EEM 206 ELECTRIC CIRCUITS LABORATORY-II

EXPERIMENT BOOKLET

Prepared By: T. Özge ÖZDİNÇ ONUR, Rıfat HACIOĞLU

BULENT ECEVIT UNIVERSITY ENGINEERING FACULTY ELECTRICAL-ELECTRONICS ENGINEERING DEPARTMENT

ZONGULDAK, 2024

INTRODUCTION

This booklet contains second class students of Electrical-Electronics Engineering Circuit Analysis-2 lecture notes. The subjects are:

- Frequency, Amplitude and Phase Measurement for Basic AC /RC Circuits
- Basic AC RC and RL Circuits
- Serial AC RLC Circuits
- Parallel RLC Circuits
- Power in AC Circuits
- Application of Transformer
- Balanced Three-Phase Y-Y Connected Circuits
- Balanced Three-Phase Δ - Δ Connected Circuits
- Passive Filters

Students must prepare the reports by obeying the rules. So, for the reports:

- There must be a covering letter and it must be contain name of the lecture, experiment number, the number of the group who prepared the report and their signatures.
- There must a purpose of the report and the study must be denoted clearly.
- The circuit realized during the experiment must be simulated by using PSpice and the obtained results must be given.
- The obtained results must be commented in the conclusion section.

GOOD LUCK ...

GENEL KURALLAR

- > Mazeretsiz olarak deneye devam şartını yerine getirmeyenlere DZ notu verilecektir.
- > Deneyler gruplar şeklinde yapılacaktır.
- Deneyler süresi içinde bitirilmek zorundadır. Bu nedenle öğrencinin deney içeriğini dikkate alarak zaman yönetimi yapması gerekir.
- Deney ön hazırlıkları ilgili deneyin başında yapılması istenen kısımdır. O hafta yapılacak olan deneyin ön çalışması deneye gelmeden önce her grup üyesi tarafından ayrı olarak hazırlanmalıdır. Deney ön hazırlığı, deneylerde yer alan "PRELIMINARY STUDY" kısmında istenilenleri içermelidir. <u>Deneylere ön çalışma hazırlamadan gelen va da denevde kullanılacak malzemeleri getirmeyen öğrenciler deneyi yapamayacaklardır.</u>
- Deney raporlarını her grup sadece kendi tecrübelerini kullanarak yazmalıdır. Başka bir grubun deney sonuçlarını veya başka kaynaklardan alınmış çıktıları getirmemelidir. Bu durumda, <u>deney rapor notu sıfır verilecektir</u>.
- > Rapor zımbalanmalıdır, ayrı bir dosya kullanılmamalıdır.
- Raporda kurulan devreler ve kullanılan elemanlar detaylı bir şekilde verilmelidir. Tüm ölçüm ve çizimlerde kullanılan birimler mutlaka yazılmalıdır.
- Raporlarda bilimsel olarak anlamlı düzgün bir dil kullanılmalıdır.
- Hazırlanan deney raporu, belirtilen tarihte laboratuar dersinin başında teslim edilebilecektir. <u>Ders saatinden ya da belirtilen tarihten ve saatten sonra getirilen</u> <u>raporlar teslim alınmayacaktır</u>.

CONTENTS

Page No

INTRODUCTION	2
CONTENTS	4
What is PSPICE ?	5
DRAWING SCHEMATIC	6
SIMULATION AND GRAPHIC ADJUSTMENTS	.12
PSPICE A/D ANALYSIS TYPES AND OPTIONS	.15
1. DC Sweep Analyse	.15
2. AC Sweep/Noise Analyses	.19
3. Time Domain (Transient) & Fourier Transform Analyses	.21
4. Bias Point (Working Point) Analysis	.23
5. Monte Carlo Analysis	.24
6. Worst Case Analysis	.25
EXAMPLES	.25
RC Circuit	.25
Full Wave Rectifier	.31
Parametric Circuit Element Usage	.32
Bode diagram of Inductive Lowpass Filter	.34
• OPAMP Usage	.36
INVESTIGATING TRANSIENT ANALYSIS FOR RL, RC AND RLC CIRCUITS	.38
LABORATORY EXPERIMENTS	.50
I. EXPERIMENT: Frequency, Amplitude, Phase Measurement for Basic AC /RC Circuits	.51
II. EXPERIMENT: Basic AC RC and RL Circuits	.52
III. EXPERIMENT: Serial AC RLC Circuits	.54
IV. EXPERIMENT: Parallel RLC Circuits	.56
V. EXPERIMENT: Power in AC Circuits	.58
VI. EXPERIMENT: Application of Transformer	.59
VII. EXPERIMENT: Balanced 3-Phase Y-Y Connected Circuits	.60
VIII. EXPERIMENT: Balanced 3-Phase Δ - Δ Connected Circuits	.62
IX. EXPERIMENT: Passive Filters	.63

What is **PSPICE** ?

SPICE (*S*imulation *P*rogram for *I*ntegrated *C*ircuits *E*mphasis) is a program for simulating electronics circuits in the computer environment. As for PSPICE is a computer aided design and simulation program of Cadence/Orcad corporation works on Windows base.

Starting PSPICE

Choose Programs>Orcad>Capture.



• Choose File>New>Project.

3	Orcad Capture - Lite Edition - [Session Log]								
3	File	View	Edit	Options	Window	Help			
e	N	ew				•	Project		
	0	pen				•	Design	Y	
· T	Si Si	ave ave As.			Cti	rl+S	Library VHDL File Text File		
							10/10/100		

• Then choose "Analog or Mixed A/D" option from the opening window and give a name for your Project. It is suggested that **not to use** Turkish characters for your projects' names. Then click on "Browse" buton for choosing the place where your designs will be saved and choose a proper file.

Name	ОК
linv_sim	Cancel
Create a New Project Using Analog or Mixed A/D C PC Board Wizard C Programmable Logic Wizard C Schematic	Help Tip for New Users Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.
C Schematic	template.

• Choose the "Create a blank project" and click on OK button.

Create PSpice Project	×
C Create based upon an existing project	OK
empty.opj	Browse
Create a blank project	Cancel
	<u>H</u> elp

DRAWING SCHEMATIC

- For activating Toolbars, click on anywhere on the design screen.
- By using "I" and "O" keys on the keyboard, you can zoom in or zoom out to the design screen.
- Keystroke to the "**P**" on the keyboard and choose the icon given in the figure from the toolbar on the left.

i	1	-	-	-	-	-	-	
	1							

• Click on the "Add Library" button.

Place Part		×
Part:		ОК
Part List:		Cancel Add Library <u>Add Library</u> Part Search <u>Filter</u>
Libraries: Design Cache	Graphic Normal Convert Packaging Parts per Pkg: 1 Part: Type:	

• Choose the libraries from the Pspice file, for choosing several elements please keystroke on the ctrl at the same time. (for the basic elements for example Resistor, Capasitor choose Analog.olb file and choose source.olb file for the sources.)

