VIT®
Vellore Institute of Technology
(Deemed to be University under section 3 of UGC Act, 1956)

SCHOOL OF ELECTRONICS ENGINEERING

ECE1001 FUNDAMENTALS OF ELECTRICAL CIRCUITS

LAB MANUAL

LIST OF EXPERIMENTS

	HARDWARE
1.	Verification of Basic Laws
2.	Node Voltage and Mesh Current Analysis
3.	Thevenin's and Maximum Power Transfer theorem
4.	Superposition and Reciprocity Theorem
5.	Transients response of First order circuit
6.	Steady state analysis of AC circuit
7.	Frequency response of series resonance circuit
	SOFTWARE
1.	Verification of Basic Laws
2.	Node Voltage and Mesh Current Analysis
3.	Thevenin's , Norton's and Maximum Power Transfer theorem
4.	Superposition and Reciprocity Theorem
5.	Transients response of First order circuit and second order circuit
6.	Steady state analysis of AC circuit
7.	Frequency response of series and parallel resonance circuit

CONTENT

Experiment No. 1a Date :	VERIFICATION OF BASIC LAWS

Aim:

To verify the following laws using discrete component.

a. Ohm’s Law.

b. Kirchhoff’s Current Law.

c. Kirchhoff’s Voltage Law.

Apparatus Required:

Sl.No.	Apparatus	Range	Quantity
1	RPS (regulated power supply)	(0-30V)	2
2	Ammeter	(0-10mA)	1
3	Voltmeter		1
4	Resistors	100Ω, 200Ω, 300Ω	3
5	Bread Board	--	--
6	Wires	--	Required

Theory:

Ohms’s Law:

Ohm's Law deals with the relationship between voltage and current in an ideal conductor. This relationship states that “The potential difference (voltage) across an ideal conductor is proportional to the current through it”.

The constant of proportionality is called the "resistance", R.

Ohm's Law is given by:

$V = I R$

Where

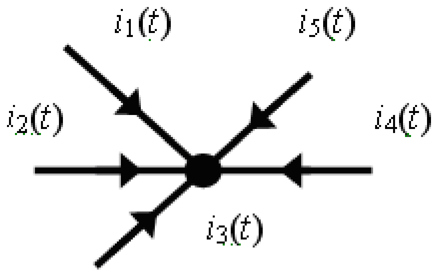
V is the potential difference between two points which include a resistance R.

I is the current flowing through the resistance.

Kirchhoff’s Current Law (KCL)

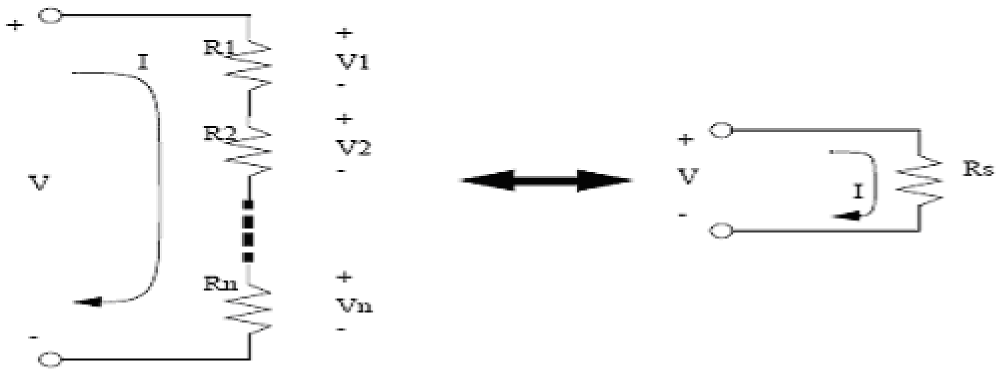
1. Sum of all currents entering a node is zero.
2. Sum of currents entering the node is equal to sum of currents leaving the node.

$$\sum_{j=1}^{n} i_j(t) = 0$$



Kirchhoff’s Voltage Law (KVL)

- Sum of voltages around any loop in a circuit is zero.



First, sum the voltages about the loop.

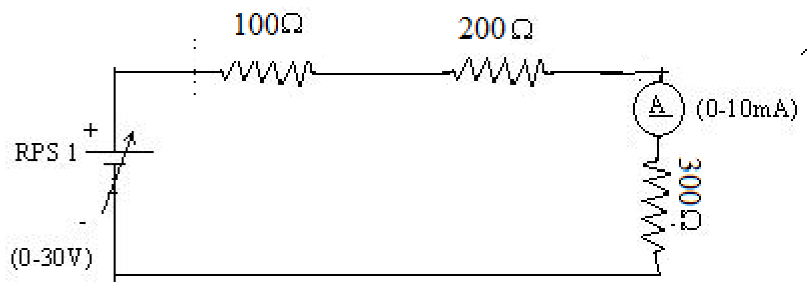
$\Sigma V = V_1 + V_2 + \ldots\ldots + V_n = 0.$

Precautions:

1. Voltage control knob should be kept at manimum position
2. current control knob of RPS should be kept at maximum position

Procedure:

1. Make the connections as per circuit diagram. All connections must be neat and tight. Take care to connect the ammeter and voltmeter with their correct polarity. (+ve to +ve and -ve to -ve).
2. Adjust the input DC voltage ranging from 0 to 10V with step range of 0.5V.
3. Record the readings of the ammeter.
4. Plot a graph with **V** along x-axis and **I** along y-axis.
5. The graph will be a straight line which verifies Ohm's law.
6. Determine the slope of the V-I graph. The reciprocal of the slope gives resistance of the wire.



Tabular Column

Voltage (Vin) in Volts	Current (I) through 300 Ω Resistor	Voltage (V) across 300 Ω Resistor	V/I

Result

It is verified that for all range of input voltage applied the current changes according and the ratio of V/I remains constant

Experiment No. 1b Date :	VERIFICATION OF BASIC LAWS

Aim:

To verify the following laws using LTSPICE tool.

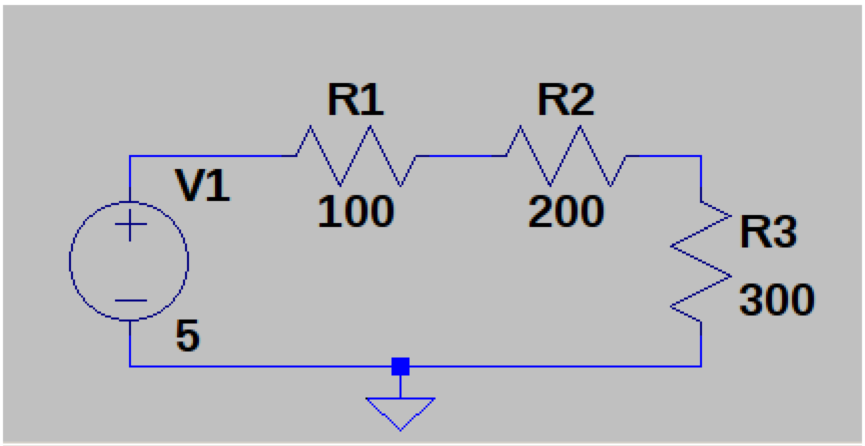
a. Ohm's Law.
b. Kirchhoff's current Law.
c. Kirchhoff's Voltage Law.

Procedure:

1. Draw the given circuit in the LTSPICE with new schematic.
2. Change the value of source and resistors.
3. Place the Ground.
4. Set the simulation profile for Operating point analysis.
5. Run the simulation and note the results

Verification of Ohm's Law

Circuit Diagram 1:



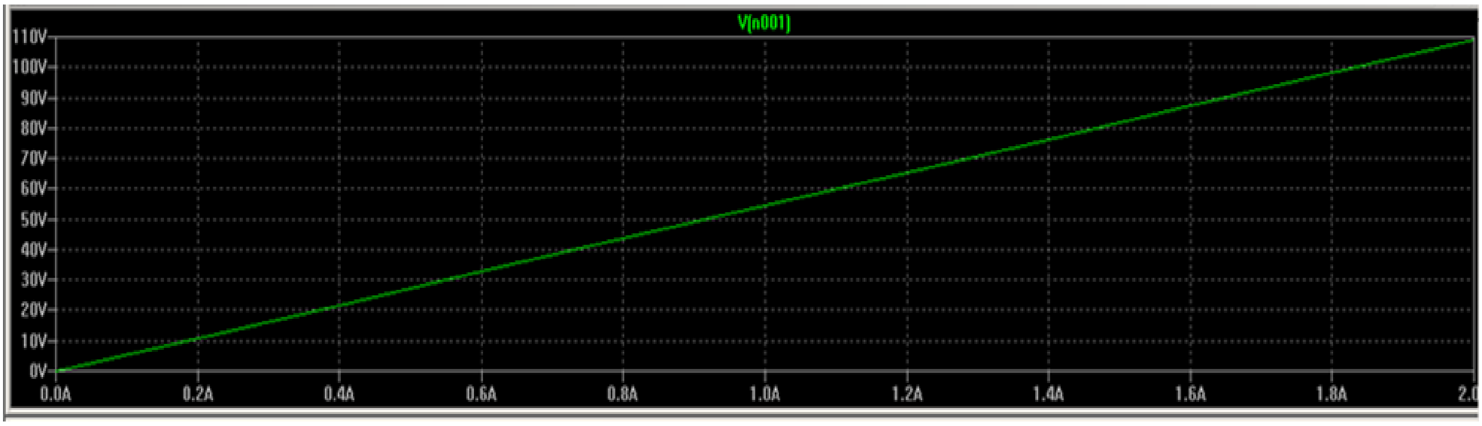
* C:\Program Files (x86)\LTC\LTspiceIV\rolano4.asc

--- Operating Point ---

V(n002):	4.16667	voltage
V(n001):	5	voltage
V(n003):	2.5	voltage
I(R3):	0.00833333	device_current
I(R2):	-0.00833333	device_current
I(R1):	-0.00833333	device_current
I(V1):	-0.00833333	device_current

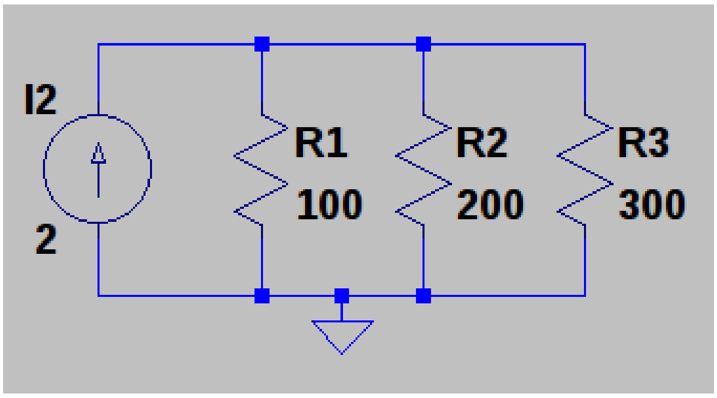
Model Calculations:

Graph 1



<u>Kirchhoff's Current Law</u>

Circuit Diagram 2:



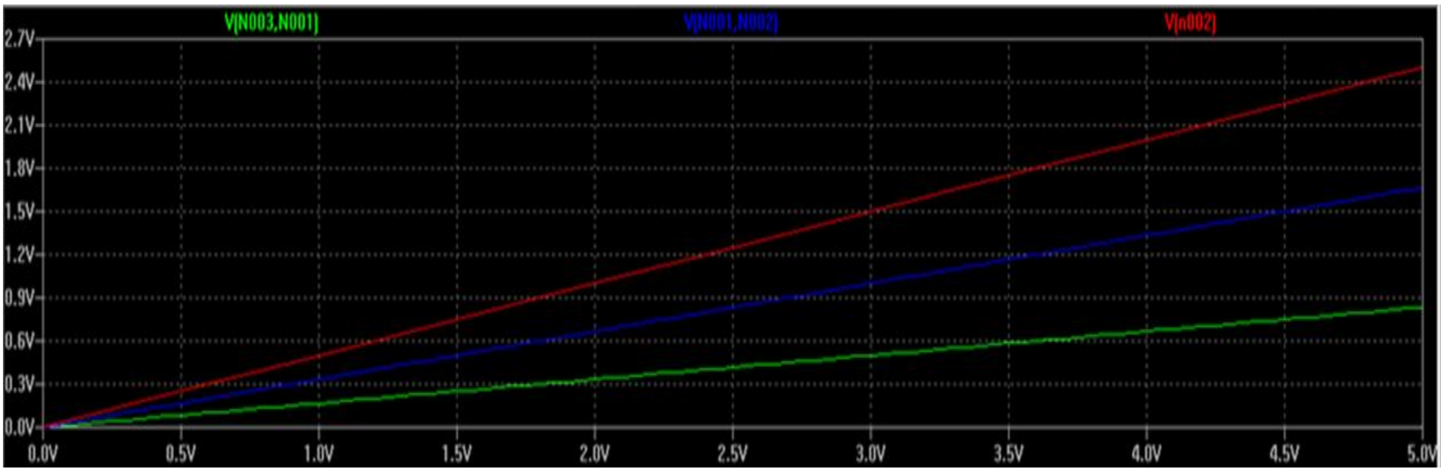
Model Calculations:

