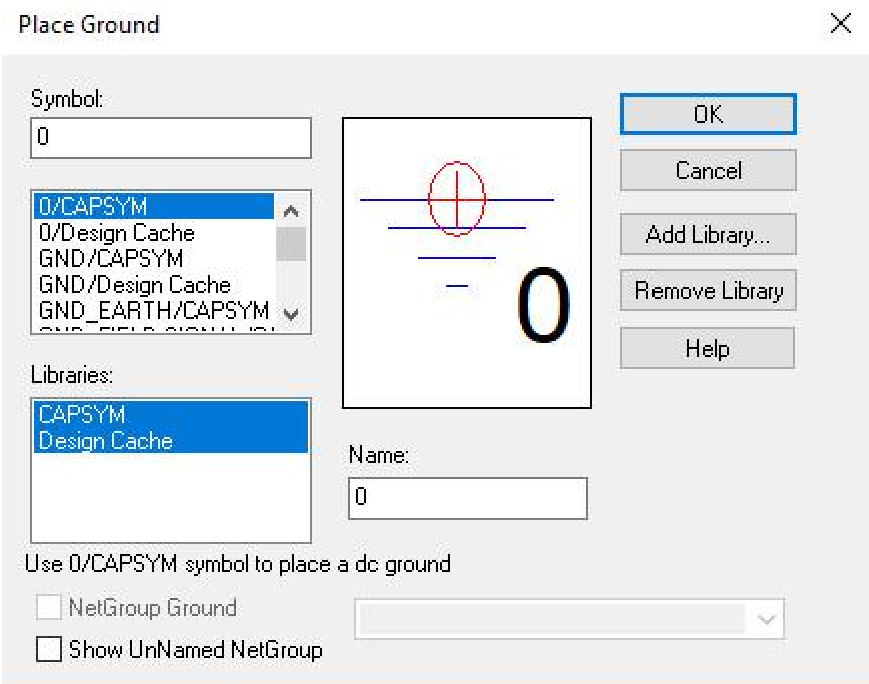
Picture 3.18. Change Value to 5Vdc

Step 20: go to Place->Ground… to place GND symbol into design page



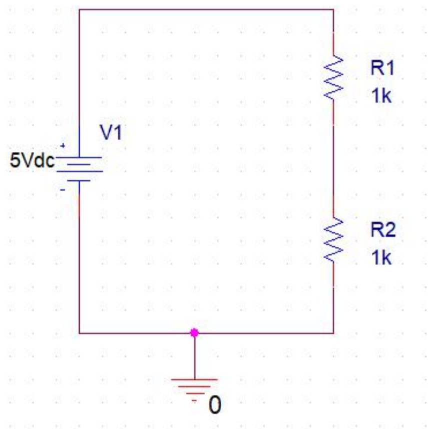
Picture 3.19. Place->Ground…

Step 21: Select 0/CAPSYM



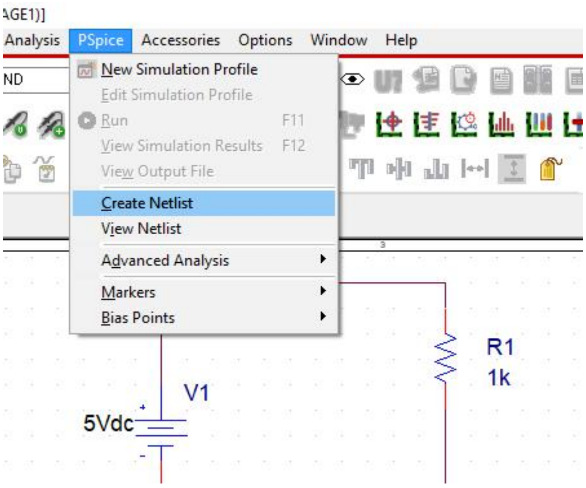
Picture 3.20. Select 0/CAPSYM

Step 22: Wiring GND to circuit



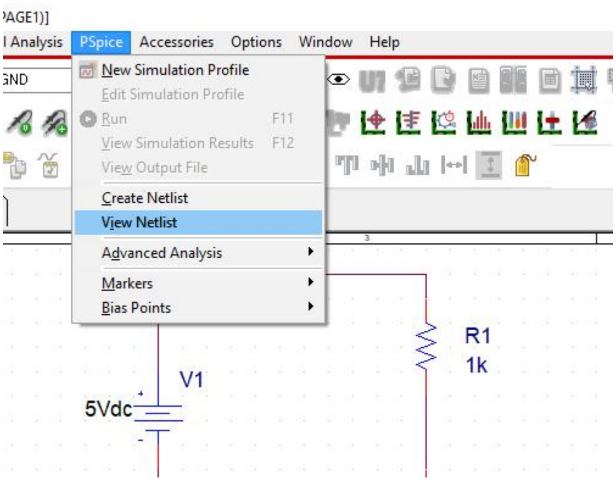
Picture 3.21. Wiring GND to circuit

Step 23: go to PSpice->Create Netlist



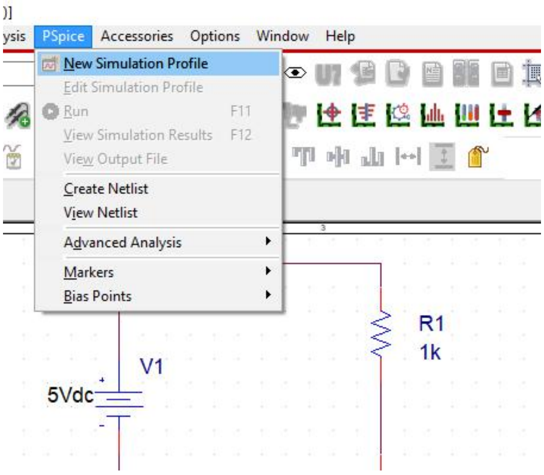
Picture 3.22. PSpice->Create Netlist

Step 24: Go to PSpice->View Netlist to view netlist of this circuit



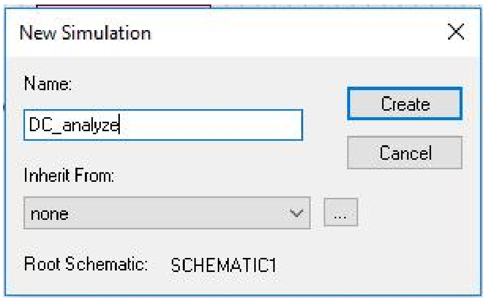
Picture 3.23: PSpice->View Netlist

Step 25: Go to PSpice->New Simulation Profile to setup simulation information



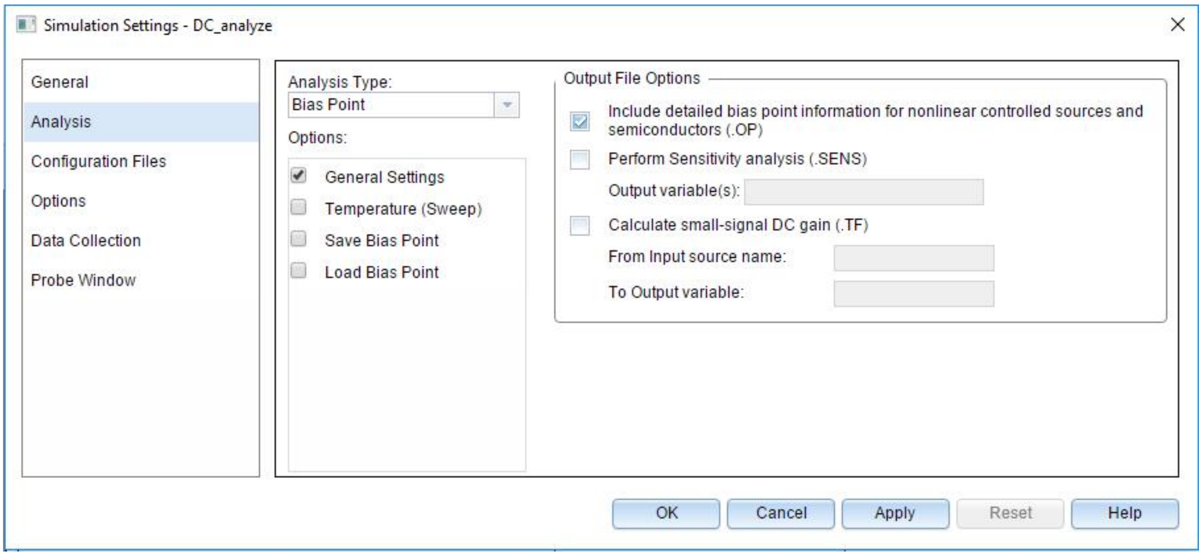
Picture 3.24. PSpice->New Simulation Profile

Step 26: set name for profile (anything you want without special characters such as space, &, #,…)



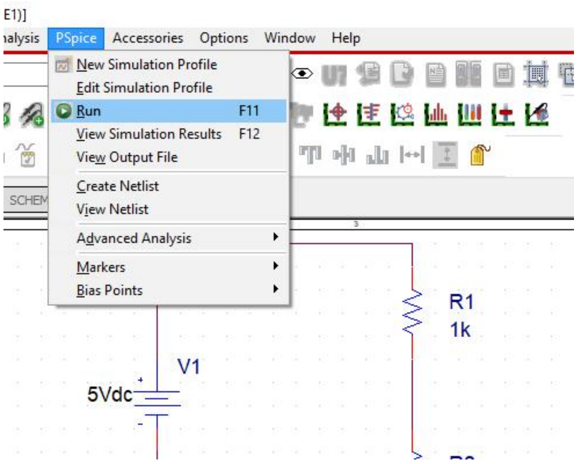
Picture 3.25. Set name for profile then click Create

Step 27: Setting as in the picture below



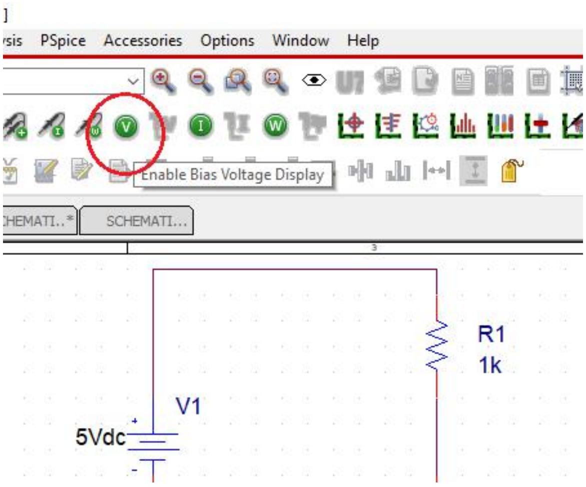
Picture 3.26. Simulation setting then click OK

Step 28: go to PSpice->Run to run simulation



Picture 3.27. Run simulation

Step 29: click V button to show results.



Step 30: Fill report

IV. PREPARATION AT HOME

Install OrCAD PCB Designer Lite on your computer.

Equation of Voltage divider?

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 3: ELECTRONIC CAD SOFTWARE

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ……………………………………………………, **student ID:** ………………..

- name: ……………………………………………………, **student ID:** ………………..

TABLE OF RESULTS		
Home question	Equation	
Step 24	Netlist	
Step 30	Schematic and Simulation results (draw schematic and values)	
Extend	R1=2k, R2=5k, V1=15Vdc Draw results	

------- END OF REPORT -------

LAB 4

DC SWEEP AND TRANSIENT IN PSPICE

I. GOAL

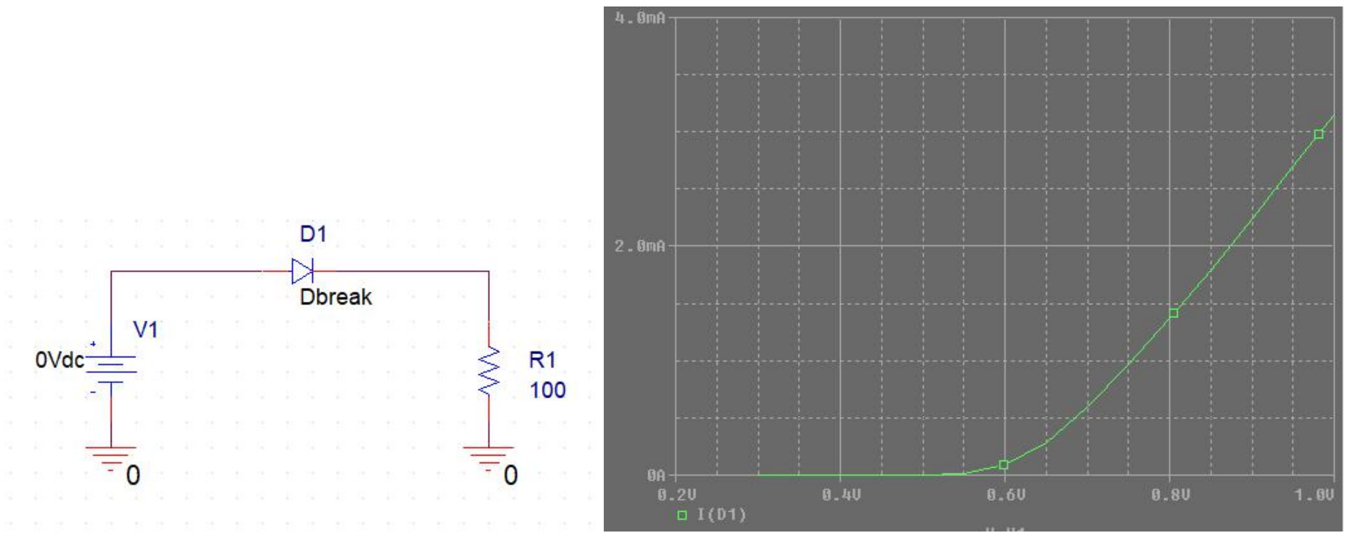
In this lab, student will have skills:

- Simulate circuits with DC sweep and transient mode in PSPICE

II. SUMMARY OF THEORY

a. DC Sweep

It is a simulation mode in DC. In this mode, DC voltage/current of suppliers will change in specific ranges with specific steps to analyze DC characteristics of circuits.