Look pr Papeles Image: Control in the second in the s	Last in [<u>.</u> - 🖛 📼 -	<u> </u>
advants 1/4-1.0b 1/4-1.0b 1/4-1.0b 1/4-1.0b 1/2 / Yac0b	Look in: [<u>⊾</u> ⊡* ⊞*	
Image: Second in the secon	advanis	III 74f.olb ⊐lb III 74b.olb	회 74s.olb [회 7400 olb]	🔟 analog_p.olb 🔟 anl. miss.olb	
Image: Pressure of the second seco	1_3Hot. 74ac.olt) 🖾 74hc.olb	I abm.olb) anig dev.olb	<u>.</u>
N 74ab.ob N 74ab.ob N ana_swit.ob N apex_pwm.ob N N 74ab.ob N 74b.ob N analog.ob N bipolar.ob N File game:	🔄 74act.o	b 🔟 74hct.olb	🔟 adv_lin.olb	apex.olb	
Image: State in the second	T4als.ol) 🖻 741.olb	ana_swit.olb	apex_pwm.olb	
File pame: Open Files of type: Capture Library(".olb) Cancel Cancel Image: Open as gead-only arg the element we want, choose the element and then click on OK. Place Part Image: Part List: OK Part List: OK Part List: Add Library Part List: Part List: Part List	🔟 74as.olt	1 74ls.olb	🔟 analog.olb	🔟 bipolar.olb	
File game:	•				
Files of type: Capture Library(".olb) Cancel Dpen as read-only at the element we want, choose the element and then click on OK. Place Part Part List Add Library Part List Part Search Part Search	File <u>n</u> ame:			<u>O</u> pen	
g the element we want, choose the element and then click on OK. g the element we want, choose the element and then click on OK. Place Part Part List Part Search Part Search	Files of tupe	Capture Libraru(* olb)		Cancel	
the element we want, choose the element and then click on OK.	These of type				
g the element we want, choose the element and then click on OK.		Open as read-only			10
g the element we want, choose the element and then click on OK.					
Place Part Image: Constraint of the system of the syst	ling the element	it we want, choose	e the element and	d then click o	n OK.
Place Part X Part: 0K I 0K Part List: Add Library R/DATACORV Add Library R/DATACONV R/DATACONV R/TIXPT/DIODE Part Search R711XPT/DIODE Filter R712XPT/DIODE Filter R712XPT/DIODE Filter R712XPT/DIODE Filter R712XPT/DIODE Filter R/AM8Kx1break/BREAKOUT Eilter RAM8Kx2break/BREAKOUT Filter RAM8Kx2break/BREAKOUT Fraphic Sources Source Sources Packaging SwitT_RAV Parts per Pkg: 1 FXTAL Type: Homogeneous It Tik					
Part: OK Image: Constraint of the system of the	Place Part				×
Image: Constraint of the system Image: Constraint of the system Part List: Image: Constraint of the system Part List: Image: Constraint of the system R/DATACONV Image: Constraint of the system R/DATACONV Image: Constraint of the system R/TAXPT/DIDDE R711XPT/DIDDE R711XPT/DIDDE R712XPT/DIDDE R711XPT/DIDDE R712XPT/DIDDE R711XPT/DIDDE Image: Constraint of the system R_var/ANALOG P RAM9Kx1break/BREAKOUT Image: Constraint of the system Libraries: SOURCE SOURCE Image: Constraint of the system SOURCE Source: SWIT_RAV Switt_REG TEX_INST Image: Constraint of the system TLINE Image: Constraint of the system TINE Image: Constraint of the system Ref: Image: Constraint of the system Ref: Image: Constraint of the system SWIT_REG Image: Constraint of the system TEX_INST Image: Constraint of the system THYRISTR Image: Constraint of the system TINE Image: Constraint of the sys	Part:				
Part List:	4				< <u> </u>
Francess Add Library R/ANALOG P R/DATACONV R/DATACONV R/FLTSUB R710XPT/DIODE R710XPT/DIODE Part Search R712XPT/DIODE Eilter R714XPT/DIODE Eilter R714XPT/DIODE Eilter R714XPT/DIODE Eilter R714XPT/DIODE Eilter R714XPT/DIODE Eilter R714XPT/DIODE Filter R714XPT/DIODE Filter R714XPT/DIODE Filter R714XPT/DIODE Filter R714XPT/DIODE Filter R714XPT/DIODE Filter R/M3Kx1break/BREAKOUT Filter RAM8Kx8break/BREAKOUT Filter Libraries: Sourace SWIT_RAG Packaging Parts per Pkg: 1 Netting Tik THYRISTR Type: Homogeneous XTAL Type: Homogeneous Filter	Part List			Can	cel
Intervention Image: Construction R/DATALOG P R/DATACONV R/DATACONV R/TIXPT/DIODE R711XPT/DIODE R711XPT/DIODE R712XPT/DIODE R714XPT/DIODE Filter R714XPT/DIODE Filter R714XPT/DIODE Filter R714XPT/DIODE Filter R_var/ANALOG P RAM8Kx1break/BREAKOUT Image: Constant Libraries: Graphic SHINDNGN Image: Constant SOURCE SOURCSTM SPECIAL Packaging Switt_RAV Packaging Switt_RAV Packaging Switt_RAV Image: Constant THYRISTR Image: Constant THYRISTR Image: Constant <td>B/ANALOG</td> <td></td> <td></td> <td>ii bbA</td> <td>prary</td>	B/ANALOG			ii bbA	prary
H7DATACUNV R/FILTSUB R710xPT/DIODE R712xPT/DIODE R712xPT/DIODE R712xPT/DIODE R712xPT/DIODE R714xPT/DIODE R714xPT/DIODE R714xPT/DIODE R714xPT/DIODE R714xPT/DIODE R714xPT/DIODE R714xPT/DIODE Ryar/ANALOG_P RAM8Kx1break/BREAKOUT Libraries: SHINDNGN SOURCE SOURCSTM SPECIAL SWIT_RAV THYRISTR TLINE XTAL ZETEX	r/ANALOG	P			
R710XPT/DIODE Part Search R711XPT/DIODE Filter R712XPT/DIODE Filter R714XPT/DIODE Filter R_var/ANALOG P RAM8Kx1break/BREAKOUT Help Libraries: Graphic SURCE SOURCE SOURCE Convert SWIT_RAV Packaging SWIT_REG Parts per Pkg: 1 THYRISTR Parts THYRISTR Type: Homogeneous XTAL Ype: Homogeneous	R/DATACU R/FILTSUE	NV I			Library
B711xF17b10bE Filter B714xF17b10bE Filter Bryar/ANALOG Filter BaM8Kx1break/BREAKOUT Filter Libraries: Surreak/BREAKOUT SURCE Source SOURCSTM Source SWIT_RAV Packaging Switz_REG Packaging Parts per Pkg: 1 Help 1k THYRISTR Type: Homogeneous XTAL Ype: Homogeneous	B710XPT/	DIODE		Part <u>S</u> e	arch
R714×PT/DIODE R_var/ANALOG R_var/ANALOG_P RAM3Kx1break/BREAKOUT RAM3Kx1break/BREAKOUT Libraries: SURCE SOURCE SOURCESTM SPECIAL SWIT_RAV SWIT_RAV SWIT_REG TEX_INST THYRISTR TLINE XTAL ZETEX	R712XPT/	DIODE		Filte	er
B_var/ANALOG_P RAM8Kx1break/BREAKOUT RAM8Kx8break/BREAKOUT Libraries: SURCE SOURCE SOURCSTM SPECIAL SWIT_RAV SWIT_REG TEX_INST THYRISTR TLINE XTAL ZETEX	B714XPT/ B_var/ANA)IODE LOG			
HAM8Kx1break/BREAKUUT ▲ elp Libraries: SHINDNGN SOURCE SOURCSTM SPECIAL © Lonvert SWIT_RAV Packaging SWIT_REG Packaging TEX_INST THYRISTR TUINE Type: Homogeneous XTAL V		LOG_P			
Libraries: SHINDNGN SOURCE SOURCSTM SPECIAL SWIT_RAV SWIT_REG TEX_INST TLINE XTAL ZETEX SURCE SOURCSTM SOURCSTM SOURCSTM SOURCE Sourcet Packaging Parts per Pkg: 1 Parts Type: Homogeneous Packaging Parts Type: Homogeneous	B_var/ANA				
SHINDNGN SOURCE SOURCSTM SPECIAL SWIT_RAV SWIT_RAV SWIT_REG TEX_INST TLINE XTAL ZETEX	R_var/ANA RAM8Kx1b RAM8Kx8b	reak/BREAKUUT reak/BREAKOUT		<u>▼</u> <u>H</u> €	ih l
SOURCE SOURCSTM SPECIAL SWIT_RAV SWIT_REG TEX.INST TLINE XTAL ZETEX SOURCSTM SPECIAL Packaging Packaging Packaging Packaging Parts per Pkg: 1 Parts Type: Homogeneous I Packaging Packaging Parts Type: Homogeneous	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries:	reak/BREAKUUT reak/BREAKOUT	anhie		
SPECIAL SWIT_RAV SWIT_REG TEX_INST THYRISTR TLINE XTAL ZETEX	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries:	reak/BREAKUUT reak/BREAKUUT	aphic Normal		
SWIT_BAV SWIT_BEG TEX_INST THYRISTR TLINE XTAL ZETEX SWIT_BAV Packaging Parts per Pkg: 1 Parts Type: Homogeneous W MARCH Packaging Packaging Parts per Pkg: 1 Type: Homogeneous	R_var/ANA RAM8Kx1b RAM8Kx8b Ljbraries: SHINDNGF SOURCE SOURCE	reak/BREAKUUT reak/BREAKOUT	aphic <u>N</u> ormal <u>D</u> onvert	<u>н</u> R?	
TEX_INST THYRISTR TLINE XTAL ZETEX TUP: Homogeneous ZETEX	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGf SOURCE SOURCE SOURCST SPECIAL	reak/BREAKOUT	aphic <u>N</u> ormal <u>C</u> onvert	R?	
TLINE XTAL ZETEX Type: Homogeneous	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGf SOURCE SOURCST SPECIAL SWIT_RAV SWIT_REC	reak/BREAKUUT reak/BREAKOUT	aphic Normal Convert ackaging	₽ <u>₩</u> R?	
ZETEX Type: Homogeneous	R_var/ANA RAM8Kx1b RAM8Kx8b Ljbraries: SHINDNGP SOURCE SOURCSTI SPECIAL SWIT_RAM SWIT_RAM SWIT_REC TEX_INST TUPESTE	reak/BREAKUUT reak/BREAKOUT	aphic <u>N</u> ormal <u>C</u> onvert ackaging arts per Pkg: 1	■ <u>H</u> R? —///- 1k	
	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGR SOURCE SOURCE SPECIAL SWIT_RAV SWIT_RAV SWIT_RAV SWIT_REG TEX_INST THYRISTR TLINE	reak/BREAKUUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 art:	■ <u>H</u> R? —///- 1k	
	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGR SOURCE SOURCE SOURCE SPECIAL SWIT_RAV SWIT_RAV SWIT_RAV SWIT_REC TEX_INST THYRISTR TLINE XTAL ZETEX	reak/BREAKUUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 ert:	■ <u>H</u> e R? —_^_ 1k	
ing the elements such as current and voltage sources, choosing she	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGF SOURCE SOURCSTI SPECIAL SWIT_RAV SWIT_REC TEX_INST THYRISTR TLINE XTAL ZETEX	reak/BREAKUUT reak/BREAKOUT	aphic <u>N</u> ormal <u>Convert</u> ackaging arts per Pkg: 1 art: ype: Homogeneous urrent and volta	R? R? 1k	choosing sho
ding the elements such as current and voltage sources, choosing shouton and choosing the "ground" icon from the toolbar on the left is enough	R-var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGR SOURCE SOURCE SOURCE SOURCE SOURCE SOURCE SWIT_RAV SWIT_RAV SWIT_RAV SWIT_REC TEX_INST THYRISTR TLINE XTAL ZETEX	reak/BREAKUUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 art: ype: Homogeneous urrent and voltation	R? R? 1k	choosing sho
ing the elements such as current and voltage sources, choosing sho on and choosing the "ground" icon from the toolbar on the left is enough	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGF SOURCE SOURCSTI SPECIAL SWIT_RAV SWIT_REC TEX_INST THYRISTR TLINE XTAL ZETEX	reak/BREAKUUT reak/BREAKOUT	aphic <u>Normal</u> <u>Convert</u> ackaging arts per Pkg: 1 art: ype: Homogeneous urrent and voltat icon from the to	R? R? 1k	choosing sho left is enough
ng the elements such as current and voltage sources, choosing sho on and choosing the "ground" icon from the toolbar on the left is enough " M " ve " Source " libraries by using "Add Library" button on the op	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGN SOURCE SOURCE SOURCST SPECIAL SWIT_RAV SWIT_SWIT_RAV S	reak/BREAKUUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 art: I I ype: Homogeneous urrent and voltation from the too y using "Add I	R? R? 1k we sources, polbar on the Library" but	choosing sho left is enough. ton on the ope
ng the elements such as current and voltage sources, choosing sho on and choosing the "ground" icon from the toolbar on the left is enough "M" ve "Source" libraries by using "Add Library" button on the op	R_var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGF SOURCE SOURCEST SPECIAL SWIT_RAV SWIT_SWIT_RAV SWIT_SWIT_SWIT_SWIT_SWIT_SWIT_SWIT	reak/BREAKOUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 arts wpe: Homogeneous urrent and voltation from the to y using "Add I	R? R? 1k we sources, oolbar on the Library" but	choosing sho left is enough. ton on the ope
ng the elements such as current and voltage sources, choosing sho on and choosing the "ground" icon from the toolbar on the left is enough M " ve "Source" libraries by using "Add Library" button on the op	R-var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGI SOURCE SOURCE SOURCSTI SPECIAL SWIT_RAV SWIT_REG TEX_INST THYRISTR TLINE XTAL ZETEX Ng the elern on and choose M" ve "So	reak/BREAKOUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 eft: ype: Homogeneous urrent and voltation from the to y using "Add I	R? R? 1k we sources, polbar on the Library" but	choosing sho left is enough. ton on the ope
ling the elements such as current and voltage sources, choosing sho ton and choosing the "ground" icon from the toolbar on the left is enough YM " ve "Source" libraries by using "Add Library" button on the op	R-var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGT SOURCE SOURCE SOURCE SOURCE SOURCE SOURCE SWIT_RAV SWIT_SWIT_RAV SWIT_RA	reak/BREAKOUT reak/BREAKOUT	aphic Normal Convert ackaging arts per Pkg: 1 art: ype: Homogeneous urrent and voltation from the to y using "Add I	R? R? 1k w w w age sources, oolbar on the Library" but	choosing sho left is enough. ton on the ope
ng the elements such as current and voltage sources, choosing sho on and choosing the "ground" icon from the toolbar on the left is enough M " ve "Source " libraries by using "Add Library" button on the op	R=var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGP SOURCE SOURCSTI SPECIAL SWIT_RAV SWIT_REC TEX_INST THYRISTR TLINE XTAL ZETEX DI and choos M " ve "S	reak/BREAKUUT reak/BREAKUUT Particular of a second reak/BREAKUUT Particular of a second reak/BREAKUUT reak/BREAKUUT Particular of a second reak/BREAKUUT rea	aphic Normal Convert ackaging arts per Pkg: 1 ert: ype: Homogeneous urrent and voltat icon from the to y using "Add I	R? R? 1k we sources, oolbar on the Library" but	choosing sho left is enough. ton on the ope
ng the elements such as current and voltage sources, choosing sho on and choosing the "ground" icon from the toolbar on the left is enough M " ve "Source " libraries by using "Add Library" button on the op	R-var/ANA RAM8Kx1b RAM8Kx8b Libraries: SHINDNGF SOURCE SOU	horntory U Ever	aphic Normal Convert ackaging arts per Pkg: 1 art: I I I ype: Homogeneous urrent and voltat icon from the to y using "Add I mont Peoclelat	R? R? 1k w w w age sources, oolbar on the Library" but	choosing sho left is enough. ton on the ope

