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

Contents

Contents	
LAB 1	
ANALOGUE MEASURING INSTRUMENT	
LAB 2	
DIGITAL MEASURING INSTRUMENT	
LAB 3	
ELECTRONIC CAD SOFTWARE	
LAB 4	
DC SWEEP AND TRANSIENT IN PSPICE	
LAB 5	
AC SIMULATION AND FREQUENCY RESPONSE IN PSPICE	
LAB 6	
P-N JUNCTION DIODE AND RECTIFIER CIRCUITS	
LAB 7	
RECTIFIER CIRCUIT WITH CAPACITOR FILTER	
LAB 8	
ZENER DIODE AND DC VOLTAGE REGULATOR	
LAB 9	
BJT TRANSISTOR AND SMALL SIGNAL AMPLIFIER	
LAB 10	
JFET AND SMALL SIGNAL AMPLIFIER	

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.
- *θ*Ω*ADJ* 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 $\partial \Omega ADJ$ 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.*: $20 \Omega X 10 = 200 \Omega$)

- X100 scale: Value = Needle position X 100 (ex.: $20 \Omega X 100 = 2000 \Omega$)
- X1k scale: Value = Needle position X 1 k Ω (ex.: 20 Ω X 1 k = 20 k Ω)
- X10k scale: Value = Needle position X 10 k Ω (ex.: 20 Ω X 10 k = 20 k Ω)



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.





Step 4: The diode is REVERSE BIASED

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:		stude	ent ID:

- name:, student ID:

		TABLE OF RESULTS
Home question	Equation	
b	Resistor values	Read: Measured: Read: Measured: Read: Measured:
c	Capacitor test	Scale:
d	Inductor Transformer	Resistance value: Primary resistance: Secondary resistance:
e	Diode test	Scale:
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)



- LCD Display
 Power Button
- 3. Rotary Switch
- 4. Input Terminals
- 5. HOLD Button



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 Ω mA 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 Ω mA 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:		

- 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:
	Oscilloscope	Period: Vpp: Vpp:
f	Function Generator	Period: Vpp: Vpp:

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:





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.

Name	OK
bai_2	Cancel
Create a New Project Using	Help
Spice Analog or Mixed A/D	Tip for New Users
O PC Board Wizard	Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.
C Programmable Logic Wizard	Learn With PSpice - Examples And AppNotes
_ocation	
C:\Users\ltkha\Documents\Projects\test	Browse

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.

Create PSpice Project		:
Create based upon an existing project		OK
AnalogGNDSymbol.opj	×	Browse
Create a blank project		Cancel
		Help

Picture 3.4. Create a blank project

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

Design Ed	view Tools Place SI Analysis PSpice Accessories Options Window Help	cādence [®] -
> 🗆 🖨	, 🖉 🏦 🦻 🥐 🔍 🔍 🔍 🔍 🗶 🔍 🖤 🎬 📴 🏢 🛅 🛄 🕀 💭 🗸 🖓 🔍 🚽 🛤 🗸	4.9
ê 😽 💭	·▶ # 탐독량 및 통 환 斷 -	
srt Page	2.opj 🖸 PAGE1	
	9 4 3 2 1	
		1
		1
		1
		1
		*
		~
		-
		1
		-
	I DEVE DEVE DEVE DEVE DEVE DEVE DEVE DEV	_
		0
		14

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.

OrCAD Capture CIS - Lite - [/ - (SCHE	MATIC1 : PAGE1)]		
File Design Edit View Tools	Place SI Analysis PSpice Pin Shift+G Pin Array Shift+J Part P	Accessories Options Window	v Help > UI 12 [] ∟ L⊕ ≢ (2
Start Page 🚺 bai_2.opj 🛐	Parameterized Part Database Part Z Wire W Auto Wire	 PSpice <u>G</u>round Capacitor Diode Inductor Resistor 	1 .]. [++]
D	L Bus B Image: Participan state J J Bus Entry E Met Alias N Image: Power F Image: Ground G	Digital Discrete Passives Source Searc <u>h</u>	
	Off-Page Connector Off-Page Connector Hierarchical Block Hierarchical Port Hierarchical Pin		

