

ELECTRICAL CIRCUITS **SIMULATION LAB**

II Year - 2 Semesters

LABORATORY MANUAL

Department of
ELECTRICAL AND ELECTRONICS ENGINEERING

2. PREAMBLE:

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 plain, 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.

3. CODE OF CONDUCT:

CODE OF CONDUCT FOR THE LABORATORIES

- ❖ All students must observe the Dress Code while in the laboratory.
- ❖ Sandals or open-toed shoes are NOT allowed.
- ❖ Foods, drinks and smoking are NOT allowed.
- ❖ All bags must be left at the indicated place.
- ❖ The lab timetable must be strictly followed.
- ❖ Be PUNCTUAL for your laboratory session.
- ❖ Experiment must be completed within the given time.
- ❖ Noise must be kept to a minimum.
- ❖ Workspace must be kept clean and tidy at all time.
- ❖ Handle all apparatus with care.
- ❖ All students are liable for any damage to equipment due to their own negligence.
- ❖ All equipment, apparatus, tools and components must be RETURNED to their original place after use.
- ❖ Students are strictly PROHIBITED from taking out any items from the laboratory.
- ❖ Students are NOT allowed to work alone in the laboratory without the Lab Supervisor
- ❖ Report immediately to the Lab Supervisor if any injury occurred.
- ❖ Report immediately to the Lab Supervisor any damages to equipment.

Before leaving the lab

- ❖ Place the stools under the lab bench.
- ❖ Turn off the power to all instruments.
- ❖ Turn off the main power switch to the lab bench.
- ❖ Please check the laboratory notice board regularly for updates

4 . OBJECTIVE & RELEVANCE:

The objective of Simulation laboratory is to impart hands on experience in verification of circuit laws and theorems, measurement of circuit parameters, study of circuit characteristics using PSpice. It also gives practical exposure to the usage of different circuits with different condition.

OUTCOME:

At the successful completion of this course, the student is expected to gain the following skills:

- Become familiar with the basic circuit components and know how to connect them to make a real electrical circuit;
- Become familiar with basic electrical measurement instruments and know how to use them to make different types of measurements;
- Be able to verify the laws and principles of electrical circuits, understand the relationships and differences between theory and practice;
- Be able to gain practical experience related to electrical circuits, stimulate more interest and motivation for further studies of electrical circuits; and
- Be able to carefully and thoroughly document and analyze experimental work.

5. List of experiments:

1. PSPICE SIMULATION OF NODAL ANALYSIS FOR DC CIRCUITS
2. PSPICE SIMULATION OF D.C. CIRCUIT FOR DETERMINING THEVENIN'S EQUIVALENT
3. PSPICE SIMULATION OF D.C. NETWORK WITH SUB CIRCUIT
4. PSPICE SIMULATION OF TRANSIENT AND PARAMETRIC ANALYSIS OF SERIES RLC CIRCUITS USING STEP AND PULSE INPUT
5. PSPICE SIMULATION OF TRANSIENT AND PARAMETRIC ANALYSIS OF SERIES RLC CIRCUITS USING SINE INPUT
6. ANALYSIS OF THREE PHASE CIRCUIT REPRESENTING GENERATOR TRANSMISSION LINE AND LOAD
7. PSPICE SIMULATION OF MAXIMUM POWER TRANSFER THEOREM FOR DC CIRCUITS
8. PSPICE SIMULATION OF RECIPROCITY THEOREM FOR DC CIRCUITS
9. PSPICE SIMULATION OF SUPERPOSITION THEOREM FOR DC CIRCUITS
10. PSPICE SIMULATION OF AC CIRCUITS.

6. TEXT AND REFERENCE BOOKS

Text Books

1. Basic Electrical Engineering- By M.S.Naidu and S. Kamakshiah – TMH.
2. Basic Electrical Engineering –By T.K.Nagasarkar and M.S. Sukhija Oxford

REFERENCES

1. Theory and Problems of Basic Electrical Engineering by D.P.Kothari & I.J.Nagrath PHI.
2. Principles of Electrical Engineering by V.K Mehta, S.Chand Publications.

7. EACH EXPERIMENT WRITE UP:**7.1.PSPICE SIMULATION OF Nodal analysis for DC circuits**

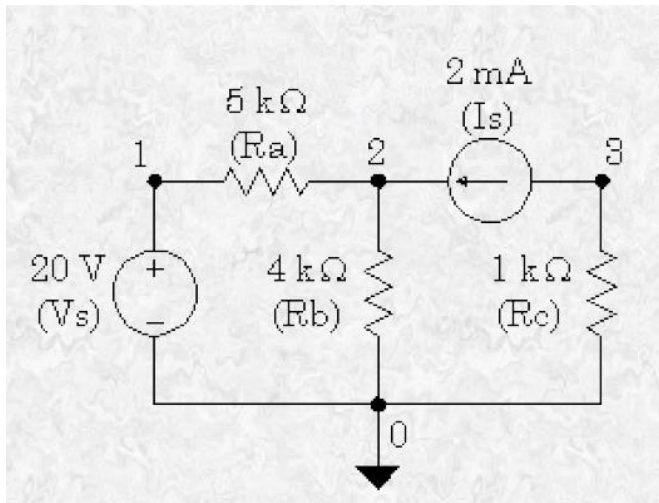
AIM: To Simulate the DC Circuit for determining the all node voltages using PSPICE.

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program
with Integrated Circuit Emphasis.

