CIRCUIT SIMULATION

LABORATORY MANUAL

B.Tech. Semester

Subject Code: BEE-351

Session: 2024-25, Odd Semester

Name:	
Roll. No.:	
Group/Branch:	

DRONACHARYA GROUP OF INSTITUTIONS
DEPARTMENT OF EEE
#27 KNOWLEDGE PARK 3

GREATER NOIDA

AFFILATED TO Dr. ABDUL KALAM TECHNICAL UNIVERSITY, LUCKNOW

Table of Contents

Vision and Mission of the Institute

Vision:

"Dronacharya Group of Institutions, Greater Noida aims to instill core human values and facilitating competence to address global challenges by providing Quality Technical Education."

Mission:

M1: Enhancing technical expertise through innovative research and education, fostering creativity and excellence in problem-solving.

M2: Cultivating a culture of ethical innovation and user-focused design, ensuring technological progress enhances the well-being of society.

M3: Equipping individuals with the technical skills and ethical values to lead and innovate responsibly in an ever-evolving digital landscape.

Vision and Mission of the Department

VISION

To be a Centre of Excellence in Globalizing Education and Research in the field of Electrical and Electronics Engineering.

MISSION

M1: To empower technocrats with state-of-art knowledge to excel as eminent electrical engineers with multi-disciplinary skills.

M2: To emphasize social values and leadership qualities to meet the industrial needs, societal problems and global challenges.

M3: To enable the technocrats to accomplish impactful research and innovations.

Programme Educational Objectives (PEOs)

PEO 1. To foster strong knowledge in basic sciences and electrical engineering that enable technocrats to have successful careers.

PEO 2. Imbibed with the state of art knowledge to adapt ever transforming technical scenario.

PEO 3. Inspire engineers to provide innovative solutions to real-world challenging problems by applying electrical and electronics engineering principles.

Programme Outcomes (POs)

PO1: Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

PO2: Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

PO3: Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.

PO4: Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

PO5: Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PO6: The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7: Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO8: Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

PO9: Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO10: Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

PO11: Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

PO12: Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

Program Specific Outcomes (PSOs)

PSO1: Graduates will be capable to gain knowledge in diverse areas of electrical and electronics engineering and apply that to a successful career, entrepreneurship and higher education.

PSO2: Enhance the competence of graduates to design and analyze systems used in advanced power applications, renewable energy, electrical drives in allied technical fields.

PSO3: Graduate will use advance tools to analyze, design and develop electrical and electronic systems for feasible operation and meet the industry requirements.

University Syllabus

1) Verification of principle of Superposition with AC sources using Multisim/ PSPICE.
2) Verification of Thevenin and Maximum Power Transfer theorems in AC Circuits using Multisim/ PSPICE.
3) Verification of Norton theorems in AC Circuits using Multisim/ PSPICE.
4) Verification of Tellegen's theorem for two networks of the same topology using Multisim/ PSPICE.
5) Determination of Z and h-parameters (DC only) for a network and computation of Y and ABCD Parameters using Multisim/ PSPICE.
6) Determination of driving point and transfer functions of a two port ladder network and verify with theoretical values using Multisim/ PSPICE.
7) Determination of transient response of current in RL and RC circuits with step voltage input.
8) Determination of transient response of current in RLC circuit with step voltage input for under damped, critically damped and over damped cases.
9) Determination of image impedance and characteristic impedance of T and Π networks, using O.C. and S.C. tests.
10) Verification of parameter properties in inter-connected two port networks: series, parallel and cascade using Multisim/ PSPICE.
11) Determination of frequency response of a Twin – T-notch filter.
12) To determine attenuation characteristics of a low pass / high pass active filters.

Course Outcomes (COs)

Upon successful completion of the course, the students will be able to:

CO 1	Apply the knowledge of basic circuital law, nodal and mesh analysis for given circuit.
CO 2	Analysis of the AC and DC circuits using simulation techniques.
CO 3	Analysis of transient response of AC circuits.
CO 4	Evaluation and analysis of two-port network parameters.
CO 5	Estimation of parameters of different filters.

CO-PO Mapping

	PO1	**PO2**	**PO3**	**PO4**	**PO5**	**PO6**	**PO7**	**PO8**	**PO9**	**PO10**	**PO11**	**PO12**
CO 1	3	-	-	-	1	-	-	-	1	2	-	2
CO 2	3	2	-	-	2	-	-	-	1	2	-	2
CO 3	2	2	2	2	-	-	-	-	1	1	-	1
CO 4	2	-	2	2	2	-	-	-	1	1	-	1
CO 5	1	2	3	2	2	-	-	-	1	1	-	2
Course Correlation mapping	**2.2**	**1.2**	**1.4**	**1.2**	**1.4**	**-**	**-**	**-**	**1**	**1.4**	**-**	**1.6**

Correlation Levels: High-3, Medium-2, Low-1

CO-PSO Mapping

	PSO1	**PSO2**	**PSO3**
CO 1	2	3	1
CO 2	2	3	1
CO 3	2	3	1
CO 4	2	3	1
CO 5	2	3	1

Course Overview

The significance of the Electrical Circuit Simulation Lab is renowned in the various fields of engineering applications. For an Electrical Engineer, it is obligatory to have the practical ideas about the Electrical Circuits and Simulation. By this perspective we have introduced a Laboratory manual cum Observation for Electrical Circuits and Simulation. The manual uses the plan, cogent and simple language to explain the fundamental aspects of Electrical Circuits and Simulation in practical. The manual prepared very carefully with our level best. It gives all the steps in executing an experiment.

List of Experiments mapped with COs

Si No.	Name of the Experiment	Course Outcome
1	Verification of principle of Superposition with AC sources using Multisim/ PSPICE.	CO 1
2	Verification of Thevenin and Maximum Power Transfer theorems in AC Circuits using Multisim/ PSPICE.	CO 1
3	Verification of Norton theorems in AC Circuits using Multisim/ PSPICE.	CO 2
4	Verification of Tellegen's theorem for two networks of the same topology using Multisim/ PSPICE.	CO 2
5	Determination of Z and h-parameters (DC only) for a network and computation of Y and ABCD Parameters using Multisim/ PSPICE.	CO 3
6	Determination of driving point and transfer functions of a two port ladder network and verify with theoretical values using Multisim/ PSPICE.	CO 2
7	Determination of transient response of current in RL and RC circuits with step voltage input.	CO 4
8	Determination of transient response of current in RLC circuit with step voltage input for under damped, critically damped and over damped cases.	CO 2
9	Determination of image impedance and characteristic impedance of T and Π networks, using O.C. and S.C. tests.	CO 3
10	Verification of parameter properties in inter-connected two port networks: series, parallel and cascade using Multisim/ PSPICE.	CO 2

DOs and DON'Ts

DOs

1. Login-on with your username and password.
2. Log off the computer every time when you leave the Lab.
3. Arrange your chair properly when you are leaving the lab.
4. Put your bags in the designated area.
5. Ask permission to print.