* C:\Program Files (x86)\LTC\LTspiceIV\rolanoparallel.asc

--- Operating Point ---

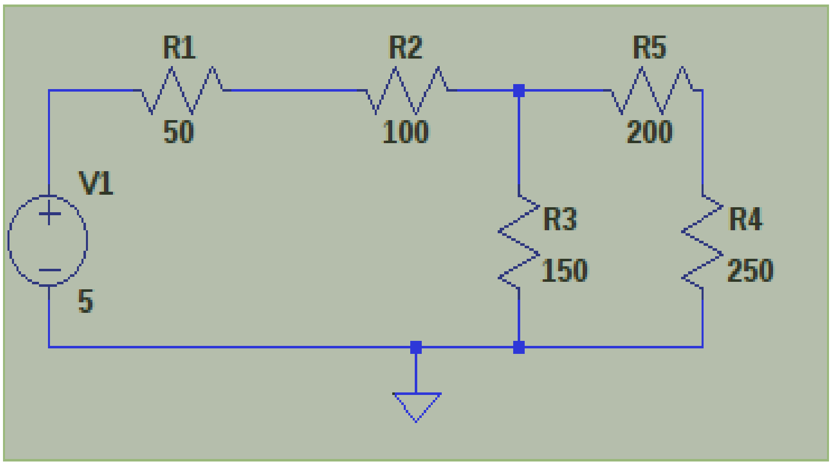
V(n001):	109.091	voltage
I(I2):	2	device_current
I(R3):	0.363636	device_current
I(R2):	0.545455	device_current
I(R1):	1.09091	device_current

Graph 2:



<u>Kirchhoff's Voltage Law</u>

Circuit Diagram 3:



* C:\Program Files (x86)\LTC\LTspiceIV\rolanoparallel.asc

--- Operating Point ---

V(n002):	4.04762	voltage
V(n001):	5	voltage
V(n003):	2.14286	voltage
V(n004):	1.19048	voltage
I(R4):	0.0047619	device_current
I(R3):	0.0142857	device_current
I(R5):	-0.0047619	device_current
I(R2):	-0.0190476	device_current
I(R1):	-0.0190476	device_current
I(V1):	-0.0190476	device_current

Experiment No. 2a **Date :**	**MESH CURRENT ANALYSIS AND NODAL VOLTAGE ANALYSIS**

Aim:

To verify the Mesh Current Analysis and Nodal Voltage Analysis

Apparatus Required:

Sl.No.	Apparatus	Range	Quantity
1	RPS (regulated power supply)	(0-30V)	2
2	Ammeter	(0-10mA)	1
3	Resistors	10□, 20□, 40□	1
4	Bread Board	--	--
5	Wires	--	Required

Statement:

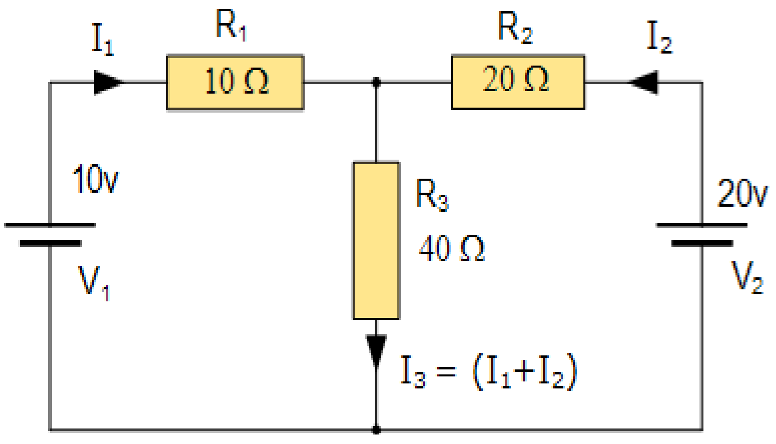
Kirchoff's first law equation, that is: *"the currents entering a node are exactly equal in value to the currents leaving the node"*

Precautions:

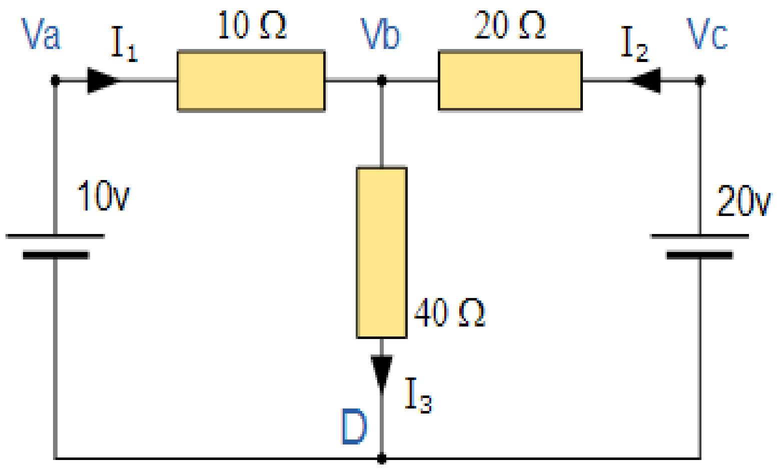
3. Voltage control knob should be kept at minimum position
4. current control knob of RPS should be kept at maximum position

Procedure:

7. Give the connections as per the diagram.
8. Set a particular voltage value using RPS_1 and RPS_2 & note down the ammeter reading
9. Set the same voltage in circuit I using RPS_1 alone and short circuit the terminals and note the ammeter reading.
10. Set the same voltage in RPS_2 alone as in circuit I and note down the ammeter reading.



CIRCUIT – 1 - Mesh Current Analysis Circuit



CIRCUIT – 2 - Nodal Voltage Analysis Circuit

TABULAR COLUMN

Theoretical Values

	RPS		Ammeter Reading (I) mA
	1	2	
Circuit – 1			
Circuit – 2			

Practical Values

	RPS		Ammeter Reading (I) mA
	1	2	
Circuit – 1			
Circuit – 2			

Model Calculations:

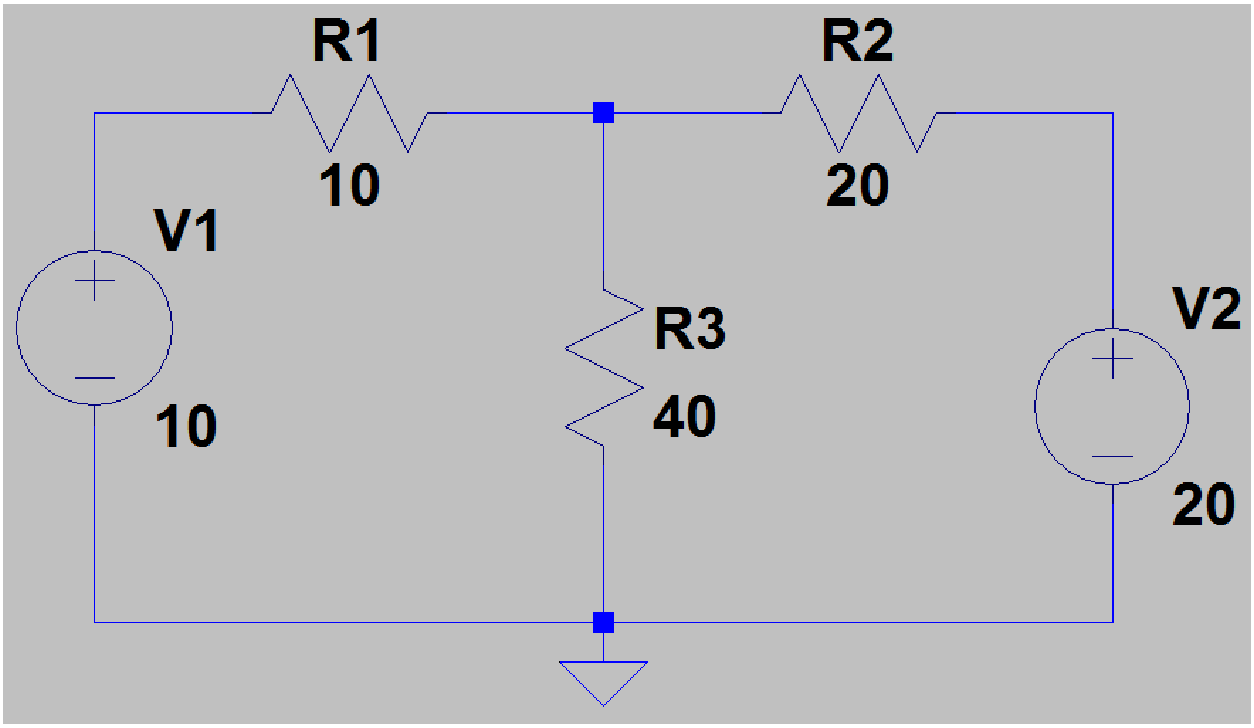
Result:

Both mesh current and nodal voltage analysis have been verified theoretically and practically.

Experiment No. 2b Date :	MESH CURRENT ANALYSIS AND NODAL VOLTAGE ANALYSIS

Aim: To verify Mesh Current Analysis and Nodal Voltage Analysis

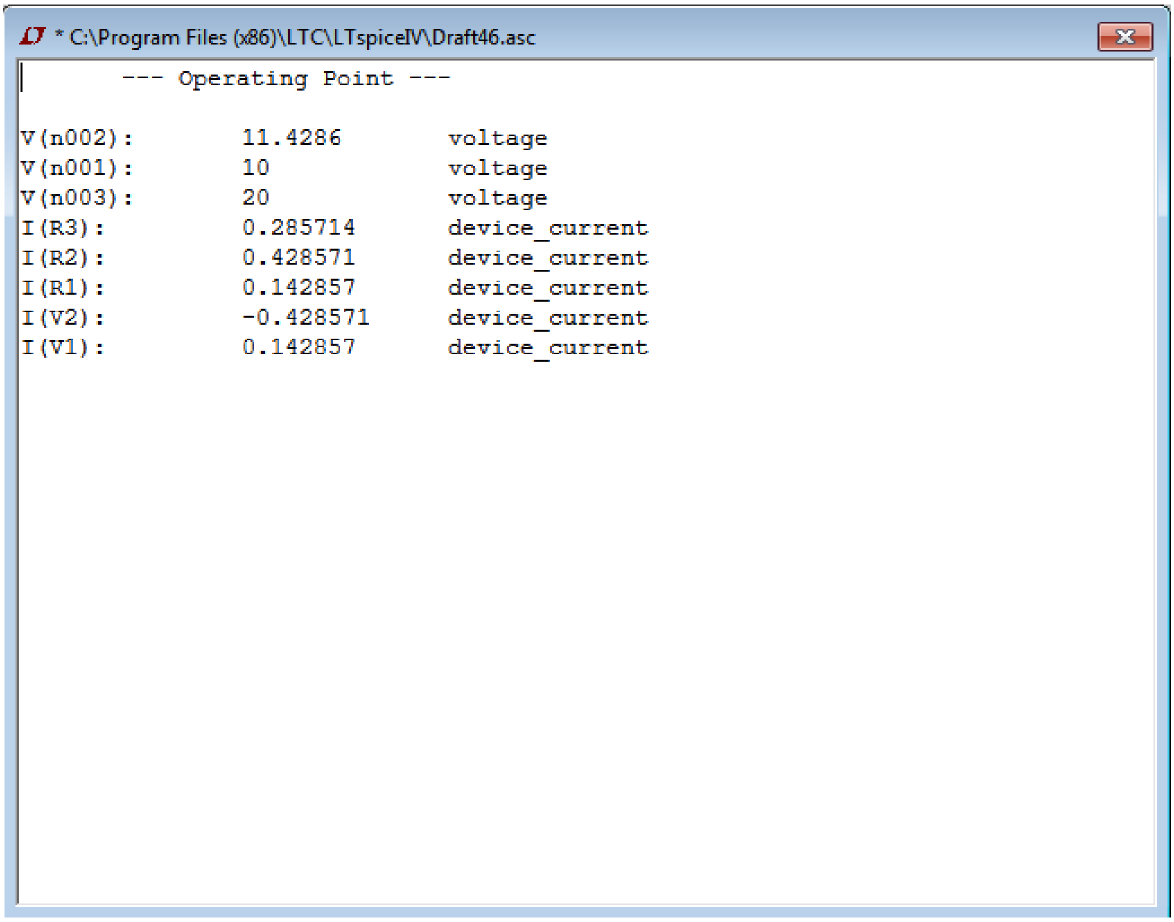
Circuit Diagram:



PROCEDURE : Using LT Spice simulation tool, create the above circuit using a resistor, a voltage source, current source and a zero potential and give the corresponding values to the resistor and the source voltage by right clicking the components.

- Click on edit simulate command under simulate section then select DC op point and then run the circuit to check the operating voltage and current at each component.
- As its name implies, **Nodal Voltage Analysis** uses the "Nodal" equations of Kirchoff's first law to find the voltage potentials around the circuit.
- At each node point write down Kirchoff's first law equation, that is: "*the currents entering a node are exactly equal in value to the currents leaving the node*" then express each current in terms of the voltage across the branch.
- For Mesh analysis use Kirchoff's Current Law equations to determine the currents, I_1 and I_2 flowing in the two resistors.

Mesh Current Analysis and Nodal Voltage Analysis Output



* C:\Program Files (x86)\LTC\LTspiceIV\Draft46.asc

```
       --- Operating Point ---

V(n002):       11.4286       voltage
V(n001):       10            voltage
V(n003):       20            voltage
I(R3):         0.285714      device_current
I(R2):         0.428571      device_current
I(R1):         0.142857      device_current
I(V2):         -0.428571     device_current
I(V1):         0.142857      device_current
```

INFERENCE: It has been observed that current through R2 due to each source independently equals current through R2 when both are active.

Experiment No. 3a Date :	VERIFICATION OF THEVENIN'S AND MAXIMUM POWER TRANSFER THEOREM

(i) Thevenin's Theorem

Aim:

To verify Thevenin's Theorem for the given electrical circuit.