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
```

```
VOLTAGE SOURCE CURRENTS
```

```
NAME CURRENT
```

```
Vs -1.333E-03 <== Current entering node 1 of Vs
```

```
TOTAL POWER DISSIPATION 2.67E-02 WATTS
```

```
JOB CONCLUDED
```

```
TOTAL JOB TIME .26
```

Result:

7.2.PSPICE SIMULATION OF D.C. CIRCUIT FOR DETERMINING THEVENIN'S AND NORTONS EQUIVALENT

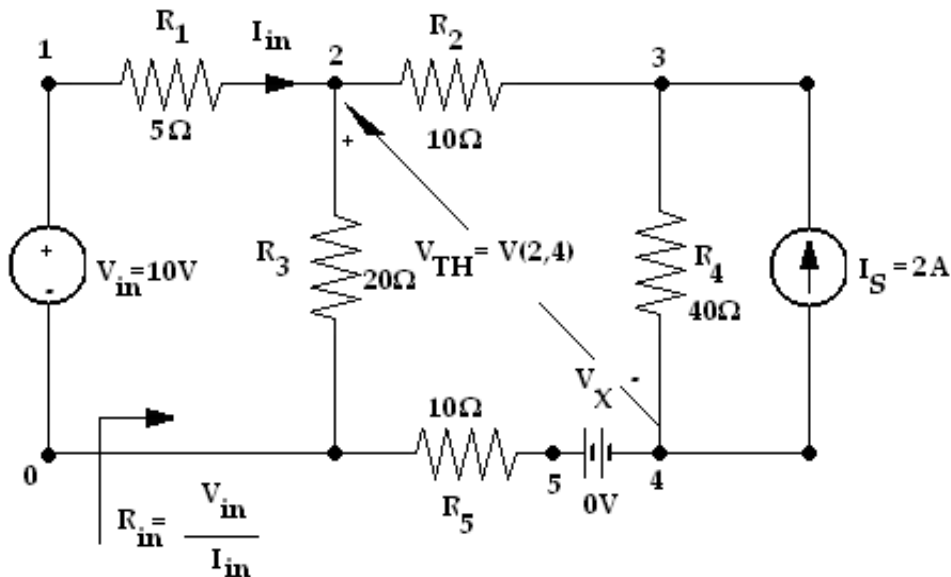
AIM: To Simulate the DC Circuit for determining the Thevenin's equivalent and nortons equivalent using PSPICE.

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

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

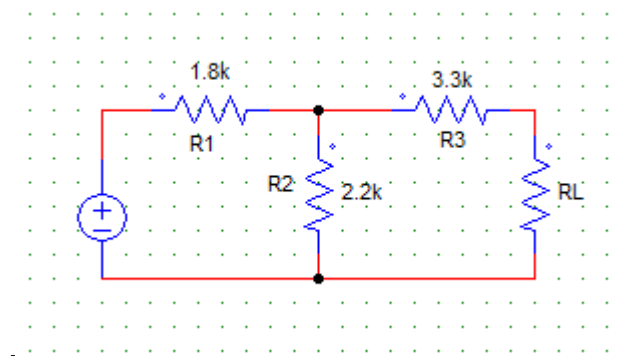
R5 5 0 10

.TF V(2,4) VIN

.END

Nortons theorem :

CIRCUIT DIAGRAM:

**OUTPUT:****RESULT:**

7.3.PSPICE SIMULATION OF D.C. network with sub circuit

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

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

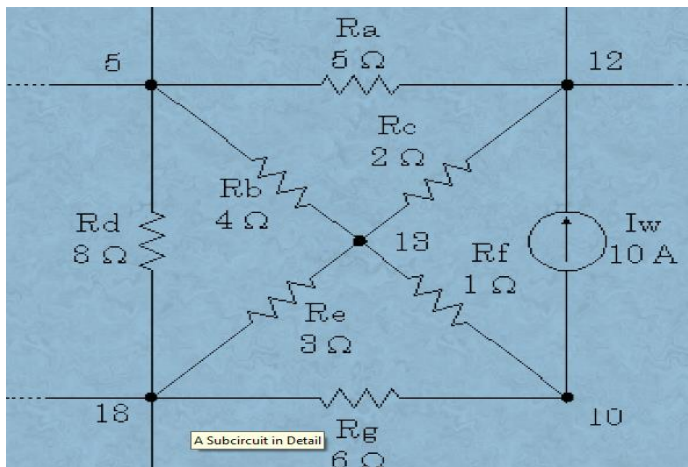
Theory :

Coding a Subcircuit

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

The initial line of a subcircuit 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 subcircuit is by example. Below, we find a cluster of components that can be combined into a subcircuit.

Sub circuit:

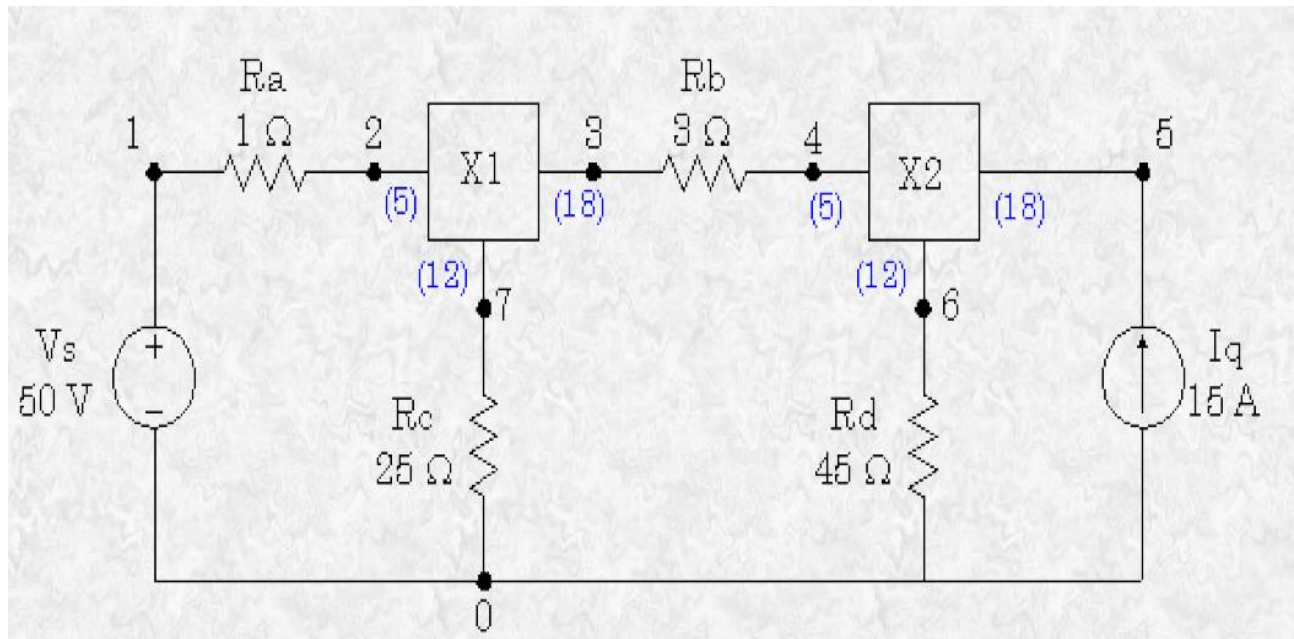


Note that nodes 5, 12 and 18 have external connections. Therefore, they must be included in the node list in the subcircuit definition. Nodes 10 and 13 do not have external connections and need not be (indeed *should* not be) included in this node list. They are internal nodes and will be used to help define the subcircuit. Now, we can code the above subcircuit as follows. Note that the code could be embedded into the rest of the code for the main circuit or could be placed in a separate *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