DON'Ts

1. Do not share your username and password.
2. Do not remove or disconnect cables or hardware parts.
3. Do not personalize the computer setting.
4. Do not run programs that continue to execute after you log off.
5. Do not download or install any programs, games or music on computer in Lab.
6. Personal Internet use chat room for Instant Messaging (IM) and Sites is strictly prohibited.
7. No Internet gaming activities allowed.
8. Tea, Coffee, Water & Eatables are not allowed in the Computer Lab.

General Safety Precautions

Precautions (In case of Injury or Electric Shock)

1. To break the victim with live electric source, use an insulator such as fire wood or plastic to break the contact. Do not touch the victim with bare hands to avoid the risk of electrifying yourself.
2. Unplug the risk of faulty equipment. If main circuit breaker is accessible, turn the circuit off.
3. If the victim is unconscious, start resuscitation immediately, use your hands to press the chest in and out to continue breathing function. Use mouth-to-mouth resuscitation if necessary.
4. Immediately call medical emergency and security. Remember! Time is critical; be best.

Precautions (In case of Fire)

1. Turn the equipment off. If power switch is not immediately accessible, take plug off.
2. If fire continues, try to curb the fire, if possible, by using the fire extinguisher or by covering it with a heavy cloth if possible isolate the burning equipment from the other surrounding equipment.
3. Sound the fire alarm by activating the nearest alarm switch located in the hallway.
4. Call security and emergency department immediately:

Emergency : 201 (Reception)

Security : 231 (Gate No.1)

Guidelines to students for report preparation

All students are required to maintain a record of the experiments conducted by them. Guidelines for its preparation are as follows: -

1) All files must contain a title page followed by an index page. ***The files will not be signed by the faculty without an entry in the index page.***

2) Student's Name, Roll number and date of conduction of experiment must be written on all pages.

3) For each experiment, the record must contain the following

(i) Aim/Objective of the experiment
(ii) Pre-experiment work (as given by the faculty)
(iii) Lab assignment questions and their solutions
(iv) Test Cases (if applicable to the course)
(v) Results/ output

Note:

1. Students must bring their lab record along with them whenever they come for the lab.

2. Students must ensure that their lab record is regularly evaluated.

Lab Assessment Criteria

An estimated 10 lab classes are conducted in a semester for each lab course. These lab classes are assessed continuously. Each lab experiment is evaluated based on 5 assessment criteria as shown in following table. Assessed performance in each experiment is used to compute CO attainment aswell as internal marks in the lab course.

Grading Criteria	Exemplary (4)	Competent (3)	Needs Improvement (2)	Poor (1)
AC1: Designing experiments	The student chooses the problems to explore.	The student chooses the problems but does not set an appropriate goal for how to explore them.	The student fails to define the problem adequately.	The student does not identify the problem.
AC2: Collecting data through observation and/or experimentation	Develops a clear procedure for investigating the problem	Observations are completed with necessary theoretical calculations and proper identification of required components.	Observations are completed with necessary theoretical calculations but without proper understanding. Obtain the correct values for only a few components after calculations. Followed the given experimental procedures but obtained results with some errors.	Observations are incomplete. Lacks the appropriate knowledge of the lab procedures.
AC3: Interpreting data	Decides what data and observations are to be collected and verified	Can decide what data and observations are to be collected but lacks the knowledge to verify	Student decides what data to gather but not sufficient	Student has no knowledge of what data and observations are to be collected
AC4: Drawing conclusions	Interprets and analyses the data in order to propose viable conclusions and solutions	Incomplete analysis of data hence the quality of conclusions drawn is not up to the mark	Cannot analyse the data or observations for any kind of conclusions.	Lacks the required knowledge to propose viable conclusions and solutions
AC5: Lab record assessment	Well-organized and confident presentation of record & ability to correlate the theoretical concepts with the concerned lab results with appropriate.	Presentation of record is acceptable	Presentation of record lacks clarity and organization	No efforts were exhibited

LAB EXPERIMENTS

LAB EXPERIMENT 1

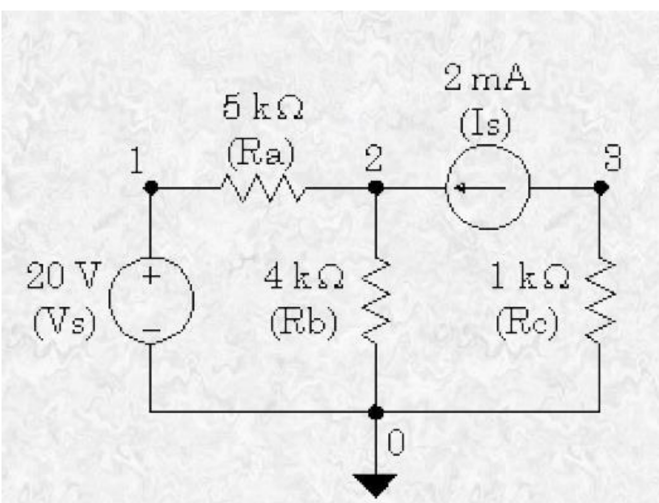
AIM: To Simulate the DC Circuit for determining the all node voltages using PSPICE.

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram **with I**ntegrated **C**ircuit **E**mphasis.

PROGRAM:

```
Vs    1    0    DC    20.0V
Ra    1    2    5.0k
Rb    2    0    4.0k
Rc    3    0    1.0k
Is    3    2    DC    2.0mA
.END
```

CIRCUIT DIAGRAM:



OUTPUT :

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE
VOLTAGE(1) 20.0000 (2) 13.3330 (3) -2.0000 <==

Results: Analysis of the DC network with sub circuit using PSPICE has been successfully completed.