Place ground)
Symbol: * \$D_HI \$D_LO 0	OK Cancel Add Library Remove Library Help
CAPSYM Design Cache Source	

<u>IMPORTANT</u>: The name of the ground part must be "0" for Pspice.

• After adding the parts, by using "w" shortcut on the keyboard or cable icon on the toolbar on the right side, you can piece with the parts. Cables can be named with "Net Alias".

OrCAD Capture			
ile Edit View Place Macro PSpice Acc	essories <u>O</u> ptions <u>V</u>	<u>/</u> indow <u>H</u> elp	
16 6 6 6 6 6	R	•	
* 	I		
/ - (SCHEMATIC1 : PAGE1)			The second se
5	+		3
			₽
			· · · · · · · · · · ·
			<u>N1</u>
R4			· · · · · · · · · · L
· · · · · · · · · · · · · · · · · · ·	• • • • • • • • • •		
· · · · · · · · · · · · · · · · · · ·			
			· · · · · · · · · · /
			· · · · · · · · · · · · · · · · · · ·
	<mark>.</mark>		GND
	 R6 		· · · · · · · · · · · · <u></u>
	-		
	· · · IK · · · · · · · ·		
			· · · · · · · · · · · · · · · · · · ·
			imu imu
			
			· · · · · · · · · · · X
			🔨
			· · · · · · · · · · · · · · · · · · ·

FREQUENTLY USED PARTS

ELEMENT	ELEMENT NAME/LIBRARY	SYMBOL
DC Voltage Source	VDC / Source	0Vdc
DC Current Source	IDC /Source	0Adc
AC Voltage Source	VAC / SOURCE	1Vac 0Vdc
Sine Wave Source	VSIN / SOURCE	VOFF = VAMPL = FREQ =
Triangle Save Source Square Wave Source	VPULSE / SOURCE	V1 = V4 V2 = TD = TR = PW = PER =
Ground (reference voltage)		
Resistor	R / ANALOG	R1 0W-0 1k
Capacitor	C / ANALOG	C1 □
Inductor	L / ANALOG	L1 100H
741 OpAmp	uA741 / EVAL	U1 - 3 - 5 052 - 5 OUT - 6 2 - 5 051 - 1 wA741 - 7
Diode	D1N4148 / EVAL	D1 DH-0 D1N4148
Zener Diode	D1N5232 / EVAL	D2 D-D-D D1N5232
npn Bipolar Junction Transistor	Q2N3904 / BIPOLAR	Q10 Q2N3904
Voltage Source with E Voltage-Control	E/ANALOG	

Current Source with F Current Control	F/ANALOG	
Current Source with G Voltage- Control	G/ANALOG	
Voltage Source with H Current-Control	H/ANALOG	

• You can obtain the parts on the list or all the parts easily by writing on the following box if their libraries are added.