```

Output:

Result:

7.4.PSPICE SIMULATION OF TRANSIENT AND PARAMETRIC ANALYSIS OF SERIES RLC CIRCUITS using step input and pulse input

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

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

a) Simulation of STEP RESPONSE Using PSPICE:

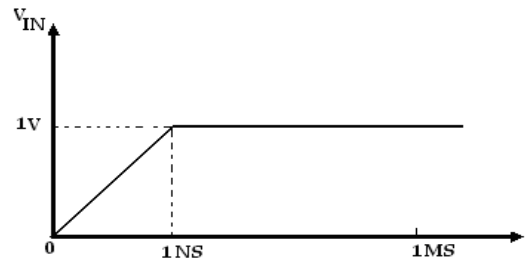
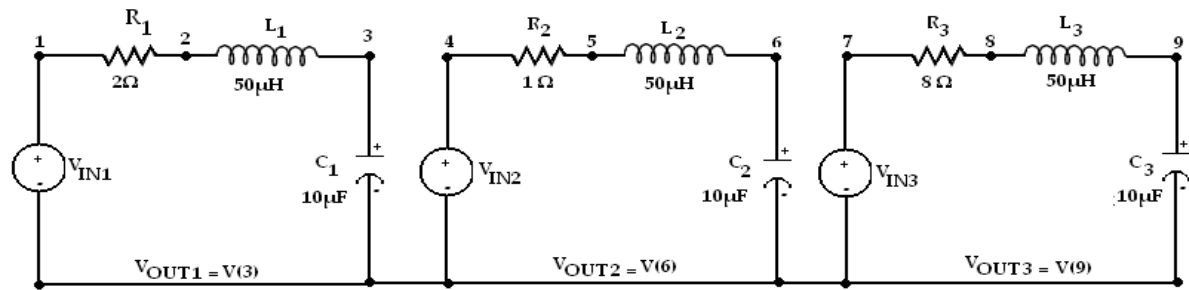
SYNTAX USED:

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 $R=2\Omega$, 1Ω , and 8Ω respectively, with L having the values of $50\mu\text{H}$ each, with C having the values of $10\mu\text{F}$ 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

OUTPUT:

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

b) Simulation of PULSE RESPONSE Using PSPICE:

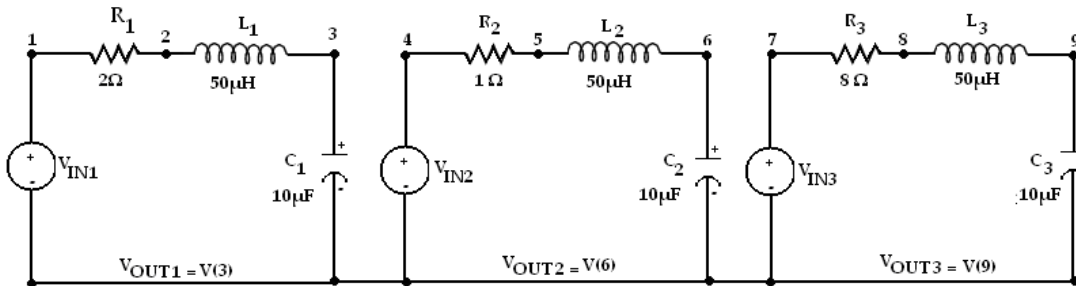
SYNTAX USED:

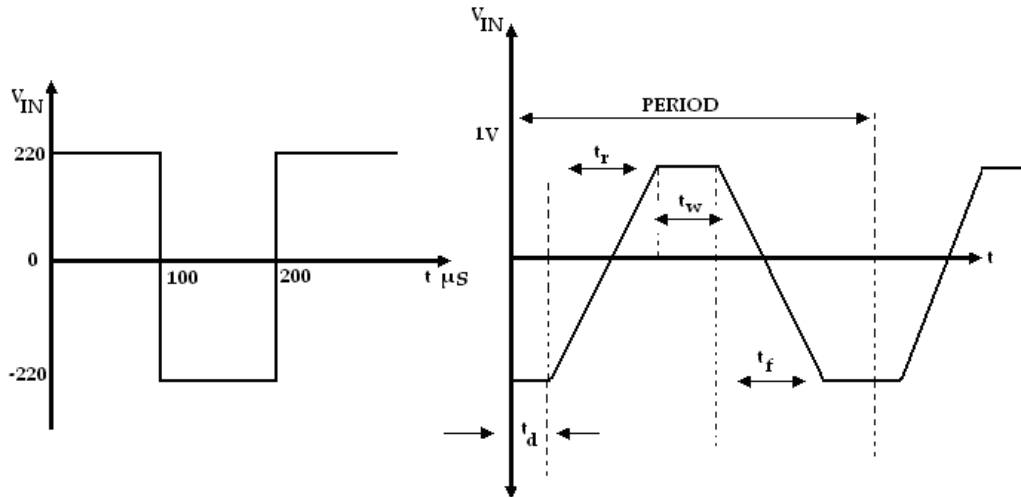
S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1.	PULSE RESPONSE	PULSE	PULSE (Initial Value) (Pulsed Value) (Delay Time)(Rise Time)(Fall Time) (Pulse Width) (period)
2.	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [Tstart TMax] [UIC]
3.	PROBE STATEMENT	.PROBE	It is a wave form analyzer
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 $R=2\Omega$, 1Ω , and 8Ω respectively, with L having the values of $50\mu\text{H}$ each, with C having the values of $10\mu\text{F}$ 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

```

OUTPUT:

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

7.5.PSPICE SIMULATION OF TRANSIENT AND PARAMETRIC ANALYSIS OF SERIES RLC CIRCUITS using sine input

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

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

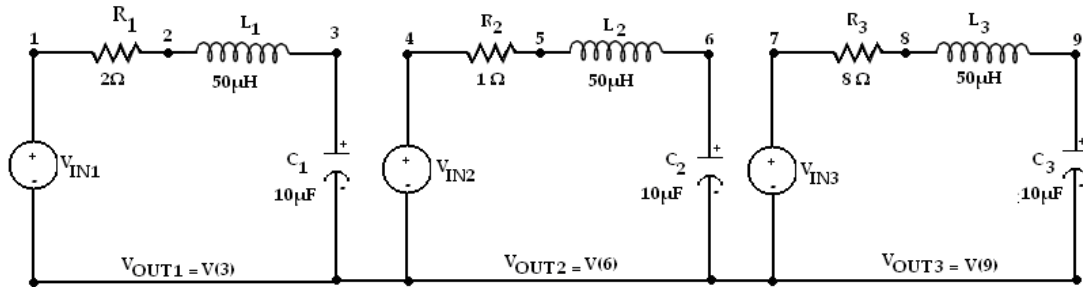
SYNTAX USED:

S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1.	SINUSOIDAL RESPONSE	SIN	SIN (Offset Value) (Peak Value) (Frequency)(Delay Time) (Damping Factor) (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 $R=2\Omega$, 1Ω , and 8Ω respectively, with L having the values of $50\mu\text{H}$ each, with C having the values of $10\mu\text{F}$ 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

```

OUTPUT:

RESULT: Analysis of Series RLC Circuit with Sinusoidal Response has been Successfully completed.

7.6.ANALYSIS OF THREE PHASE CIRCUIT REPRESENTING GENERATOR TRANSMISSION LINE AND LOAD

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 – Personal Computer Simulated Program with Integrated Circuit Emphasis.

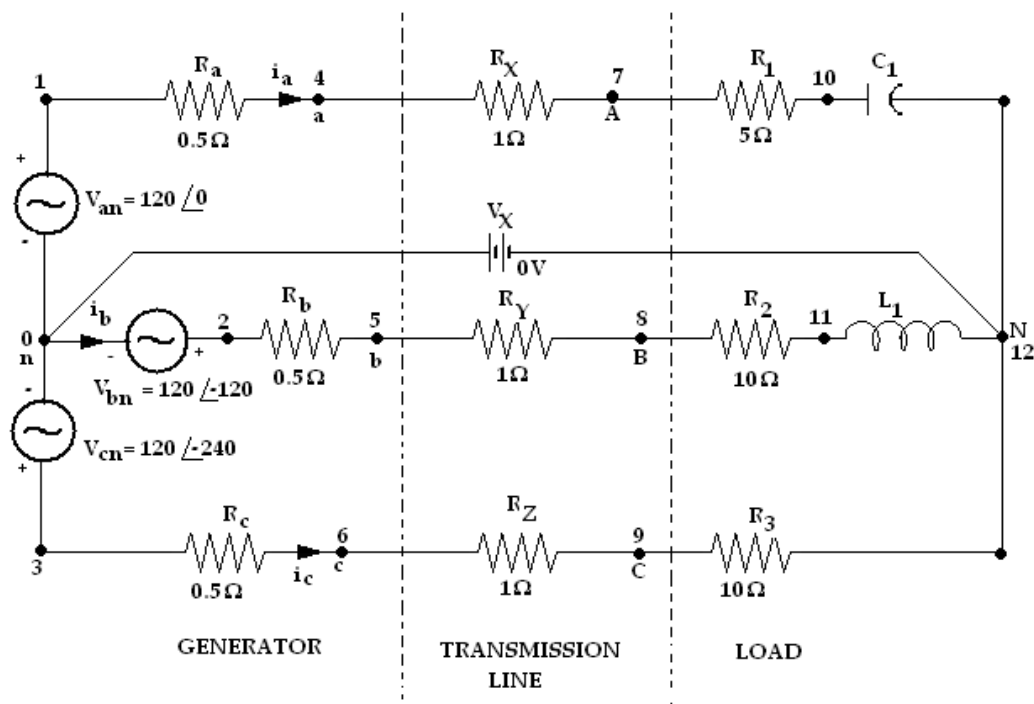
SYNTAX USED:

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 1Ω and consists of loads having $R_1=10\Omega$, $R_2=10\Omega$, and $R_3=10\Omega$ respectively, $L_1=120\text{mH}$ and $C_1=120\mu\text{F}$. 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° . Use PSPICE to plot the instantaneous currents. Plot the transient response from 0 to 50mseconds with an increment of $5\mu\text{second}$.