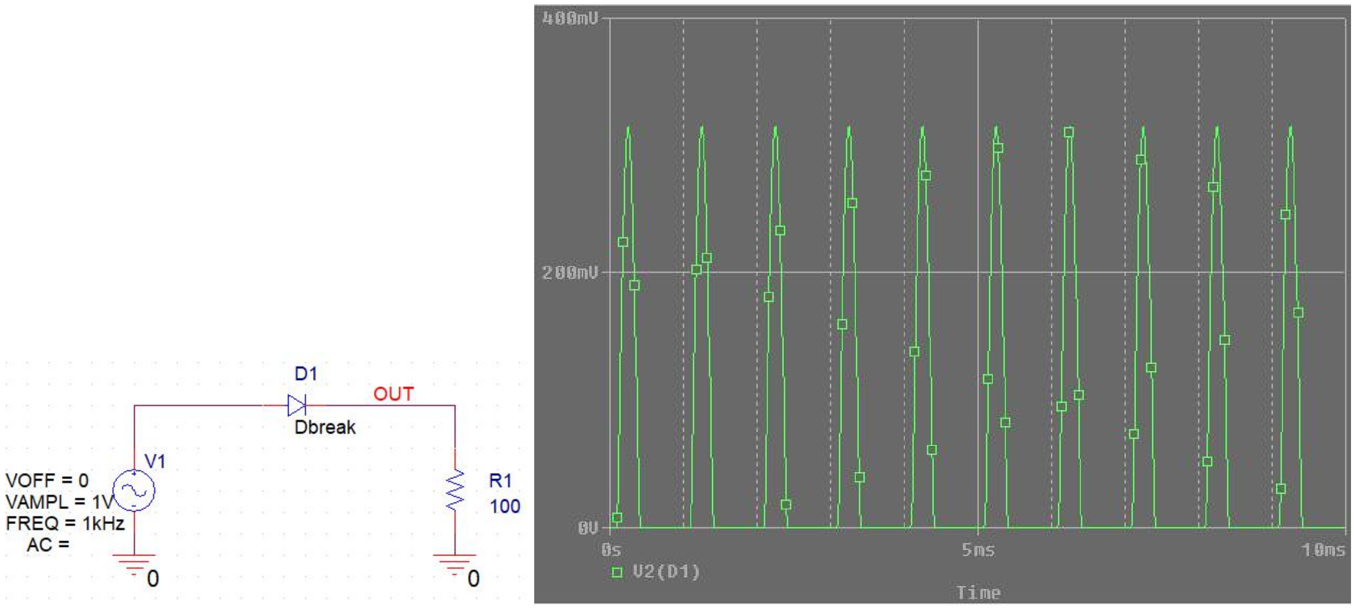


Picture 4.1 DC Voltage-Ampere characteristic of a diode

In the circuit of picture 4.1, voltage of V1 changes in the range from 0.2V to 1.0V with step is 0.05V, the changes of DC current through D1 is plotted to show the voltage-ampere relation in D1.

b. Transient (Time domain)

This mode of simulation shows how a circuit operates its function. Time to analyze is the most important thing. It requires start time that is usually zero and stop time. The smaller step of simulation time, the higher accuracy of simulation.



Picture 4.2. Output voltage on wire "OUT" in transient simulation

A transient simulation example is showed in picture 4.2. An AC source with sine wave is connected to a diode circuit to work as a single phase rectifier. Stop time of the simulation is 10ms with the step of 10us. Output waveform shows the result of operation of this circuit.

Transient simulation is one of the most important simulation mode not only to check and learn but also to prove functions of a circuit.

III. PRACTICE

a. Equipment

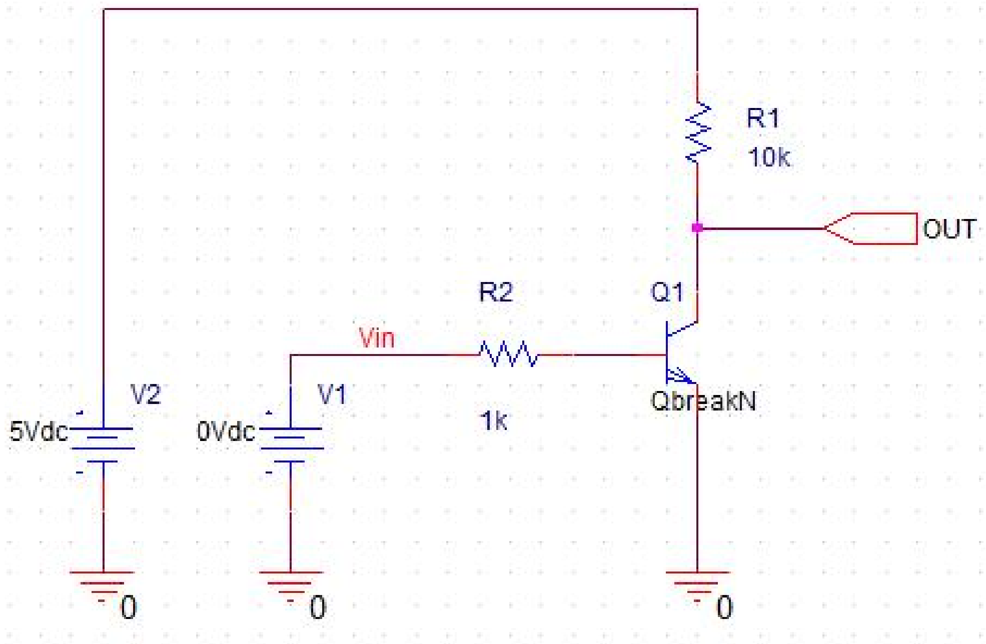
- OrCAD PCB Designer Lite

b. DC sweep

Step 1: open Capture CIS Lite and draw a circuit as in the picture 4.3.

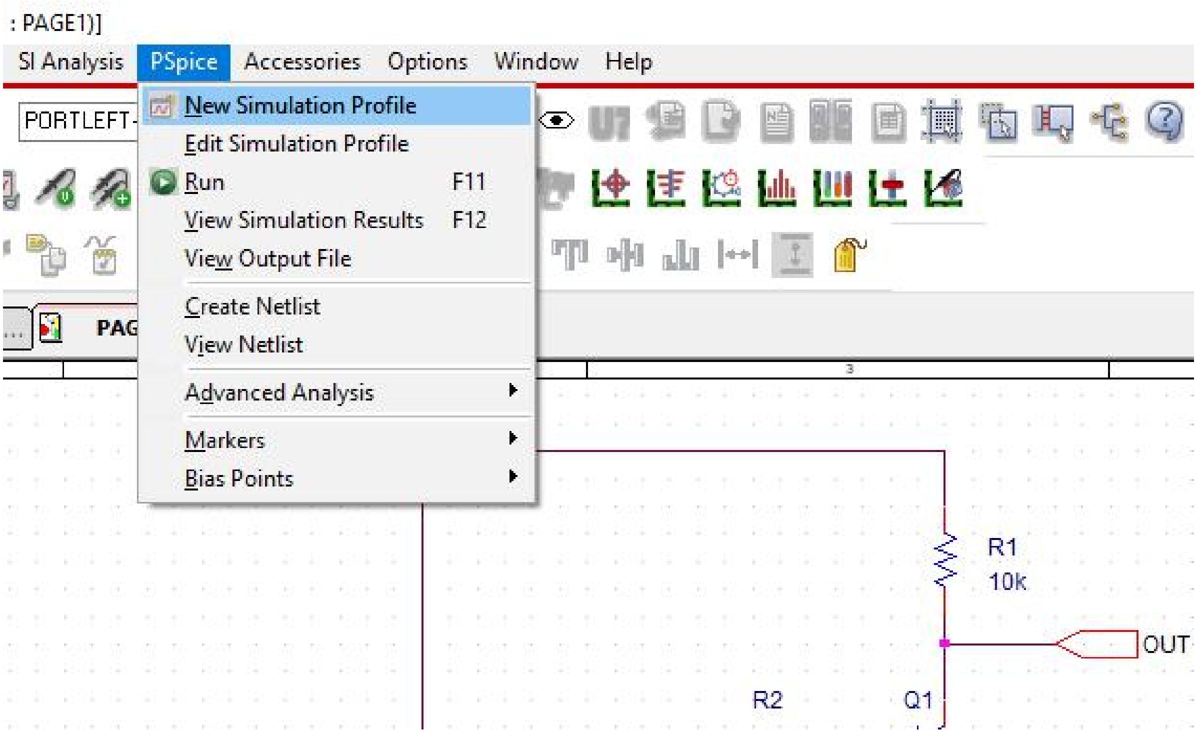
- To place name for a wire such as Vin:
  - Place->Net Alias…
  - Write a name such as Vin then click OK
  - Click on a wire on schematic to place the name above for that wire.
- To place a port such as OUT:
  - Place->Hierarchical Port…
  - Select a port type such as PORTLEFT-L then click OK

- Click on schematic page to place that port.
- Wiring a wire to the port.
- Double-click on PORTLEFT-L label to change port name such as change to OUT.



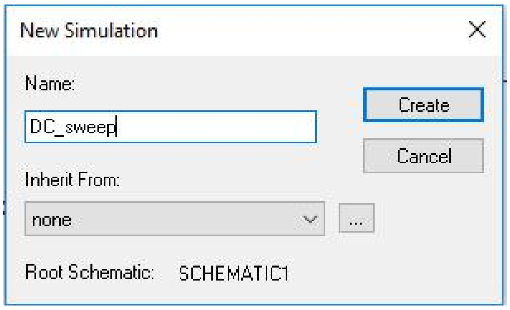
Picture 4.3 NPN transistor testing circuit

Step 2: select PSpice->New Simulation Profile



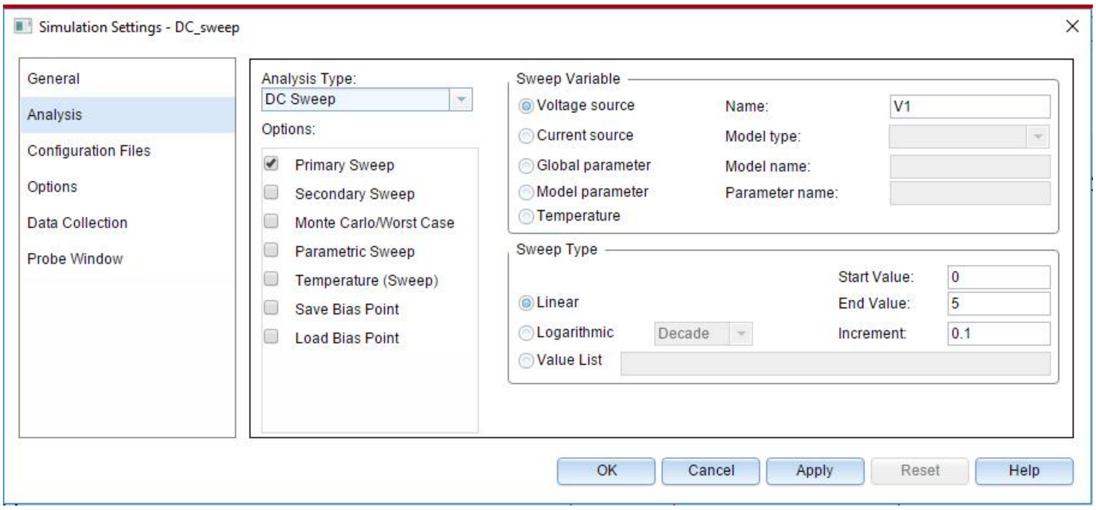
Picture 4.4. Setup a simulation

Step 3: setup name of simulation



Picture 4.5 Setup name of simulation

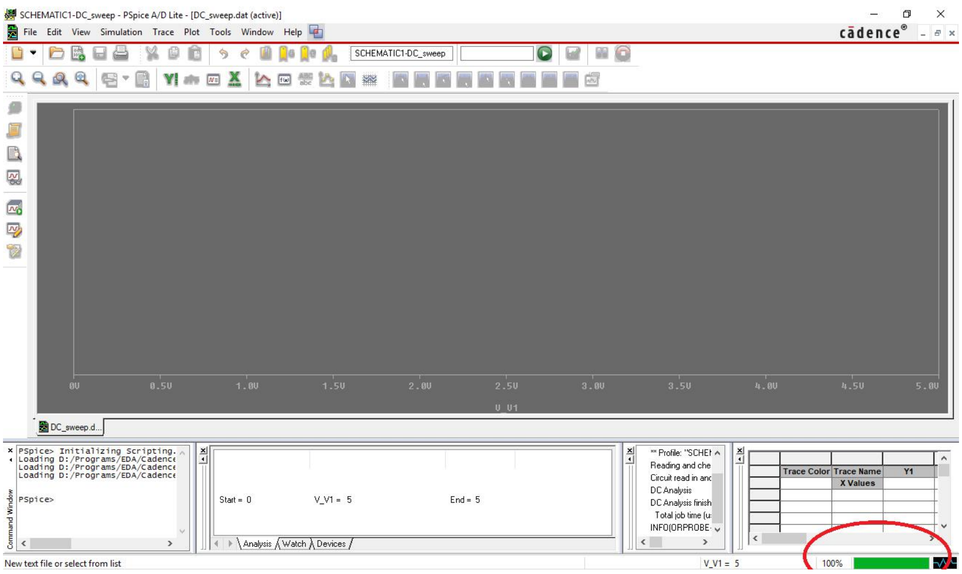
Step 4: configure simulation options as in the picture below then click OK