LAB EXPERIMENT 2

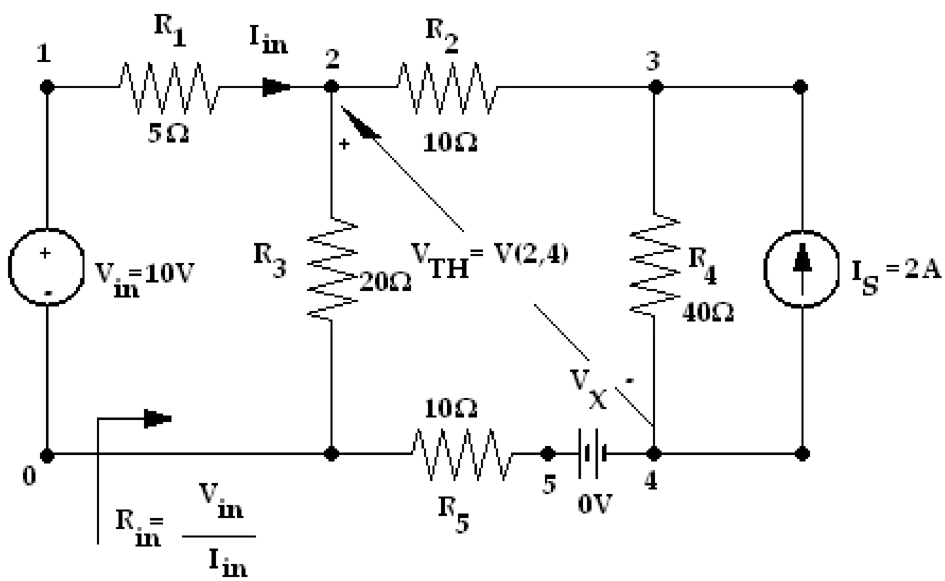
AIM: To Simulate the DC Circuit for determining the Thevenin's equivalent and Norton's equivalent using PSPICE.

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram **with I**ntegrated **C**ircuit **E**mphasis.

DATA REQUIRED FOR DRAWING CIRCUIT DIAGRAM:

A DC Circuit is as shown in the figure. It Consists of Voltage Source whose Value is **10V;**the Current source has the Value of **2A.** It has the resistance values as **5Ω, 10Ω, 20Ω, 40Ω, and 10Ω** respectively. Use PSPICE to plot and calculate the Thevenin's Equivalent Circuit across the nodes **2** and **4.**Obtain the transfer function between the two nodes **2** and **4.**

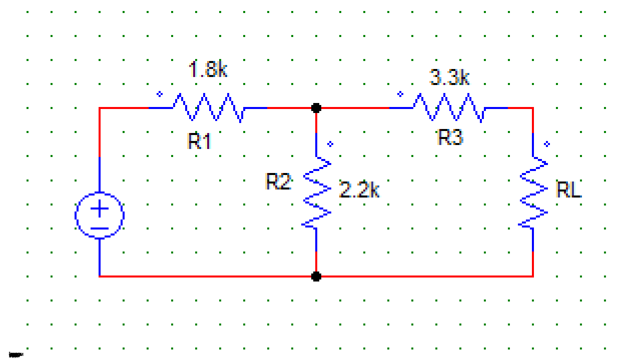
CIRCUIT DIAGRAM:



PROGRAM:
Thevenins theorem :

```
*
VIN 1 0 DC 10V
IS 4 3 2A
VX 4 5 DC 0V
R1 1 2 5
R2 2 3 10
R3 2 0 20
R4 3 4 40
R5 5 0 10
.TF V(2,4) VIN
.END
```

CIRCUIT DIAGRAM:



RESULT

1. the DC Circuit for determining the Thevenin's equivalent and Norton's equivalent using PSPICE has been successfully completed

LAB EXPERIMENT 3

AIM: To Simulate the DC network with sub circuit using PSPICE.

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram **with I**ntegrated **C**ircuit **E**mphasis.

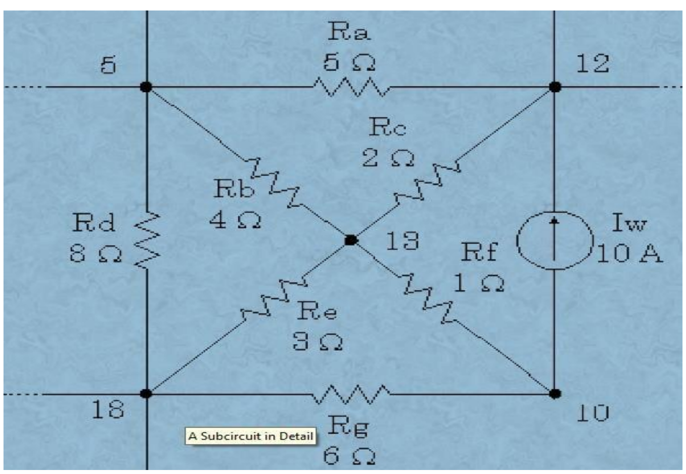
THEORY :

Coding a Sub-circuit

Each sub-circuit used in a study must have a unique name. This is true of any othercircuit element. Also, there must be a list of at least two nodes that can be connected to elements external to the sub-circuit. A sub-circuit can have many external node connections, if needed. Later, we will find that parameters can be passed to a sub-circuit in order to allow unique behavior and responses from an instance of a sub-circuit.

The initial line of a sub-circuit section must begin with ".SUBCKT," followed by the name and then the external node list. After that, optional features (not to be discussed yet) can be added. The best method of understanding the use of a sub-circuit is by example. Below, we find a cluster of components that can be combined into a sub-circuit.

Sub circuit:



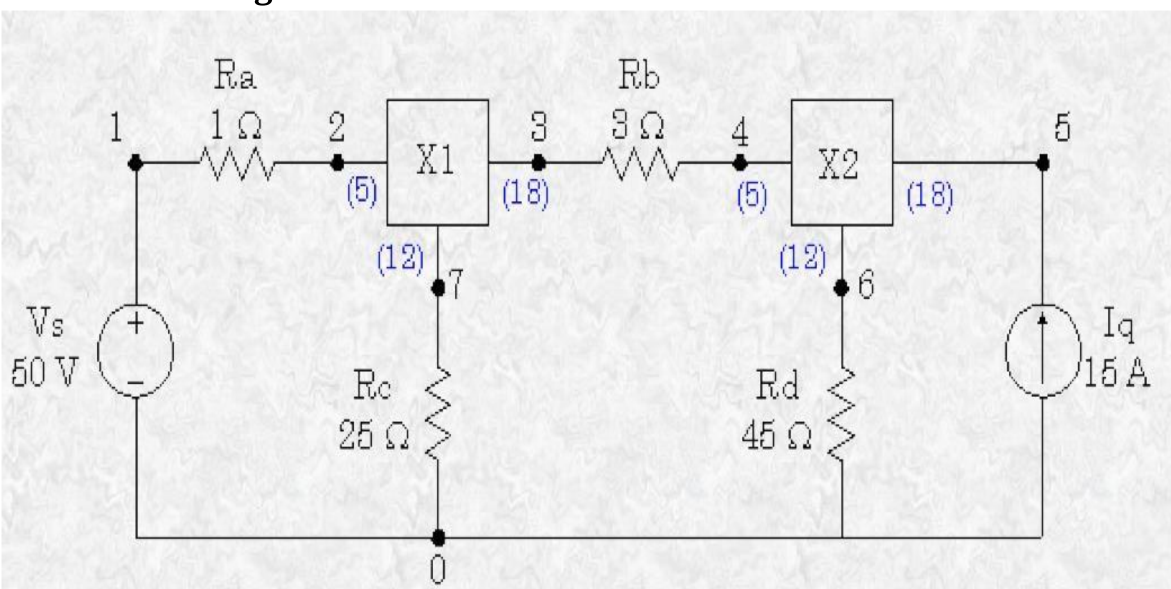
Note that nodes 5, 12 and 18 have external connections. Therefore, they must beincluded in the node list in the sub-circuit definition. Nodes 10 and 13 do not haveexternal connections and need not be (indeed *should* not be) included in this node list. They are internal nodes and will be used to help define the sub-circuit.