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

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

-	P	AG	E1'	*																																																																	
	6	5	-	-	·	-	-		-	-	-	-	-	-	-	-	4	-	-	-	-	-	-	-	-	Т	-	-	-	-	-	-	-	-	-	3	-	-	-	-	-	-	-	-	T	-	-	-	-	-	-	-	-	2	-	-	-	-	-	-	-	-	Т	-	-	-	-	-	-
																																																																					1
																																																																					212
																																																																					122
																																																																					1
																																																																					2164
																																				1.																																	
																																		1		F	1																																
																																		in the second	-	N	iA.	_																															
																																		1		4	Ľ																																
																																		i.		-	2	÷	A.																														
																																																																					914
																																																																					1
																																																																					262
																																																																					12
																																																																					2.2
																																																																					51

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

Pin Shift+G Pin Array Shift+G Parameterized Part P Database Part Z Wire W Auto Wire Pin Array Bus B Junction J Bus Entry E Bus Entry E Bus Entry E Bus Array Controlled Sources Source Controlled Sources Current Sources Current Sources Ground G Off-Page Connector Modeling Application Hierarchical Port Sine Hierarchical Port Sine Filest Symbol Shift+X Title Block Exponential Bus Entry E Modeling Application K Prime Array Sine	回道也吗~& (吐坯		PSpice Ground Capacitor Diode Inductor	Shift+G Shift+J P ent art Z	Bin Pin Array Part PSpice Compon Parameterized P	
Pin Array Shift+J Parameterized Part P Database Part Z Mire Wire Auto Wire Inductor Bus Bit Junction Junction J Bus Entry E Stround G Goff-Page Connector F Hierarchical Block Hierarchical Block </td <td></td> <td></td> <td>PSpice Ground Capacitor Diode</td> <td>Shift+J P ent art</td> <td>Pin Array Pin Array Part PSpice Compon Parameterized P</td> <td></td>			PSpice Ground Capacitor Diode	Shift+J P ent art	Pin Array Pin Array Part PSpice Compon Parameterized P	
Part P PSpice Component P Parameterized Part PSpice Ground Parameterized Part Capacitor Database Part Z Diode Inductor Auto Wire N Auto Wire N Bus Entry E Bus Entry E Parameterized Part Diode Diode Inductor Auto Wire N Bus Entry E Parameterized Part Digital Parameterized Part Digital Source Controlled Sources Voltage Sources AC Off-Page Connector Modeling Application Hierarchical Block Hierarchical Block Hierarchical Block Exponential Hierarchical Block K If EE Symbol Shift+X Title Block F Bokmark F	l± 1∕£	1€ (≝ 122 (± 111) (± 11 11-11) Ξ (ff)	PSpice <u>G</u> round Capacitor Diode	P ent art Z	Part PSpice Compon Parameterized P	2
PSpice Component PSpice Ground Parameterized Part Capacitor Database Part Z Wire W Auto Wire N Bus Entry E Bus Entry E Merence Diode Bus Entry E Bus Entry E Bus Entry E Source Controlled Sources Quirential Block F Hierarchical Block Modeling Application Hierarchical Block F Hierarchical Block F ItEE Symbol Shift+X Title Block F ItEE Symbol Shift+X		4 11 H+I I 👔 🏠	 PSpice <u>G</u>round Capacitor Diode Inductor 	ent art Z	P <u>S</u> pice Compon Parameterize <u>d</u> P	
Parameterized Part Database Part Database Part Wire Wire Auto Wire Parameterized Part Bus Bus Junction Junction Jusc Entry Bus Entry Power Power <td></td> <td></td> <td>Capacitor Diode Inductor</td> <td>art Z</td> <td>Parameterize<u>d</u> P</td> <td></td>			Capacitor Diode Inductor	art Z	Parameterize <u>d</u> P	
Database Part Z Diode Wire W Inductor Auto Wire Resistor 2 Bus Bus Digital Junction J Discrete Bus Entry E Passives Net Alias N Source Controlled Sources Ground G Off-Page Connector Modeling Application Hierarchical Block Modeling Application It Pulse Hierarchical Block Exponential Sine IEEE Symbol Shift+X Title Block Modeling Application			Diode	Z		K
Image: Wire with the sector w			Inductor		Database Part	
Auto Wire Resistor Auto Wire Resistor Bus Digital Junction J Juscrete Discrete Market Passives Source Controlled Sources Ground G Ground G Hierarchical Block Modeling Application Hierarchical Block Bus Entry Hierarchical Pin R2 Ko Connect X IEEE Symbol Shift+X Title Block F Bokmark F				W	1 Wire	
Lgus Bus Bus Digital Junction J Discrete Bus Entry E Bus Entry E Passives Controlled Sources Source Current Sources Ground G Off-Page Connector Modeling Application Hierarchical Block Hierarchical Port Hierarchical Port R2 IEEE Symbol Shift+X Title Block EM Sine	2	3	Resistor	,	Auto Wire	_
		▶ 21 MON 1995 1995	Digital	В	1. <u>B</u> us	
Image: Buse Entry E Power F Image: Buse Entry F Source Controlled Sources Image: Buse Entry F Search Yoltage Sources Image: Buse Entry F Search Yoltage Sources Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry Image: Buse Entry <td></td> <td>,</td> <td>Discrete</td> <td>J</td> <td>- Junction</td> <td></td>		,	Discrete	J	- Junction	
Image: Source Search Source Controlled Sources Image: Source Search Current Sources Image: Source Search Yoltage		 E1000000000000000000000000000000000000	Passives	E	Bus Entry	
Pgwer F Ground G Ground G Godeling Application Yoltage Sources Modeling Application AC DC Pulse Pulse Sine R2 Exponential Exponential EM Sine	•	Controlled Sources	Source	N	📇 Net Alias	
Image: Search in the search	• 61 2353 2353 23	Current Sources	C 1	F	Power	
Modeling Application Image: Construction of the second secon	AC	Voltage Sources	searcn	G	🚽 Ground	
Image: Second	DC	1 5 1K	Modeling Application	ctor	📴 Off-Page Conne	
Weight Hierarchical Port Sine Hierarchical Pin File No Connect X Itz Itz Itz FM Sine FM Sine FM Sine	Pulse	<u> </u>		ck	Kierarchical Blo	
Hierarchical Pin R2 Exponential No Connect X International State EM Sine IEEE Symbol Shift+X Title Block Bookmark	Sine	φ	1222122212222222	t	Hierarchical Por	
Image: No Connect X If EE Symbol Shift+X Title Block Bookmark	Exponential	₹ R2			📲 Hierarchical Pin.	
IEEE Symbol Shift+X Title Block Bookmark	EM Sine	Į	12221222122212221222	Х	X No Connect	
Title Bloc <u>k</u> Book <u>m</u> ark				Shift+X	lEEE Symbol	
Bookmark					Title Block	
					Bookmark	
에 가는 NM 부사님 이 이 것 같은			- 1101 1101 1101 110	-		