CIRCUIT DIAGRAM:

**PROGRAM:**

*

```

VIN1 1 0 SIN(0 169.7V 50)
VIN2 2 0 SIN(0 169.7V 50 0 120)
VIN3 3 0 SIN(0 169.7V 50 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

```

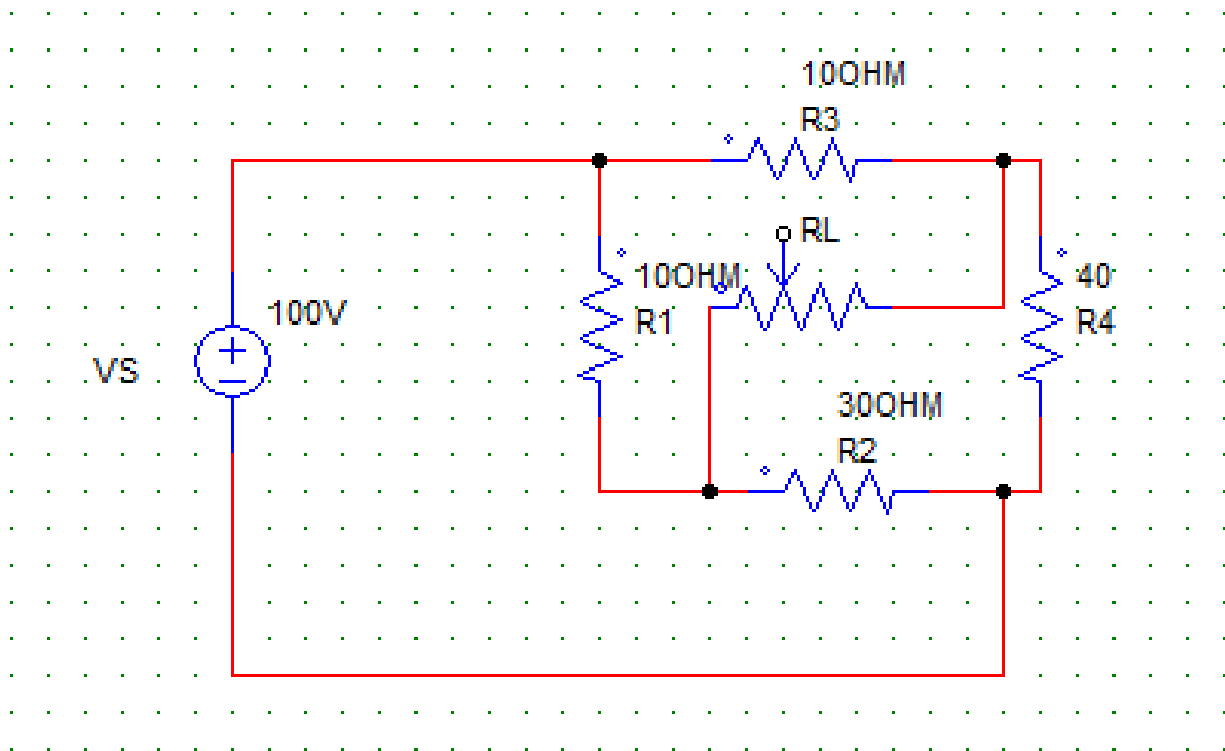
OUTPUT:**Result:**

7.7.PSPICE SIMULATION OF MAXIMUM POWER TRANSFER THEOREM FOR DC CIRCUITS

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

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

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
```

Output:

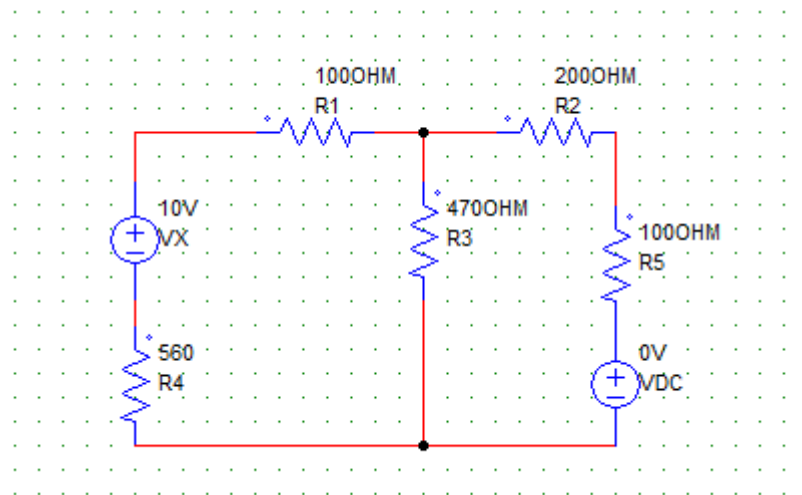
Result:

7.8.PSPICE SIMULATION OF RECIPROCITY THEOREM FOR DC CIRCUITS

AIM: To verify the reciprocity theorem for dc circuits

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

Circuit diagram:



Program:

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
```

Output:

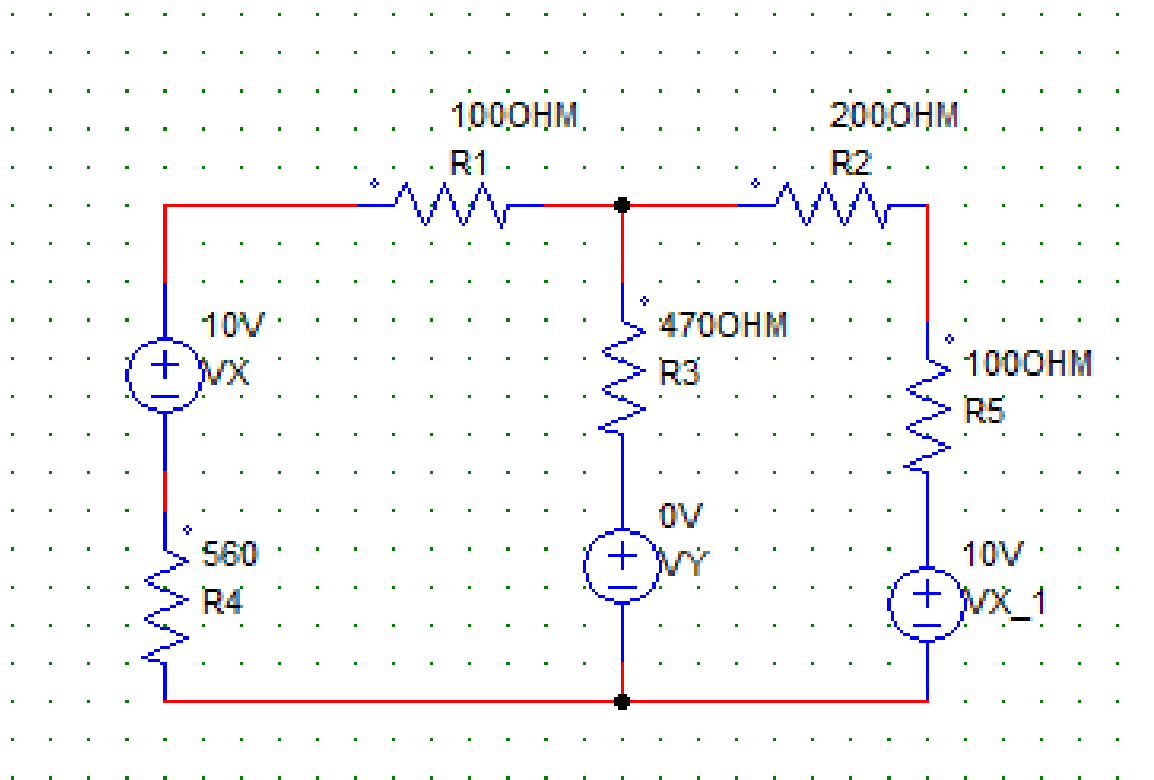
Result:

7.9.PSPICE SIMULATION OF SUPERPOSITIO THEOREM FOR DC CIRCUITS

AIM: To verify the superposition theorem for dc circuits

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

Circuit diagtam:



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
```

Output:

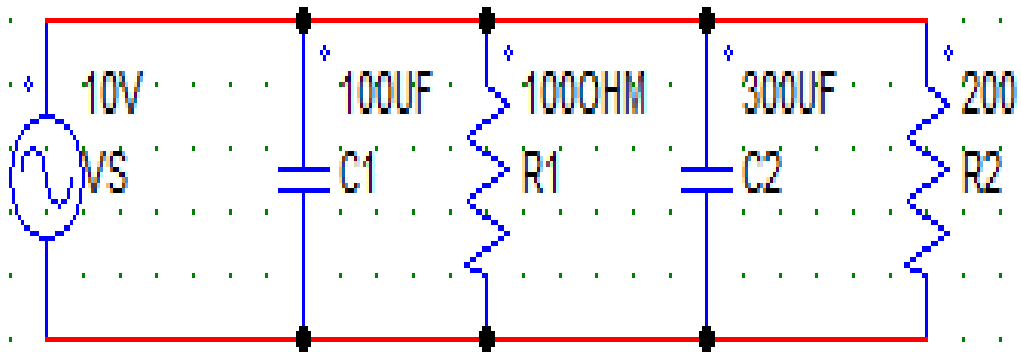
Result:

7.10.PSPICE SIMULATION OF AC CIRCUITS

AIM: To calculate the response for the ac circuits.

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

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
```

Output:

Result:

8. CONTENT BEYOND SYLLABUS

- 1) Simulation of networks theorem using MATLAB.
- 2) Simulation of compensation theorem using pspice.
- 3) Simulation of filters using pspice.
- 4) Simulation of cascaded twoport network using pspice.
- 5) Simulation of Tellegen's theorem using pspice.

9. SAMPLE VIVA VOCE QUESTIONS**Exp:1**

- 1 The frequency of the voltage in your house is 60 Hz. How much time is required for the waveform to complete three cycles?
- 2- What is the difference between AC and DC coupling of the oscilloscope? Explain how to use them to measure the average value of any periodic signal.
- 3- Some meters are calibrated to read r.m.s. Value of sinusoidal waveforms from the basic unit that responds to the peak value of the waveform. In terms of the peak value, V_p , the meter will read $2V_p$ which is the correct r.m.s value for a sinusoidal signal. Can this meter be used to read the correct rms value for other waveforms like square, triangular, etc? Comment.

Exp:2

1. Define thevenin's theorem.
2. What are the conditions to apply the thevenin's theorem?
3. What is the importance of Thevenin's theorem?

Exp:3

1. What is sub circuit.
2. What is the importance of sub circuit in simulation.
3. Write the program for the sub circuit.
4. Write the program for the sub circuit of thyristor.

Exp:4

1. For a resistance and capacitance in series with a voltage source, show that it is possible to draw a phasor diagram for the current and all voltages from magnitude measurement of these quantities only. Illustrate your answer graphically
2. The equivalent impedance of a capacitor in series with an inductor is equivalent to a short circuit (i.e. equal to zero) at a certain frequency. Derive an expression for this frequency.
3. The equivalent impedance of a capacitor in parallel with an inductor is equivalent to an open circuit (i.e. equal to infinity) at a certain frequency. Derive an expression for this frequency

Exp:5

1. What is transient analysis.
2. What is steady state analysis.
3. Explain about rlc ckt.
4. Draw the characteristics of RL .
5. Draw the characteristics of RC.
6. Draw the characteristics of RLC

Exp:6

1. What is different between line and phase.
2. What is line voltage and phase voltage for star and delta.
3. How to measure three phase power using two wattmeter method.
4. How to measure three phase power using single wattmeter method.
5. How to measure three phase power using three wattmeter method.