Now, we can code the above subcircuit as follows. Note that the code could beembedded into the rest of the code for the main circuit or could be placed in aseparate *include* file.

Program for sub circuit:

```
.SUBCKT Example_1    5    12    18
Iw    10    12    DC    10A
Ra     5    12    2.0
Rb     5    13    5.0
Rc    12    13    2.0
Rd     5    18    8.0
Re    13    18    3.0
Rf    10    13    1.0
Rg    10    18    6.0
.ENDS
```

Main circuit diagram:



Main program:

```
6.0.ENDS
Vs 1 0 DC 50V
Ra 1 2 1.0
Rb 3 4 3.0
```

```
Rc 7 0 25.0
Rd 6 0 45.0
X1 2 7 3
X2 4 6 5
.END
```

RESULT:

Analysis of the DC network with sub circuit using PSPICE has been successfully completed.

LAB EXPERIMENT: 4

AIM: To find out the transient response and parametric analysis by simulationof RLC circuits Using Pulse, and Step response

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

a) Simulation of STEP RESPONSE Using PSPICE:

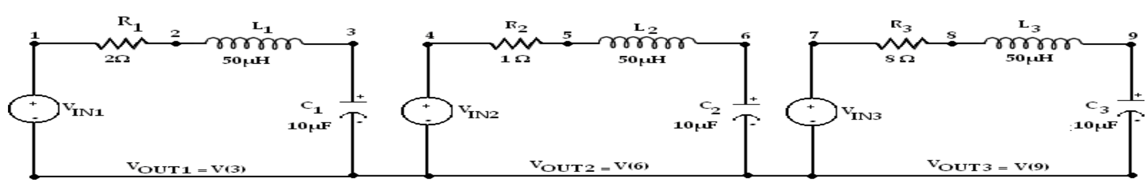
SYNTAX USED:

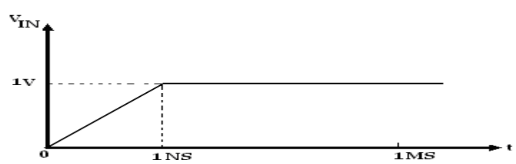
S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1.	STEP RESPONSE	PWL	STEP (Time at a Point) (Voltage at a Point)
2.	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [TStart TMax] [UIC]
3.	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4.	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit Value), (Upper Limit Value)}

DATA REQUIRED FOR DRAWING THE CIRCUIT DIAGRAM:

For example, Three RLC circuits with R=**2Ω**, **1 Ω**, and **8 Ω** respectively, with L having thevalues of **50µH** each, with C having the values of **10µF** each. The inputs are identical **Step Response.** The Step having the Time at points as **1nsec** and **1msec** respectively and Voltage at a point as **1V** respectively. Use PSPICE to plot and calculate the transient response from **0** to **400µseconds** with an increment of **1µsecond**. Plot the voltages across the capacitors

CIRCUIT DIAGRAM:





PROGRAM:

```
*
VIN1 1 0 PWL(0 0 1NS 1V 1MS 1V)
VIN2 4 0 PWL(0 0 1NS 1V 1MS 1V)
VIN3 7 0 PWL(0 0 1NS 1V 1MS 1V)
R1 1 2 2
R2 4 5 1
R3 7 8 8
L1 2 3 50UH
L2 5 6 50UH
L3 8 9 50UH
C1 3 0 10UF
C2 6 0 10UF
C3 9 0 10UF
.TRAN 1US 400US
.PLOT TRAN V(3) V(6) V(9)
.PROBE
.END
```

RESULT: Analysis of Series RLC Circuit with STEP Response has been successfully completed.

b) Simulation of PULSE RESPONSE Using PSPICE:

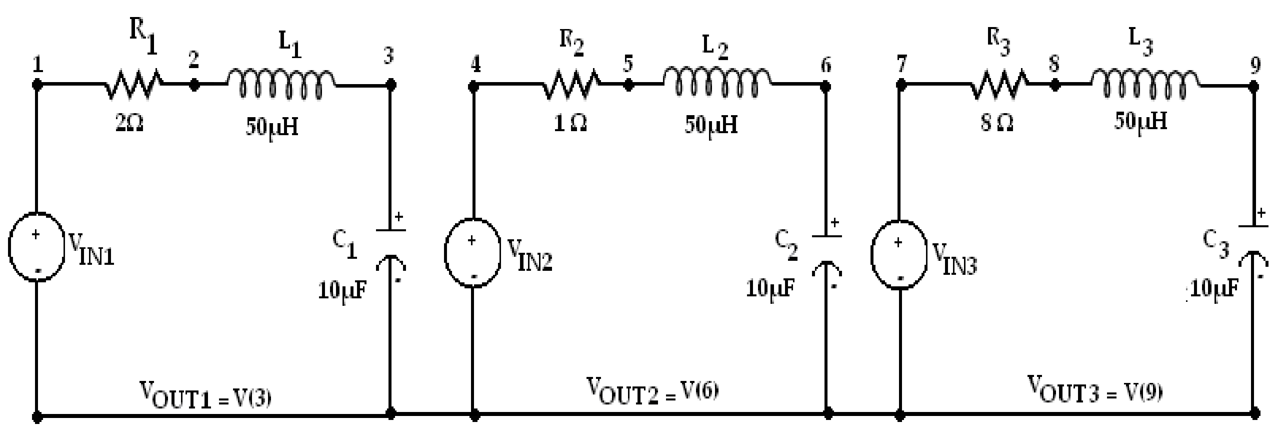
SYNTAX USED:

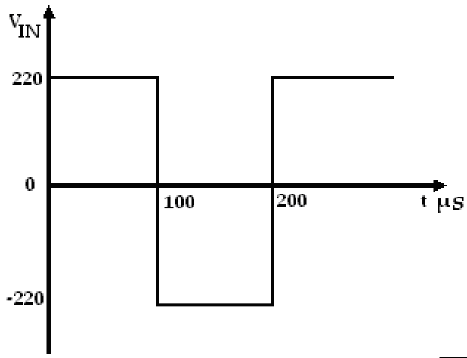
S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1.	PULSE RESPONSE	PULSE	PULSE (Initial Value) (Pulsed Value) (Delay Time)(Rise Time)(Fall Time) (Pulse Width) (period)
2.	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [TStart TMax] [UIC]
3.	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4.	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit Value), (Upper Limit Value)}

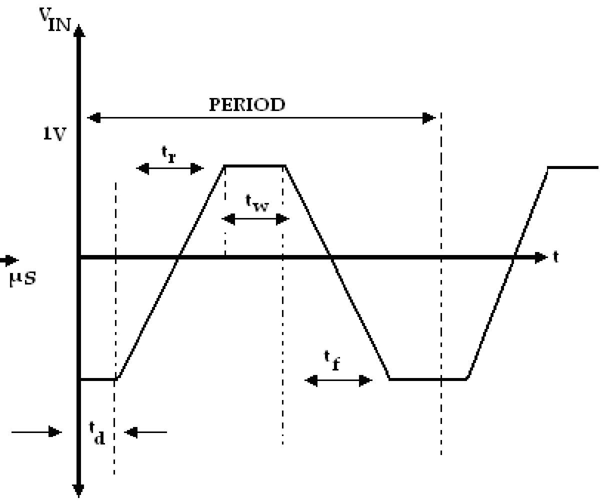
DATA REQUIRED FOR DRAWING THE CIRCUIT DIAGRAM:

For example, Three RLC circuits with R=**2Ω**, **1 Ω**, and **8 Ω** respectively, with L having thevalues of **50µH** each, with C having the values of **10µF** each. The input is **Pulse Response.** The Pulse having the Initial voltage as **-10V**, Pulsed Voltage as **10V,** Delay Time as **1nsec,** Rise Time as **1nsec,**Fall Time as **1nsec,** Pulse Width as **100µSeconds**, and Period as **200µseconds**.Use PSPICE to plot and calculate the transient response from **0** to **400µseconds** with an increment of**1µsecond**. Plot the voltages across the capacitors.

CIRCUIT DIAGRAM:







PROGRAM:

```
*
VIN1 1 0 PULSE(-220 220 0 1NS 1NS 100US 200US)
VIN2 4 0 PULSE(-220 220 0 1NS 1NS 100US 200US)
VIN3 7 0 PULSE(-220 220 0 1NS 1NS 100US 200US)
R1 1 2 2
R2 4 5 1
R3 7 8 8
L1 2 3 50UH
L2 5 6 50UH
L3 8 9 50UH
C1 3 0 10UF
C2 6 0 10UF
C3 9 0 10UF
.TRAN 1US 400US
.PLOT TRAN V(3) V(6) V(9)
.PROBE
.END
```

RESULT: Analysis of Series RLC Circuit with PULSE Response has beensuccessfully completed.

LAB EXPERIMENT 5

AIM: To find out the transient response and parametric analysis by simulationof RLC circuits Using Sinusoidal Responses.

SOFTWARE REQUIRED: **PSPICE** – **P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

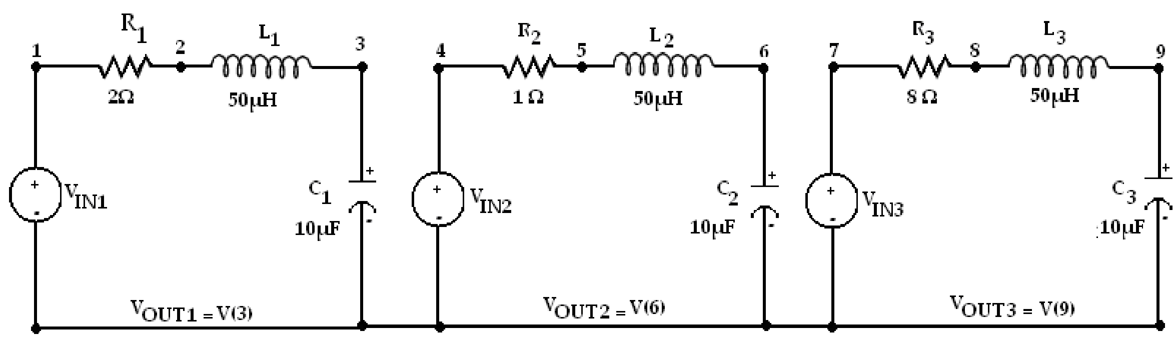
SYNTAX USED:

S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1.	SINUSOIDAL RESPONSE	SIN	SIN (Offset Value) (Peak Value) (Frequency)(Delay Time) (DampingFactor) (Phase Delay)
2.	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [TStart TMax]
3.	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4.	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit Value), (Upper Limit Value)}

DATA REQUIRED FOR DRAWING THE CIRCUIT DIAGRAM:

For example, Three RLC circuits with R=**2Ω**, **1 Ω**, and **8 Ω** respectively, with L having the values of **50µH** each, with C having the values of **10µF** each. The inputs are identical **Sinusoidal Response.** The Sinusoidal response having the offset voltage as **0V,** RMS voltage as **120V** and the frequency as **50Hz**. Use PSPICE to plot and calculate the transient response from **0** to **60mseconds** with an increment of **1µsecond**. Plot the voltages across the capacitors.

CIRCUIT DIAGRAM:



PROGRAM:

```
*
VIN1 1 0 SIN(0 169.7V 50)
VIN2 4 0 SIN(0 169.7V 50)
VIN3 7 0 SIN(0 169.7V 50)
R1 1 2 2
R2 4 5 1
R3 7 8 8
L1 2 3 50UH
L2 5 6 50UH
L3 8 9 50UH
C1 3 0 10UF
C2 6 0 10UF
C3 9 0 10UF
.TRAN 1US 400US
.PLOT TRAN V(3) V(6) V(9)
.PROBE
.END
```

RESULT: Analysis of the transient response and parametric analysis by simulation of RLC circuits Using Sinusoidal Responses has been successfully completed.

LAB EXPERIMENT 6

AIM: To analyse three phase currents and the neutral current by the analysis of three phase circuit representing the Generator, Transmission line and loads using PSPICE.

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram **with I**ntegrated **C**ircuit **E**mphasis.