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.

(SCHE	MATIC1	: PAGE1)]							
Tools	Place	SI Analysis	PSpice	Accessories	Options	Window	Help		
Ĉ	r∰ <u>P</u> in }} Pin	 Array	Shift+G Shift+J	~ @	9,0	، ک	17 🗐	C D	
IN.	B Par	t	P	D 14	01	0 1	世世		111 L±
	P <u>S</u> Par	<mark>pice Compon</mark> ameterize <u>d</u> P	i ent Part	• R	6 4	-1 1	le phat	++ [
	. <u>D</u> at	tabase Part	Z						
M	1 <u>W</u> in	re	W						
	Au	to Wire		• 4			3		
	1. <u>B</u> us	5	В	12212					
	📥 Jun	iction	J	1111					
	A Bus	s <u>E</u> ntry	E					3	
	<u> </u>	t Alias	N	1.111		Q V5		₹ 1k	
	Por	wer	F	102513	ova	슬레이			
	🕁 <u>G</u> ro	ound	G	1111					
	Off	-Page Conne	ector	100012				₹ R2	
	📕 <u>H</u> ie	ararchical Blo	ck	1.1.1.1				1 1k	
	G Hie	archical Por	t	102013					
	📲 Hie	rarchical Pin		1111					
		<u>C</u> onnect E Symbol	X Shift+X						

Picture 3.12. Place->Wire





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





Picture 3.17. Double click on "0Vdc"

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

Name: DC	Font Arial 7 (default)
Value: 5Vdc	Change Use Default
Display Format	Color
Value Only	Default 🗸
O Name and Value	Rotation
O Name Only	● 0° ○ 180°
Both if Value Exists	○ 90° ○ 270°
	Text Justification
	Default 🗸 🗸