Apparatus Required:

S.No	Apparatus	Range	
1	Regulated power supply	-	1
2	Ammeter	(0-20) mA	1
3	Voltmeter	(0-30) V	1
4	Resistors	1kΩ	4
5	Breadboard	-	1
6	Connecting Wires	-	As required

Theory:

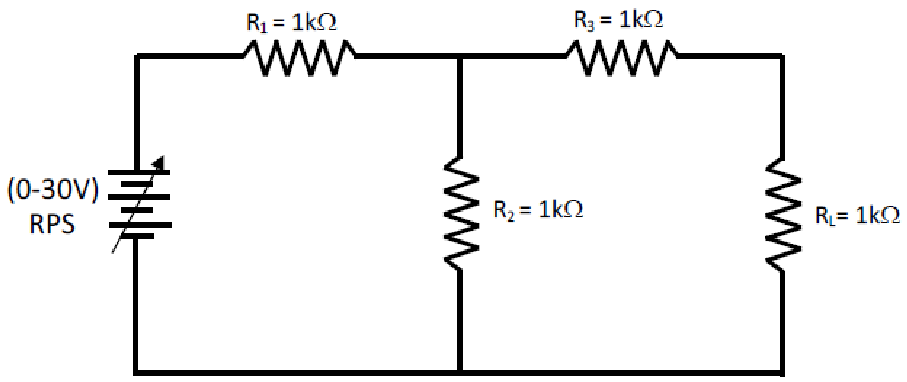
Thevenin's Theorem:

Any linear active network with output terminals A and B can be replaced by an equivalent circuit with a single voltage source Vth (thevenin's voltage) in series with Rth (thevenin's resistance)

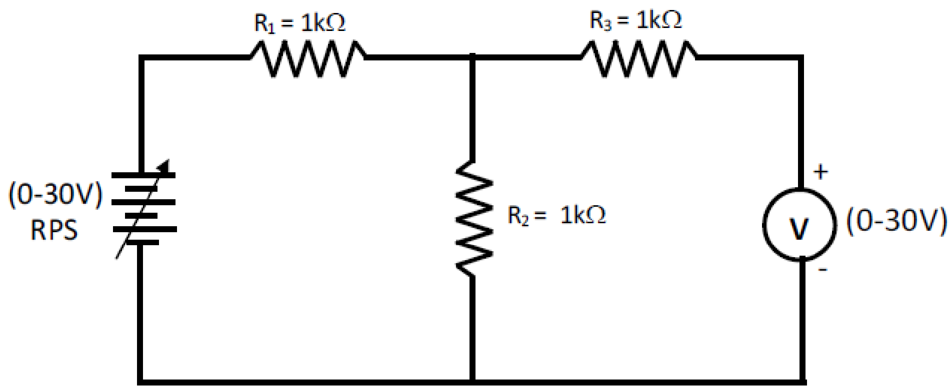
Vth - open circuit voltage across terminal A & B

Rth - equivalent resistance obtained by looking back the circuit through the open circuit terminal A and B

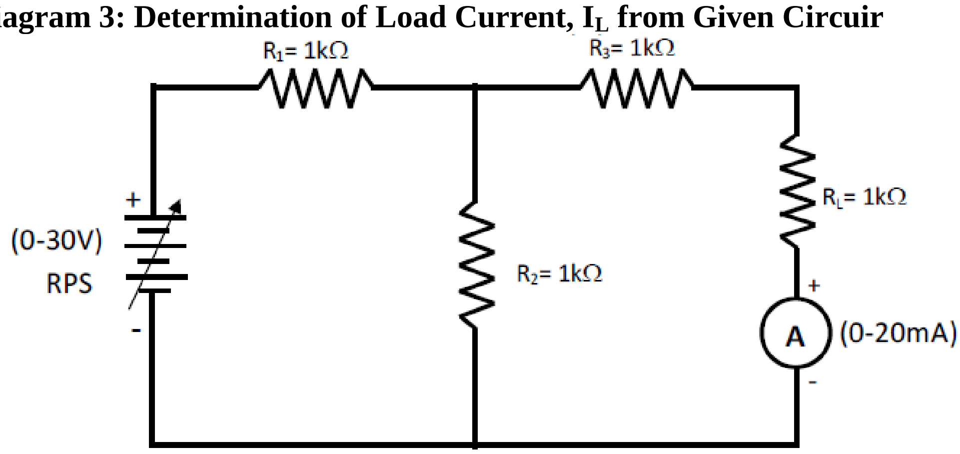
Circuit Diagram 1:



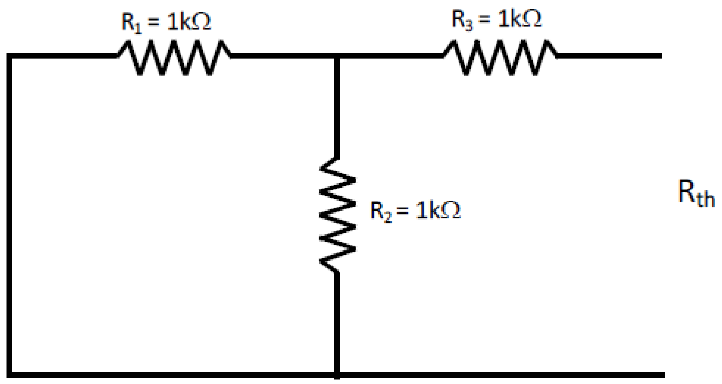
Circuit Diagram 2: Practical Determination of Thevenin's Voltage, Vth



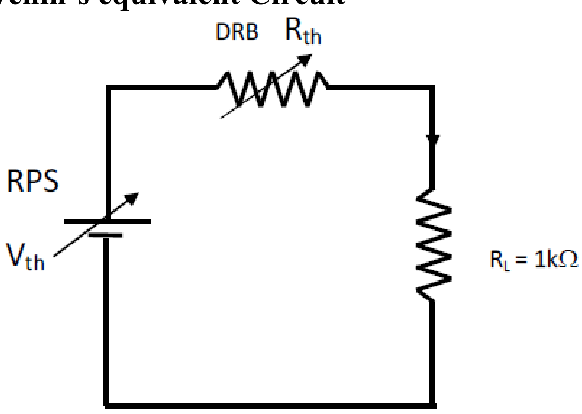
Circuit Diagram 3: Determination of Load Current, I_L from Given Circuir



Circuit Diagram 4: Determination of Thevenin's Resistance, Rth



Circuit Diagram 5: Thevenin's equivalent Circuit



Procedure:

(i) Determination of Thevenin's Voltage, Vth:

1. Make the connections as per the circuit diagram 2.
2. Switch on the power supply

3. Vary the regulated power supply to a specified voltage and note down the corresponding voltmeter readings
4. Switch off the power supply

(ii) Determination of Load Current, I_L:

1. Make the connections as per the circuit diagram 3.
2. Switch on the power supply
3. Vary the regulated power supply to a specified voltage (as in procedure (i))and note down the corresponding ammeter readings
4. Switch off the power supply

(iii) Determination of Thevenin's Resistance, Rth:

1. Make the connections as per the circuit diagram 4.
2. Determine the Resistance value seen from the open terminals.

(iv) Verification of Thevenin's Theorem:

1. Make the connections as per the circuit diagram 5.
2. Determine the current across the Load Resistor.
3. Compare the value with the value obtained in Procedure (ii).

Theoretical Calculations:

(i) Calculation for Vth:

(ii) Calculation of Rth:

Tabulation:

Supply Voltage (V)	Thevenin's Voltage		Thevenin's resistance		Load current	
	Vth (V) (Theoretical)	Vth (V) (Practical)	Rth (Ω) (Theoretical)	Rth (Ω) (Practical)	I_L(mA) (From Given Circuit)	I_L(mA) (From Thevenin's Circuit)

Result:

Thus from the load current values, it is proved that Thevenin's Theorem holds good for the given electrical circuit.

(ii) Maximum Power Transfer Theorem

Aim:

To verify maximum power transfer theorem for the given circuit

Apparatus Required:

Sl.No.	Apparatus	Range	Quantity
1	RPS	(0-30V)	1
2	Voltmeter	(0-10V) MC	1
3	Resistor	1KΩ, 1.3KΩ, 3Ω	3
4	DRB	--	1
5	Bread Board & wires	--	Required

Statement:

In a linear, bilateral circuit the maximum power will be transferred to the load when load resistance is equal to source resistance.

Precautions:

1. Voltage control knob of RPS should be kept at minimum position.
2. Current control knob of RPS should be kept at maximum position.

Procedure:

Circuit – I

1. Connections are given as per the diagram and set a particular voltage in RPS.
2. Vary R_L and note down the corresponding ammeter and voltmeter reading.
3. Repeat the procedure for different values of R_L & Tabulate it.
4. Calculate the power for each value of R_L.

To find V_{TH}:

5. Remove the load, and determine the open circuit voltage using multimeter (V_{TH})

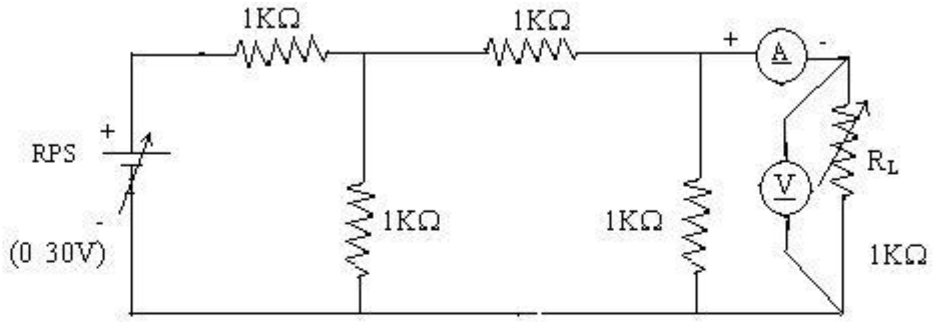
To find R_{TH}:

6. Remove the load and short circuit the voltage source (RPS).
7. Find the looking back resistance (R_{TH}) using multimeter.

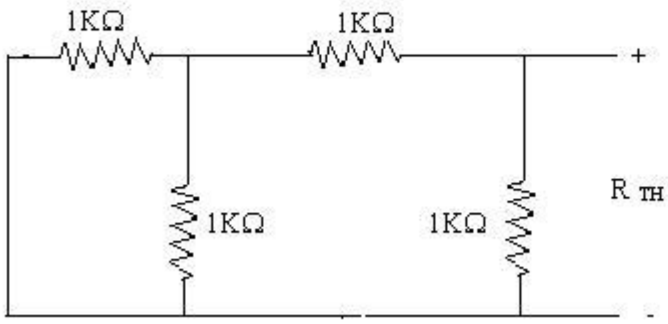
Equivalent Circuit:

8. Set V_{TH} using RPS and R_{TH} using DRB and note down the ammeter reading.
9. Calculate the power delivered to the load ($R_L = R_{TH}$)
10. Verify maximum transfer theorem.

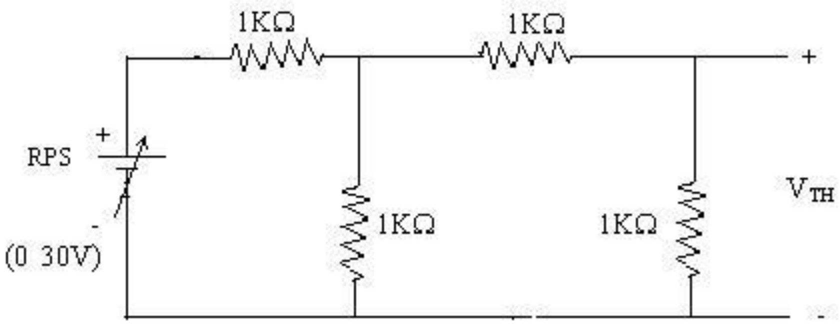
Circuit - 1



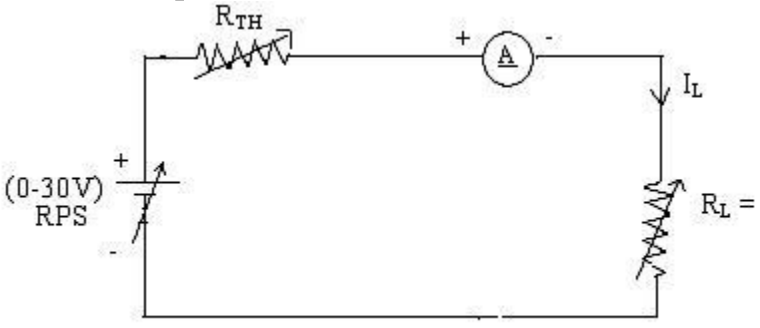
To find V_{TH}



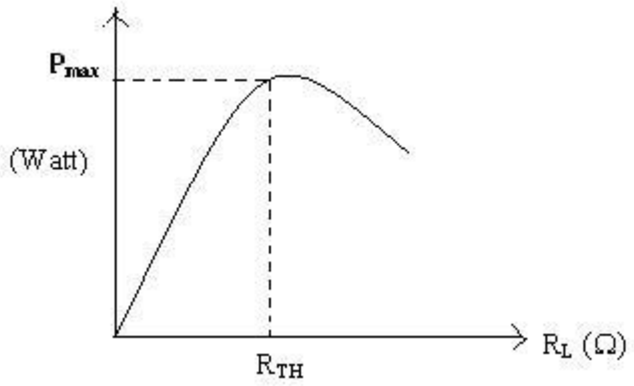
To find R_{TH}



Thevenin's Equation Circuit



Power V_S R_L



Circuit – I

Sl.No.	RL (Ω)	I (mA)	V(V)	P=VI (watts)
1				
2				
3				
4				
5				
6				
7				
8				

To find Thevenin's equivalent circuit

	V_{TH} (V)	R_{TH} (Ω)	I_L (mA)	P (milli watts)
Theoretical Value	2002	1320	0.758	0.759
Practical Value				

Model Calculations:

Result:

Thus maximum power theorem was verified both practically and theoretically

Experiment No. 3b Date :	**VERIFICATION OF THEVENIN'S , NORTON'S AND MAXIMUM POWER TRANSFER THEOREM USING LTSPICE**

(i) VERIFICATION OF THEVENIN'S THEOREM

Aim:

To verify Thevenin's Theorem for the given electrical circuit using LTSPICE Simulation.