Picture 4.6. Simulation options for DC sweep

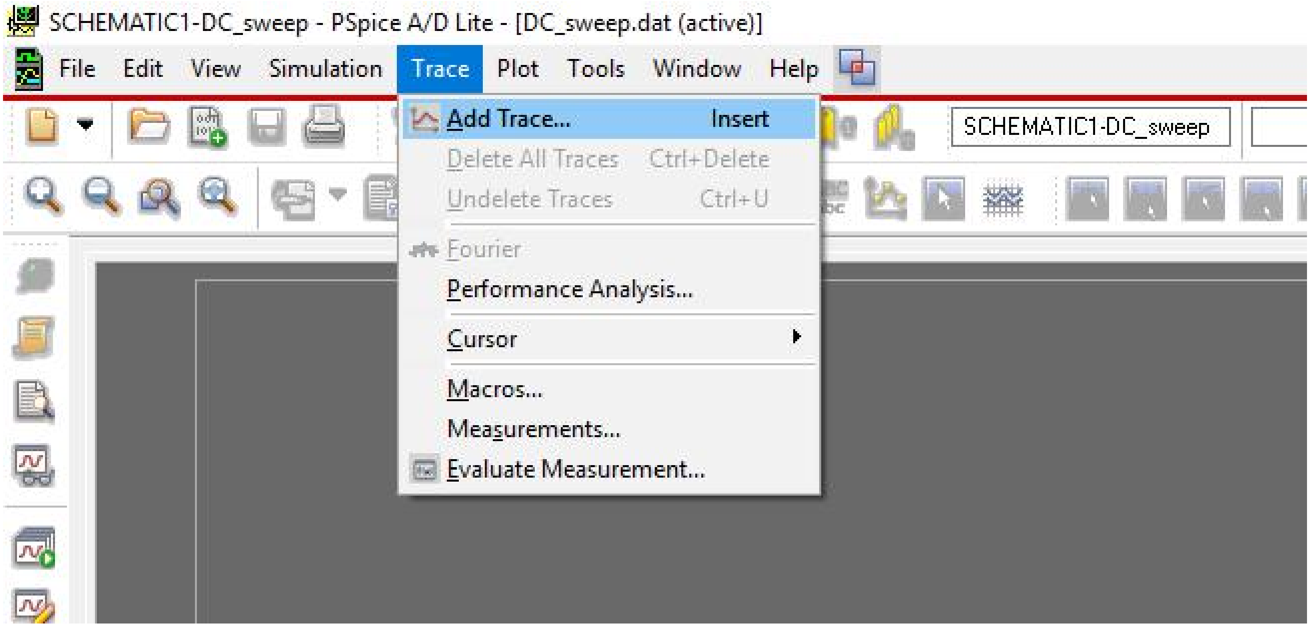
Step 5: Create netlist of the circuit

Step 6: Run simulation (PSpice->Run) and wait for PSpice A/D Lite window shows 100%



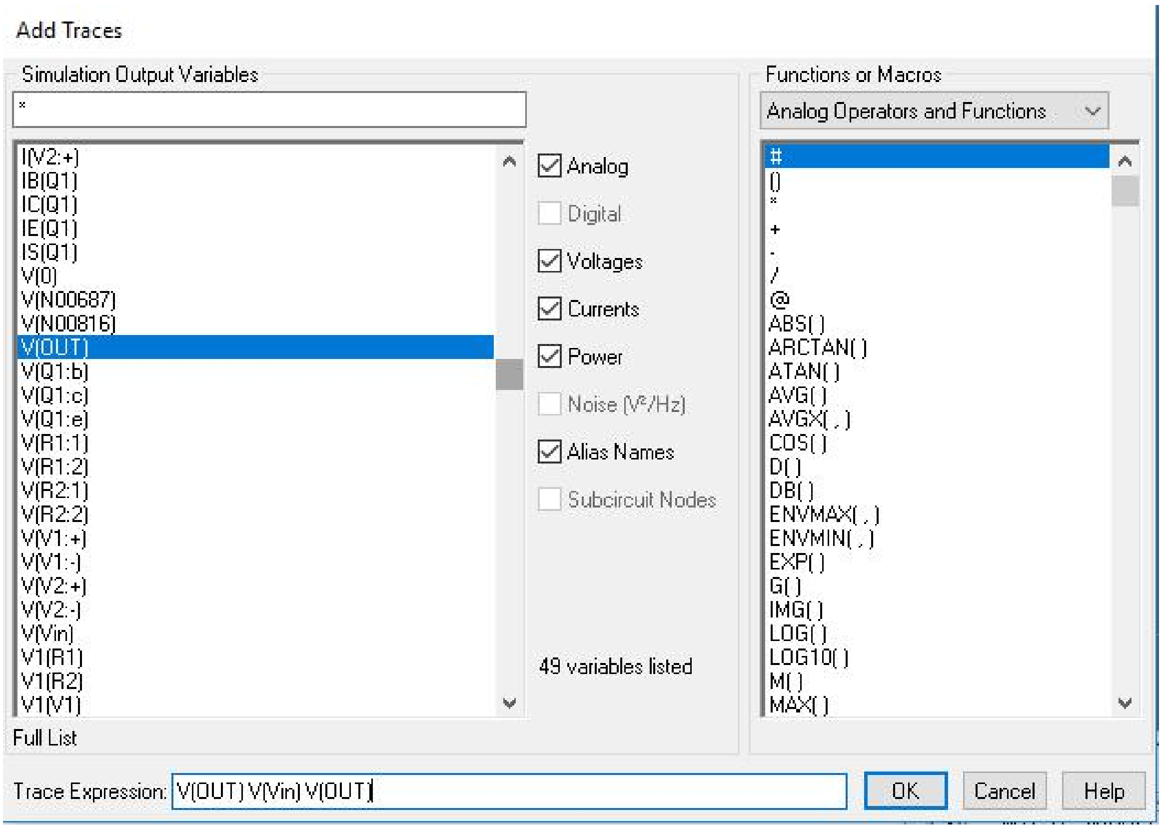
Picture 4.7. PSpice A/D Lite

Step 7: go to Trace->Add Trace…



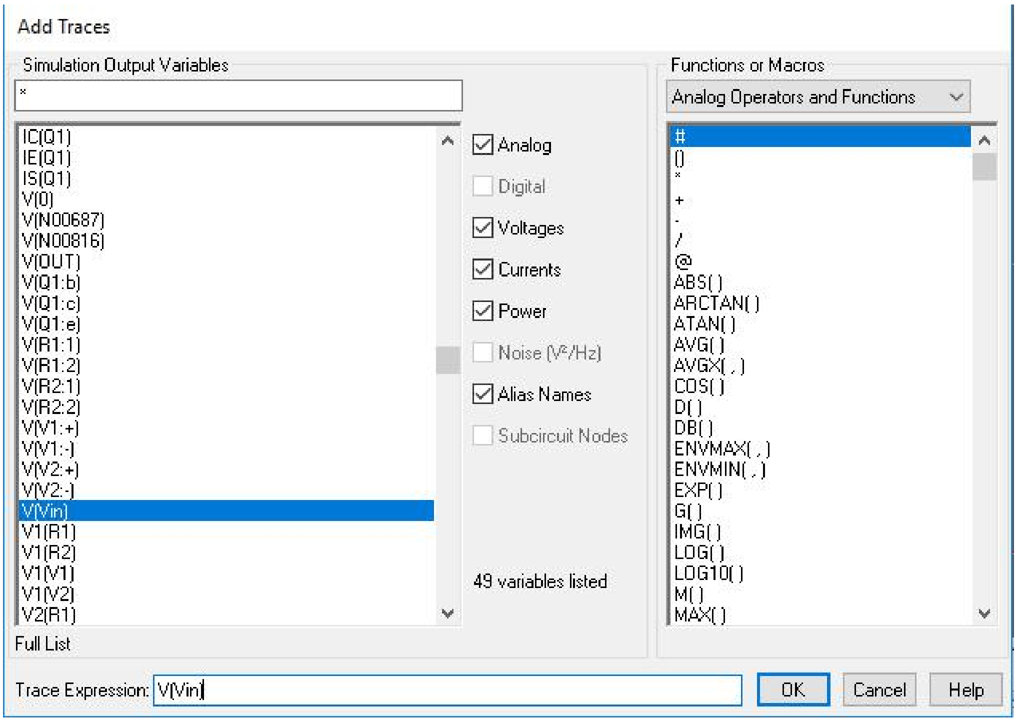
Picture 4.8. Add trace

Step 8: Find and select V(OUT) to plot



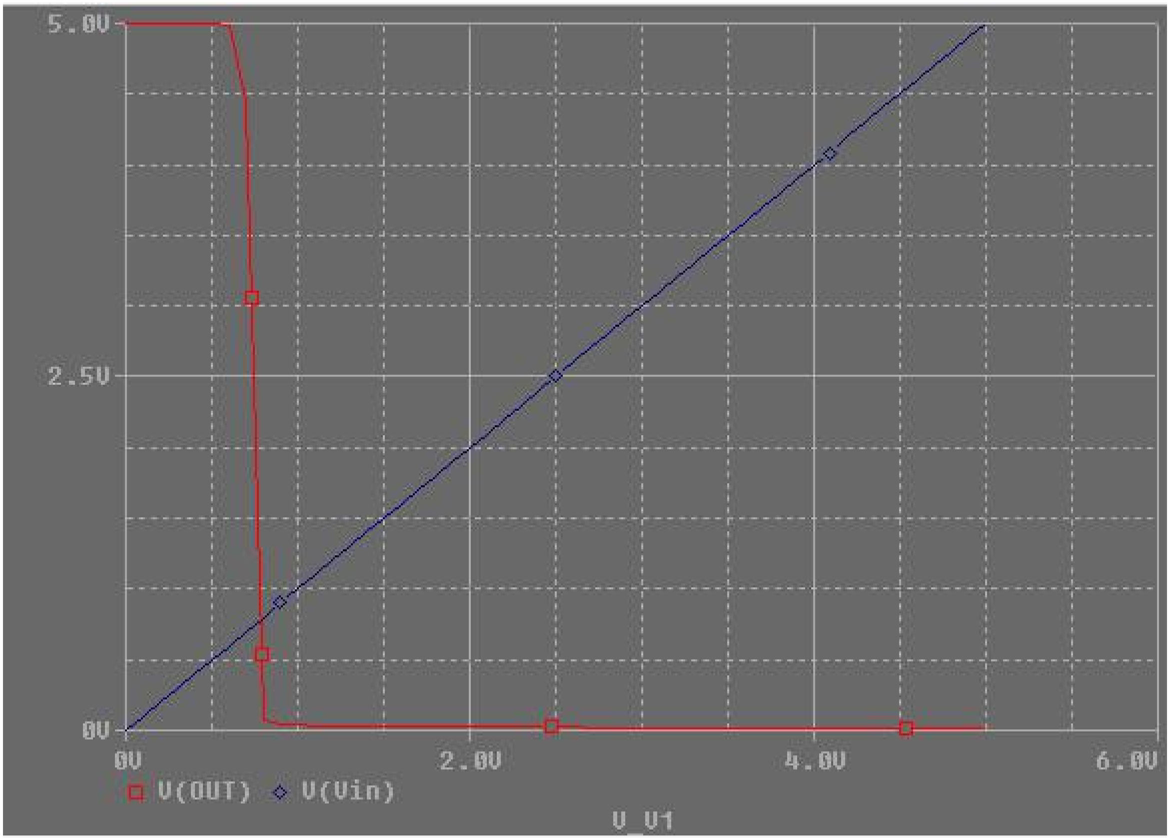
Picture 4.9. Select V(OUT) then click OK

Step 9: do step 8 again to plot V(Vin)



Picture 4.10. Select V(Vin) to plot then click OK

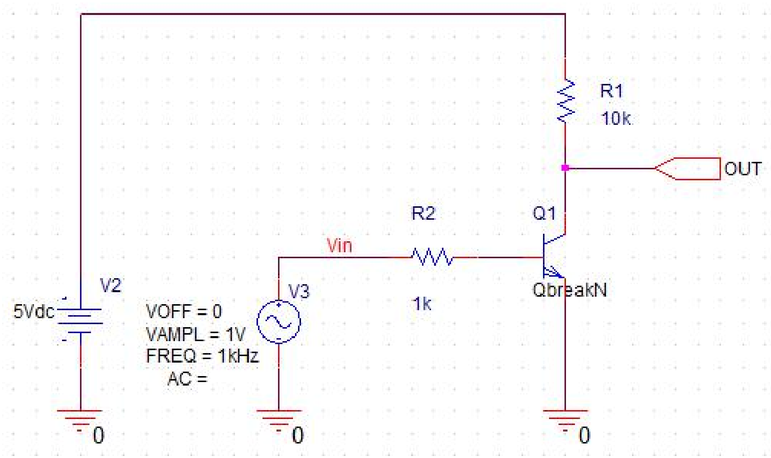
Step 10: result



Picture 4.11. Waveform result

c. Transient

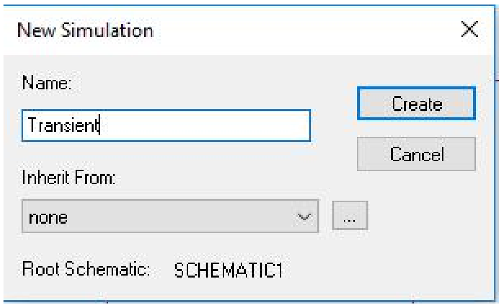
Step 11: change circuit to the circuit below



Picture 4.12. New circuit

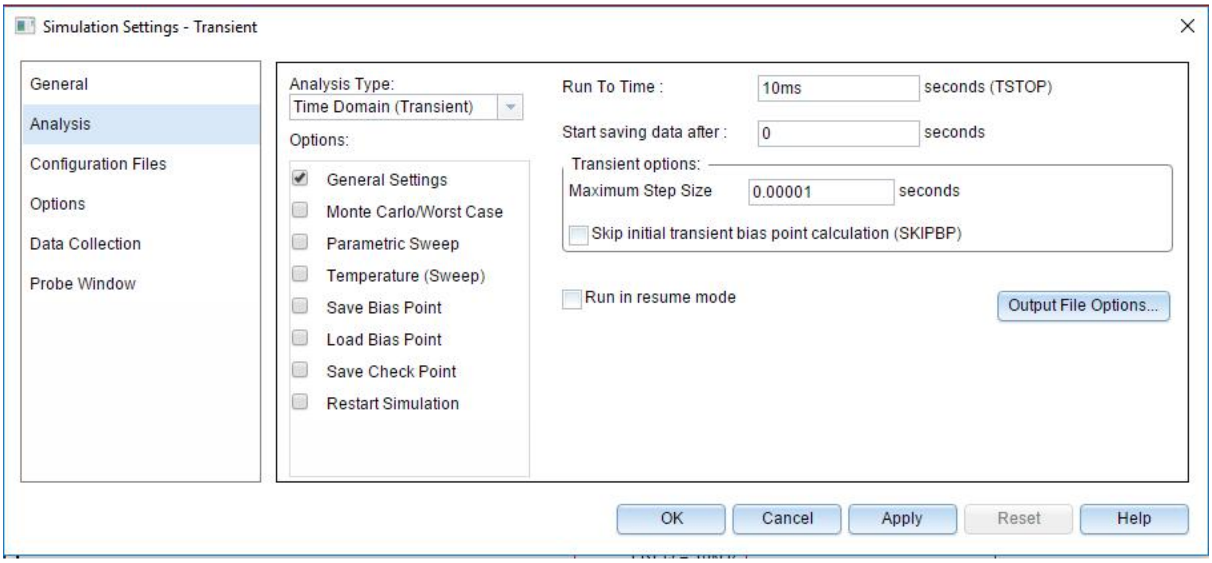
Step 12: select PSpice->New Simulation Profile

Step 13: setup name of simulation



Picture 4.13 Setup name of simulation

Step 14: configure simulation options as in the picture below then click OK



Picture 4.14. Simulation options for transient

Step 15: Repeat from step 5 to step 10 to get result

IV. PREPARATION AT HOME

Simulating the circuit in the picture 4.1 using R = 100 Ω, Diode = 1N4001, DC voltage varies from -10 V to 10 V.