	•
- <u>*</u> =) \$ @ @ @ @ @	

SIMULATION AND GRAPHIC ADJUSTMENTS

• It is necessary doing simulation adjustments after the desired circuit are set up. For this, it is enough to click on the simulation buton shown in the following.

Macro	PSpice	Accessories	Options
è 🔁	20	C	
	I 🛓		g 🖉 🤅
		∿5 New Simulation	n Profile

• By using "New Simulation" window, you can give any name for your simulation. Do not use Turkish characters.

New Simulation Name: sweep1 Inherit From: none Root Schematic: SCHEMATIC1	
• Choose the proper analysis type by using "Analysis Type" list in the "Ana	ılysis" tab.
Simulation Settings - sweep1 X General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window Analysis type: Time Domain (Transient) Start saving data after: Output Start saving data after:	Enter the time value which
AC Sweep/Noise Bias Point Monte Larlo/Worst Lase AC Sweep/Noise Maximum step size: seconds	 what time the simulation will take.
The nodes wanted to be watched are signed by using "Voltage/Level Market	er".
• Simulation is started by clicking on the Run Spice.	
• For signing the wanted points in the graphic, you must find the point "toogle cursor" and then you can sign it with "mark".	nt by using
EEM206 Electric Circuits Laboratory II Experiment Booklet	13



• Also, you can make mathematical operations with the variables in the simulation by using "Add Trace" buton and you can transfer the results on the graphic.



• After you clicked on the "Add Trace" button, on the opening window you can see the variables on the left list and mathematical functions on the right list. For example, if you want to write the sum of the values of the currents on the R1 resistor and R2 resistor on the graphic, you must write "I(R1) + I(R2)" on the "Trace expression" area and click OK.



PSPICE A/D ANALYSIS TYPES AND OPTIONS

1. DC Sweep Analyse:

You can realize the simulation by making the current and voltage values constant in the circuit. Also, you can make simulations when there are current and voltage changings between certain two values. For **DC** analysis, you can make browsing as from initial value up to final value step by step. For every input voltage value, circuit is analysed and results are saved. The **DC** transfer characteristics of the circuit is extracted. Scanning can be linear, logarithmic or as part of certain values. Also you can realize, Secondary **DC Sweep** Analysis, **Monte Carlo**/Worst Case Analyse, **Parametric** Analysis, **Bias Point** Analysis as choices of **DC** analysis.

General Analysis Include File	s Libraries Stimulus	Options Data Collection P	robe Window
Analysis type:	Sweep variable		
DC Sweep 💌	Voltage source	Name:	
Options:	 Current source Global parameter 	Model type:	-
Primary Sweep	O Model parameter	Model name:	
Monte Carlo/Worst Case	C Temperature	Parameter name:	
Parametric Sweep	Sweep type		
Save Bias Point	• Linear	Start value:	
Load Bias Point		End value:	
Lineer tarama	C Logarithmic Dec	ade 🔽 Increment:	
Lugantinik tarama		increment.	
Değerler listesi	C Value list		

If it is containing multiple source DC scanning it is used **VSRC**, otherwise VDC is used for voltage. Similarly, **IDC** is used for current and if it contains multiple source DC scanning **ISRC** is used.

1.1. Primary Sweep:

We can tell this by using an example:

Let's scan V1 source between 0-20 V with 0,1 V value intervals and observe the changes on R1:



• After set up the circuit, click on the Edit Simulation Settings 🗖 button and choose the DC Sweep choice.

- Then, choose Voltage Source on Sweep Variable and write the source name on the Name part.
- Choose Linear from the Sweep Type part and enter the initial, final and increase values. Click OK.

Simulation Settings - pamukk	ale 🛛 📉
General Analysis Include Fil	es Libraries Stimulus Options Data Collection Probe Window
Analysis type:	Sweep variable
DC Sweep 💌	Voltage source Name: V1
Options:	O Current source O Global parameter
Primary Sweep	O Model parameter Model name:
Secondary Sweep Monte Carlo/Worst Case	O Temperature Parameter name:
Parametric Sweep Temperature (Sweep)	Sweep type
Save Bias Point	 Linear Start value: 0
Load Blas Point	C Leastitutia Decada El End value: 20
	Increment: 0.1
	C Value list
	Tamam İptal Uygula Yardım

• Circuit is simulated by clickin on the **Run Pspice .** The current change of R1 can be seen on the graphic. If the scattered source is current source instead of the voltage source, it must be used **Isrc**. You can add the graphics you want by using **Add Trace**.



1.2. Secondary Sweep:

Let's give an example. Let's scan the current and voltage source values together. Scan the V1 source between 0-20 V with 0,1V value intervals and I1 source 0-5A with 1A intervals. Observe the current changes of **R1**.



- After you set up the circuit, in addition to the steps of primary sweep, choose the **Secondary Sweep** in the **Simulation Settings** window.
- When click on the Secondary Sweep and choose Current Source from the Sweep Variable. Then, write the source name on the Name section. Choose Linear from the Sweep Type part and enter the *initial*, *final* and *increase* values.
- OK is selected and exit there. By clicking on the Run Pspice buton, simulation is realized and current changes of R1 can be seen on the graphic. By using Add Trace, the desired graphics can be added.

onnalation octango panlata	kale 📃 🔜
General Analysis Include F	iles Libraries Stimulus Options Data Collection Probe Window
Analysis type:	Sweep variable
DC Sweep 💌	O Voltage source Name: [1
Options:	Current source Model type:
Primary Sweep	O Global parameter Model name:
Secondary Sweep	O Temperature Parameter name:
Parametric Sweep	
Temperature (Sweep)	Sweep type
Load Bias Point	Einear End value: 5
	C Logarithmic Decade
	O Value list
	I amam Iptal Uygula Yardim
	1
4.0A	
2.0A	<u></u>
2 . 0A	<u></u>
2 . 0A	
2 . 0A	

2. AC Sweep/Noise Analyses:

The frequency response of the circuit can be extracted or changing of impedance with frequency by using AC scanning. Also, as options of AC analysis, Monte Carlo/Worst Case Analyses, Parametric Analysis, Bias Point Analysis can be done. For voltage in AC scanning VAC is used and for multiple sources VSRC is used. Similarly, in AC scanning IAC is used for current and for multiple source ISRC is used. Let's investigate AC analysis on a RLC circuit:

	R1 L1 50 1mh C1 1n
After set up the circuit, ch AC scanning. Choose OK button.	noose AC Sweep choice on Simulation Settings window for S. Simulation is realized by clicking on the Run Pspice
Simulation Settings - pamukka	ile 📃
Analysis type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	AC Sweep Type Linear Start Frequency: 10 Logarithmic End Frequency: 10MEG Decade Points/Decade: 50 Noise Analysis Enabled Output Voltage: I/V Source: Interval: Output File Options Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)
	Tamam İptal Uygula Yardım



- Voltage value on the capacitor changes according to the frequency on the graphic. This variation can be seen by using **Voltage Marker** placed on the graphic. Here frequency change is between 10Hz and 10MHz.
- For Noise analysis, choose the analizi için **Enabled** choice on the **Noise Analysis** section. Write the output voltage that you want to measure on the **Output Voltage** choice.
- Write a free input voltage or current that will be a input value for the measuring on the I/V Source section. Write frequency interval on the Interval choice.

3. Time Domain (Transient) & Fourier Transform Analyses:

If you want to see the changing of the variables in the circuit according to the time, this simulation mode must be used. At the end of the simulation, values of the variables are obtained as a function of time. Time domain analysis always starts from **t=0** and finals at the **Tstop** value with the steps given by user. By using time domain analysis, responses of the circuits to the input signals like sinusoidals or pulse can be examined (e.g rectifier, clipper, amplifier). As options of **Time Domain (Transient)** analysis, Monte Carlo/Worst Case Anaysis, Parametric Analysis, Bias Point Analysis can be done.