Apparatus Required:

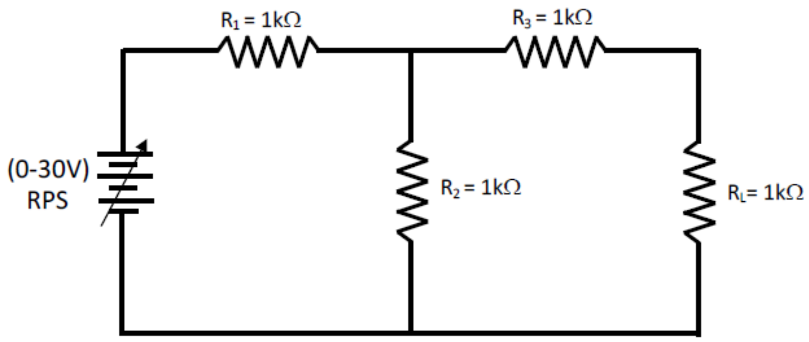
LTSPICE Software

Theory: Thevenin's Theorem:

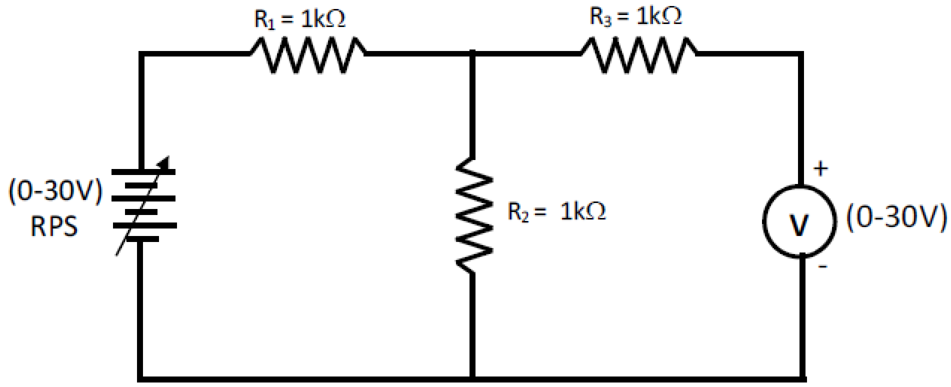
Any linear active network with output terminals A and B can be replaced by an equivalent circuit with a single voltage source Vth (thevenin's voltage) in series with Rth (thevenin's resistance)

Vth - open circuit voltage across terminal A & B
Rth - equivalent resistance obtained by looking back the circuit through the open circuit terminal A and B

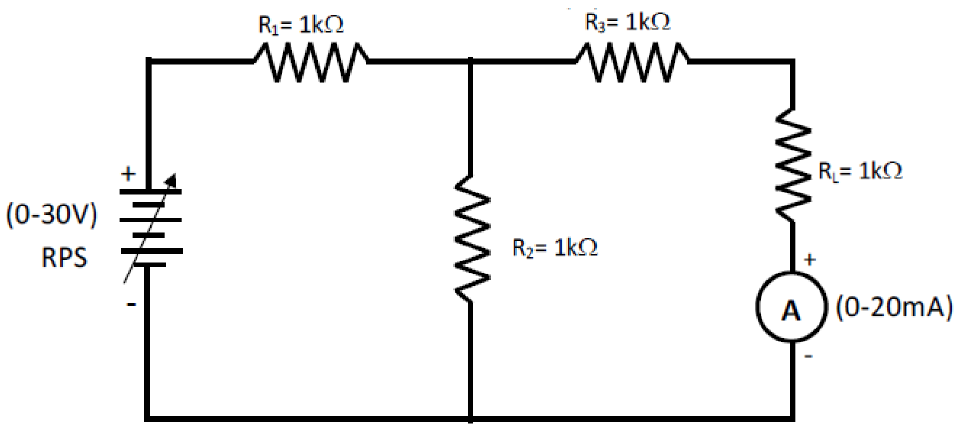
Circuit Diagram 1:



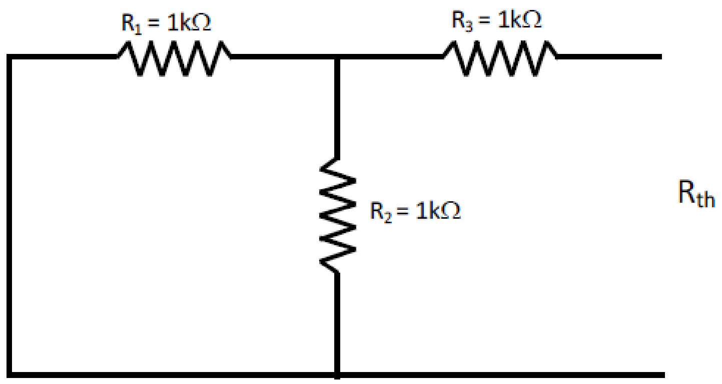
Circuit Diagram 2: Determination of Thevenin's Voltage, Vth



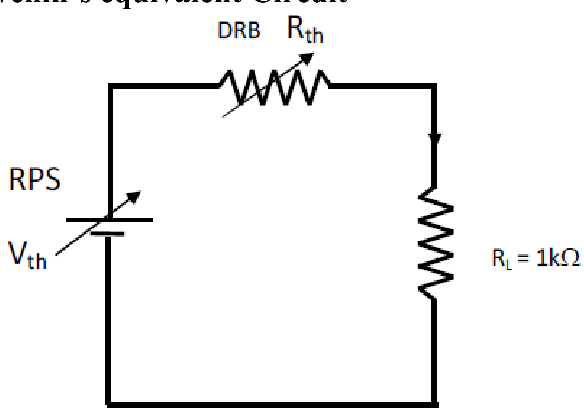
Circuit Diagram 3: Determination of Load Current, I_L from Given Circuir



Circuit Diagram 4: Determination of Thevenin's Resistance, Rth



Circuit Diagram 5: Thevenin's equivalent Circuit



Procedure:

(v) **Determination of Thevenin's Voltage, Vth:**

5. Design a circuit using LTSPICE as per the circuit diagram 2.
6. Note down the voltage using voltage probe.

(vi) **Determination of Load Current, I_L:**

5. Make the connections as per the circuit diagram 3.
6. Note down the current value.

(vii) **Determination of Thevenin's Resistance, Rth:**

3. Make the connections as per the circuit diagram 4.
4. Determine the Resistance value seen from the open terminals.

(viii) Verification of Thevenin's Theorem:

4. Make the connections as per the circuit diagram 5.
5. Determine the current across the Load Resistor.
6. Compare the value with the value obtained in Procedure (ii).

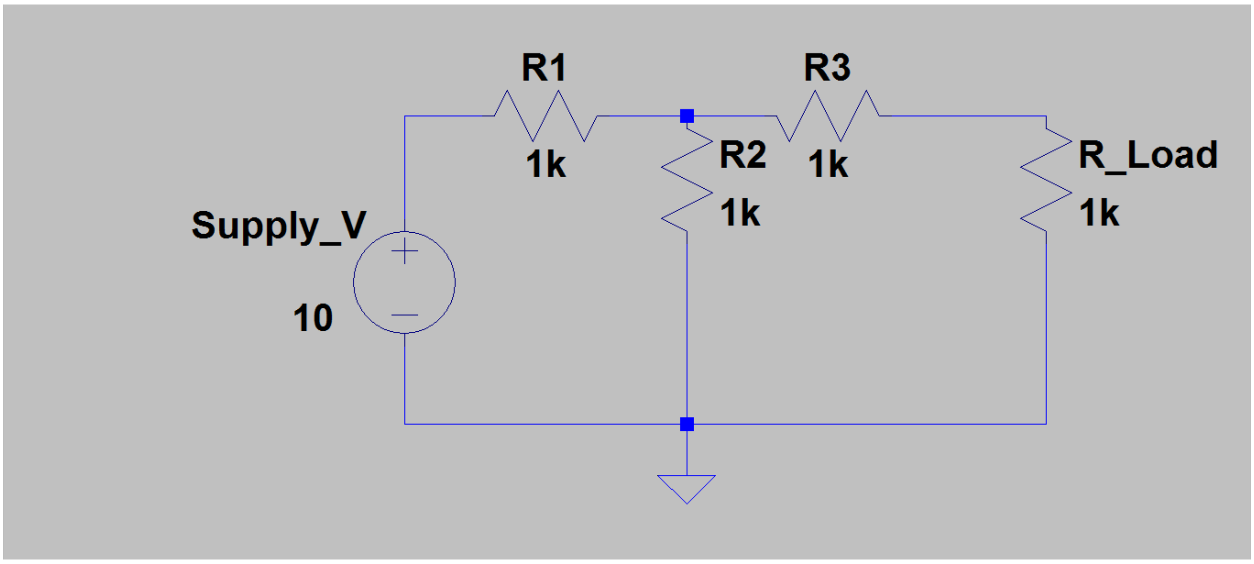
Theoretical Calculations:

(iii) Calculation for Vth:

(iv) Calculation of Rth:

Simulation Steps:

Circuit Diagram 1:

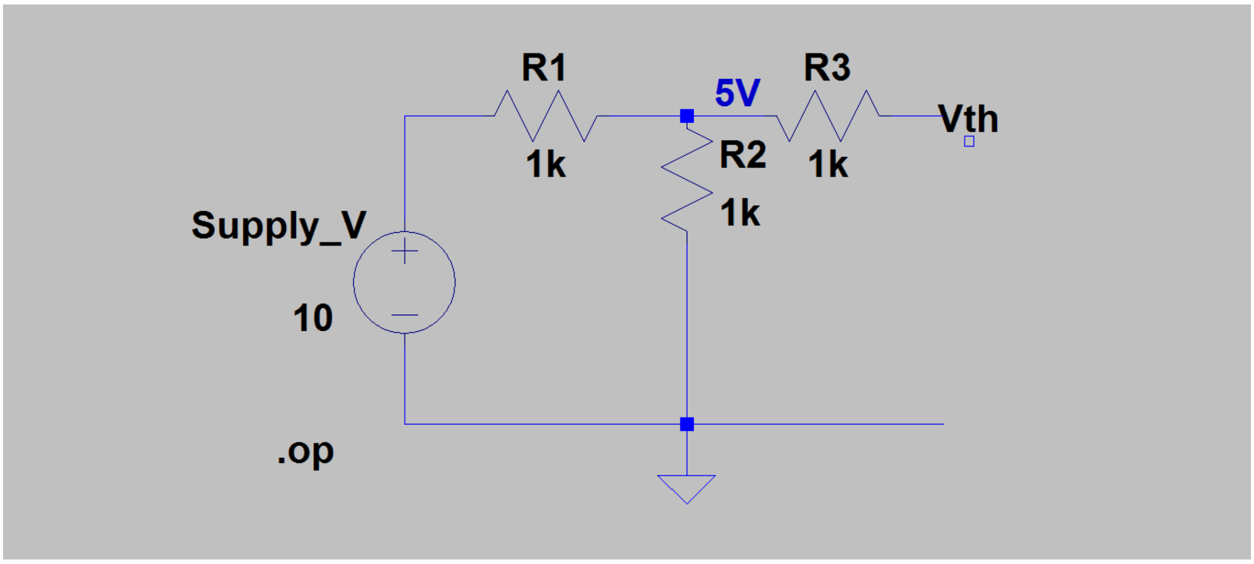


--- Operating Point ---

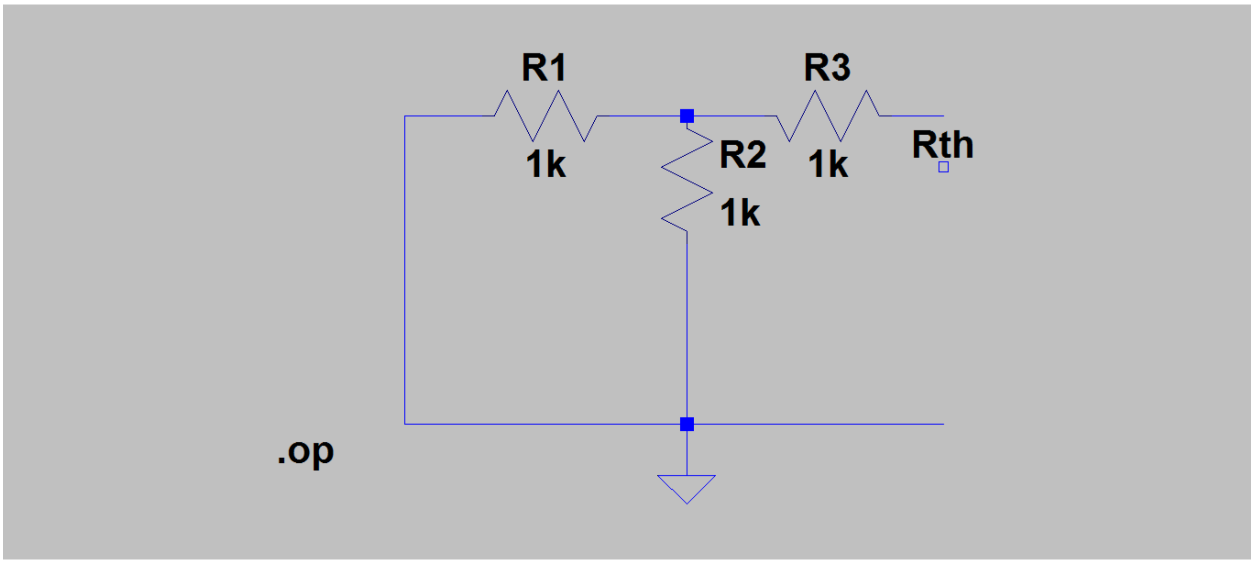
V(n001):	10	voltage
V(n002):	4	voltage
V(n003):	2	voltage
I(R_load):	0.002	device_current
I(R3):	-0.002	device_current