Picture 3.18. Change Value to 5Vdc

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

Tools	Place	SI Analysis	PSpice	Ac	cess	ories	; (Opti	ions	١	Vind	dow	Help						
Ĉ	<mark>-® ⊵</mark> in]} Piౖn	 Array	Shift+G Shift+J		~	0	0	6	Q	0	6	۲	U7 1	B	C				ì
17/	Par PSr	t pice Compon	P ent		Ø	Įv	Ĩ) '	ĮI	Ŵ		b		Ŧ	¢	dlı.		! [t
	Par	ameterize <u>d</u> P	art					E				η	州	J	ŀ	 4-4	1	r	
-	<u>D</u> at	abase Part	Z																
	1. <u>W</u> ir	e	W																
t 2	Aut	to Wire		+	-	T	2	a.			2		54 - 43	2	. (T	 at.	24		=
	1. <u>B</u> us	5	В																
	🔶 Jun	ction	J		Ι.														
	A Bus	: <u>E</u> ntry	E													-			
	🚔 <u>N</u> et	t Alias	N		Ι.														
	Pov	wer	F										1.1						
	<u> </u>	ound	G																
	Off	-Page Conne	ctor																
	📕 <u>H</u> ie	rarchical Blo	ck		Ι.							4	V	1					
	Hie	rarchical Por	t							5	Vd	C	-						
	📲 Hie	rarchical Pin.			Ι.							-	T						
	X No	<u>C</u> onnect	Х																
	IEEI	E Symbol	Shift+X																
	Titl	e Block											120						
		e oroc <u>k</u> m																	

Picture 3.19. Place->Ground...



1	— ОК
	Cancel
	Add Library
- C	Remove Library
	Help
Name:	
0	
] - C Name: 0

Picture 3.20. Select 0/CAPSYM

Step 22: Wiring GND to circuit



Picture 3.21. Wiring GND to circuit

Analysis	PSpice A	cessories	Option	s Wind	dow	Help				
ND	<u>N</u> ew Sin Edit Sim	nulation Pro	ofile ofile		•	n (g	OE			
AA	O Run		F	11	10 B	♦ ₹	10. 1	h. 1		1+
	<u>V</u> iew Sir	nulation Re	esults P	12				_	-	-
è è	Vie <u>w</u> Ou	itput File			du	에 내] [++] [\$		
	Create M	Vetlist								
	V <u>i</u> ew Ne	tlist								
8 st 6	A <u>d</u> vanc	ed Analysis		•			D = S = D	65	27	65
	Markers			•			1 2 2			
	<u>B</u> ias Poi	nts			· · ·		(S) 93			
	-					4	< R	1		
		S. S.				4	5 1	<		
		V1								
	5Vdc-	<u> </u>					0 93 			
		T					1.1.1			
		1 1					12 - 55 12 - 55			

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

	and the second second							-						-	_	_
_	Mew New	Simu	lation	Profil	e				Ð	117		P	N	100	圖	Ť
-	Edit	Simu	lation P	rofile							1	-	land	-	head	100
R	🖸 <u>R</u> un					F11		10	1	+	E	Q	. du		L±	L
2	View	Sim	lation	Resul	ts	F12	2	E		- 10			1.100	-	21	
2	Vie <u>w</u>	Out	out File					Ľ	Ju	mili		1 1*	1			
	Crea	te Ne	tlist					E								
	V <u>i</u> ew	Netl	ist					L								
10	Adva	incec	Analys	is			•	F	10	3	10	5 T	10	10.13	12	3
	Mark	ers					≁	H				1				
	<u>B</u> ias	Point	s				•					1				
								1				5	R1			
											-	2	1k			
		+	V1													
	5Vdc	-	-													
		- 1	-													

Picture 3.24. PSpice->New Simulation Profile

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

Name:	-
DC analyze	Create
Inherit From:	Cancel
none 🗸	

Picture 3.25. Set name for profile then click Create

Step 27: Setting as in the picture below

General Analysis Configuration Files Options Data Collection Probe Window	Analysis Type: Bias Point Options: General Settings Temperature (Sweep) Save Bias Point Load Bias Point Calculate small-signal DC gain (.TF) From Input source name: To Output variable:
--	---

Picture 3.26. Simulation setting then click OK

E1)] nalysis ice Accessories Options Window Help Mew Simulation Profile 👁 UT 🕼 🖸 🗎 E Edit Simulation Profile F11 🖢 🕂 🖾 🕍 🛄 🕂 🖋 3 12 🕞 Run View Simulation Results F12 ግ ት ሴ ፦ 💽 👚 Ň View Output File Create Netlist SCHE View Netlist Advanced Analysis ۶ Markers Þ Bias Points . **R1** 1k V1 5Vdc

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

Picture 3.27. Run simulation

Step 29: click V button to show results.