In the **Time Domain** analysis; *for voltage:* VSRC, VEXP, VPULSE, VPWL, VPWL_RE_FOREVER, VPWL_F_RE_FOREVER, VPWL_N_TIMES, VPWL_F_N_TIMES, VSFFM, VSIN sources, *for current:* ISRC, IEXP, IPULSE, IPWL, IPWL_RE_FOREVER, IPWL_F_RE_FOREVER, IPWL_F_RE_FOREVER, IPWL_N_TIMES, IPWL_F_N_TIMES, ISFFM, ISIN sources can be used.



Let's examine Time Domain analysis for a RC circuit.

- After set up the circuit, Analysis type is chosen as Time Domain on the Simulation Settings section.
- Then, for determining simulation time, write the duration of simulation in terms of seconds on **Run to Time**.

You can determine when the data will be taken after the simulation started by using **Start saving data after**.

General Analysis Include File Analysis type: Time Domain (Transient)	es Libraries Stimulus Options Data Collection Probe Window Run to time: 5 seconds (TSTOP)
Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	Start saving data after: 2 seconds Transient options Maximum step size: seconds Skip the initial transient bias point calculation (SKIPBP) Output File Options
	Tamam İptal Uygula Yardım
Click OK and exit. Start s	imulation by clicking on Run Pspice button.
206 Electric Circuits Labora	atory II Experiment Booklet

22



You can write the maximum allowable time step size on the part of **Transient Options** of **Maximum Step Size**. Also, you can see **Fourier analysis option** in the **Time Domain** (**Transient**) Analysis type of **Output Files icon**. Fourier Analysis calculates the **DC** and **Fourier** components of Transient analysis.

4. Bias Point (Working Point) Analysis:

Bias point saves the detailed Bas point data fort he simulation exit files. Also, as an option of Bias Point Analysis, it can realize temperature analysis. The reports data fort he exit files are: It lists all of the analog and digital point voltages, and it shows the currents between all of the voltage sources and their powers as you can see in the **Output Files** section. It realizes sensitivity analysis. It lists small signal parameters for all of the elements.

Bias Point	Uutput File Option	ins ailed bias point informati I semiconductors (.0P)	ion for nonlinear	controlled
General Setting ☐ Temperature (So ☐ Save Bias Point ☐ Load Bias Point	veep) Perform Ser Output varia Calculate sr From Input s To Output v	isitivity analysis (.SENS ible(s): nall-signal DC gain (.TF) :ource name: ariable:]	

5. Monte Carlo Analysis

It calculates the response of the circuit by changing the values of elements randomly which are described in a certain interval. You can run this analysis for defined number, it uses nominal values for first run and variable values for the others.

Options:	C Worst-case/Sensitiv Monte Carlo options	ity Oulput	variable: Vio	ut)	
General Settings	Number of runs	20			
Monte Carlo/Worst Case Parametric Sweep	Use distribution Bandom number seed:		Distribution	18 Dad/Save Monte Carlo Parame	der File
Temperature (Sweep) Save Bias Point	Save data from	All			
	Worst-case/Sensitivity Vary devices that have	options both DEV a	hdLOT 🚽	X/2.mcp	Browse
	Limit devices to type(s)	, ,		🔽 Save parameter values in file:	
	Save data from eac	h sensitivity ru	n 	abc.mcp	Biowse
		L Load/Save.		DK Cance	
	Tamam	İptal	Uygula		

6. Worst Case Analysis

It calculates the worst result for the maximum or minimum values of the elements with the closest value to the nominal. Addition to the circuit definition, it can determined that what is the ''worst'' definition and parameter tolerations. Tolerations data are defined in .MODEL definition, "worst" definition is defined on the 'Simulation Settings' window for Pspice model.

Analysis type: Time Domain (Transient) 💌 Indians	⊂ Monte Carlo	
General Setting: Conte CarloAWorst Cose Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	Colating Function The collating function is performed on an output variable (for example, V(1)). The result is listed in the output (.OUT) file only. Find the greatest difference from the nominal run (YM4X) T Threshold value: Evaluate only when the sweep variable is in the range: Vorst-Case direction Worst-Case direction The Low	OK Cancel

EXAMPLES

4 RC Circuit

Fort his example, a basic RC circuit will be impelemented.

• Set up the circuit in the following by considering the knowledge given before.



- Capacitor has an initial value for t=0 time. For entering this value give as 1 Volt click on the capacitor two times, so "Property Editor" is opened.
- For assigning a initial value with "Property Editor", find "IC" value and right click on the free space opposite of it and click on the "Display" choice.

<u> </u>		2 C	
SCHEMATIC1-eem	- <u>*</u>		Ø 99 Ø
New Row App	ly Display De	lete Property Fi	lter by: < Curre
	A		
		PAGE1	
Color	Default		
Designator			
Graphic	C.Normal		
IC	///////////////////////////////////////	77777	_
ID		Pivot	
Implementation		Edit	
Implementation Path		Delete Proper	ty
Implementation Type	<none></none>	Display	
Location X-Coordinate	410		
Location Y-Coordinate	110		
Name	101504		
Part Reference	C3		
PCB Footprint	RAD/CK05		
Power Pins Visible			
Primitive	DEFAULT		
PSpiceTemplate	C^@REFDES %1 %	2.9TOLE	
Reference	C3		
Source Library	D:\CADENCE\ORC	AD	
Source Package	C		
Source Part	C.Normal		
TOLERANCE			
Value	1n		

• After marking "Name and Value" choice in the incoming window, then confirm it by using OK buton.

Name: IC	Font Arial 7
Value:	Change Use Default
Display Format C Do Not Display C Value Only	Color Default
 Name and Value Name Only Both if Value Exists 	Rotation
OK	Cancel Help

EEM206 Electric Circuits Laboratory II Experiment Booklet

27

• After these steps, you can see the circuit as given in the following. By clicking on "IC" two times, you can write the desired 1V value.



• Simulation adjustments must be as seen below.

Simulation Settings - onu	r 🔀
General Analysis Include F Analysis type: Time Domain (Transient) Include F Options: Options: Include F Options: Include F Include F Include F Include F Include F Options: Include F Include F Include F Include F	Idea Libraries Stimulus Options Data Collection Probe Window Run to time: 0.1s seconds (TSTOP) Start saving data after: 0 seconds Transient options Maximum step size: 0.001 seconds Skip the initial transient bias point calculation (SKIPBP) Output File Options
	Tamam İptal Uygula Yardım

• After yo urun the simulation, you can obtain the graphic given in the following.



Replace the "Vdc" source with "Vsin" in the same circuit and adjust the value of capacitor as "10u". Place a "Voltage Marker" on the source for observing input source. The values of Vsin source as given below.



 If you adjust the "Run time" for 2 seconds in the simulation adjustments the graphic in the following can be obtained. Here, V(C2:1) is the value of capacitor voltage and V(R1:1) is the voltage value of input source.



• Finally, by replacing "Vsin" source with "Vpulse" source, we will be able to observe the behaviour of RC circuit under square pulse. After adding "Vpulse" source, adjustments must be as shown below.



• You can obtain this graphic when yo urun the simulation.



Full Wave Rectifier

Set up the circuit given below. If you are using full version PSPICE, you can add diodes from DIODE library.



Simulation adjustments

CUE: If you can not obtain smooth graphics, it means that you don't take enough sensitive value for "maximum step size". If you obtain irrelevant graphics although this, you may be taking the simulation duration too long or too short.



4 Parametric Circuit Element Usage

In this example, resistor value will change in the interval of desired values. Since the value of resistor changes, it is a potentiometer simulation.



- Firstly, take the value for the circuit element which we want to change its value as {rl}, the important thing is writing it in the braces, you can use any name. (be sure that you didn't change the "value" instead of name)
- Parça ekleme menüsünden (kısayol P) PARAM/SPECIAL'I çizim ekranına ekleyin
 . (SPECIAL kütüphanesi eklenmemişse, "add library " butonunu kullanarak ekle
 yebilirsiniz.)
- **3.** Open the properties window by two times clicking on the PARAM part. Here, write "RL" on the name place and 1k on the value part by using "New Column" buton.
- Choose Rl coloumn by clicking one time and then by right clicking open "display" menu. Enter the "Name and Value" choice.
- 5. Now, simulation adjustment will be done.