I(R2):	0.004	device_current
I(R1):	-0.006	device_current
I(Supply_v):	-0.006	device_current

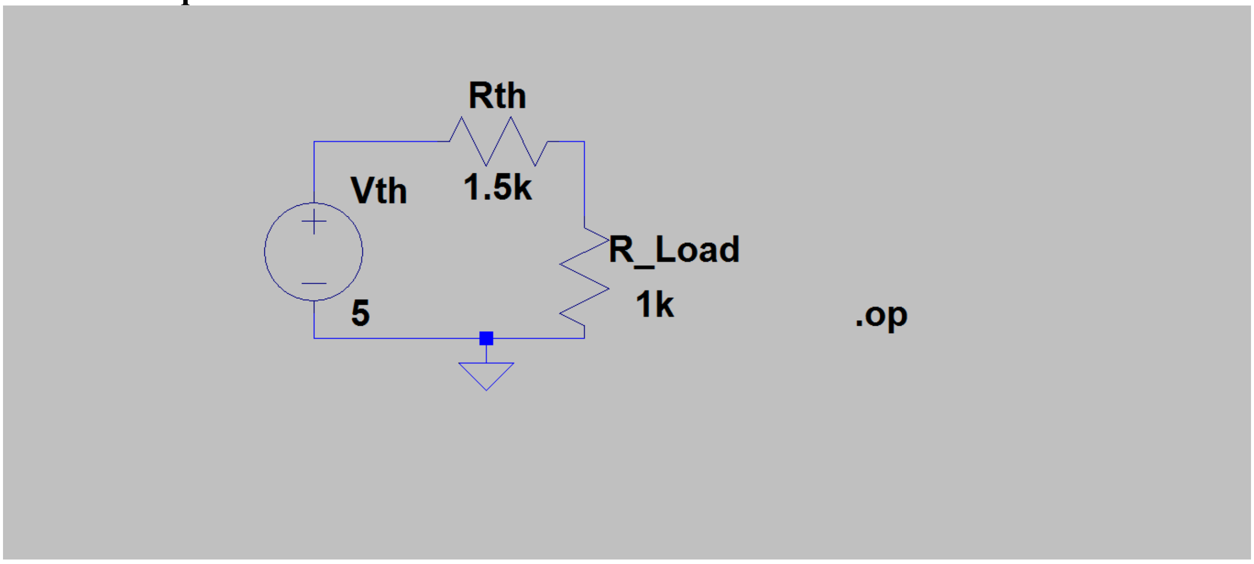
Circuit Diagram 2: Determination of Thevenin's Voltage, Vth



Circuit Diagram 3: Determination of Thevenin's Resistance, Rth



Thevenin's equivalent Circuit



--- Operating Point ---

V(n002):	2	voltage
V(n001):	5	voltage
I(R_load):	-0.002	device_current
I(Rth):	-0.002	device_current
I(Vth):	-0.002	device_current

Supply Voltage (V)	Thevenin's Voltage		Thevenin's resistance		Load current	
	Vth (V) (Theoretical)	Vth (V) (Simulation)	Rth (Ω) (Theoretical)	Rth (Ω) (Simulation)	I_L(mA) (From Given Circuit)	I_L(mA) (From Thevenin's Circuit)

Tabulation:

Result:

The reciprocity theorem was verified for given network with the theoretical calculation.

(ii) VERIFICATION OF NORTON'S THEOREM

Aim:

To verify Norton's Theorem for the given electrical circuit using LTSPICE Simulation.

Apparatus Required:

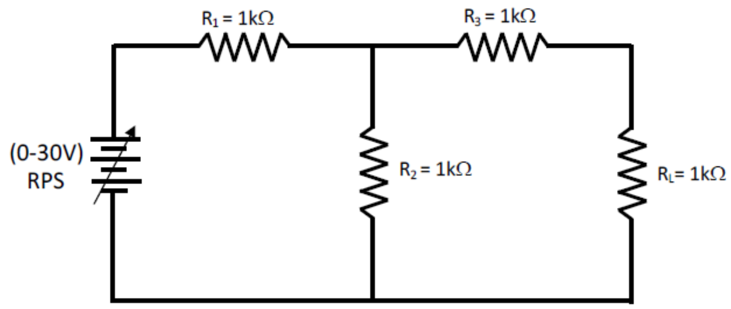
LTSPICE Software

Theory: Norton's Theorem:

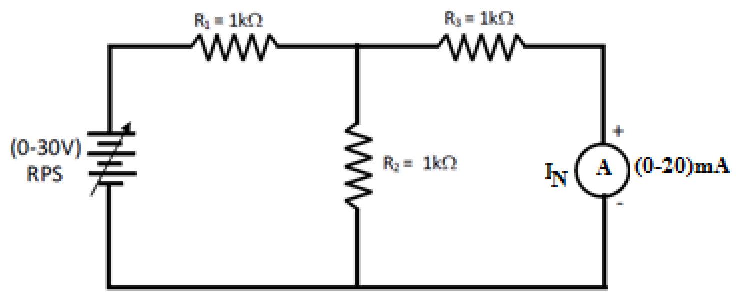
Any linear active network with output terminals A & B can be replaced by an equivalent circuit with a single current source I in parallel with Rth (Thevenin equivalent resistance)

Where Rth is the equivalent resistance obtained by looking back the circuit through the open terminal A & B

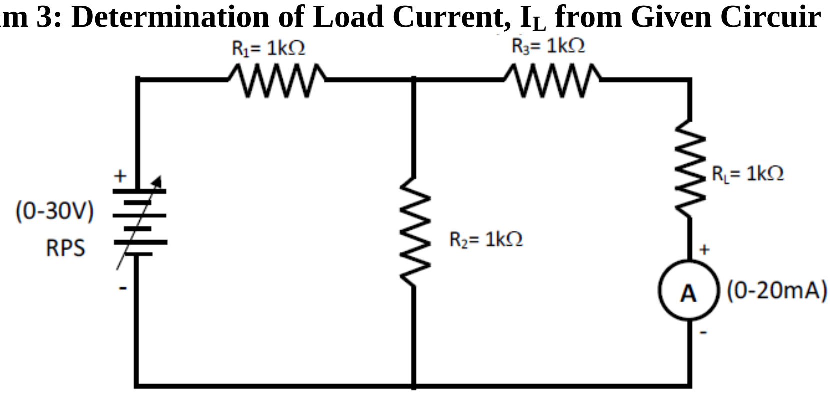
Circuit Diagram 1:



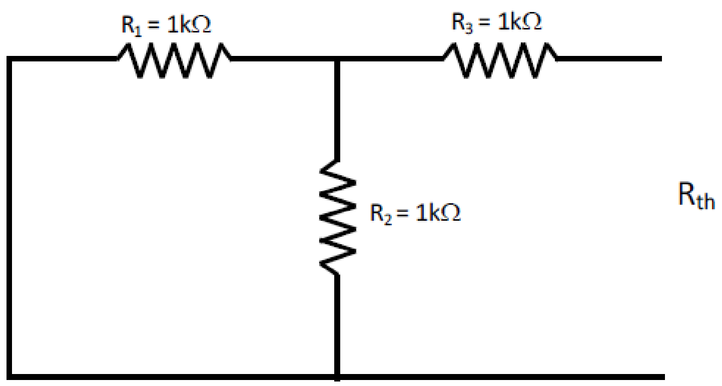
Circuit Diagram 2: Determination of Norton's Current, I_N



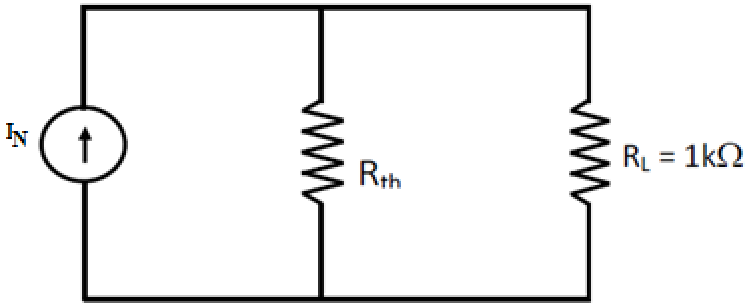
Circuit Diagram 3: Determination of Load Current, I_L from Given Circuir



Circuit Diagram 4: Determination of Thevenin's Resistance, Rth



Circuit Diagram 5: Norton's equivalent Circuit

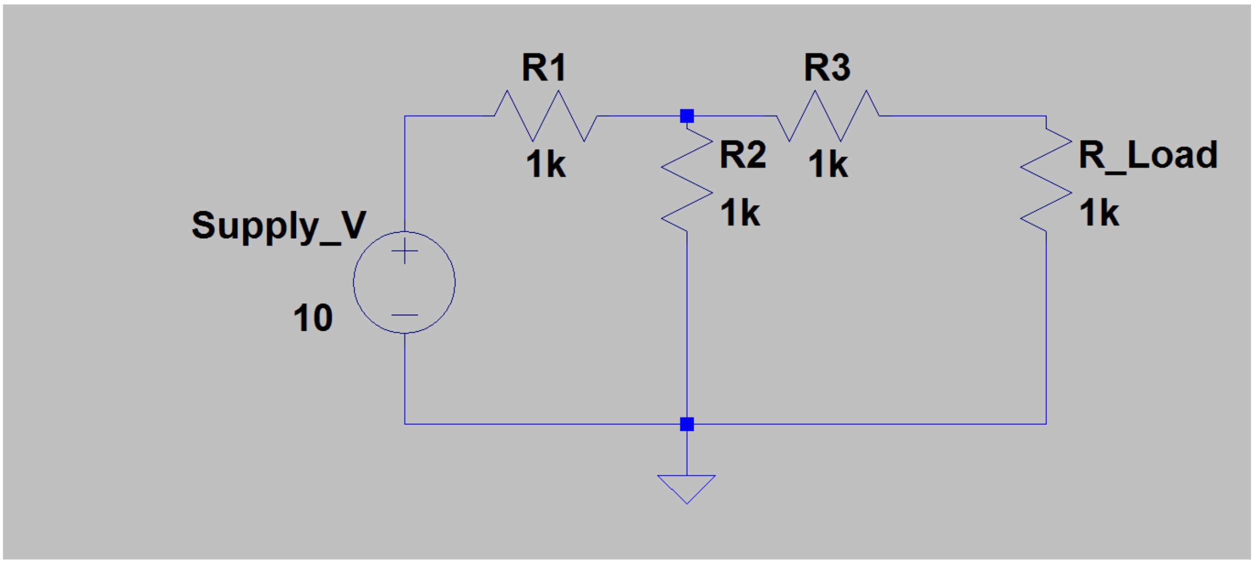


Procedure:

1. Give connections as per the circuit diagram.
2. Measure the current through RL.
3. Short Circuit the output terminals and measure the short circuit current.
4. Find out the Norton's Resistance viewed from the output terminals.
5. Give connections as per the Norton's Equivalent circuit.
6. Measure the current through RL.
7. Verify Norton's theorem by comparing currents in RL directly and that obtained with the equivalent circuit.

Simulation Steps:

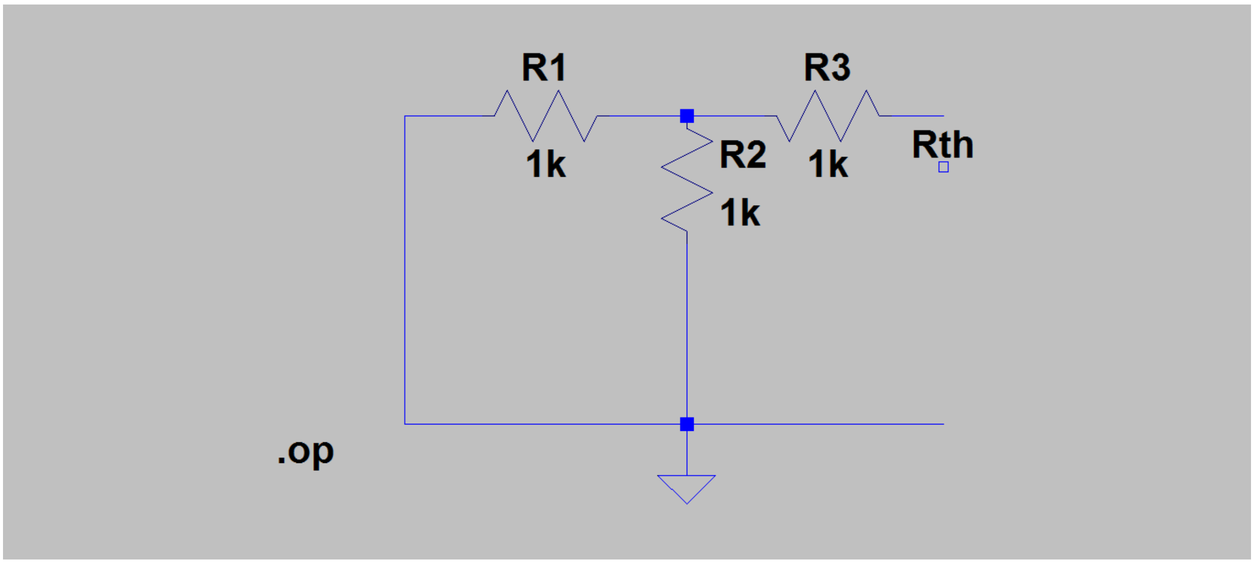
Circuit Diagram 1:



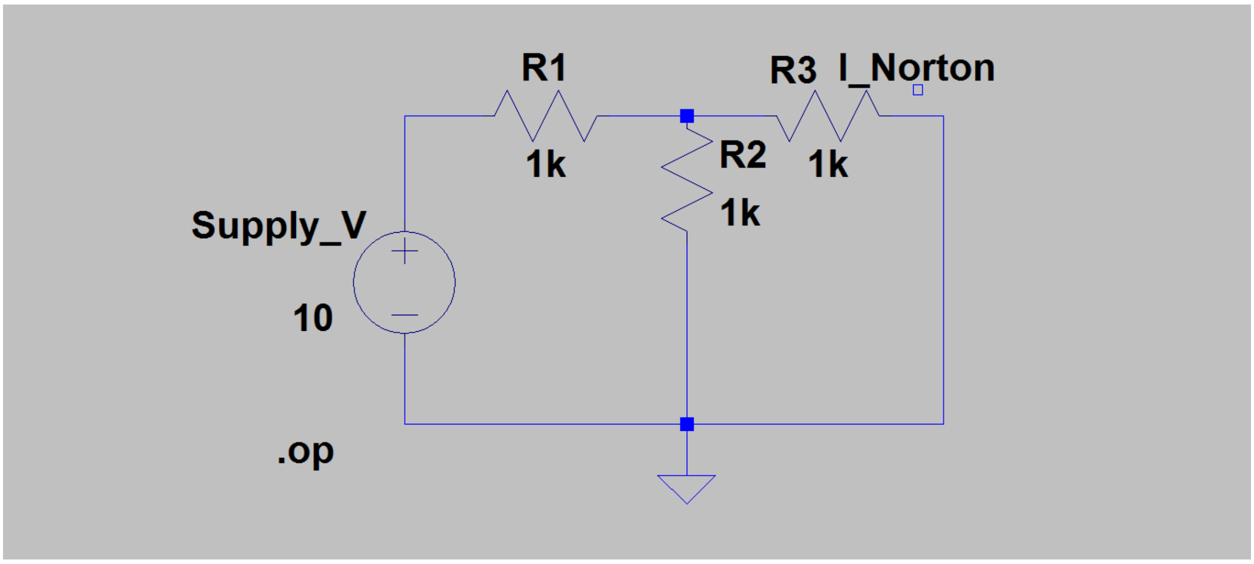
--- Operating Point ---

V(n001):	10	voltage
V(n002):	4	voltage
V(n003):	2	voltage
I(R_load):	0.002	device_current
I(R3):	-0.002	device_current
I(R2):	0.004	device_current
I(R1):	-0.006	device_current
I(Supply_v):	-0.006	device_current

Circuit Diagram 2: Determination of Thevenin's Resistance, Rth



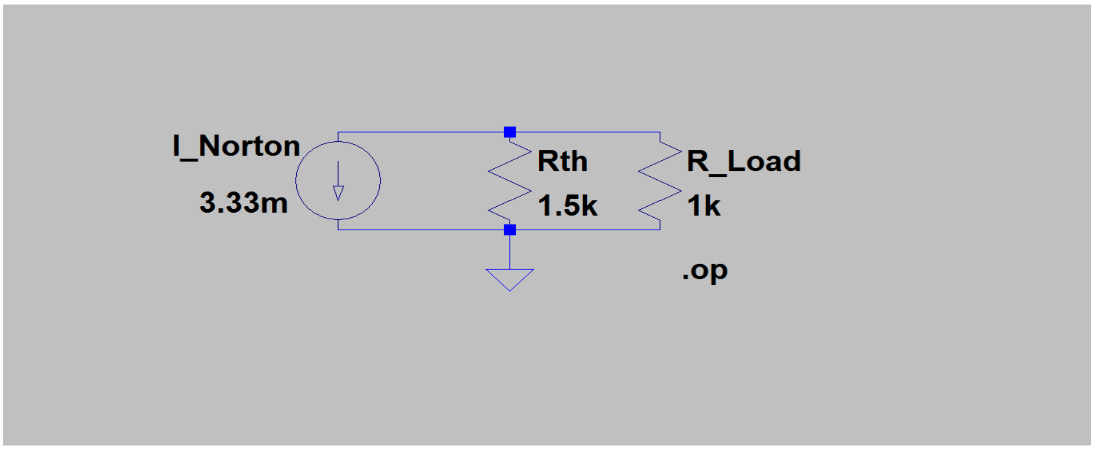
Circuit Diagram 3: Determination of Norton's Current, I_N



--- Operating Point ---

V(n001):	10	voltage
V(n002):	3.33333	voltage
I(R3):	-0.00333333	device_current
I(R2):	0.00333333	device_current
I(R1):	-0.00666667	device_current
I(Supply_v):	-0.00666667	device_current

Norton's equivalent Circuit



--- Operating Point ---

V(n001):	-1.998	voltage
I(I_norton):	0.00333	device_current
I(R_load):	-0.001998	device_current
I(Rth):	-0.001332	device_current

Tabulation:

Supply Voltage (V)	Norton's Current		Thevenin's resistance		Load current	
	I_N (mA) (Theoretical)	I_N (mA) (Simulation)	Rth (Ω) (Theoretical)	Rth (Ω) (Simulation)	I_L(mA) (From Given Circuit)	I_L(mA) (From Norton's Circuit)

Result:

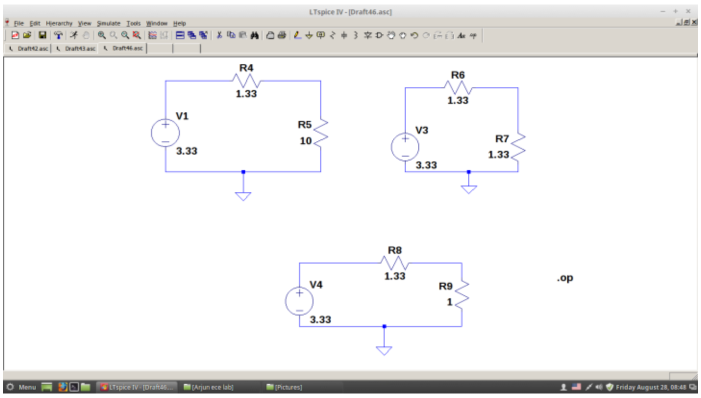
Thus from the load current values, Norton's theorem was verified for the given circuit.

.

(iii) Maximum Power Transfer

AIM: - To veriry Maximum power transfer theorem.

CIRCUIT DIAGRAM : -



PROCEDURE:-

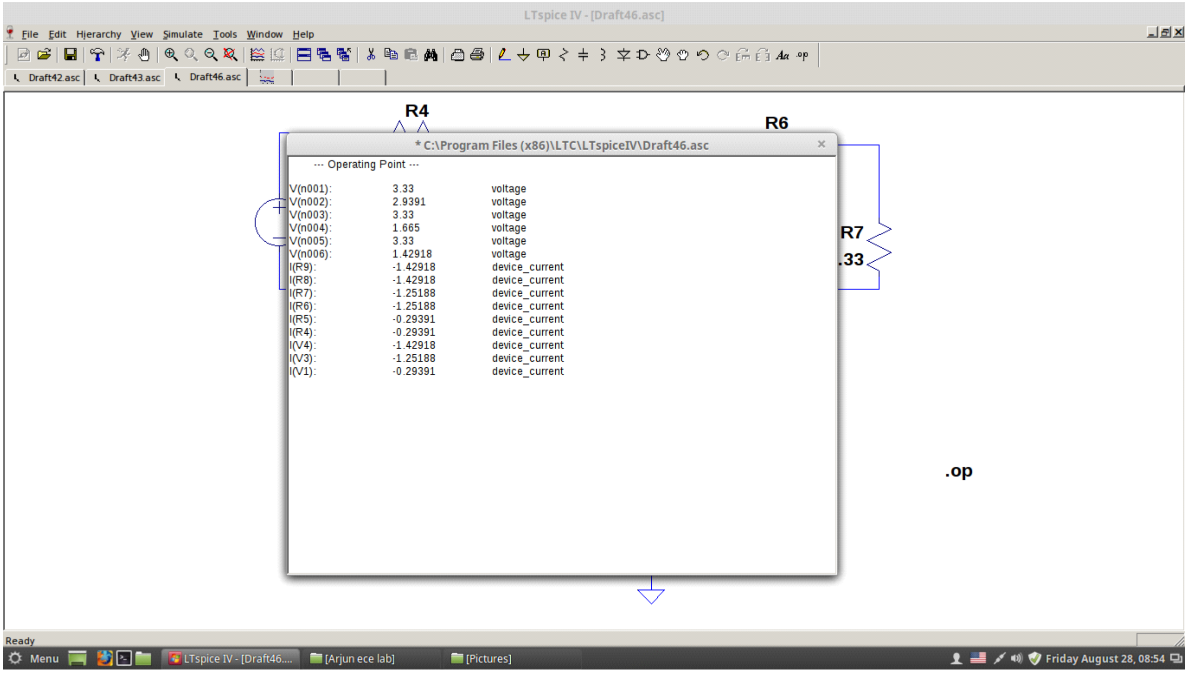
1. Take the Thevenin's Equivalent circuit from the previous experiment.

2. Take three values for the load resistance. One equal to R_{th}, one greater and one smaller.

3. Click on edit simulate command under simulate section then select DC op pnt.

4. Run the simulation by clicking on the run command and obtain the output.

OUTPUT : - After giving the values to the resistors, current source and voltage sources , the following output has been obtained.



The formula used for power calculation = $(V_{th})^2/4R_{th}$

We find that the max. power is transfered when $R_{load}=R_{th}$ .We use the formula $(V_{th})^2/4R_{th}$.

INFERENCE: - From this experiment Maximum power transfer theorem has been verified and it has been noted that the maximum power has been transferred when $R_{load}=R_{th}$.

Experiment No. 4a Date :	VERIFICATION OF SUPERPOSITION THEOREM

Aim:

To verify the superposition theorem for the given circuit.

Apparatus Required:

Sl.No.	Apparatus	Range	Quantity
1	RPS (regulated power supply)	(0-30V)	2
2	Ammeter	(0-10mA)	1
3	Resistors	1k□, 330□, 220□	3
4	Bread Board	--	--
5	Wires	--	Required

Statement:

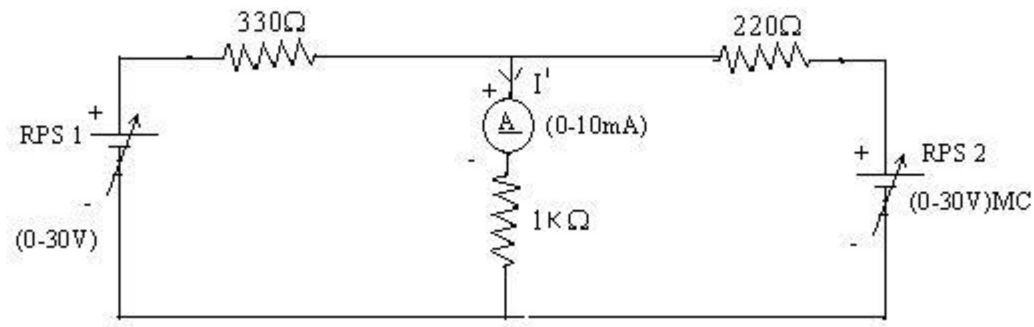
Superposition theorem states that in a linear bilateral network containing more than one source, the current flowing through the branch is the algebraic sum of the current flowing through that branch when sources are considered one at a time and replacing other sources by their respective internal resistances.

Precautions:

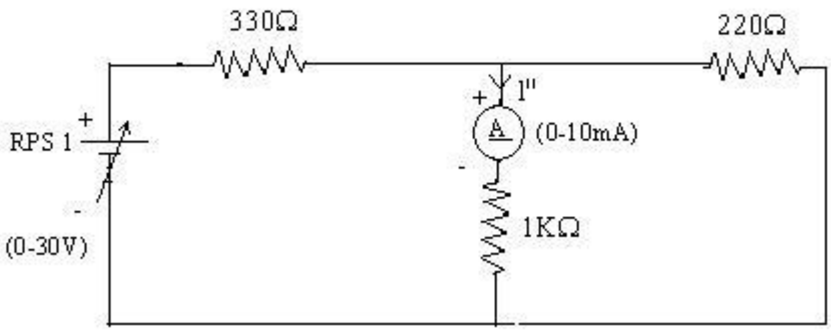
5. Voltage control knob should be kept at manimum position
6. current control knob of RPS should be kept at maximum position

Procedure:

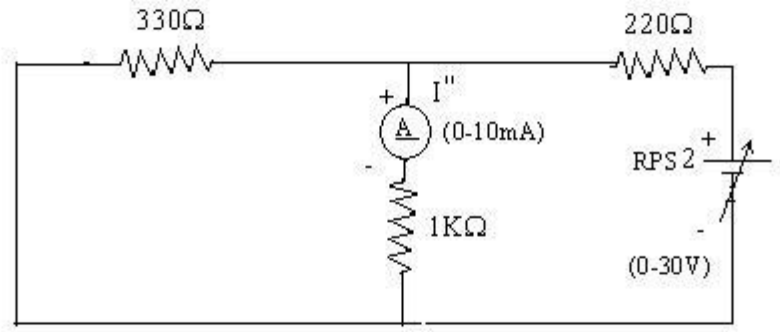
11. Give the connections as per the diagram.
12. Set a particular voltage value using RPS_1 and RPS_2 & note down the ammeter reading
13. Set the same voltage in circuit I using RPS_1 alone and short circuit the terminals and note the ammeter reading.
14. Set the same voltage in RPS_2 alone as in circuit I and note down the ammeter reading.
15. Verify superposition theorem.