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:		, stude	ent 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	

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...
 - $\circ~$ 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

Name:		-	
DC sweed			Create
о <u>-</u> опоор		1	Cancel
nherit From:		-	
none	~		

Picture 4.5 Setup name of simulation

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

General	Analysis Type:	Sweep Variable						
Inclusio	DC Sweep 👻	Voltage source	Name:	V1				
Analysis	Options:	Current source	Model type:		+			
Configuration Files	Primary Sween	Global parameter	Model name:					
Options	Secondary Sween	Model parameter	Parameter name:					
Data Collection	Monte CarloWorst Case	 Temperature 	i didileter idile.					
Data Collection	Parametric Sween	. Sweep Type						
Probe Window		Oweep Type	Start	Value	0			
	Temperature (Sweep)	Inear	End	Value:	5			
	Save Blas Point		do – Incro	mont	0.1			
	Load Bias Point	O Valua Lint	incle incle	ment.	0.1			

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



Simulation Output Variables			Functions or Macros	
			Analog Operators and Functions	~
(V2:+) R(01)	^ E	Analog	#	
C(Q1)	T	Digital	U ×	
E(Q1) S(Q1)			+	
(0)	Ŀ	∠ Voltages	1	
/(N00687) /(N00816)	5	Currents 🛛	(ABS()	
(OUT)	F	刁 Power	ARCTAN()	
(Q1:6) ((Q1:c)			ATAN() AVG()	
(Q1:e)	-L	Noise (V*/Hz)	AVGX(,)	
(B1:1) (B1:2)	E	🛛 Alias Names		
((11.2) ((R2:1)		Subcircuit Nodes	DB()	
((R2:2)			ENVMAX(,)	
(v1.+) (V1:-)			ENVMIN(,) EXP()	
((V2:+)			G()	
(V2:-) (Vin)				
/1(Ř1)	2	9 variables listed	LOGIÓ()	
/1(82) /1(/1)				
ull List			Tricoll	

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

Name:	
Transient	Create
Inherit From:	Cancel
none 🗸 🗸	

Picture 4.13 Setup name of simulation

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

General	Analysis Type: Time Domain (Transient)	Run To Time :	10ms	seconds	(TSTOP)
Analysis	Options:	Start saving data after :	0	seconds	
Configuration Files Options Data Collection	General Settings Monte Carlo/Worst Case Parametric Sweep	Transient options: Maximum Step Size Skip initial transient I	0.00001 bias point calcul	seconds ation (SKIPBP)	
Probe Window	 Temperature (Sweep) Save Bias Point Load Bias Point Save Check Point Restart Simulation 	Run in resume mode		Output File Options	

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 $R = 100 \Omega$, 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

MATIC1 : PAGE1)] Place SI Analysis	PSpice Accessories Options V	Window Help
> < □ □ ∰ 16 /8 } ¥ ₽ % %	New Simulation Profile Edit Simulation Profile Run F11 View Simulation Results F12 View Output File	○ U
PAGE1*	<u>C</u> reate Netlist V <u>i</u> ew Netlist	3
	A <u>d</u> vanced Analysis <u>M</u> arkers Bias Points	; <u>R1</u>
	Vin	
	1/4	100
VOFF =	0	⊥ C1

Picture 5.4. Setup a simulation

Step 5: Setup a name for simulation profile

New Simulation			~
Name:		1 E	Create
AC			Cicato
Inherit From:			Cancel
none	\sim		
Boot Schematic: SCHEMATIC	1		

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)

General	Analysis Type:	AC Sweep Type		200
Analysis	AC Sweep/Noise 👻	💿 Linear	Start Frequency:	1
hindiyələ	Options:	Logarithmic	End Frequency:	1e6
Configuration Files	General Settings	Decade	Points/Decade:	100
Options	Monte Carlo Worst Case	Noise Applysis		
Data Callection		Noise Analysis		
Jata Collection		Enabled Output	t voltage:	
Probe Window	Temperature (Sweep)	I/V Sou	urce:	
	Save Bias Point	Interva	al:	
	Load Bias Point	Output File Options		
		semiconductors (.OP)	formation for nonlinear controll	ed sources and
		3		