Plot the I curve of the diode.

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 4: DC SWEEP AND TRANSIENT IN PSPICE

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ………………………………………………, **student ID:** ………………..

- name: ………………………………………………, **student ID:** ………………..

TABLE OF RESULTS		
Step 5	Netlist	
Step 10	Find the intersection voltage of waveform	
Step 11	Netlist of new circuit	
Step 15	Drawing Waveforms of new circuit in Transient simulation	

------- END OF REPORT -------

LAB 5

AC SIMULATION AND FREQUENCY RESPONSE IN PSPICE

I. GOAL

In this lab, student will have skills:

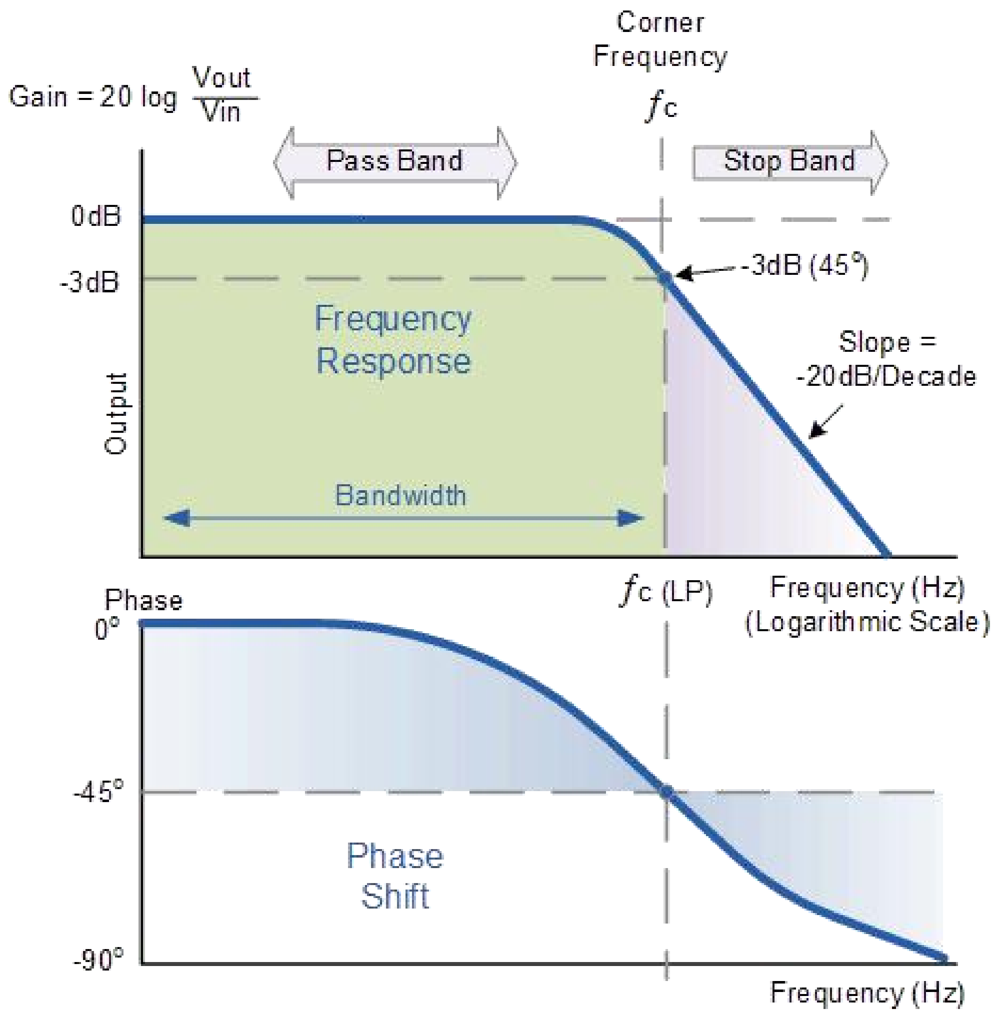
- Simulate frequency response of circuits using AC mode in PSPICE

II. SUMMARY OF THEORY

a. Frequency response

Frequency response is used to analyze dynamic characteristics of a circuit or system. It measures the ratio between output and input when changing frequency of input signal. The result tell us how fast of the circuit or system.

Results of frequency response are usually plotted in Bode-plot as in picture 5.1.

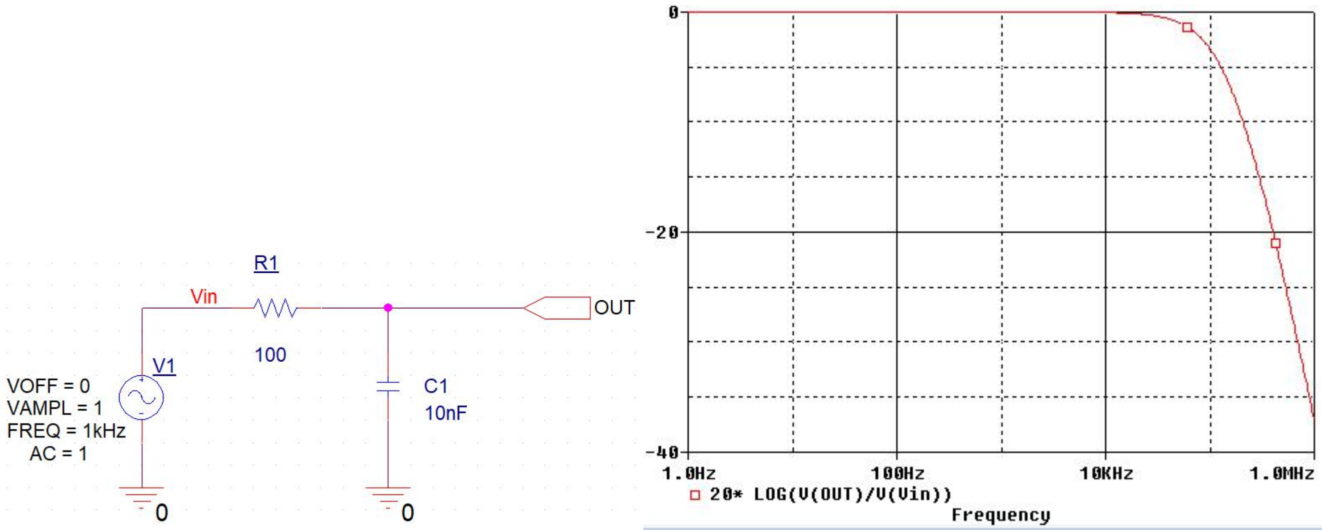


Picture 5.1 Bode plot

In the picture above, **corner frequency** (or **cuttof frequency**) is the frequency of input signal that decreases the ratio line amount of -3dB (or the frequency when the ratio between Vout/Vin decreases amount of $1/\sqrt{2}$).

b. AC simulation

AC simulation in PSPICE is a simulation mode that allows changing frequency of input signals and analyzes characteristics of circuits or system in frequency domain.

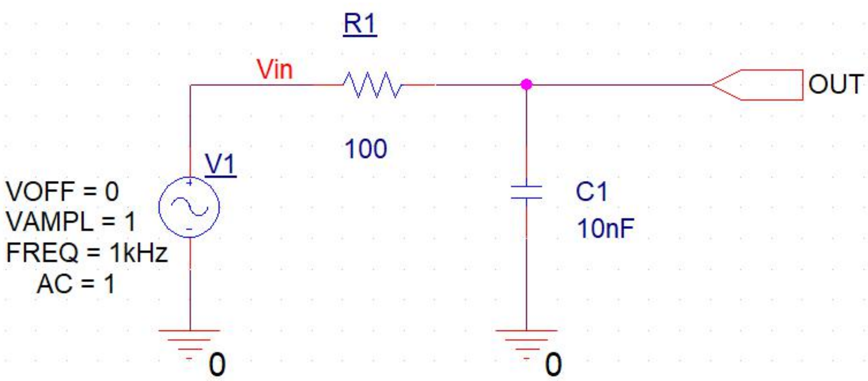


Picture 5.2 Bode plot of a simple lowpass filter citcuit

III. PRACTICE

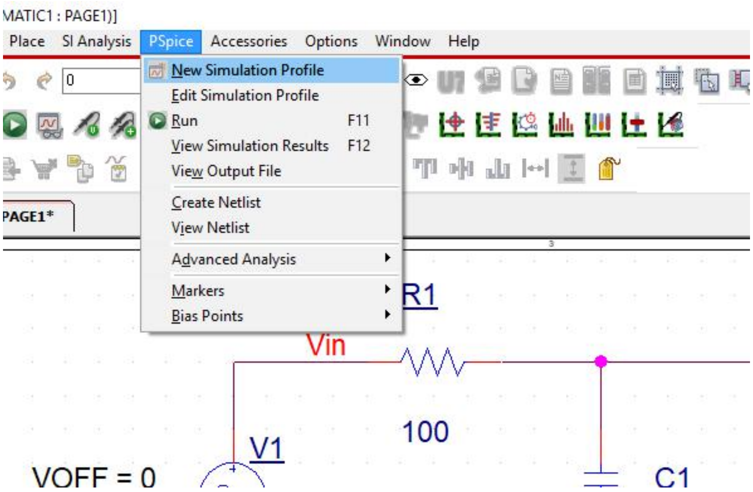
a. Simple Low-pass circuit

Step 1: open Capture CIS Lite and draw a circuit as in the picture 5.3.



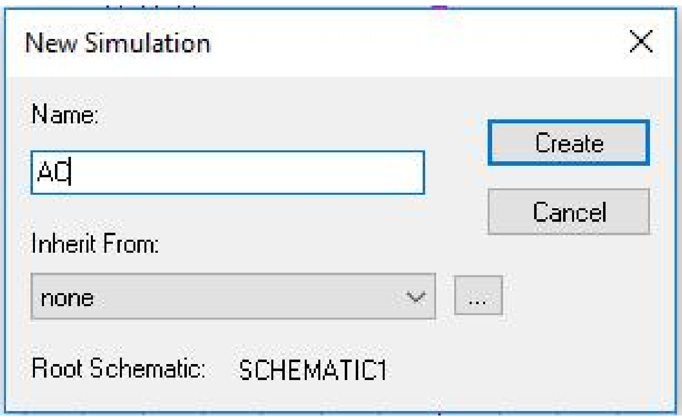
Picture 5.3 Simple low-pass filter circuit

Step 2: select PSpice->New Simulation Profile



Picture 5.4. Setup a simulation

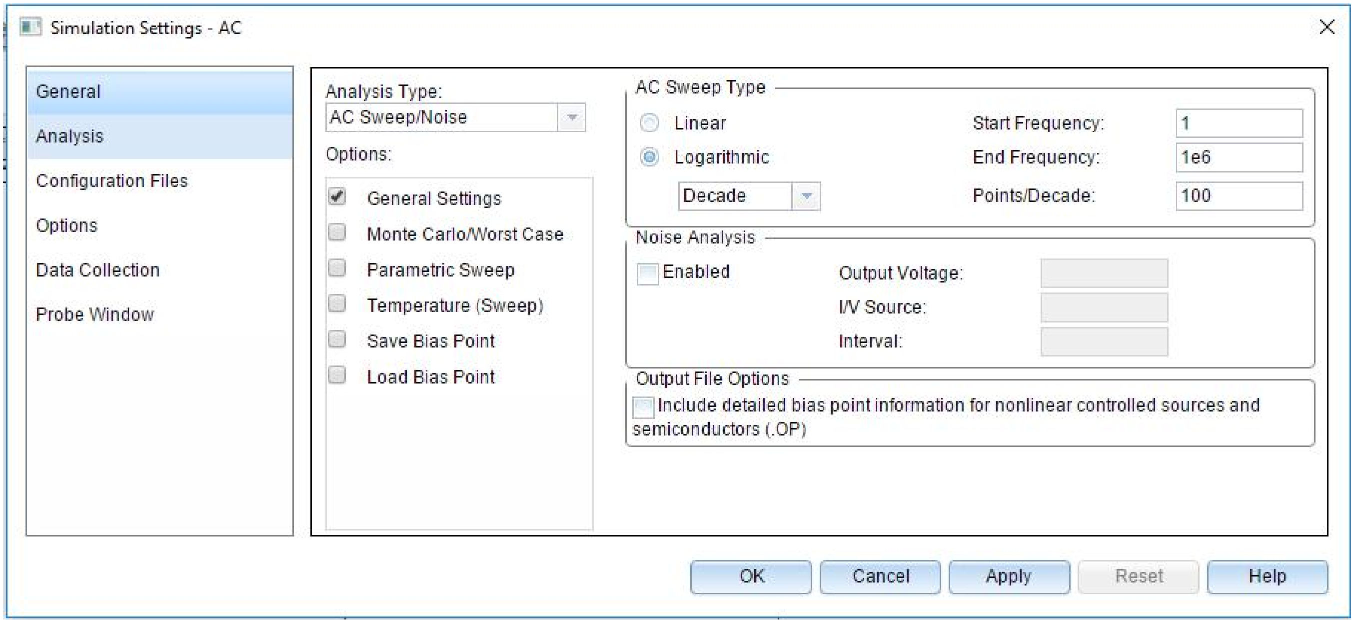
Step 5: Setup a name for simulation profile



Picture 5.5 Setup a name

Step 6: Configure simulation parameters as in picture 5.6

- Analysis Type: AC Sweep
- AC Sweep Type: Logarithmic
- Start Frequency: 1 (it means 1 Hz)
- End Frequency: 1e6 (it means 1×10^6 Hz or 1 MHz)
- Points/Decade: 100 (more points, more accurate but slow simulation time)