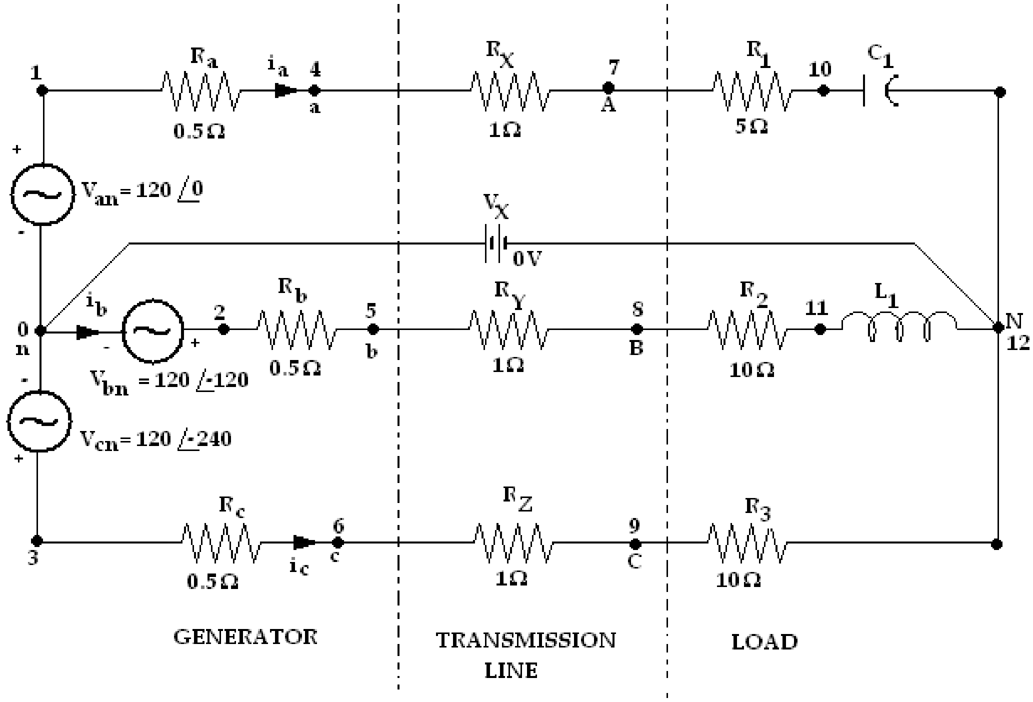
SYNTAX USED:

S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1.	SINUSOIDAL RESPONSE	SIN	SIN (Offset Voltage) (Peak Voltage)(Frequency)(Delay Time) (Damping Factor) (Phase Delay)
2.	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop
3.	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4.	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit Value), (Upper Limit Value)}

DATA REQUIRED FOR DRAWING CIRCUIT DIAGRAM:

For example, the circuit consists of Generators, transmission lines and loads. It is fed with a three phase balanced supply. Arrange the generators in STAR connection and connect the Transmission lines and loads to it. The generator is having a resistance of **0.5Ω** and the transmission line is having a resistance of **1 Ω** and consists of loads having **R_1=10Ω, R_2=10Ω,** and **R_3=10Ω** respectively, **L_1=120mH** and **C_1=120µF**. The Sinusoidal having the offset voltage as **0V,** RMS voltage as **120V,** the frequency as **60Hz,** the Delay Time and the Damping Factor are given as **0** and the Phase angle as **120^0**. Use PSPICE to plot the instantaneous currents. Plot the transient response from **0** to **50mseconds** with an increment of **5µsecond**.

CIRCUIT DIAGRAM:



PROGRAM:

```
*
VIN1 1 0 SIN(0 169.7V 50)
VIN2 2 0 SIN(0 169.7V 50 0 0 120)
VIN3 3 0 SIN(0 169.7V 50 0 0 240)
RA 1 4 0.5
RB 2 5 0.5
RC 3 6 0.5
RX 4 7 1
RY 5 8 1
RZ 6 9 1
R1 7 10 5
R2 8 11 10
R3 9 12 10
VX 12 0 DC 0V
.TRAN 5US 50MS
.PLOT TRAN I(RA) I(RB) I(RC)
.PROBE
.END
```

RESULT.:.

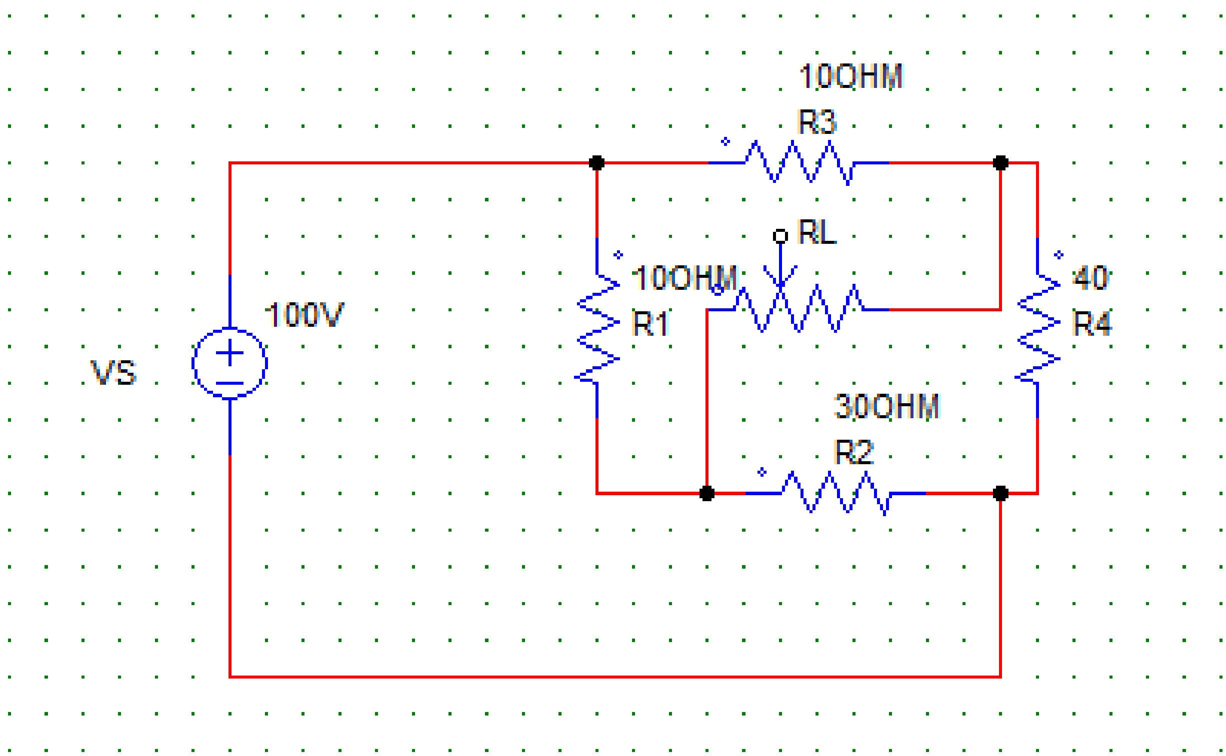
Three-phase circuit representing the generator transmission line and load. three phase currents and neutral current are plotted using PSPICE.