Exp:7

1. State the maximum power transfer theorem.
2. What are the applications of maximum power transfer theorem?
3. What is the difference in the application of maximum power transfer theorem to AC and DC?
4. Is it possible to find maximum power in any circuit without drawing its equivalent thevenin's circuit?

Exp:8

1. What is reciprocity theorem.
2. What are the advantages of reciprocity theorem.
3. What are the applications of reciprocity theorems.
4. What are the disadvantages of reciprocity theorems.

Exp:9

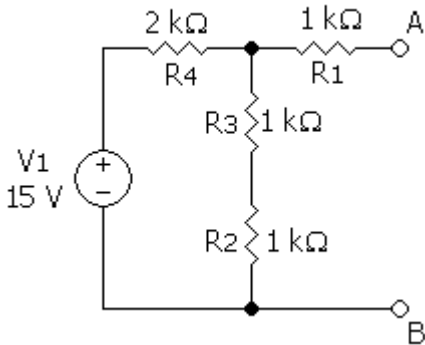
- 1) Where we can apply superposition theorem?
- 2) What is the importance of reciprocity theorem?
- 3) what is bilateral property of the element?
- 4) What is the node in the circuit? Define branch in the circuit?
- 5) State the superposition theorem and reciprocity theorem.

Exp:10

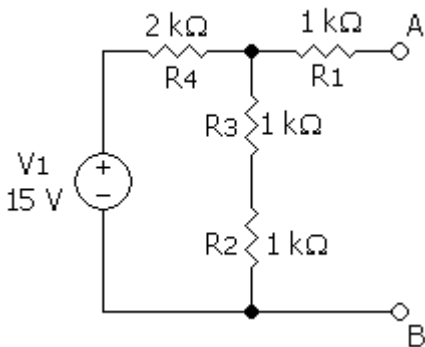
1. What is AC circuits.
2. Draw the phasor diagram of RL CKT.
3. Draw the phasor diagram of RC CKT.
4. Draw the phasor diagram of RLC CKT.

10 SAMPLE QUESTION PAPER OF THE LAB EXTERNAL

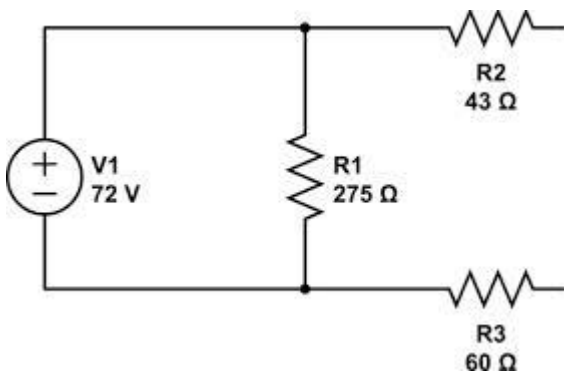
1) Verify the Thevenin's theorem USING PSPICE



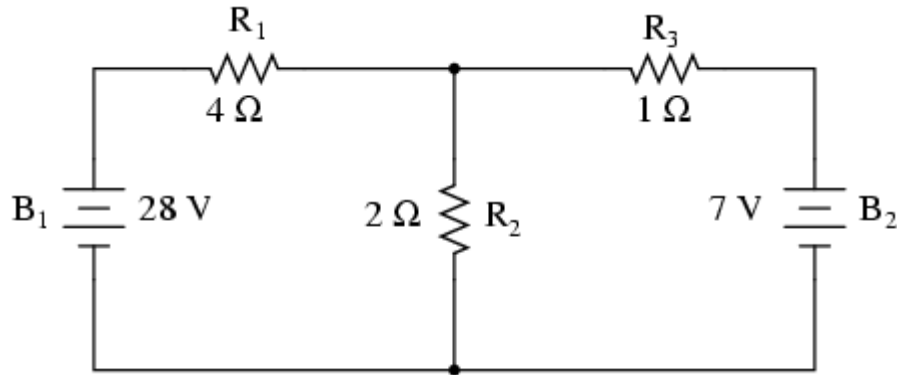
2) Verify the Norton's theorem USING PSPICE



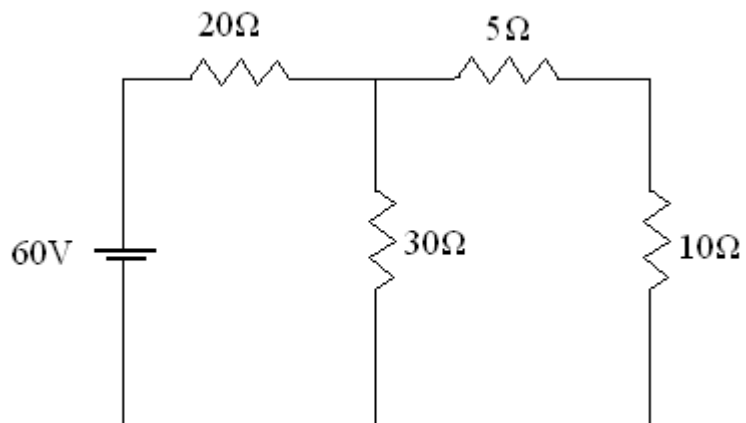
3) Verify the Maximum Power Transfer Theorems USING PSPICE



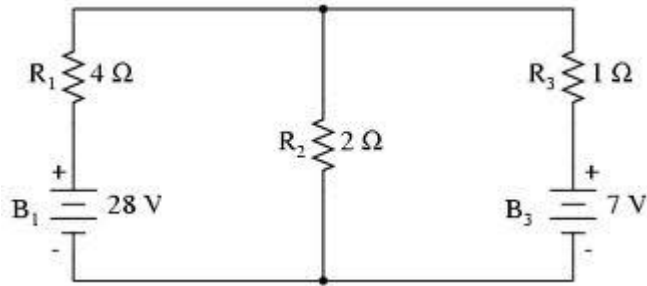
4) Verify the Super position Theorem and RMS value of Complex wave USING PSPICE