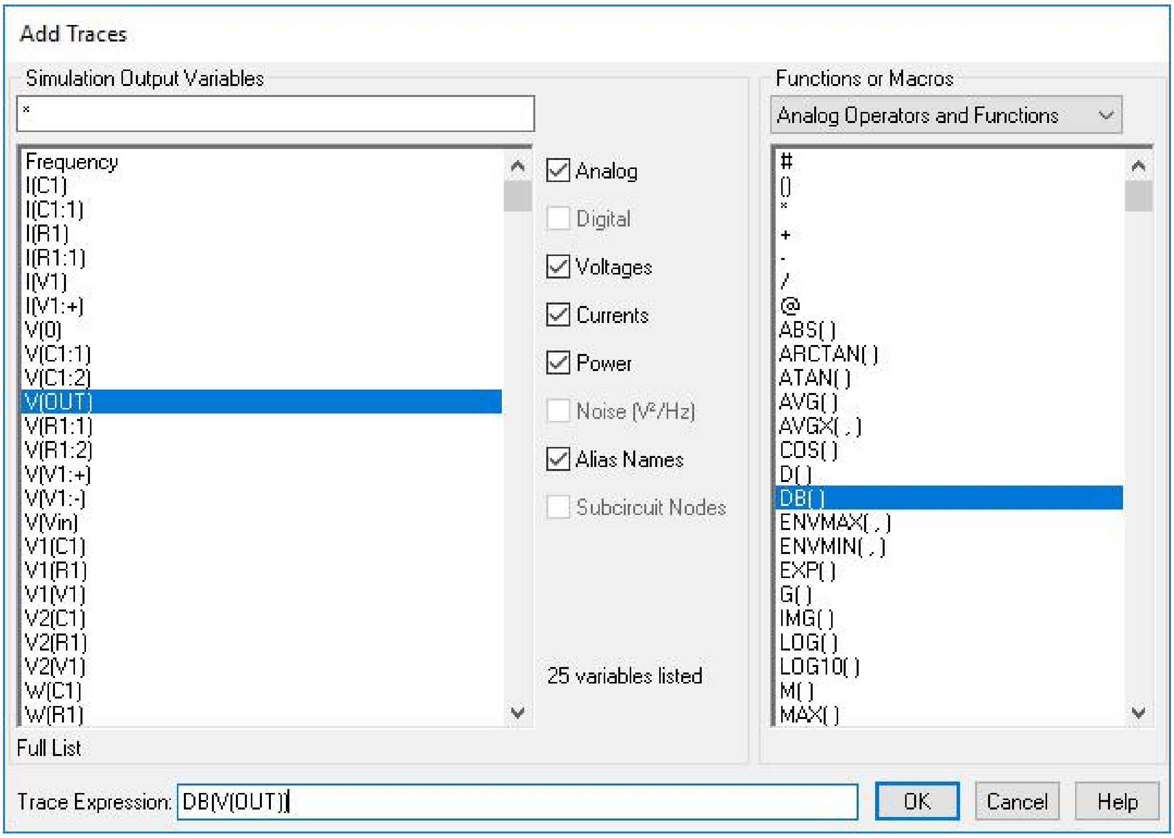
Picture 5.6 Simulation parameters for AC Sweep

Step 7: Create netlist of the circuit

Step 8: Run simulation (PSpice->Run) and wait for PSpice A/D Lite window shows 100%

Step 9: In PSpice A/D Lite, go to Trace->Add Trace…

Step 10: In Add Traces window, type function as in picture 5.7 then click OK



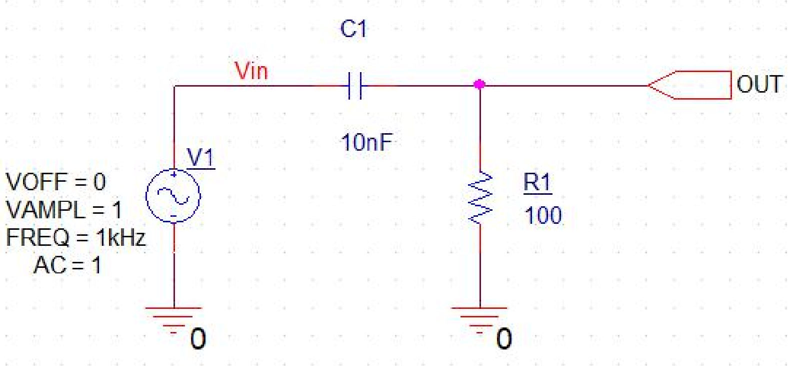
Picture 5.7 Add signal to plot in the function of Decibel (DB)

Step 11: Right click on the plot and select Cursor On to display cursor on plot

Step 12: Using arrow keys to move cursor to -3dB and find out frequency at that position.

b. Simple High-pass circuit

Step 13: Change the schematic to the circuit in picture 5.8.

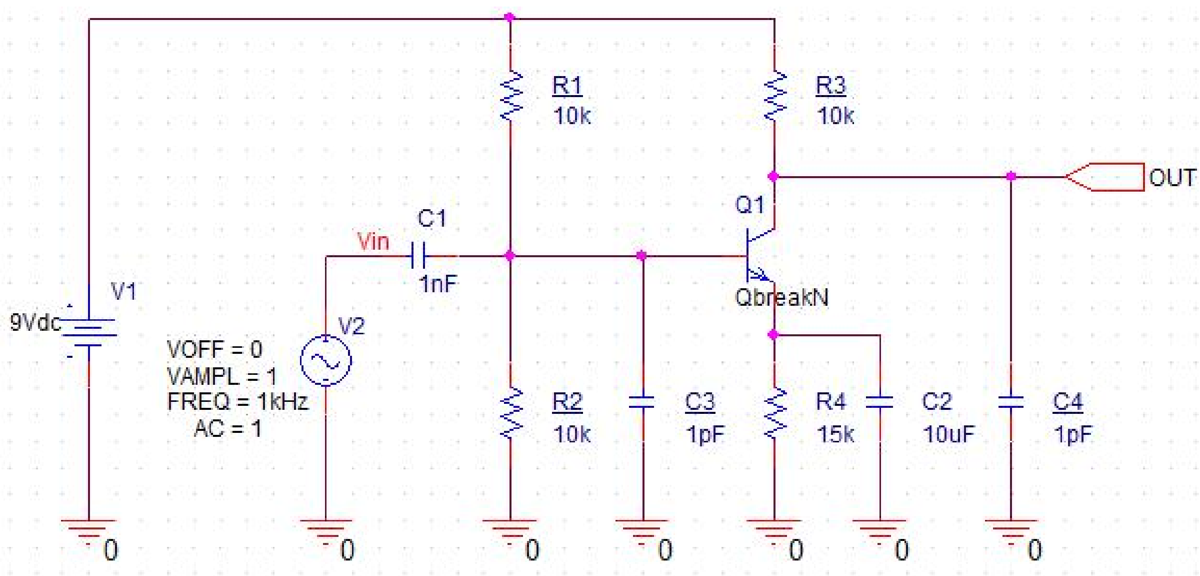


Picture 5.8 Simple High-pass circuit

Step 14: Repeat step 2 to step 12 to find cutoff frequency of this circuit.

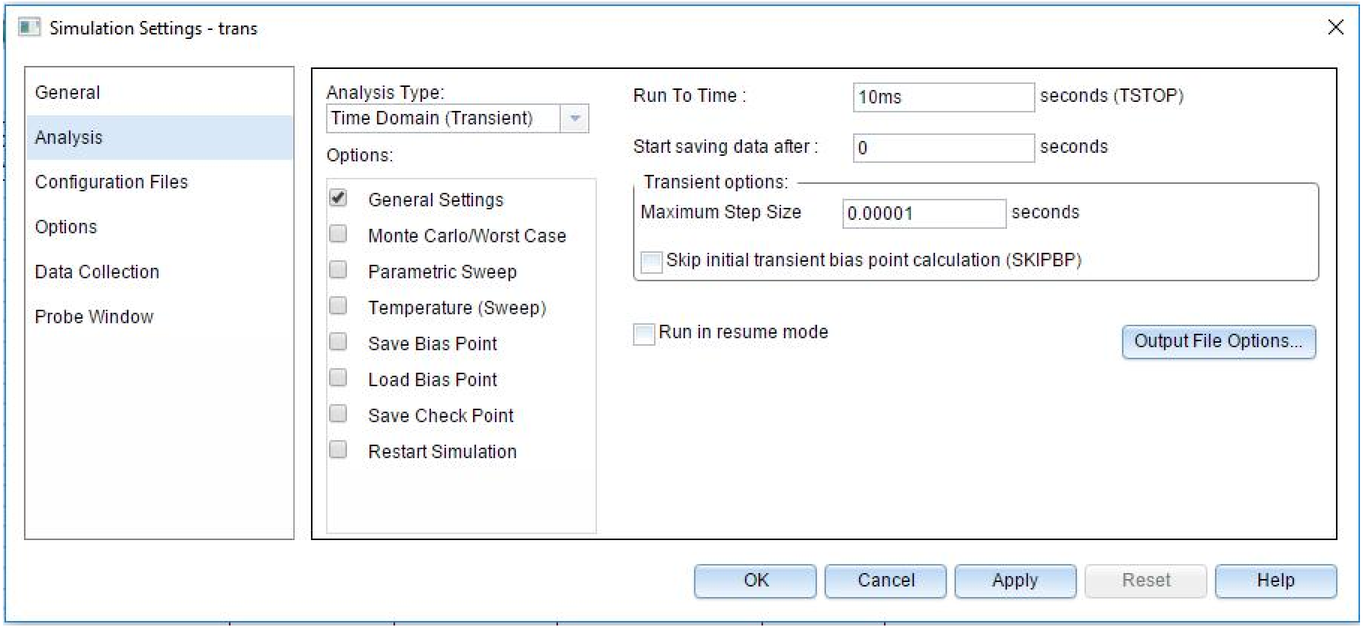
c. Simple BJT Amplifier

Step 15: Create new project for the circuit in picture 5.9.



Picture 5.9 Simple BJT Amplifier

Step 16: Setup Transient simulation profile as in picture 5.10

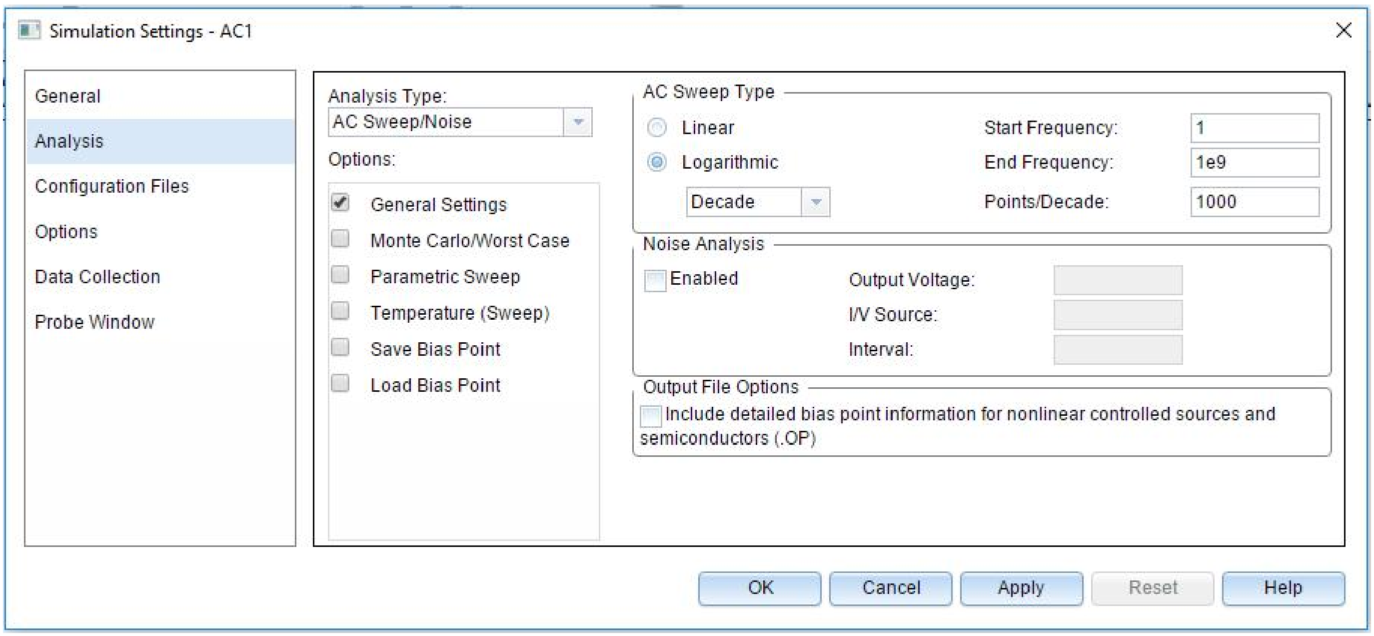


Picture 5.10 Transient simulation parameters

Step 17: Run simulation and plot signals V(OUT) and V(Vin).

Step 18: Repeat step 2 to step 12 using parameters as in picture 5.11 to find cutoff frequency of this circuit.

- Start Frequency: 1 Hz
- End Frequency: 1e9 (it means 1 GHz)
- Points/Decade: 1000



Picture 5.11 AC Sweep parameters

IV. PREPARATION AT HOME

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 5: AC SIMULATION AND FREQUENCY RESPONSE IN PSPICE

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ………………………………………………, **student ID:** ………………..

- name: ………………………………………………, **student ID:** ………………..

TABLE OF RESULTS		
Step 7	Netlist	
Step 11	Drawing Bode plot	Cutoff frequency = ...
Step 15	Netlist of new circuit	
Step 17	Drawing Bode plot	Cutoff frequency = ...

------- END OF REPORT -------

LAB 6

P-N JUNCTION DIODE AND RECTIFIER CIRCUITS

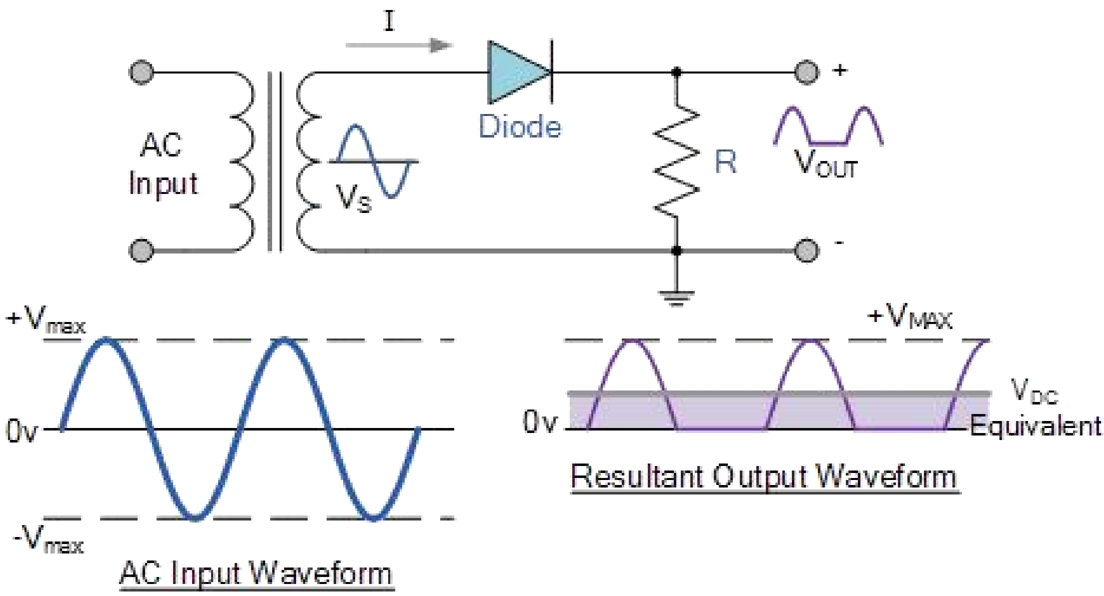
I. GOAL

In this lab, student will have skills:

- Making three basic types of rectifier circuits.
- Calculate characteristics of rectifier circuits.