CIRCUIT – 1



CIRCUIT - 2



CIRCUIT - 3

TABULAR COLUMN

Theoretical Values

	RPS		Ammeter Reading (I) mA
	1	2	
Circuit – 1	10 V	10 V	I = 8.83
Circuit – 2	10 V	0 V	I’= 3.5
Circuit – 3	0 V	10 V	I”= 5.3

I = I’ □□I” = 8.83

Practical Values

	RPS		Ammeter Reading (I) mA
	1	2	
Circuit – 1	10 V	10 V	
Circuit – 2	10 V	0 V	
Circuit – 3	0 V	10 V	

Model Calculations:

Result:

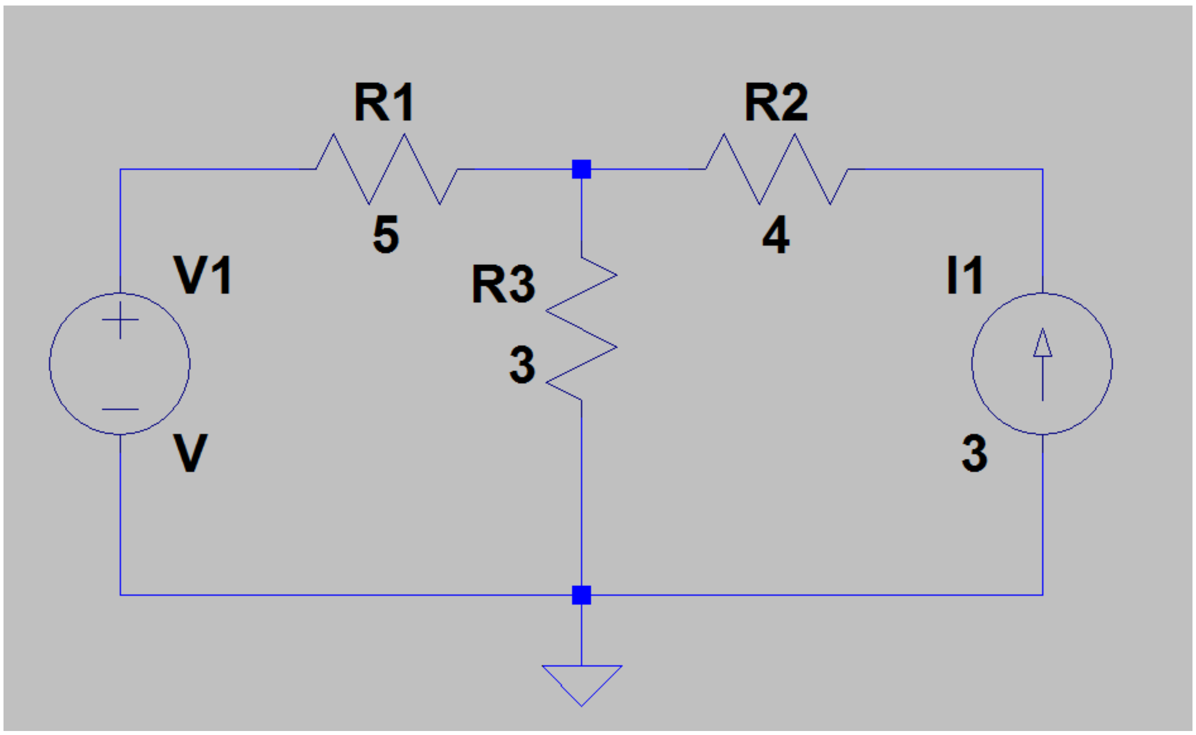
Superposition theorem has been verified theoretically and practically.

Experiment No. 4b Date :	VERIFICATION OF SUPERPOSITION AND RECIPROCITY THEOREM USING LTSPICE

(i) Superposition Theorem

Aim: To verify superposition theorem

Circuit Diagram:

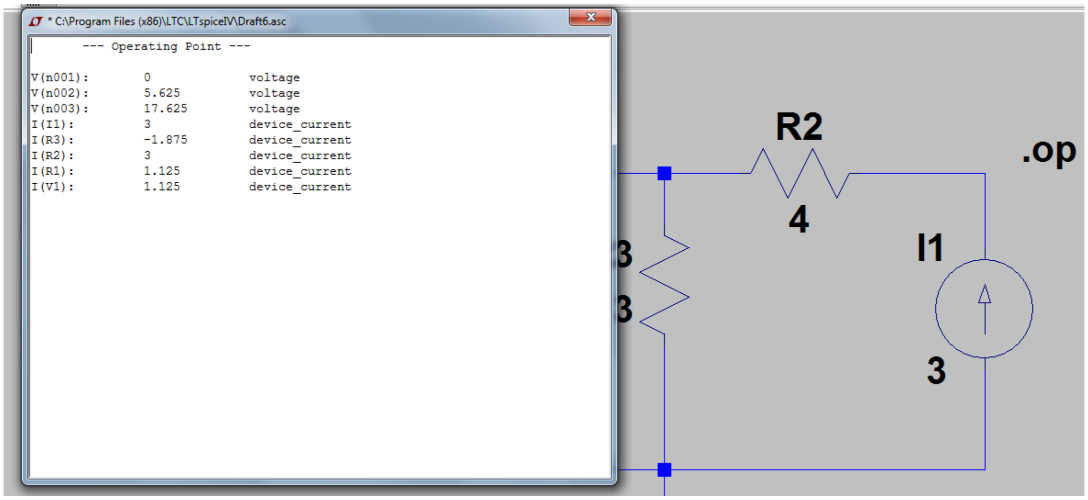


PROCEDURE : - Using LT Spice simulation tool, create the above circuit using a resistor, a voltage source, current source and a zero potential and give the corresponding values to the resistor and the source voltage by right clicking the components.

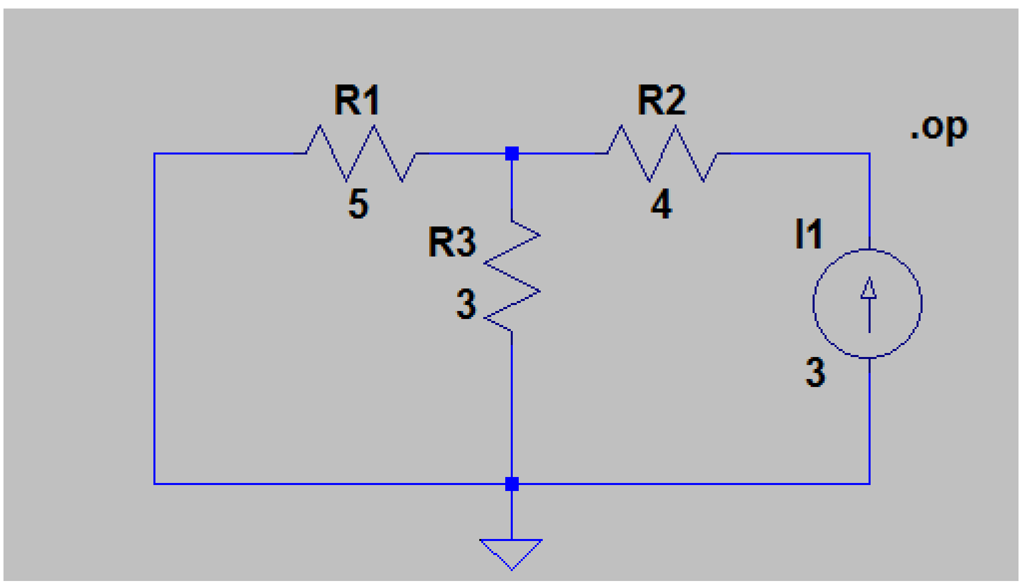
• Click on edit simulate command under simulate section then select DC op point and then run the circuit to check the operating voltage and current at each component. Current across R2 is 3A.

- Repeat the same measurement by considering only one independent source at a time.

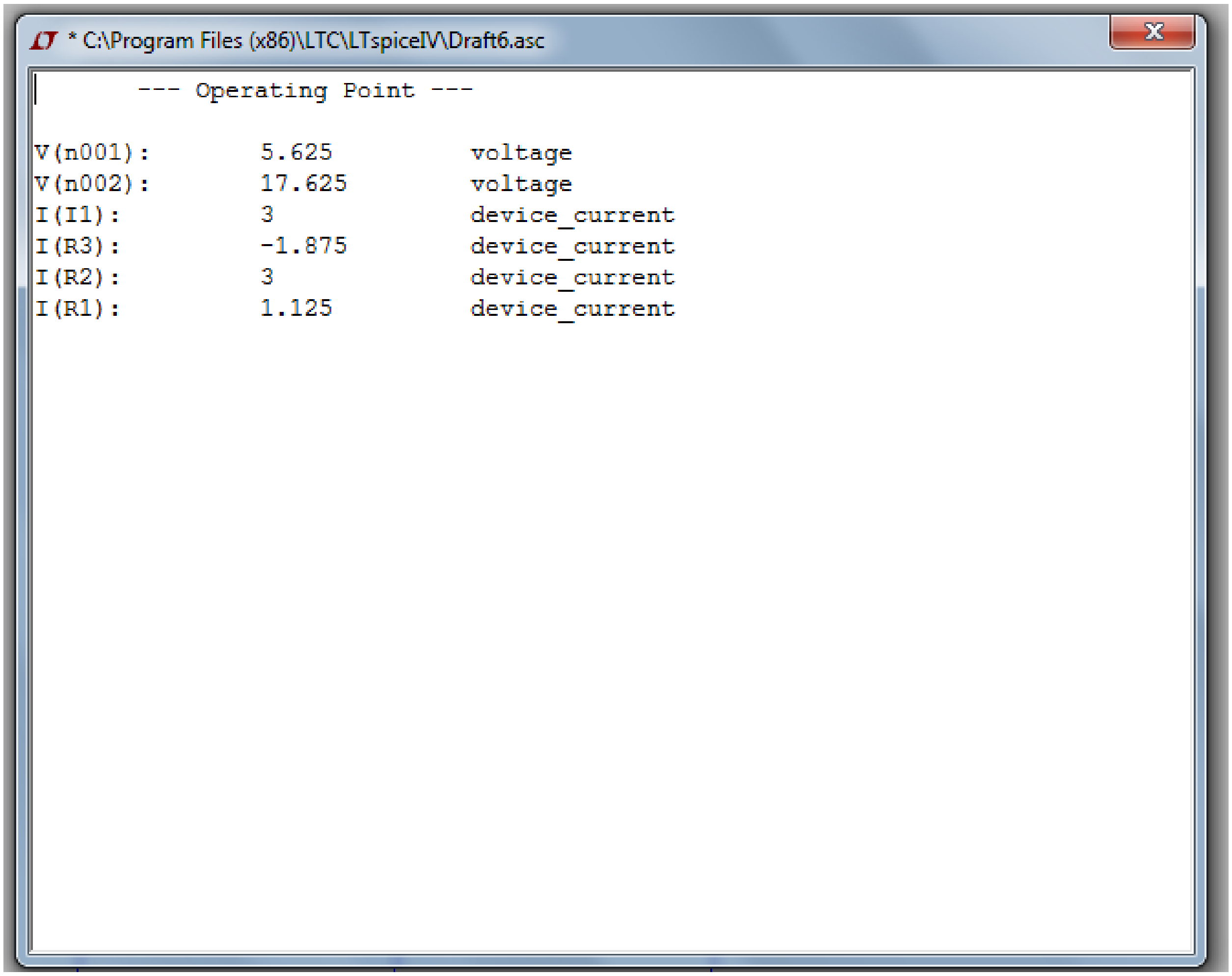
Theorem Proof: Current at R2 when both the independent sources considered = Summation of current at R2 when the independent sources considered separately [Voltage source is short-circuited and current source open circuited].



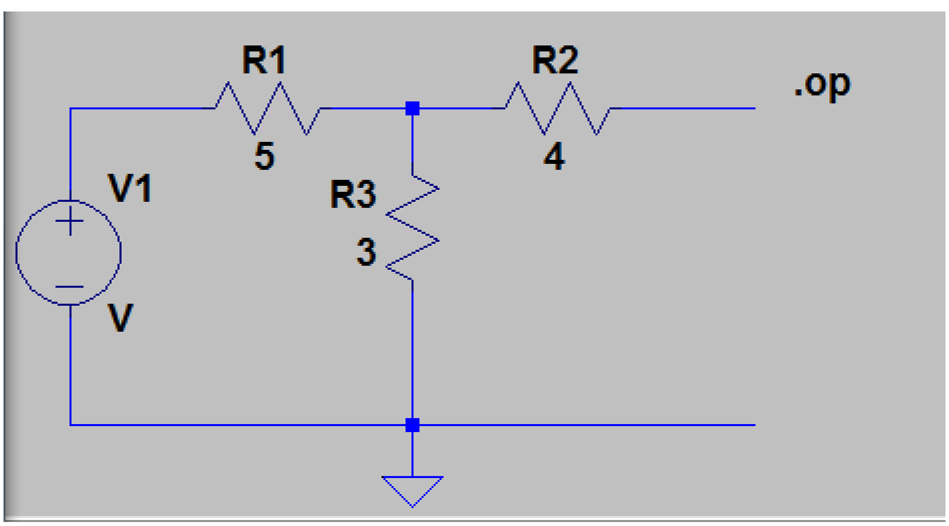
Consider only current source



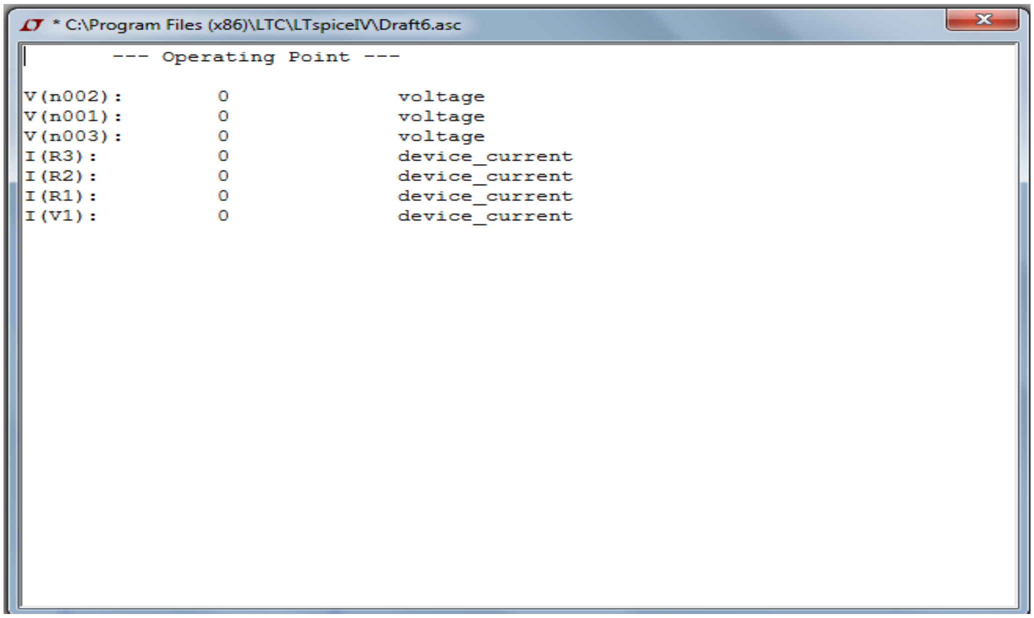
Measure current across R2.