LAB EXPERIMENT 7

AIM: To find out the unknown resistance and maximum power for dc circuits

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

CIRCUIT DIAGRAM:



Program:

```
*
VS 1 0 DC 100
R1 1 2 10
R2 2 0 30
R3 1 3 20
R4 3 0 40
RL 2 3 RLOAD 1
```

```
.MODEL RLOAD RES(R=25)
.
DC RES RLOAD(R) 0.001 40 0.01
.TF V(2,3) VS
.PROBE
.END
```

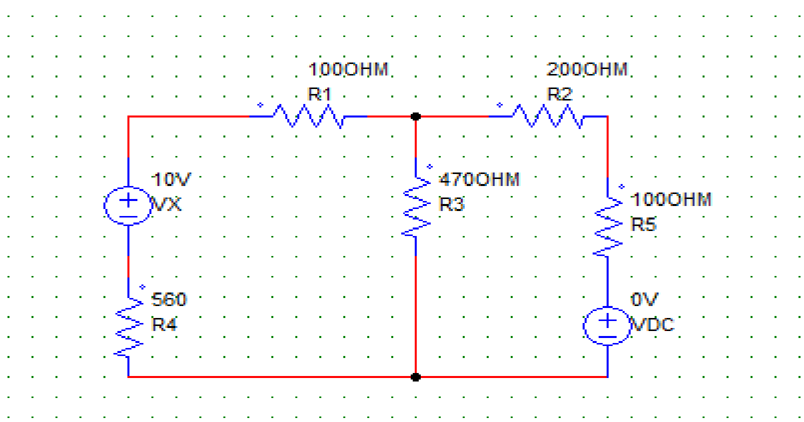
RESULT: Analysis of unknown resistance and maximum power for dc circuits using PSPICE has been successfully completed .

LAB EXPERIMENT 8

AIM: To verify the reciprocity theorem for dc circuits

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

Circuit diagram:



FOR MAIN CIRCUIT:

FOR CIRCUIT 1:

*****
VX 1 2 DC 0V

R1 1 3 100
R2 3 4 200
R3 3 0 470
R4 2 0 560
R5 4 5 100
VDC 5 0 DC 10V
.OP
.END

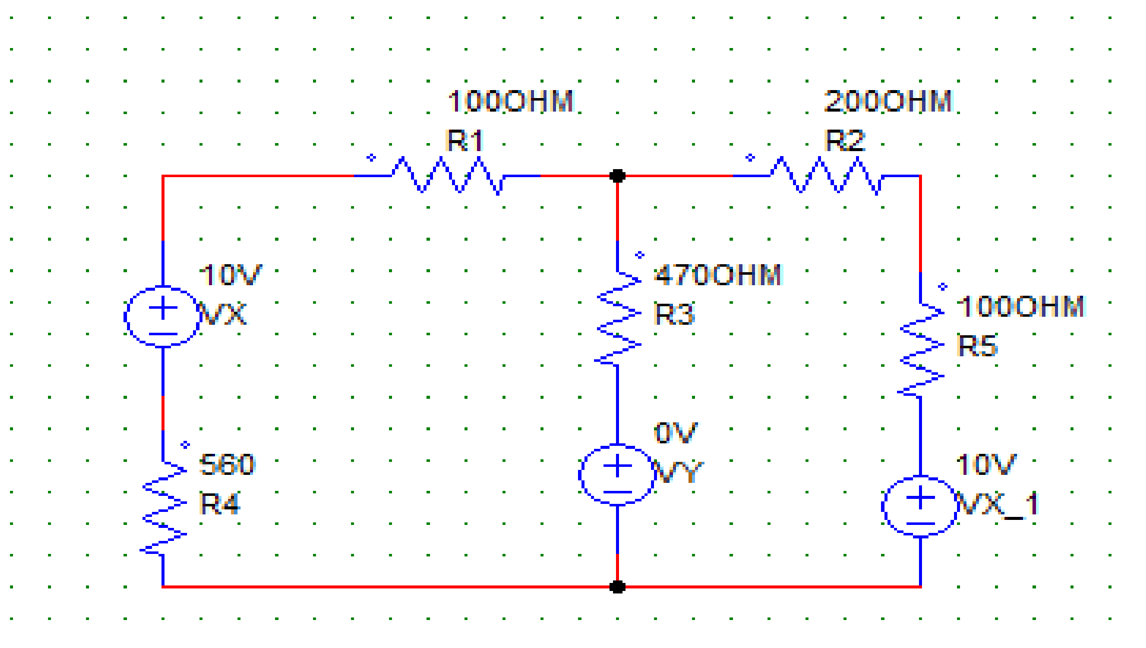
RESULT: Analysis of reciprocity theorem for dc circuit using PSICE has been successfully completed .

LAB EXPERIMENT 9

AIM: **T**o verify the superposition theorem for dc circuits

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

CIRCUIT DIAGRAM:



Program:
FOR MAIN CIRCUIT :

```
*
VDC 1 2 DC 10V
R1 1 3 100
R2 3 4 200
R3 3 6 470
R4 2 0 560
R5 4 5 100
VX 5 0 DC 10V
VY 6 0 DC 0V
.OP
.END
```

FOR CIRCUIT 1:

```
*
VDC 1 2 DC 10V
R1 1 3 100
R2 3 4 200
R3 3 6 470
R4 2 0 560
R5 4 0 100
VY 6 0 DC 0V
.OP
.END
```

FOR CIRCUIT 2:

```
*
VDC 1 2 DC 0V
R1 1 3 100
R2 3 4 200
R3 3 6 470
R4 2 0 560
R5 4 5 100
VX 5 0 DC 10V
VY 6 0 DC 0V
.OP
.END
```

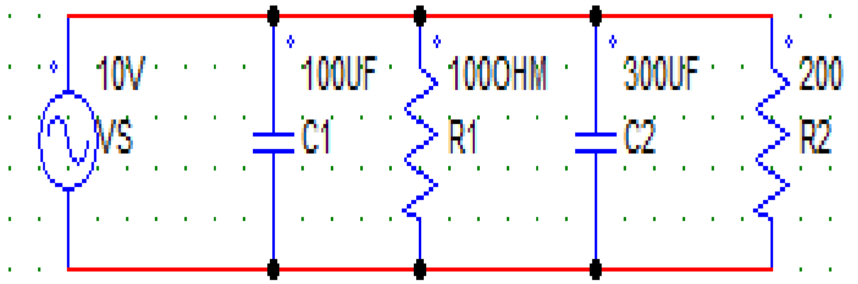
RESULT: Analysis of the superposition theorem for dc circuits using PSPICE has been successfully completed .