II. SUMMARY OF THEORY

a. Half-wave Rectifier



Picture 6.1. Half-wave Rectifier

Average DC voltage is calculated from below equations:

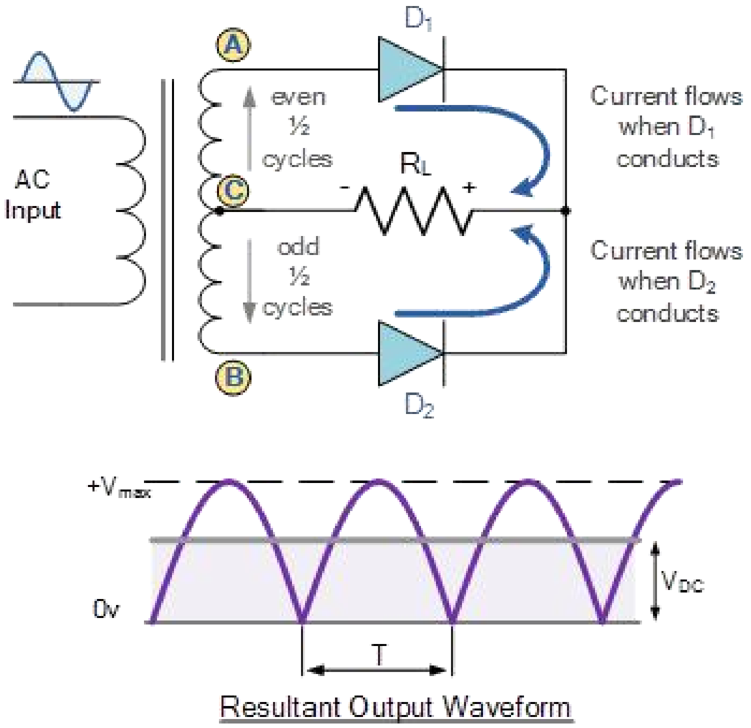
$$V_{dc} = \frac{V_{max}}{\pi} = 0.318 \times V_{max} = 0.45 \times V_S \quad (1)$$

Load current is

$$I_{dc} = \frac{V_{dc}}{R} \quad (2)$$

b. Full-wave Rectifier

i. Two diodes



Picture 6.2. Full-wave Rectifier with two diodes

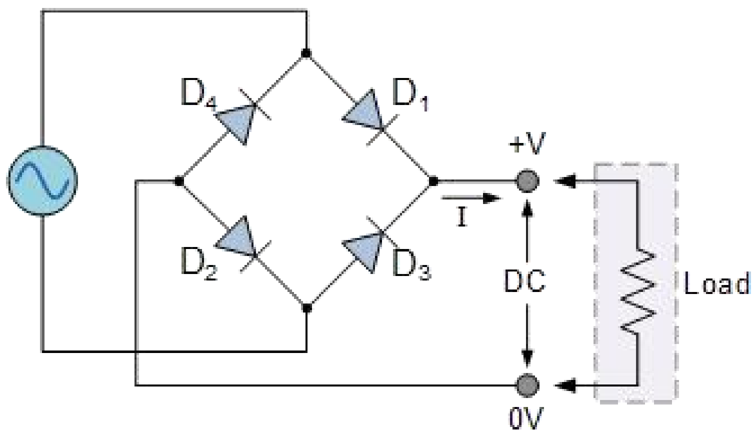
Average DC voltage is calculated from below equations:

$$V_{dc} = \frac{2\times V_{max}}{\pi} = 0.636 \times V_{max} = 0.9 \times V_S \quad (3)$$

Load current is

$$I_{dc} = \frac{V_{dc}}{R} \quad (4)$$

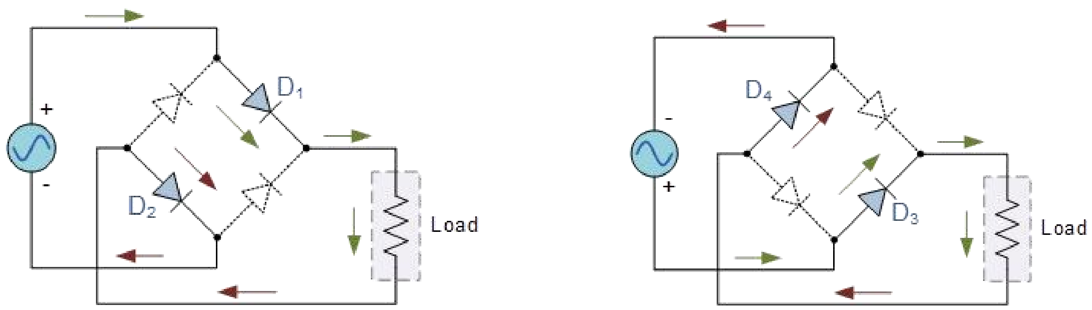
ii. Four diodes (bridge)



Picture 6.3. Full-wave Rectifier with diode bridge

The circuit has two working cycles:

- Positive half-cycle: D3 and D4 open because of reverse bias, D1 and D2 conduct in series
- Negative half-cycle: D1 and D2 open because of reverse bias, D3 and D4 conduct in series



a) Positive cycle *b) Negative cycle*

Picture 6.4. Working cycles of full-wave Rectifier with diode bridge

Equations of this kind of circuit are same with full-wave Rectifier with two diodes.

III. PRACTICE

a. Half-wave Rectifier

Step 1: wiring a circuit as in picture 6.1 with AC voltage is 6VAC-50Hz, RL is 1kΩ.

Step 2: capture waveform on RL using oscilloscope, find Vpp and frequency.

Step 3: measure V_{LDC} using Digital VOM.

Step 4: calculate I_{LDC} on RL.

b. Full-wave Rectifier with two diodes

Step 1: wiring a circuit as in picture 6.2 with AC voltage is 6VAC-50Hz, RL is 1kΩ.

Step 2: capture waveform on RL using oscilloscope, find Vpp and frequency.

Step 3: measure VLDC using Digital VOM.

Step 4: calculate ILDC on RL.

c. Full-wave Rectifier with diode bridge

Step 1: wiring a circuit as in picture 6.3 with AC voltage is 6VAC-50Hz, RL is 1kΩ.

Step 2: capture waveform on RL using oscilloscope, find Vpp and frequency.

Step 3: measure VLDC using Digital VOM.

Step 4: calculate ILDC on RL.

IV. PREPARATION AT HOME

Simulate circuit in picture 6.3 and generate its netlist.

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 6: P-N JUNCTION DIODE AND RECTIFIER CIRCUITS

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ………………………………………………, **student ID:** ………………..

- name: ………………………………………………, **student ID:** ………………..

<table>
<tr><th colspan="3">TABLE OF RESULTS</th></tr>
<tr><td>Preparation at home</td><td colspan="2"></td></tr>
<tr><td rowspan="2">Half-wave Rectifier</td><td>Output waveform</td><td></td></tr>
<tr><td>Voltage</td><td>Vp(secondary) = ………………V_{LDC} = ……………………
R_{Load} = ………………………… I_{LDC} = ...
Frequency = ...</td></tr>
<tr><td rowspan="2">Full-wave Rectifier with two diodes</td><td>Output waveform</td><td></td></tr>
<tr><td>Voltage</td><td>Vp(secondary) = ………………V_{LDC} = ……………………
R_{Load} = ………………………… I_{LDC} = ...
Frequency = ...</td></tr>
<tr><td rowspan="2">Full-wave Rectifier with diode bridge</td><td>Output waveform</td><td></td></tr>
<tr><td>Voltage</td><td>Vp(secondary) = ………………V_{LDC} = ……………………
R_{Load} = ………………………… I_{LDC} = ...
Frequency = ...</td></tr>
</table>

------- END OF REPORT -------

LAB 7

RECTIFIER CIRCUIT WITH CAPACITOR FILTER

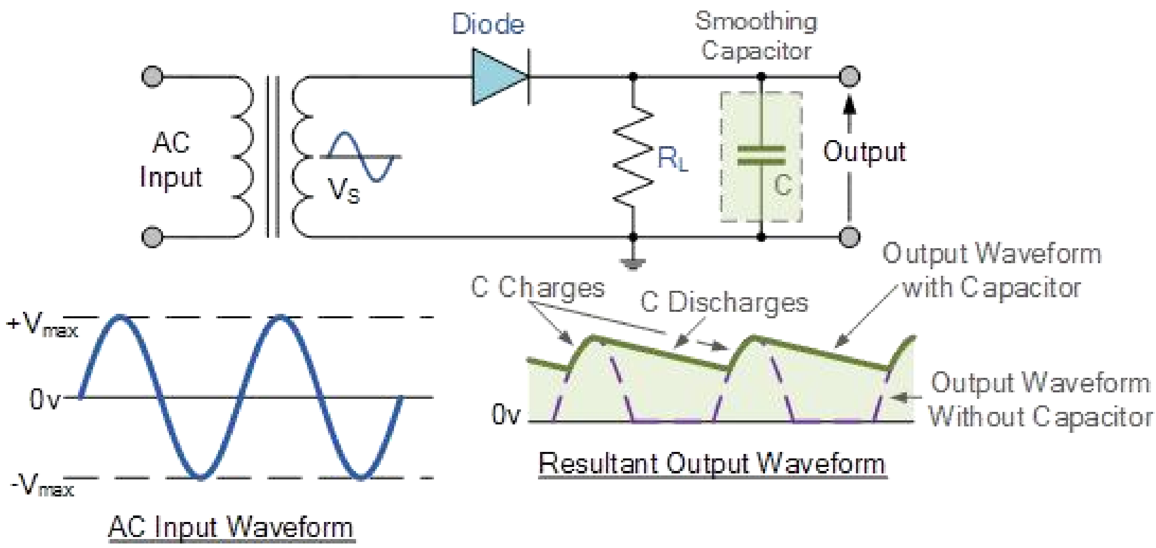
I. GOAL

In this lab, student will have skills:

- Making basic types of rectifier circuits with capacitor filter (smoothing capacitor).
- Calculate characteristics of rectifier circuits with capacitor filter.

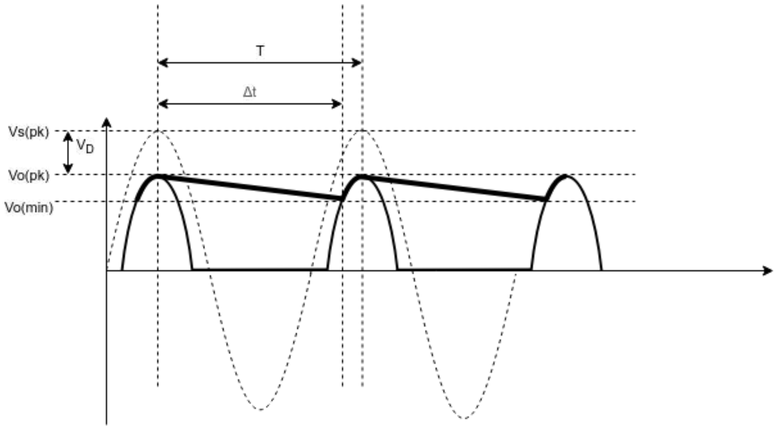
II. SUMMARY OF THEORY

a. Half-wave Rectifier



Picture 7.1. Half-wave Rectifier with smoothing capacitor

To simplify, the charging time of capacitor goes to 0 because of very large $R_L C$ time constant. The maximum voltage on the capacitor after charging is $V_{Cmax} = V_{o(pk)} = V_{max} - V_D$. After that, this voltage is slowly discharged to $V_{Cmin} = V_{o(min)}$ through R_L in a period of discharging Δt.