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

Simulation Uutput Variables			Functions or Macros	
×			Analog Operators and Functions	~
Frequency I(C1) I(C1:1) I(R1:1) I(R1:1) I(M1) I(V1:+) V(0) V[C1:1] V[C1:2] V[01] V[R1:2] V[V1:+] V[V1:+] V[V1:+] V[V1:+] V[V1:1] V1(C1) V2[R1] V2[V1] V(V1)	^	 Analog Digital Voltages Currents Power Noise (V^e/Hz) Alias Names Subcircuit Nodes 	# () * + - / @ ABS() ARCTAN() ATAN() AVG() AVG() AVG() COS() D() D() DE() ENVMAX(,) ENVMIN(,) EXP() G() IMG() LOG() LOG() LOG() LOG() M()	
W(R1) Full List	*		MAX()	~

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

General	Analysis Type:	Run To Time :	10ms	seconds (TSTOP)	
nalysis	Time Domain (Transient) - Options:	Start saving data after :	0	seconds	
Configuration Files	General Settings	Transient options: — Maximum Step Size	0.00001	seconds	
Data Collection	Monte Carlo/Worst Case Parametric Sweep	Skip initial transient	bias point calcul	ation (SKIPBP)	
Probe Window	 Temperature (Sweep) Save Bias Point Load Bias Point 	Run in resume mode		Output File Optio	ons
	Save Check Point Restart Simulation				

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

General	Analysis Type:	AC Sweep Type
Analysis	AC Sweep/Noise	Linear Start Frequency:
	Options:	Logarithmic End Frequency: 1e9
Configuration Files	General Settings	Decade Points/Decade: 1000
Options	Monte Carlo/Worst Case	Noise Analysis
Data Collection	Parametric Sweep	Enabled Output Voltage:
	Temperature (Sweep)	
Probe Window	Save Bias Point	Interval:
	Load Pigs Point	Output File Options
	Load Blas Folin	Include detailed bias point information for nonlinear controlled sources and
		semiconductors (.OP)

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:

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

Load current is

$$I_{dc} = \frac{V_{dc}}{R} \tag{2}$$

b. Full-wave Rectifier

i. Two diodes



Picture 6.2. Full-wave Rectifier with two diodes

Average DC voltage is calculated from below equations:

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

Load current is

$$I_{dc} = \frac{V_{dc}}{R} \tag{4}$$

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 $1k\Omega$.

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

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

Step 4: calculate ILDC 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 $1k\Omega$.

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 $1k\Omega$.

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:	, st	udent ID:

- name:, student ID:

TABLE OF RESULTS		
Preparation		
at home		
	Output	
	Output	
	waveloim	
Half-wave		
Rectifier		
	Voltage	$Vp(secondary) = \dots V_{LDC} = \dots$
		$R_{Load} = \dots I_{LDC} = \dots$
		Frequency =
	Output	
	waveform	
Full-wave		
Rectifier		
with two		
diodes	Voltage	$V_{n(secondary)} = V_{rns} =$
	voltage	$v_{\text{DC}} = 1$
		$R_{Load} = \dots $ $I_{LDC} = \dots$
	Output	
	waveform	
T U -		
run-wave		
Kectiller		
with aloae		
briage	Voltage	$Vp(secondary) = \dots V_{LDC} = \dots$
		$R_{Load} = \dots I_{LDC} = \dots$
		Frequency =
		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_LC 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

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

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

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

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

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

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

b. Full-wave Rectifier with diode bridge



Picture 7.3. Full-wave Rectifier with smoothing capacitor



Picture 7.4. Waveform

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

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

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

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

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

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

c. Ripple factor

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

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

Ripple factor:

$$r = \frac{V_{rp(eff)}}{V_{dc}} = \frac{V_{o(pk)}}{\sqrt{3}(4fR_LC)V_{dc}}$$
(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 $1k\Omega$.

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 ILDC 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 $1k\Omega$.