Adjustments

Analysis type: DC SweepOptions: Primary Sweep (Not Parametric Sweep!)Sweep variable: Global parameterParameter name:RL (name in the braces)Sweep Type: Linear

Enter the values of Start, End, and Increment. These values determine the value interval that parametric variable will take and the increment of the value. 0,1,10k,100 are used respectively.

CUE: Be carefull about that resistor value doesn't start from 0.

6. Finally, by placing a "probe" anywhere you want, you can observe the unstable value.



4 Bode diagram of Inductive Lowpass Filter

Lowpass filter transfers the signals under the cut off frequency and doesn't transfer above the cut off frequency ideally. Bode diagram is the drawing of gain versus frequency. For obtaining this graphic in PSPICE environment, "AC Sweep" simulation will be used. Firstly, set up the circuit given in the following.

1Vac 0Vdc	$ \begin{array}{c} L1 \\ 1 \\ 3H \\ V \\ V1 \\ \hline 0 \\ \end{array} $ $ \begin{array}{c} R1 \\ 1k \\ \hline 0 \\ \end{array} $
Simulation Settings - Bode	
General Analysis Include Files Analysis type: AC Sweep/Noise Options: Options: Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	Libraries Stimulus Options Data Collection Probe Window AC Sweep Type

In Ac Sweep analysis, for obtaining bode diagram, you must choose logarithmic option and then you must enter the frequency values which will be scanned.

CUE: You must be carefull about "Start Frequency" value shouldn't be değerinin 0 ! The obtained graphic is as given below.



OPAMP Usage

For this example, it will be designed a complementer OPAMP system which has a gain about 1. This means that the input signal will be multiplied by -1 in the output. Set up the circuit given in the following. You can find LM741 in the OPAMP library for full version PSPICE.



CUE: Here, supply voltage is given for 7 and 4 numbered jacks of LM741 OPAMP. Be carefull that the 12V source is negative in the 4 numbered input. Also, two "voltage probes" are used in order to see input and output simultaneously.

Simulation settings

As similar to other examples, "Transient Analysis" will be used.

Simulation Settings - opar	mp	×
General Analysis Include Fill Analysis type: Time Domain (Transient) Image: Comparison of the second	Files Libraries Stimulus Options Data Collection Pro Run to time: 2s seconds (TSTO) Start saving data after: 0 seconds Transient options Maximum step size: 0.1 seconds Skip the initial transient bias point calculation (SK Output File	be Window 2) IPBP) Options
<u> </u>	Tamam İptal Uygula	Yardım

In the graphic, V(V1) is input signal and V(R1) is output signal.



INVESTIGATING TRANSIENT ANALYSIS FOR RL, RC AND RLC CIRCUITS

If you want to investigate an electrical circuit in time domain, you must write either entegrated differential equations or state equations of that circuit. As it is known, the solution for differential equations has two parts: The first part of the solution is determined by the initial conditions in the circuit and the second part of the solution is determined by the sources in the circuit. These parts of the solution are named as Natural and Forced Solution respectively. For an asymptotically stable circuit in other words for a circuit which as $t \rightarrow \infty$ state transition matrix f(t)goes to zero, natural solution arrives to zero and forced solution arrives to natural solution. In a clear manner, for an asymptotically stable circuit after a while for investigating the circuit, exact solution is nearly equal to the particular solution. For an asymptotically stable circuit, exact solution is composed of transient solution and steady solution. For an asymptotically stable circuit, the homogen solution of differential equation system is named as transient, particular solution is named as steady solution. Although the transient solution is very large at first, after a while for circuit running, it gets smaller and approaches to zero. Steady solution is the solution which will continue so far as sources present in the circuit. Although its effect is too short, transient solution is important for determining the circuit's elements. It is important investigating circuits in the time domain in terms of understanding their principles and functions.

After that, we will consider simple *RC*, *RL* and *RLC* circuits and investigate the solutions of these circuits in case of they are stimulated by step, pulse and square wave sources.

RC Circuit:

Let's consider RC circuit given in Figure 1. State equations of this circuit are,



If e(t)=Eu(t) is step function in the equation (1), the solution of the circuit is;

$$v_C(t) = e^{-t/RC} v_C(0) + E(1 - e^{-t/RC})$$
(2)

If $v_C(0) = 0$, the variance of voltages at the *C* and *R* are shown in Figure 2.





If you remove e(t) source and its place is short circuited, (e(t) = 0 in equation (1)), the solution of equation (1),

$$v_{C}(t) = e^{-t/RC} v_{C}(0)$$
(3)

Variance of this voltage according to time is given in the Figure 3. *RC* is the time constant of the circuit in equation (2) and (3); if you put *R* as ohm and *C* as farad, its unit is seconds.



e(t) source is a pulse source shown in the Figure 4a, (e(t) = E[u(t) - u(t-D)]), vC(0) = 0 and voltage of the capacity;

$$v_C(t) = E(1 - e^{-t/RC})u(t) - E(1 - e^{-(t - \Delta)/RC})u(t - \Delta)$$
(4)

The voltages of C capacity and R resistor are shown as Figure 4b and Figure 4c.



If e(t) source is a pulse source as shown in Figure 4a, one can think circuit in Figure 1 as Figure 5.



Figure 5

For this case, it is necessary to indicate that A key in Figure 5 is kept on 1 position for D time, then it is taken to position 2. When A key is on position 1, in the interval $0 \le t \le \Delta$, the voltage expression of C capacity is defined by curve from 0 to Δ shown in Figure 4b. If $t = \Delta$, the voltage of the capacity is $v_C \Delta = v_C(\Delta)$. When t equals to Δ , the capacity will start to decharge since A key is taken to position 2. The curve given in Figure 3 can be used for t ³ Δ but it is necessary to

shift this curve by Δ and replace $v_C(0)$ with $v_C(\Delta)$. The operation done on the Figure 3 is equal to replacing t with $(t - \Delta)$ and $v_C(0)$ with $v_C(\Delta)$ in equation (3). These explanations can be summarized by the equation given below;

$$v_{C}(t) = \begin{cases} E(1 - e^{-t/RC}) & ; 0 \le t < \Delta \\ v_{C}(0) e^{-(t - \Delta)/RC} & ; t \ge \Delta \end{cases}$$
(5)

In equation (5), $v_C(0) = E(1 - e^{-(t-\Delta)/RC})$. If E, R, C and t, Δ is given as numerically, $v_C(0)$ can be computed as numerically, also. Similarly, $v_R(t)$ can be found as given below;

$$v_{R}(t) = \begin{cases} E e^{-t/RC} & ; 0 \le t < \Delta \\ v_{R}(t) = () \\ v_{c}(0) e^{-(t - \Delta) / RC} & ; t \ge \Delta \end{cases}$$

Let's assume that e(t) source in Figure 1 is a rectangular wave source as in Figure 6. It is necessary to compare the R.C time constant with the period of rectangular pulse when it is started to investigate the voltage changing on the C capacity.



a) If $R.C \ll T$, capacity is charged during T1 time with primary pulse and it decharges until the second pulse in the T2 time interval. Since the time constant is small, it can be assumed that voltage in the capacity edges reaches to zero in the T2 time. The phenomena is repeated for second and afterward pulses as the same manner; voltage of the capacity changes as in Figure 6b periodically. By remarking that this voltage is periodic and replacing Δ with T1, it can be defined with equation (5). The variance of voltage on the R resistors edges is as shown in Figure 6c. b) If *RC* time constant is enough large to compare with the period, the variance of v_c(t) will be as Figure 7. Capacity will charge with first pulse, during T2 time, second pulse will come before

the capacity discharge exactly. This situation will continue so for the pulses in the beginning. After a certain number of pulses, variance of the voltage on the capacity edges will be periodic.