LAB EXPERIMENT 10

AIM: To calculate the response for the ac circuits.

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

CIRCUIT DIAGRAM:



Program:

```
*
VS 1 0 AC 10V
C1 1 0 100U
R1 1 0 100
C2 1 0 300U
R2 1 0 200
.AC LIN 1 50 100
.PRINT AC IM(VS) IP(VS) IM(C1) IP(C1)
.END
```

RESULT: Analysis the response for the ac circuits using PSPICE has been successfully completed

LAB EXPERIMENT 11

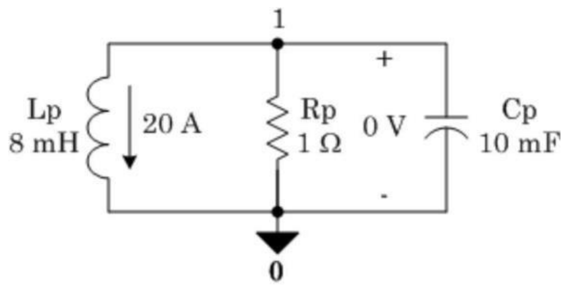
AIM: To obtain transient response of a parallel R-L-C circuit for step currentinput.

SOFTWARE REQUIRED:

PSPICE VERSION 8

CIRCUIT & PROGRAM:

One of the most interesting aspects of circuit analysis is the study of natural and step responses of circuits and the responses of circuits to time-varying sources. To perform these analyses we introduce another group of "dot" commands



Use of the .TRAN command

This is the command that passes the user's parameters for performing the transient analysis ona circuit to the PSpice program. There are four time parameters and an instruction to use the initial conditions rather than calculated bias point values for starting conditions. First, we show a sample .TRAN statement and then we will describe its parameters.

```
* prt_stp t_max prt_dly max_stp
.TRAN 20us 20ms 8ms 10us UIC
```

In the above statement, the "20us" value labeled "prt_stp" *(print step)* is the frequency with which data is saved. In this case, the system variables are stored each 20µs of simulation time. The actual time steps used by PSpice may be different from this. The second parameter,"20ms," labeled as "t_max" *(final time)* is the value of time at which the simulation will be ended. Since PSpice starts at t = 0, there will be a total of 20ms time span of simulation forthe circuit. The third parameter, "8ms," labeled as "prt_dly" *(print delay)* is the print delay time. In some cases, we do not want to store the data for the entire time span of the simulation. In our sample statement shown above, we ignore the data from the first 8ms of simula- tion and then store the data for the last 12ms. Most of the time, this parameter is set to zero ornot used. The fourth parameter, "10us," labeled as "max_stp" *(max step)* is the maximum time step size PSpice is allowed to take during the simulation. Since PSpice automatically adjusts its time step size during the simulation, it may increase the step size to a value greater than desirable for displaying the data.

When the variables are changing rapidly, PSpice short- ens the step size, and when the variables change more slowly, it increases the step size. Use of this parameter is optional. The last parameter in our list is "UIC." It is an acronym for "UseInitial Conditions."

Unless you include this parameter, PSpice will ignore the initial conditions you set for your inductors and capacitors and will use its own calculated bias point in- formation instead. Note that the use of the letter "s" after the numbers in the .TRAN statement is optional. PSpice assumes these values are seconds and actually ignores the "s." However, it is recommended that you use units until you are extremely familiar with all of these commands and definitions.
Now, we will examine some more .TRAN examples.

```
.TRAN 10ns 500us
```

In the above example, PSpice will save data at each 10ns interval of the simulation starting att = 0 until the final time of 500μs. I.e., there is no print delay and the user has given full con- trol of the calculation step size to PSpice. In addition, PSpice will calculate its own initial conditions for any inductors and capacitors, ignoring any initial conditions set by the user.

```
.TRAN 50m 2.5 0 10m UIC
```

In the above statement, PSpice collects the data at each 50ms time interval starting from zero up to 2.5s. A zero was required as as a placeholder for the print delay parameter since the maximum step size of 10ms was specified. PSpice will use the designated initial conditionsof capacitor voltage and inductor current. Notice that the units were left off the numbers in this statement. Only the prefixes which size the values are needed.

Use of the .PROBE command

In addition to specifying the time parameters for a transient solution of a circuit problem, we need to specify how the data is to be saved. In most cases, this simply means that we includea line in the *.CIR file consisting of ".PROBE." This instructs PSpice to create a data file and store the data it calculates. If we create a circuit listing named "CIRCUIT1.CIR" containing a ".TRAN" statement and a ".PROBE" statement, PSpice will create a file named "CIR-CUIT1.DAT" holding the data as well as the usual "CIRCUIT1.OUT" file with basic information about the circuit. By default, the data file created by PSpice is a binary data file; i.e., you can't read it with a text editor. This is the most efficient way of saving the data. However,there is an optional parameter (/CSDF) for the .PROBE statement that causes PSpice to save the data in a Common Simulation Data Format which is a text format that allows you to look at the raw data with a text editor.

However, it will take up more space and PROBE doesn't load it for graphing. You will need to make a second run without the /CSDF parameter if you want to plot the data.
Also by default, .PROBE causes *all* the circuit variables to be saved, including all the variables inside each instance of each sub-circuit. In some cases, this can amount to a lot of data.If you simulate a large complex circuit with many parts and need to save data at short time intervals over a very long time span, you can easily create gigabyte-size "DAT" files. To avoid this, you can specify the values you want to save. If the

".PROBE" command is issued without any parameters, everything is saved. If you specify thequantities you want saved, *only* those quantities will be saved. We will now examine some PROBE statements.

```
.PROBE
```

All the above statement does is the enable PSpice to save everything in a binary DAT file.

```
.PROBE/CSDF
```

The above statement enables PSpice to save everything in a CSDF file that can be opened (and edited) with a text editor. You can

```
.PROBE V(5,23) I(Rx) I(L4)
```

The above statement tells PSpice to save only the voltage drop between nodes 5 and 23, the current through resistor, Rx, and the current through inductor, L4, all in binary format. No other data will be saved.

RESULTS: The response of RLC circuit for step input is simulated using PSPICE .

This lab manual has been updated by

Ms. Anuradha Yadav

Cross checked
byHOD-EEE

Verified By
Director, DGI Greater Noida

Please spare some time to provide your valuable feedback.