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 OF RESULTS		
Preparation at home		
Half-wave Rectifier	Output waveform Voltage	V_{LDC} = R_{Load} = I_{LDC} = I_{LDC} =
Full-wave Rectifier	Output waveform Voltage	$V_{LDC} = \dots R_{Load} = \dots I_{LDC} = \dots C filter = \dots Vripple = \dots Vr$

----- 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:

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

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:

$$V_{LDC} = V_Z - V_{BE}$$

$$I_1 = I_Z + I_B$$

$$I_1 = \frac{V_{iDC} - V_Z}{R_1}$$

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

$$I_B = \frac{I_E}{B+1} = \frac{I_L}{B+1}$$

$$I_Z = I_1 - I_B = I_1 - \frac{I_L}{B+1}$$

Power dissipations:

$$P_{D} = V_{CEQ} \times I_{CQ} = (V_{iDC} - V_{LDC}) \times I_{CQ}$$

$$= (V_{iDC} - V_{LDC}) \times BI_{B} = (V_{iDC} - V_{LDC}) \times \frac{BI_{E}}{B+1}$$

$$= (V_{iDC} - V_{LDC}) \times I_{L} < P_{D(max)}$$

$$P_{Z} = V_{Z} \times I_{Z} < P_{Z(max)}$$

$$P_{R1} = R_{1} \times I_{1}^{2}$$

Output impedance:

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

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

 $k\Omega$, $RL = 390 \Omega$ 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 OF RESULTS		
Preparation		
at home		
	Schematic	
DC Voltage		
Regulator		
using zener		
diode		
	Measurement	$V_{iDC} = \dots I_Z = \dots R_S = \dots I_S = \dots$
		$V_{\text{LDC(cal)}} = \dots V_{\text{LDC(real)}} = \dots$
		$R_{LOAD} = \dots I_{LDC} = \dots$
	Schematic	
Transistor		
Series		
Voltage		
Regulator		
0		
	Measurement	$V_{iDC} = I_{Z} = R_{1} = I_{1} =$
		$V_{\rm LDC(cal)} = \dots V_{\rm LDC(real)} = \dots$
		$R_{LOAD} = \dots I_{LDC} = \dots I_{B} = \dots h_{FF} = \dots$
l	1	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:

$$V_{BB} = \frac{R_2}{R_1 + R_2} V_{CC}$$
$$R_B = \frac{R_1 R_2}{R_1 + R_2}$$
$$I_B = \frac{V_{BB} - V_{BE}}{R_B + (B + 1)R_E}$$
$$I_C = \beta I_B$$
$$V_{CE} = V_{CC} - (R_C + R_E)I_C$$

Temperature stabilized factor:

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

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:

$$Z_{O} = \frac{V_{O}}{I_{O}}\Big|_{V_{S}=0, Z_{L\to\infty}} = \frac{1}{h_{OP}} \approx \infty$$

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

III. PRACTICE

Step 1: wiring circuit as in picture 9.1 using 2N2222A, $R1 = R2 = 10 \text{ k}\Omega$, RC = 1.5

 $k\Omega$, RE = 1 $k\Omega$, C1 = Cout = 10 μ F, CE = 47 μ F

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:*

* Session: * Group:
* Session: * Group:

Members: - name:, student ID:

- name:, student ID:

Preparation at home
at home
$V_{\text{PP}(\text{rad})} = V_{\text{PP}(\text{rad})} = V_{\text{PP}(\text{rad})} =$
Step 2 $I_{D(x)} = I_{C(x)} =$
$\frac{1}{1} \frac{1}{1} \frac{1}$
$\nabla CE(cal) = \dots \nabla CE(real) = \dots$
Output waveloini
Step 4
$V_{pp(output)} = \dots A_{V_0/V_i} = \dots$
Step 5
Step 6 $F_{Cutoff} = \dots$
Step 11 Input impedance =
Step 16 Output impedance =
Output waveform
Step 18
$V_{pp(output)} = \dots A_{Vo/Vi} = \dots$

----- 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:

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

- 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, $RD = 2.2 \text{ k}\Omega$, $RS = 1.5 \text{ k}\Omega$,

 $R1 = R2 = 1 M\Omega$, $RE = 1 k\Omega$, 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		
Star 2	$\mathbf{V}_{G(cal)} = \dots \mathbf{V}_{G(real)} = \dots \mathbf{I}_{D(cal)} = \dots$	
Step 2	$V_{GS(real)} = \dots V_{GS(real)} = \dots$	
	$V_{DS(cal)} = \dots V_{DS(real)} = \dots$	
	Output waveform	
Step 4		
	$V_{pp(output)} = \dots A_{Vo/Vi} = \dots$	
Step 5		
	Output waveform	
	Output waveloini	
Sten 7		
Step /		
	$V_{pp(output)} = \dots A_{Vo/Vi} = \dots$	