Figure 7

The time T_k when the phenomena became periodic can be considered as beginning. Because of this, for explain the phenomena equations (2) and (3) can be used;

$$v_{C}(t) = \begin{cases} e^{-t/RC} E_{1} + E(1 - e^{-t/RC}) ; 0 \le t < T_{1} \\ E_{2} e^{-(t - T_{1})/RC} ; t \ge T_{2} \end{cases}$$
(6)

If it is assumed that $v_C(T1) = E2$ at t = T1 time and $v_C(T) = E1$ at t = T1+T2 time, by using equation (6);

$$E_{2} = E_{1}e^{-T1/RC} + E(1 - e^{-T1/RC})$$

$$E_{1} = E_{2}e^{-T2/RC}$$
(7)

From this;

$$E_{2} = E \quad \frac{1 - e^{-T1/RC}}{1 - e^{-T/RC}} \qquad E_{1} = E \quad \frac{e^{-T2/RC} - e^{-T/RC}}{1 - e^{-T/RC}}$$
(8)

is found. If the pulse duration of rectangular wave is T1, period is T, amplitude is E and numerical values of R and C are given by using equations in (8), E1 and E2 are calculated numerically and also (6) equations which give the variation of $v_{C}(t)$ with time can be found numerically. If $v_{C}(t)$ is clear, $v_{R}(t)$ can be found as given in Figure 8.



c) If RC >> T, (6) and (8) equations are valid again. In other words, if RC >> T,

 $e^{-t/RC} \cong 1 - t/RC$ $0 \le t < T_1$

$$e^{-(t-T_1)/RC} \cong 1 - (t-T_1)/RC$$
 $T_1 \le t < T_2$

can be written. So, the variation of $v_{C}(t)$ will be linear as shown in Figure 9.



RL Circuit :

Let's consider RL circuit in Figure 10.



The structure of equation (9) is same as equation (1). You can apply all steps for RL circuit like as RC circuit. The circuit shown in Figure 11 can be used for modelling an energy distribution system simply. Here, e(t) is a source with 50Hz frequency.



In this case, particular solution of equation (9) is;

$$i_L(t) = \frac{\sqrt{2E}}{Z} \sin(\omega t - \psi)$$
(11)

This equation is;

$$Z = \sqrt{R^{2} + \omega^{2}L^{2}}$$

$$\psi = \operatorname{arctg}(\frac{\omega L}{R})$$

$$(12)$$

The solution of the homogen part of the (9) equation is,

 $i_L(t) = I_h e^{-(RL)t}$ (13)

Exact solution is;

$$i_L(t) = I_h e^{-(R/L)t} + \frac{\sqrt{2E}}{Z} \sin(\omega t - \psi)$$
(14)

Before it is short circuited, there is no current transition on the inductance, $i_L(0)$ is 0. This first condition is yield in equation (4),

$$I_h(t) = \frac{\sqrt{2E}}{Z} \sin \psi \tag{15}$$

is obtained. So, equation (14) can be written as;

$$i_L(t) = \left(\frac{\sqrt{2E}}{Z}\sin\psi\right)e^{-(R/L)t} + \frac{\sqrt{2E}}{Z}\sin(\omega t - \psi)$$
(16)

First term is transient solution and second term is steady solution in equation (16). After a time as five times of (L/R), transient solution can be ignored. After a time stated from the short circuit current is appeared, short circuit current pass whose effective value of the circuit equals to (E/Z). $i_L(t)$ gets its maximum value for $\omega t = \pi/2 + \psi$. This value of current is named as short circuit pulse current and defined as equations 17a and 17b;

$$I_{s} = \frac{\sqrt{2}E}{Z} \left(1 + \frac{\omega L}{Z} e^{-\frac{R}{\omega L} \left(\frac{\pi}{2} + \psi\right)} \right)$$
(17a)

$$\mathbf{I}_{s} = \sqrt{2} \mathbf{I}_{k}^{"} \mathbf{x} \tag{17b}$$

Here, I_k " is the value of steady short circuit current and equals to E/Z.

$$x = 1 + \left(1/\sqrt{\left(\frac{R}{\omega L}\right)^2 + 1}\right) e^{-\left(\frac{R}{\omega L}\left(\frac{\pi}{2} + \psi\right)\right)}$$
(18)

As seen from equation (18), coefficient x is a constant dependent to R/(wL). If R=0, x value will be 2. So, the current passes in the circuit because of the transient solution is the twice of steady short circuit current. Consider that here, generator is modelled with a voltage source and an internal resistance and an internal inductance connected series to it.

RLC Circuit:

Let's consider RLC circuit given in Figure 12.



Figure 12

State equations of this circuit are,

$$\frac{d}{dt} \begin{bmatrix} v_{C}(t) \\ i_{L}(t) \end{bmatrix} = \begin{bmatrix} 0 & (1/C) \\ (-1/L) & (-R/L) \end{bmatrix} \begin{bmatrix} v_{C}(t) \\ i_{L}(t) \end{bmatrix} + \begin{bmatrix} 0 \\ (1/L) \end{bmatrix} e(t)$$
(19)

Characteristic equation of this equation system is,

$$p^2 + 2\zeta \,\omega_o p + \omega_o^2 = 0 \tag{20}$$

$$\omega_o = \frac{1}{LC} \quad , \quad \zeta = \frac{R}{2} \sqrt{\frac{C}{L}} \tag{21}$$

By taking $v_c(0) = 0$, $i_L(0) = 0$, for three situation the variance of $v_c(t)$ with time is drawn in Figure 13 rudely.



As seen in Figure 13, the voltage on the capacity edges goes to E for both of the three condition. Let's consider e(t) is a source in the form of pulse as E(u(t)-u(t-D)). In this case, it will be easy to interpret the circuit as Figure 14. After A key is kept on (1) position for D time, it will be taken to position (2). Assume that the initial conditions are $v_C(0)$, $i_L(0)$. During the key is kept on position (1), in the interval of $0 \le t < \Delta$, by considering value of z, one can use either equation (23) or (24) and (25). At the time $t= \Delta$, we can find $i_L(\Delta)$ and $v_C(\Delta)$ by using these equations. If key is on the position (2), since there is no source in the circuit, only natural solution is obtained. For $t > \Delta$, in order to find $v_C(t)$ and $i_L(t)$, it can be used one of the equations of (23), (24), (25) in terms of z values. For $t > \Delta$, in order to determine $v_C(t)$ and $i_L(t)$, it is enough to write $(t - \Delta)$ instead of t; $v_C(0)$ and $i_L(0)$ instead of $v_C(\Delta)$ and $i_L(\Delta)$; zero instead of E. If we assume that Δ is big enough, the change of $v_C(t)$ in terms of ζ will be as shown in Figure 15.



LABORATORY EXPERIMENTS

EXPERIMENT-I: Frequency, Amplitude and Phase Measurement for Basic
AC /RC CircuitsEXPERIMENT-III: Basic AC RC and RL CircuitsEXPERIMENT-III: Serial AC RLC CircuitsEXPERIMENT-IV: Parallel RLC CircuitsEXPERIMENT-V: Power in AC CircuitsEXPERIMENT-VI: Application of TransformerEXPERIMENT-VIII: Balanced Three-Phase Y-Y Connected CircuitsEXPERIMENT-VIIII: Balanced Three-Phase Δ-Δ Connected CircuitsEXPERIMENT-VIIII: Passive Filters

I. EXPERIMENT: Frequency, Amplitude and Phase Measurement for Basic AC /RC Circuits

PURPOSE: The purpose of this experiment is to investigate amplitude and phase for serial AC RL and RC circuits. Theorical knowledge about this subject is given in EEM 204 lecture.

PRELIMINARY STUDY: All formulas will be derivatized and each values will be calculated for the circuit given below. So you can compare the theorical values calculated and experimental values measured during the experiment and write conclusion at the end of the report about the differences between these values you obtained.

EXPERIMENT:

- 1. Generate $v_s(t)=10\cos(2000\pi t)$ signal by using function generator and show it on the oscilloscope screen (channel 1).
- 2. Design the circuit given in the figure: Show Is, V_{C1} and V_{R1} on Channel 2.
- **3.** Draw I_s, Vs, V_{C1} ve V_{R1}. Determine the frequency, period, amplitude and phase values by measuring from oscilloscope.
- 4. Measure the RMS values of I_s , Vs, V_{C1} and V_{R1} by using multimeter.

<u>CIRCUIT DIAGRAM</u> :



 $R_1=330 \Omega$ $C_1=1,5 \mu F$

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

II. EXPERIMENT: Basic AC RC and RL Circuits

PURPOSE: The purpose is to learn about basic RC and RL AC circuits analysis in this experiment.