5) Verify the Reciprocity, USING PSPICE

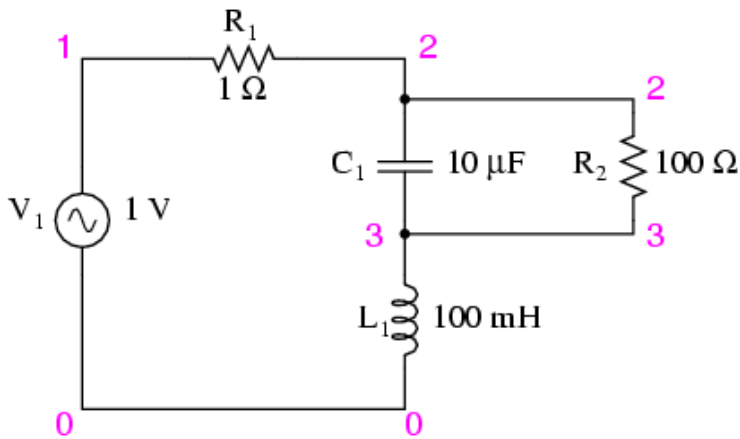


6) Verify the Millman's theorem USING PSPICE

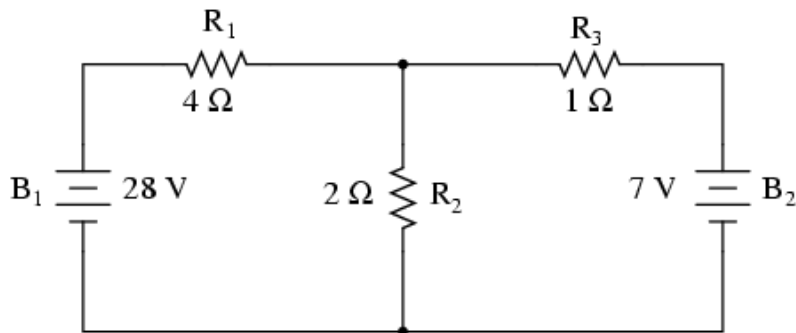


7) Draw the response of Series and parallel resonance

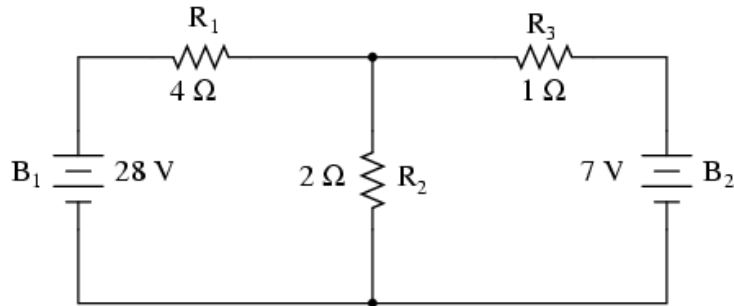
Series LC with resistance in parallel with C



8) Calculate the Z & Y parameters USING PSPICE



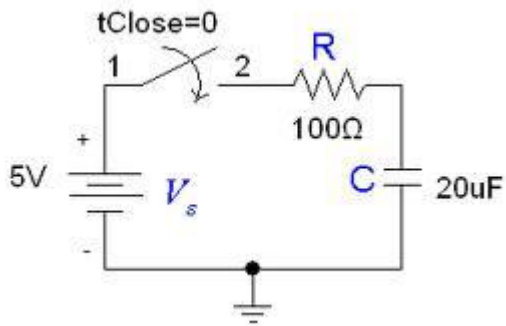
9) Calculate the Transmission & hybrid parameters USING PSPICE



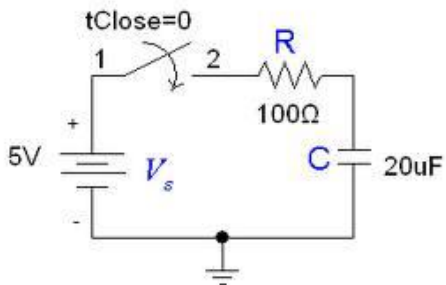
10) Measure the active power for star and delta connected balanced loads USING PSPICE

11) Measure the active power for star and delta connected balanced loads USING PSPICE

12) Simulate the DC Circuits USING PSPICE



13) Simulate the DC Transient Response USING PSPICE



11. APPLICATIONS OF THE LABORATORY

1. Getting electrical knowledge in home appliances as well as basic theorems in Electrical engineering.
2. They will get knowledge about different types of supply systems and different types of motors which fall in AC and DC.
3. Simulation of some dc and ac circuits.

13. PRECAUTIONS TO BE TAKEN WHILE CONDUCTING THE LAB**SAFETY – 1**

- Power must be switched-OFF while making any connections.
- Do not come in contact with live supply.
- Power should always be in switch-OFF condition, EXCEPT while you are taking readings.
- The Circuit diagram should be approved by the faculty before making connections.
- Circuit connections should be checked & approved by the faculty before switching on the power.
- Keep your Experimental Set-up neat and tidy.
- Check the polarities of meters and supplies while making connections.
- Always connect the voltmeter after making all other connections.
- Check the Fuse and it's ratify.
- Use right color and gauge of the fuse.
- All terminations should be firm and no exposed wire.
- Do not use joints for connection wire.

SAFETY – II

1. The voltage employed in electrical lab are sufficiently high to endanger human life.
2. Compulsorily wear shoes.
3. Don't use metal jewelers on hands.
4. Do not wear loose dress

Don't switch on main power unless the faculty gives the permission