Picture 7.2. Waveform

$$V_{dc} = \frac{V_{Cmax}+V_{Cmin}}{2} = \frac{V_{o(pk)}+V_{o(min)}}{2} \tag{5}$$

$$V_{Cmin} = V_C(\Delta t) = V_{o(pk)}e^{-\frac{\Delta t}{RC}} \approx V_{o(pk)}\left(1-\frac{\Delta t}{R_LC}\right) \tag{6}$$

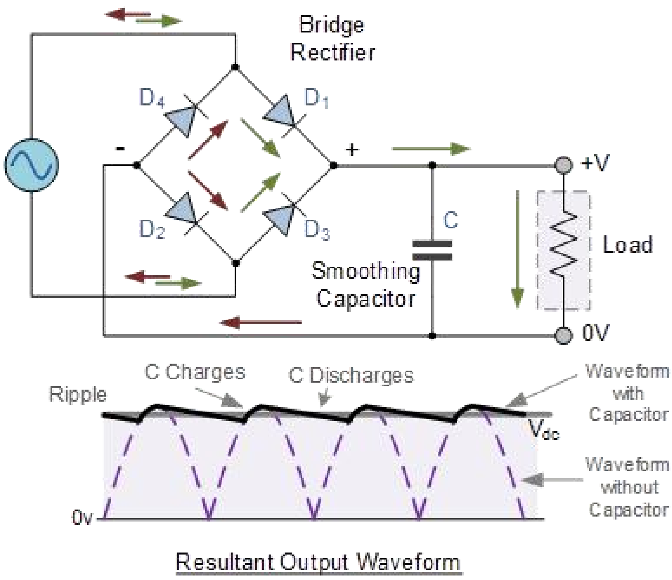
$$V_{ripple(pk-pk)} = V_{Cmax} - V_{Cmin} \tag{7}$$

$$V_{rp} = \frac{V_{ripple(pk-pk)}}{2} = \frac{V_{o(pk)}}{2fR_LC} \tag{8}$$

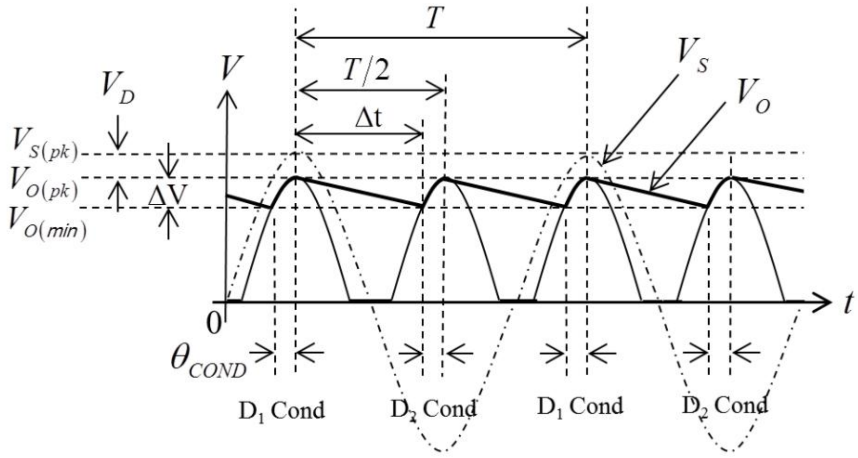
$$V_{dc} \approx V_{o(pk)}\left(1-\frac{\Delta t}{2R_LC}\right) \tag{9}$$

$$\approx V_{o(pk)}\left(1-\frac{1}{2fR_LC}\right) \tag{10}$$

b. Full-wave Rectifier with diode bridge



Picture 7.3. Full-wave Rectifier with smoothing capacitor



Picture 7.4. Waveform

$$V_{dc} = \frac{V_{Cmax}+V_{Cmin}}{2} = \frac{V_{o(pk)}+V_{o(min)}}{2} \tag{11}$$

$$V_{Cmin} = V_C(\Delta t) = V_{o(pk)}e^{-\frac{\Delta t}{RC}} \approx V_{o(pk)}\left(1-\frac{\Delta t}{R_L C}\right) \tag{12}$$

$$V_{ripple(pk-pk)} = V_{Cmax} - V_{Cmin} \tag{13}$$

$$V_{rp} = \frac{V_{ripple(pk-pk)}}{2} = \frac{V_{o(pk)}}{4fR_L C} \tag{14}$$

$$V_{dc} \approx V_{o(pk)}\left(1-\frac{\Delta t}{4R_L C}\right) \tag{15}$$

$$\approx V_{o(pk)}\left(1-\frac{1}{4fR_L C}\right) \tag{16}$$

c. Ripple factor

Because of large smoothing capacitor, the discharge curve can be considered as linear. The effective ripple voltage (full-wave) follows equation:

$$V_{rp(eff)} = \frac{V_{rp}}{\sqrt{3}} = \frac{V_p}{\sqrt{3}(4fR_L C)} \tag{17}$$

Ripple factor:

$$r = \frac{V_{rp(eff)}}{V_{dc}} = \frac{V_{o(pk)}}{\sqrt{3}(4fR_L C)V_{dc}} \tag{18}$$

The smaller *r*, the higher filter quality.

VI. PRACTICE

a. Half-wave Rectifier

Step 1: wiring a circuit as in picture 7.1 with AC voltage is 6VAC-50Hz, RL is 1kΩ.

Step 2: use a 100µF capacitor to the circuit.

Step 3: capture waveform on RL using oscilloscope.

Step 4: measure V_{LDC} using Digital VOM.

Step 5: calculate I_{LDC} on RL.

Step 6: calculate Vrp.

b. Full-wave Rectifier with diode bridge

Step 1: wiring a circuit as in picture 7.3 with AC voltage is 6VAC-50Hz, RL is 1kΩ.

Step 2: use a 100µF capacitor to the circuit.

Step 3: capture waveform on RL using oscilloscope.

Step 4: measure VLDC using Digital VOM.

Step 5: calculate ILDC on RL.

Step 6: calculate Vrp and ripple factor.

VII. PREPARATION AT HOME

Simulate circuit in picture 7.3 and generate its netlist.

VIII. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 7: RECTIFIER CIRCUIT WITH CAPACITOR FILTER

Date:……………………… Time: ………………………

Class: ……............................ *** Session:** .... *** Group:** ……………..

Members: - name: …………………………………………………, **student ID:** ………………..

- name: …………………………………………………, **student ID:** ………………..

<table>
<tr><th colspan="3">TABLE OF RESULTS</th></tr>
<tr><td>Preparation at home</td><td colspan="2"></td></tr>
<tr><td rowspan="2">Half-wave Rectifier</td><td>Output waveform</td><td></td></tr>
<tr><td>Voltage</td><td>V_{LDC} = …………… R_{Load} =……………. I_{LDC} = …………
C filter = …………………… Vripple = …………………...</td></tr>
<tr><td rowspan="2">Full-wave Rectifier</td><td>Output waveform</td><td></td></tr>
<tr><td>Voltage</td><td>V_{LDC} = …………… R_{Load} =……………. I_{LDC} = …………
C filter = …………………… Vripple = …………………...</td></tr>
</table>

------- END OF REPORT -------

LAB 8
ZENER DIODE AND DC VOLTAGE REGULATOR

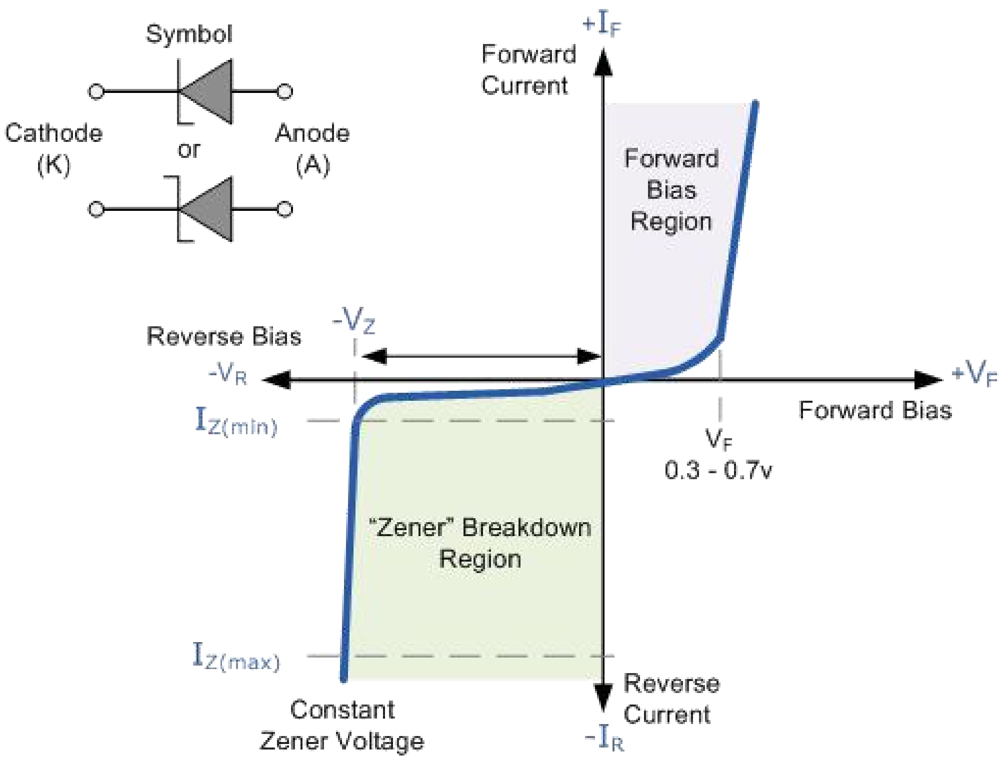
I. GOAL

In this lab, student will have skills:

- Calculating and making basic Zener diode circuit.
- Designing a simple DC voltage regulator using Zener diode.

II. SUMMARY OF THEORY

a. Zener Diode



Picture 8.1. Voltage-Ampere characteristic of zener diode

Operation conditions:

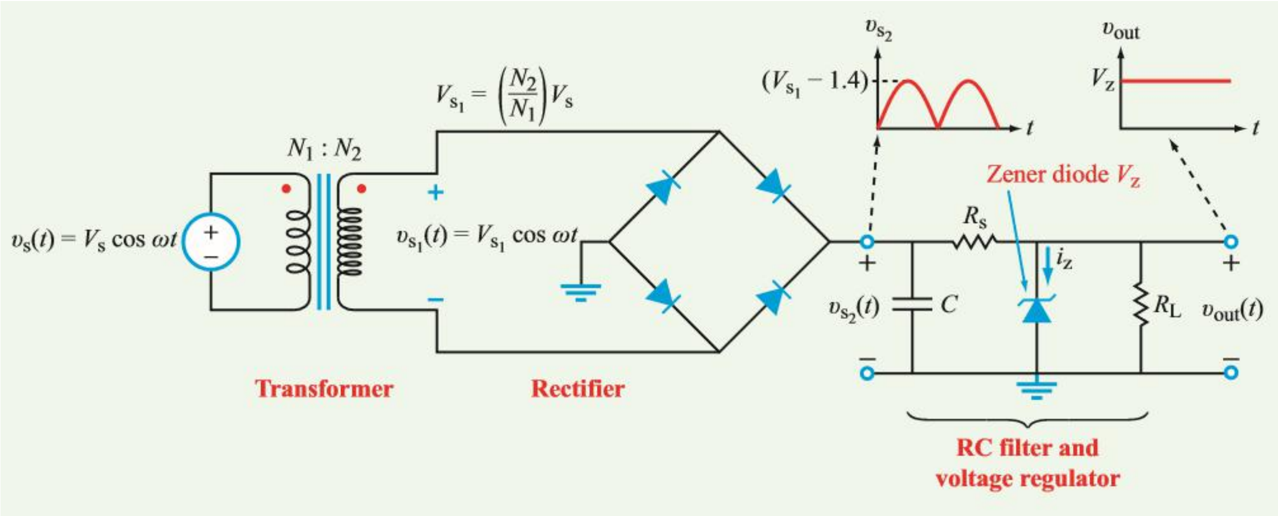
- Input DC voltage must be greater than Zener voltage:

$$V_{iDC} > V_Z$$

- Current going through the diode must be:

$$I_{Z(min)} < I_Z < I_{Z(max)}$$

b. DC Voltage Regulator



Picture 8.2. DC Voltage Regulator using zener diode

In the circuit on Picture 8.2, Rs is limited current resistor for zener diode circuit. The DC input voltage is:

$$V_{iDC} = V_2 = V_1 - 1.4$$

We have the following equations:

$$V_{LDC} = V_Z$$

$$I_S = I_Z + I_L$$

$$I_L = \frac{V_{LDC}}{R_L}$$

$$I_S = \frac{V_{LDC} - V_Z}{R_S}$$

$$I_Z = I_S - I_L$$

To ensure the operation of zener circuit and protect it from damage ($I_Z > I_{Z(max)}$), value of Rs must be in the range as equation below:

$$\frac{V_{iDC(max)} - V_Z}{I_{Z(max)} + I_{L(min)}} \leq R_S \leq \frac{V_{iDC(\min)} - V_Z}{I_{Z(min)} + I_{L(max)}}$$

In the case there is no R_L (I_L=0A), I_Z will be equal to I_S, therefore:

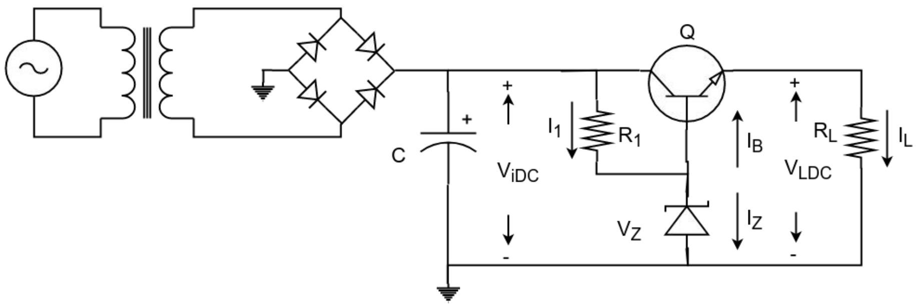
$$I_Z = I_S < I_{Z(max)}$$

Power dissipations will be:

$$P_Z = V_Z \times I_Z < P_{Z(max)}$$

$$P_{Rs} = R_s \times I_S^2$$

c. Transistor Series Voltage Regulator



Picture 6.3. Transistor Series Voltage Regulator

Equations:

$$V_{LDC} = V_Z - V_{BE}$$
$$I_1 = I_Z + I_B$$
$$I_1 = \frac{V_{iDC} - V_Z}{R_1}$$
$$I_L = \frac{V_{LDC}}{R_L}$$
$$I_B = \frac{I_E}{\mathrm{B}+1} = \frac{I_L}{\mathrm{B}+1}$$
$$I_Z = I_1 - I_B = I_1 - \frac{I_L}{\mathrm{B}+1}$$

Power dissipations:

$$P_D = V_{CEQ} \times I_{CQ} = (V_{iDC} - V_{LDC}) \times I_{CQ}$$
$$= (V_{iDC} - V_{LDC}) \times \mathrm{B} I_B = (V_{iDC} - V_{LDC}) \times \frac{\mathrm{B} I_E}{\mathrm{B}+1}$$
$$= (V_{iDC} - V_{LDC}) \times I_L < P_{D(max)}$$
$$P_Z = V_Z \times I_Z < P_{Z(max)}$$
$$P_{R1} = R_1 \times I_1^2$$

Output impedance:

$$R_o = \frac{V_O}{I_O}\bigg|_{V_i \to 0, R_L \to \propto} = \frac{r_2}{\mathrm{B}} + r_e = r_e$$

III. PRACTICE

a. DC Voltage Regulator

Step 1: calculating Rs value for circuit as in picture 8.2 using Zerner 3.3 V / 1 W, C = 1000 uF and RL = 390 Ω.

Step 2: wiring circuit in step 1.

Step 3: measuring VLDC.

Step 4: changing RL to 39 Ω and measuring VLDC again.

b. Transistor Series Voltage Regulator

Step 5: wiring circuit in picture 6.3 using Zerner 3.3 V / 1 W, C = 1000 uF, R1 = 1 kΩ, RL = 390 Ω and BJT is 2N2222A

Step 6: measuring VLDC.