PRELIMINARY STUDY:

1. Find the V_R and V_C voltages of basic AC-RC circuit given in Figure 1 by using phasor representation. Input AC voltage' frequency and values of elements used in the circuit are given in the table. $V_S=10Cos(2\pi ft)$



f (frequency)	100 Hz, 1kHz, 5kHz
С	1µF
R ₁	1 kΩ, 10 kΩ

Figure 1. RC Circuit

2. Find the V_R and V_L voltages of basic AC-RC circuit given in Figure 1 by using phasor representation. Input AC voltage' frequency and values of elements used in the circuit are given in the table. $V_s=10Cos(2\pi ft)$



f (frequency)	100 Hz, 1kHz, 2kHz, 5kHz
L	100 mH
R	330 kΩ

Figure 2. RL Circuit

EXPERIMENT :

Design the circuit given in Figure 1 on the breadboard. Show the voltage V_S with V_L,V_R voltages on the oscilloscope screen by changing input AC voltage frequencies for 100 Hz,1kHz, 2kHz, 5kHz. Determine how measures differ if R and C values are changed.

2. Set up the circuit given in Figure 2 on the breadboard. Show the voltages V_L,V_R with V_S on the oscilloscope screen by changing frequency values of input AC voltage for 100 Hz,1kHz, 2kHz, 5kHz. Draw the graphics on the squared paper. Determine how the measurements differ by changing R and C values.

CONCLUSION and COMMENTS

Compare and interpret the results you obtained from experiments. Express the results clearly.

III. EXPERIMENT: Serial AC RLC Circuits

PURPOSE: The purpose of this experiment is to apply voltage divider rule for serial AC circuits. The theorical knowledge about this subject is given in the content of EEM 204 lesson. Voltage divider circuit will be analysed first mathematically, then experimentally. Lastly, the mathematically and experimental results will be compared.

PRELIMINARY STUDY:

In the circuit below $V_s(t)$ is defined as $V_s(t)=5cos(1000\pi t)$.

- Calculate the amplitude and phase angles for V₁, V₂, V_{R2} by using the values given in the table.
- 2) Calculate the amplitude and phase angles for current Is.



Resistor	Value
R ₁	1 kohm
R ₂	0.82 kohm
L ₁	100 mH,1mH
C ₂	0.2 μF

EXPERIMENT:

- 1. Set up the circuit in the figure and obtain Vs voltage on the oscilloscope.
- 2. Measure the amplitude or phase angles of V_2 and V_{R2} according to Vs (Channel 1). Note: Connect the ground node of oscilloscope to E node.
- **3.** Measure the amplitude or phase angles of V₁ according to Vs (Channel 1). Note: Connect the ground node of oscilloscope to A node.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

IV. EXPERIMENT: Parallel RLC Circuits

PURPOSE: The purpose of this experiment is to apply voltage divider rule for parallel AC circuits. The theorical knowledge about this subject is given in the content of EEM 204 lesson. Voltage divider circuit will be analysed first mathematically, then experimentally. Lastly, the mathematically and experimental results will be compared.

PRELIMINARY STUDY:

In the circuit below $V_s(t)$ is defined as $V_s(t)=5\cos(1000\pi t)$.

- 1. Calculate the amplitude and phase angles for V₁, V₂, V_{R2} by using the values given in the table.
- 2. Calculate the amplitude and phase angles for current Is.



Resistor	Value
R ₁	1 kohm,10kohm
R ₂	10 kohm
L ₁	1mH,100mH
C ₂	1 μF,10μF

EXPERIMENT:

- 1. Set up the circuit in the figure and obtain Vs voltage on the oscilloscope.
- 2. Measure the amplitude or phase angles of V_2 and V_{R2} according to Vs (Channel 1).

3. Measure the amplitude or phase angles of V_1 according to Vs (Channel 1).

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

V. EXPERIMENT: Power in AC Circuits

PURPOSE: The purpose of this circuit is investigating the power analysis in AC circuits.

EXPERIMENT :

- 1. Set up the circuit given in Figure 1 except capacitor. Use values in the table. $V_s(t)=10cos(2000\pi t)$.
- 2. Calculate voltage Vs and current I_s . Calculate phase difference between V_{R1} and Vs.
- 3. Connect capacitor as seen in the figure and repeat step 2.
- 4. Interpret the results.

R ₁	100 ohm
R ₂	1 kohm
L_1	100 mH
C ₁	1 μF





CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

VI. EXPERIMENT: Application of Transformer

PURPOSE: In this experiment, voltage, current and power analysis of basic transformer circuit will be done theorically and experimentally.

PRELIMINARY STUDY:

- 1. In the figure below Vs(t) is defined as Vs(t)= $5\cos(2000\pi t)$. Find V₂ and I₂.
- 2. Determine the average power and reactive power which source generated.
- 3. Determine the average power and reactive power consumed on the load.



EXPERIMENT:

- 1. Set up the circuit given in the figure. Measure the amplitude, phase and frequency of V_2 , I_2 .
- 2. Measure the average power and reactive power that source generated.
- 3. Measure the average power and reactive power consumed on the load.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

VII. EXPERIMENT: Balanced 3-Phase Y-Y Connected Circuits

PURPOSE: The purpose of this experiment is to investigate voltage, current and power analysis for 3-phase Y-Y connected circuits experimentally and theoretically.

PRELIMINARY STUDY:



- **1.** F is 2000 Hz for the 3-phase circuit given in the figure. Determine the line and phase voltage and current for the load (as amplitude and phase).
- 2. Find average power that each load used up. Find the average power that load used up.
- **3.** Find the total average power that load used up by using 2-wattmeter method. Compare the result with you found in step 2.
- **4.** Find the percent ratio of average power that got lost in the transmitting line (single phase) to the power generated by source (single phase).

EXPERIMENT:

- **1.** F is 2000 Hz for the 3-phase circuit given in the figure. Determine the line and phase voltage and current for the load (as amplitude and phase).
- 2. Measure the power that each phase load used up (According to the ground). Measure the total average power that load used up.
- **3.** Find the total power that load used up by using 2-wattmeter method. Compare the result with you found in step 2.

4. Measure the average power lost in the transmitting line (single phase) and power generated by source (single phase). Find the loss ratio.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

VIII. EXPERIMENT: Balanced 3-Phase Δ - Δ Connected Circuits

PURPOSE: The purpose of this experiment is to investigate voltage, current and power analysis for 3-phase Δ - Δ connected circuits experimentally and theoretically.



PRELIMINARY STUDY:

For the 3-phase circuit in the figure, f is 2000 Hz.

- 1. Determine the currents I_{AB} , I_{BC} , I_{CA} , I_{aA} , I_{bB} , I_{cC} , I_{ba} , I_{cb} , I_{ac} .
- 2. Find the voltages V_{AB} , V_{BC} , V_{CA} .
- **3.** Calculate the power in each phase of the load.

EXPERIMENT:

- 1. Measure the currents I_{AB} , I_{BC} , I_{CA} , I_{aA} , I_{bB} , I_{cC} , I_{ba} , I_{cb} , I_{ac} .
- **2.** Measure voltages V_{AB} , V_{BC} , V_{CA} .
- **3.** Measure the power in each phase of the load.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.

IX. EXPERIMENT: Passive Filters

PURPOSE: The purpose of this experiment is to investigate the responses of passive filters for different amplitudes and frequencies.

PRELIMINARY STUDY:

Determine the transfer function for the circuits 1, 2, 3 and 4 and calculate the cut-off frequencies.

EXPERIMENT:

A) Low-Pass Filter

- Design a low-pass filter circuit by using 10 kohm and 1µF for R and C, respectively. Adjust the amplitude of signal source for 4 V (p-p), measure the output voltages for 1, 2, 4, 6, 8, 10, 12, 14, 16, 18, 20, 40, 60, 80 and 100 (kHz) input frequencies.
- 2) Convert the voltage values you measured to decibel by using the formula given below,

dB=20 log (V_0/V_i) (V_i is constant at 4 V (p-p))

- 3) Draw the decibel values according to frequency by using semilog graphic paper.
- 4) Determine the actual cut off frequency by using drawing.

B) High-Pass Filter

Design a high-pass filter circuit with C=1 μ F and R=4,7 kohm and realize the process steps 2-4 given above.

C) Band-Pass Filter

Design a band-pass filter circuit with $C_1=1 \mu F$, $C_2=1 \mu F$ and $R_1=10$ kohm, $R_2=10$ kohm and realize the process steps 2-4 given above.

D) Band Stop Filter

Design a band-stop filter circuit C_1 =1000 pF, C_2 =1000 pF, C_3 =2000 pF and R_1 =47 kohm, R_2 =47 kohm, R_3 =47 kohm and realize the process steps 2-4 given above.

CONCLUSION and COMMENTS

1. Report all the procedures and measurements done in the experiment section. Add your comments.

2. Compare the theorical and measurement values of the circuit.