Step 7: changing RL to 39 Ω and measuring VLDC again.

IV. PREPARATION AT HOME

Simulating circuit in picture 8.3 and generating its netlist.

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 8: ZENER DIODE AND DC VOLTAGE REGULATOR

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ………………………………………………, **student ID:** ………………..

- name: ………………………………………………, **student ID:** ………………..

<table>
<tr><th colspan="3">TABLE OF RESULTS</th></tr>
<tr><td>Preparation at home</td><td colspan="2"></td></tr>
<tr><td rowspan="2">DC Voltage Regulator using zener diode</td><td>Schematic</td><td></td></tr>
<tr><td>Measurement</td><td>V_{iDC} = ……..…. I_Z = ………… R_S = …..……. I_S = ………
$V_{LDC(cal)}$ = ………………… $V_{LDC(real)}$ = …………………
R_{LOAD} = …………………… I_{LDC} = ……………………</td></tr>
<tr><td rowspan="2">Transistor Series Voltage Regulator</td><td>Schematic</td><td></td></tr>
<tr><td>Measurement</td><td>V_{iDC} = ……..…. I_Z = ………… R_1 = …..……. I_1 = ………
$V_{LDC(cal)}$ = ………………… $V_{LDC(real)}$ = …………………
R_{LOAD} = ………… I_{LDC} = ……..… I_B = ……… h_{FE} = ……</td></tr>
</table>

------- END OF REPORT -------

LAB 9

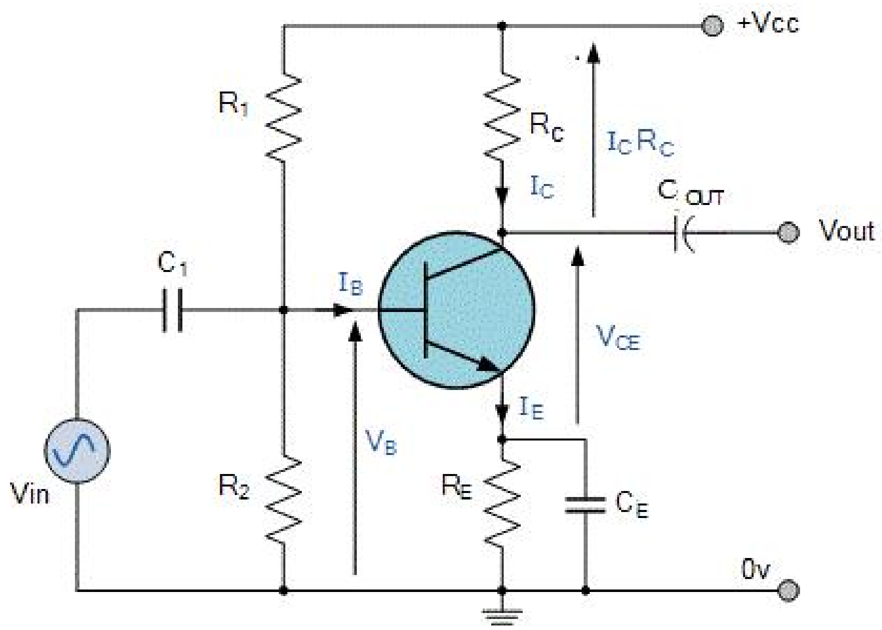
BJT TRANSISTOR AND SMALL SIGNAL AMPLIFIER

I. GOAL

In this lab, student will have skills:

- Examining operation of BJT transistor.
- Investigating AC characteristics of BJT transistor in a small signal amplifier circuit.

II. SUMMARY OF THEORY



Picture 9.1. Small signal amplifier using BJT transistor in common Emitter mode

DC characteristics:

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

$$R_B = \frac{R_1 R_2}{R_1 + R_2}$$

$$I_B = \frac{V_{BB} - V_{BE}}{R_B + (\mathrm{B} + 1) R_E}$$

$$I_C = \beta I_B$$

$$V_{CE} = V_{CC} - (R_C + R_E) I_C$$

Temperature stabilized factor:

$$S_i = (1 + \mathrm{B}) \times \frac{R_B + R_E}{R_B + (1 + \mathrm{B})R_E} \approx 1 + \frac{R_B}{R_E}$$

Small signal characteristics (with bypass capacitor CE):

- Input impedance from base:

$$Z_i = \frac{dV_{be}}{dI_b} = h_{ie}$$

- Input impedance from source of signal:

$$Z_{iS} = \frac{dV_i}{dI_i} = \frac{R_B h_{ie}}{R_B + h_{ie}}$$

- Voltage gain:

$$A_v = -\frac{h_{fe} Z_C}{h_{ie}} = -\frac{h_{fe} R_C}{h_{ie}}$$

- Current gain:

$$A_i = \frac{I_c}{I_b} = h_{fe}$$

$$A_{iB} = \frac{I_c}{I_i} = A_i \times \frac{R_B}{R_B + h_{ie}}$$

- Output impedance:

$$Z_O = \frac{V_o}{I_o}\bigg|_{V_s=0,\ Z_{L\to\infty}} = \frac{1}{h_{oe}} \approx \infty$$

$$Z_o = r_o || R_C \approx R_C$$

III. PRACTICE

Step 1: wiring circuit as in picture 9.1 using 2N2222A, R1 = R2 = 10 kΩ, RC = 1.5 kΩ, RE = 1 kΩ, C1 = Cout = 10 uF, CE = 47 uF

Step 2: measuring VBB, VBE and VCE.

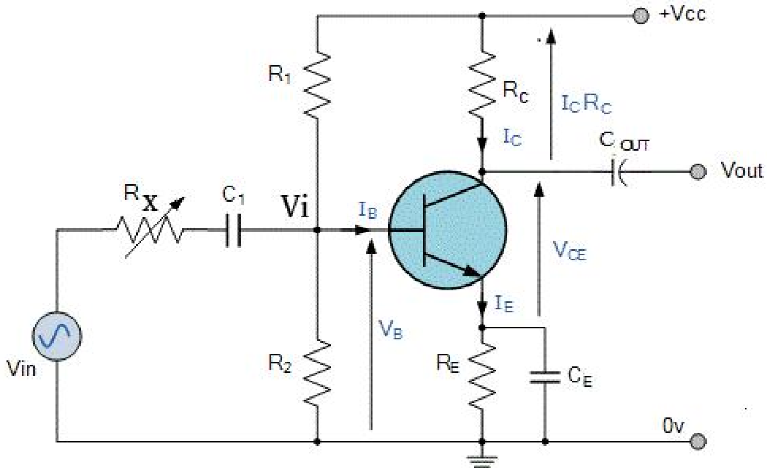
Step 3: applying a sine wave 100 mV Vpp - 1 kHz to C1.

Step 4: measuring output waveform using oscilloscope.

Step 5: calculating voltage gain.

Step 6: changing frequency of input signal until voltage gain decreases 3 dB.

Step 7: adding potentiometer Rx as in picture 9.2 below



Picture 9.2. Adding Rx to measure input impedance.

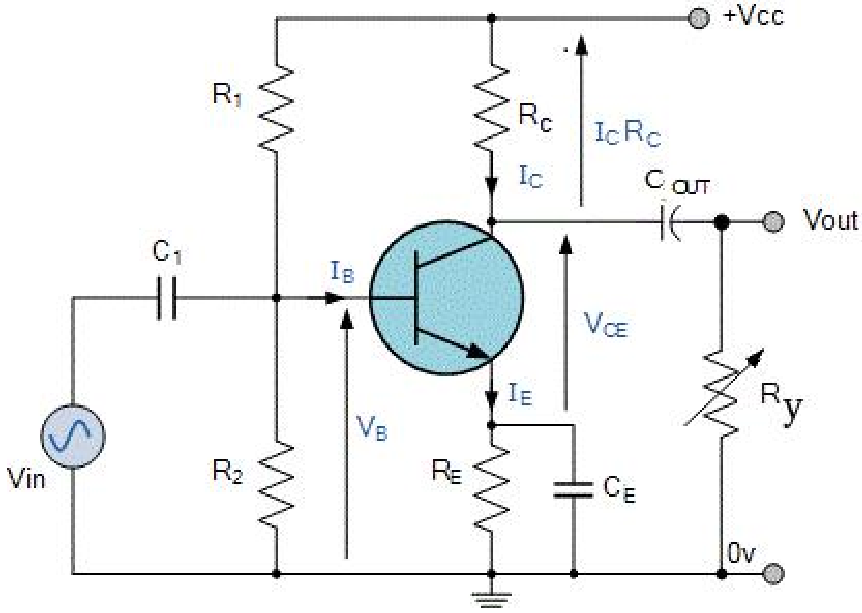
Step 8: applying a sine wave 100 mV Vpp - 1 kHz to Rx.

Step 9: tuning Rx until Vi = Vin / 2.

Step 10: removing Rx out of the circuit.

Step 11: measure Rx value.

Step 12: adding potentiometer Ry as in picture 9.3 below



Picture 9.3. Adding Ry to measure output impedance.

Step 13: applying a sine wave 100 mV Vpp - 1 kHz to C1.

Step 14: tuning Ry until output voltage decreases a half of output value when there is no Ry in the circuit.

Step 15: removing Ry out of the circuit.

Step 16: measure Ry value.

Step 17: removing Ry and CE from the circuit.

Step 18: repeat step 4 and 5.

IV. PREPARATION AT HOME

Generating netlist of the circuit in picture 9.2

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 9: BJT TRANSISTOR AND SMALL SIGNAL AMPLIFIER

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: …………………………………………………, **student ID:** ………………..

- name: …………………………………………………, **student ID:** ………………..

TABLE OF RESULTS	
Preparation at home	
Step 2	$V_{BB(cal)}$ = ……………. $V_{BB(real)}$ = …………….$V_{BE(real)}$ = ………………… $I_{B(cal)}$ = ……………. $I_{C(cal)}$ = …………… $V_{CE(cal)}$ = ……………. $V_{CE(real)}$ = ……………
Step 4	Output waveform
Step 5	$V_{pp(output)}$ = ……………. $A_{Vo/Vi}$ = ……………
Step 6	F_{Cutoff} = …………….
Step 11 **Step 16**	Input impedance = ... Output impedance = ...
Step 18	Output waveform $V_{pp(output)}$ = ……………. $A_{Vo/Vi}$ = ……………

------- END OF REPORT -------

LAB 10

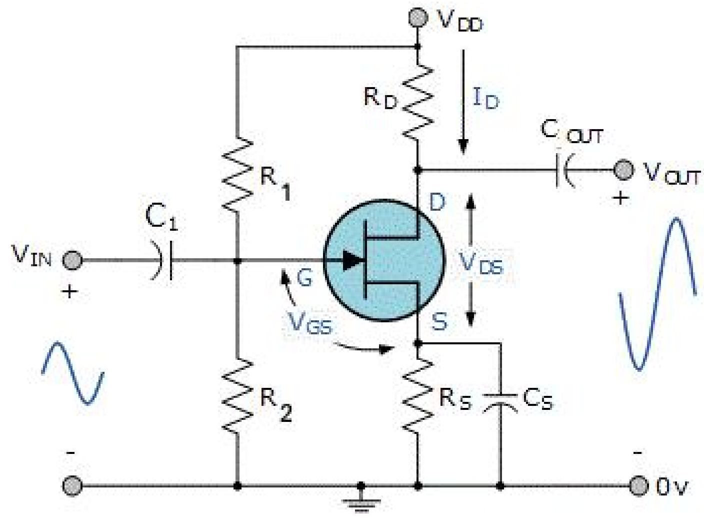
JFET AND SMALL SIGNAL AMPLIFIER

I. GOAL

In this lab, student will have skills:

- Examining operation of JFET transistor.
- Investigating AC characteristics of JFET transistor in a small signal amplifier circuit.

II. SUMMARY OF THEORY



Picture 10.1. Small signal amplifier using JFET transistor in common Source mode

DC characteristics:

$$V_{GS} = -V_S = -R_S I_D$$

$$I_D = I_{DSS}\left(1 - \frac{V_{GS}}{V_{GSoff}}\right)^2$$

$$V_{DS} = V_{DD} - (R_D + R_S) I_D$$

Small signal characteristics (with bypass capacitor Cs):

- Transconductance:

$$g_m = \frac{dI_D}{dV_{GS}} = \left|\frac{2I_{DSS}}{V_{GSoff}}\right|\left(1 - \frac{V_{GS}}{V_{GSoff}}\right) = g_{m0}\left(1 - \frac{V_{GS}}{V_{GSoff}}\right)$$

- Voltage gain:

$$A_v = -g_m Z_L = -g_m R_D$$

- Input impedance:

$$Z_i = R_G = \frac{R_1 R_2}{R_1 + R_2}$$

- Output impedance:

$$Z_O = r_d$$

$$Z_o = r_d || R_D \approx R_D$$

III. PRACTICE

Step 1: wiring circuit as in picture 10.1 using 2SK30, RD = 2.2 kΩ, RS = 1.5 kΩ, R1 = R2 = 1 MΩ, RE = 1 kΩ, C1 = Cout = 10 uF, CS = 47 uF

Step 2: measuring VG, VGS and VDS.

Step 3: applying a sine wave 100 mV Vpp - 1 kHz to C1.

Step 4: measuring output waveform using oscilloscope.

Step 5: calculating voltage gain.

Step 6: removing CS from the circuit.

Step 7: repeat step 3 to step 5.

IV. PREPARATION AT HOME

Generating netlist of the circuit in picture 10.1.

V. REPORT

Filling the practice results into template of report in the next page.

LABOTORY REPORT

LAB 10: JFET AND SMALL SIGNAL AMPLIFIER

Date:……………………… Time: ………………………

Class: ……….......................... *** Session:** .... *** Group:** ……………..

Members: - name: ……………………………………………, **student ID:** ………………..

- name: ……………………………………………, **student ID:** ………………..

TABLE OF RESULTS	
Preparation at home	
Step 2	$V_{G(cal)}$ = ……………. $V_{G(real)}$ = …………......…$I_{D(cal)}$ = ………………......... $V_{GS(rcal)}$ = ……..........……………$V_{GS(real)}$ = ……………………................. $V_{DS(cal)}$ = ……...............……… $V_{DS(real)}$ = ………………........................
Step 4	Output waveform
Step 5	$V_{pp(output)}$ = ……………. $A_{Vo/Vi}$ = …………..
Step 7	Output waveform $V_{pp(output)}$ = ……………. $A_{Vo/Vi}$ = …………..

------- END OF REPORT -------