Consider only voltage source



Measure current across R2.



```
* C:\Program Files (x86)\LTC\LTspiceIV\Draft6.asc
       --- Operating Point ---

V(n002):        0              voltage
V(n001):        0              voltage
V(n003):        0              voltage
I(R3):          0              device_current
I(R2):          0              device_current
I(R1):          0              device_current
I(V1):          0              device_current
```

INFERENCE: It has been observed that current through R2 due to each source independently equals current through R2 when both are active.

(ii) VERIFICATION OF RECIPROCITY THEOREM

Aim:

To verify the reciprocity theorem for the given electrical network using LTSPICE simulation.

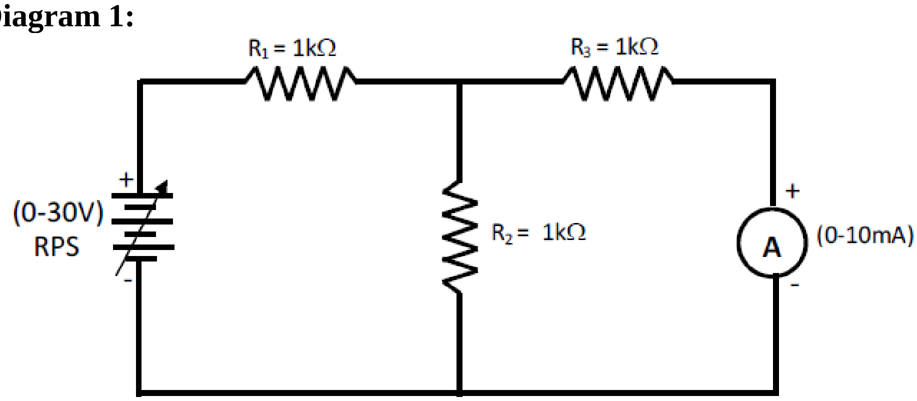
Apparatus Required: **LTSPICE software**

Theory:

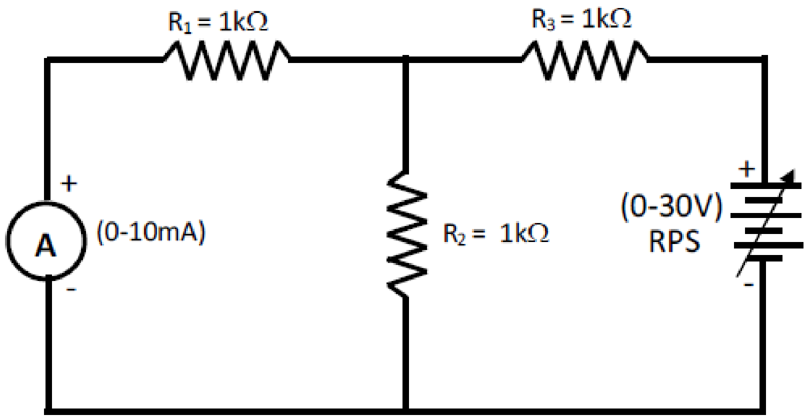
Reciprocity Theorem:

In any linear bilateral network the ratio of voltage to current response, is constant even when the position of the input and output are interchanged.

Circuit Diagram 1:



Circuit Diagram 2:

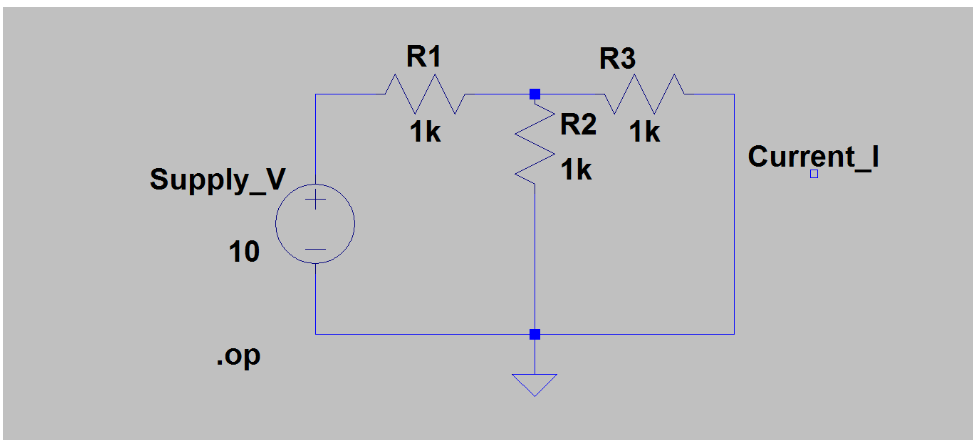


Procedure:

1. Connections are made as per the circuit diagram.
2. Note down the ammeter reading and find the ratio of output current and input voltage.
3. Interchange the position of ammeter and voltage source.
4. Note down the ammeter reading and find the ratio.
5. Compare this value with the value obtained in step – 2.

Simulation Steps:

Circuit Diagram 1:

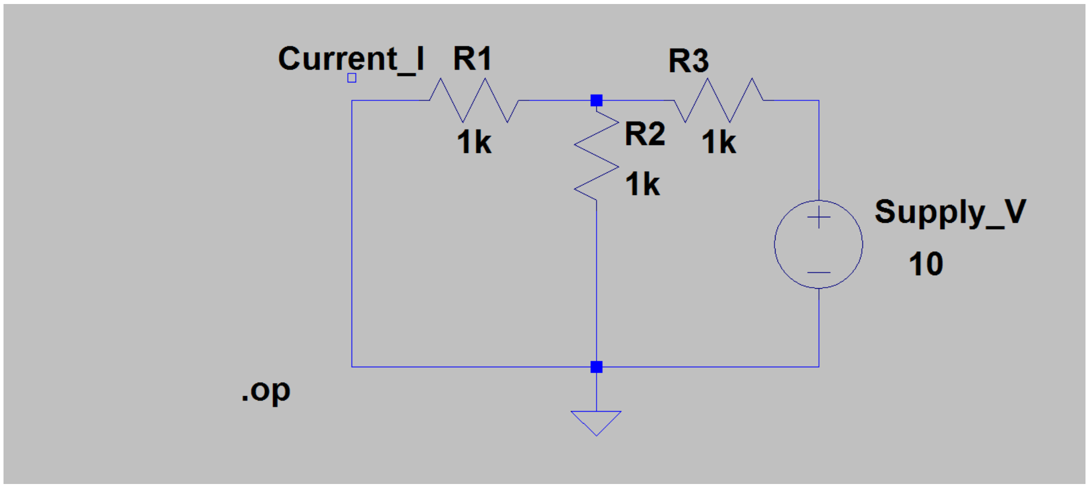


--- Operating Point ---

V(n001):	10	voltage
V(n002):	3.33333	voltage
I(R3):	-0.00333333	device_current
I(R2):	0.00333333	device_current
I(R1):	-0.00666667	device_current
I(Supply_v):	-0.00666667	device_current

Ratio : Supply_V / Current_I = 3000.003

Circuit Diagram 2:



--- Operating Point ---

V(n002):	10	voltage
V(n001):	3.33333	voltage
I(R3):	0.00666667	device_current
I(R2):	0.00333333	device_current
I(R1):	0.00333333	device_current
I(Supply_v):	-0.00666667	device_current

Ratio : Supply_V / Current_I = 3000.003

Tabulation:

S.No	Voltage (V)	Case A		Case B	
		Voltage V_1 (V)	Current I_2 (mA)	Voltage V_2 (V)	Current I_1 (mA)

Result: The reciprocity theorem was verified for given network.

Experiment No. 5a Date :	FIRST ORDER RL AND RC CIRCUITS

Aim: To observe the response of the first order R-C and R-L circuits.

Apparatus Required:

Sl.No.	Apparatus	Range	Quantity
1	Function Generator	-	1
2	CRO	-	1
3	Decade Resistance Box	-	1
4	Capacitor	0.1uF	1
5	Inductor	100mH	1
6	Bread Board	--	--
7	Wires	--	Required

Theory

Natural Response of the First Order Circuits.

The natural response of an RC circuit, is given by (1).

$$v(t) = v_0 \,.\, e^{-t/r}$$

(1)

Where, v_o is the value of the capacitor voltage v at t=0s, τ=RC is the time constant of the circuit. The time constant gives the rate at which the voltage decays to zero. In circuits, this decay response is due to ohmic losses.

The time constant can be measured by one of the following graphical methods.

(a) The Tangential Line. A line tangential at a certain point of the response curve is drawn. The line intersects the time axis exactly in one time constant from the point of tangent. Figure A shows the graph of the response from (1). Note that the response begins at t= 0.05 s. With reference to Figure A, a line is drawn tangential to the curve at t= 0.05 s (that is at the onset of the decay) . The line is extended until it intersects the time axis. This occurs at t= 0.25 s. Therefore, the time constant of the response is

$$\tau = 0.25 - 0.05 = 0.2 \text{ s} \qquad (2)$$

This method is suitable if a hard copy of the response graph is available.

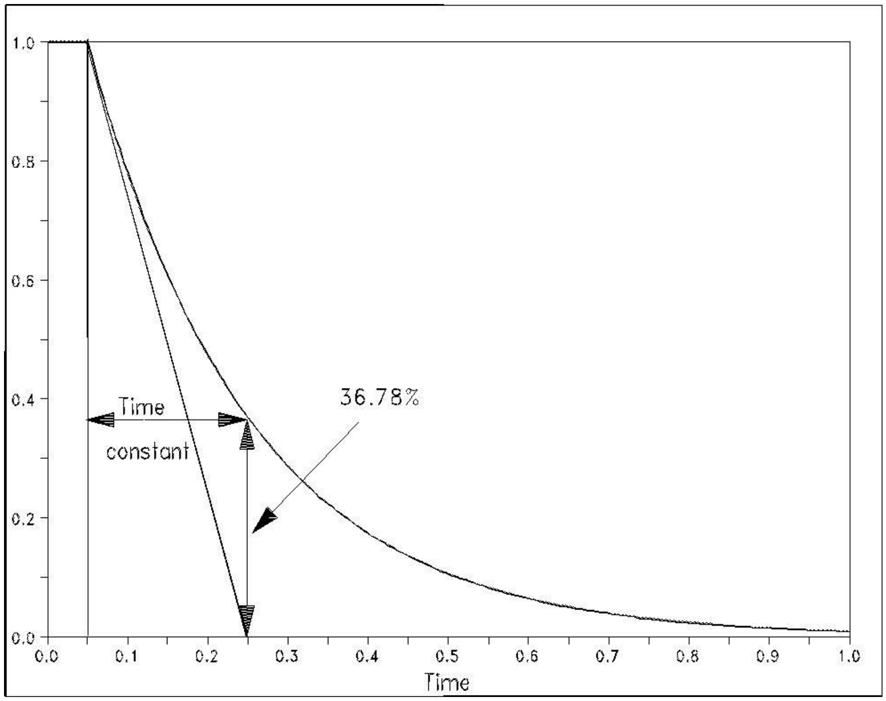


Fig. A. Measurement of the time constant of an exponentially decaying response.

(b) The 63.22% Decay. The second method is suitable for measurements on the oscilloscope. From (1), every time interval equal to one time constant the response decays by 63.22%. Equivalently, at the end of every interval equal to one time constant, the response is at the 100-63.22= 36.78 % of its value at the beginning of the interval. This method of measurement is demonstrated in Figure B. The response begins its decay at t=0.05 s, at that point the value of the curve is 1. The response reaches 36.78 % of its initial value at t= 0.25 s. Thus the time constant is

$$t = 0.25 - 0.05 = 0.2 \text{ s} \tag{3}$$

The results of (3) and (2) are in agreement.

The Step Response of First Order Circuits.

The step response of the first order circuit RC circuit is given by (4).

$$v(t) = v_f\left(1 - e^{-\frac{t}{\tau}}\right) \tag{4}$$

Where, v_f is the final value of the response (also, called the constant steady state of the response). The step response of the circuit is characterized by the same time constant as the natural response. The time constant affects the step response in a similar manner as the natural response. Therefore, the methods discussed previously apply in this case, as well.

On the oscilloscope the time constant of the step response can be measured by measuring the time the output requires to reach 63.22% of its final value.

The natural and step responses of a first order R-L circuit are the same as in (1) and (4) given for the circuit current i. The circuit time constant is τ=L/R.

Observations.

Figure B shows the response of a first order circuit to a periodic square pulse. The first part of the circuit response is the step response described by (4). The second part of the circuit response, when the excitation becomes zero, is the natural response described by (1). If the response is allowed to decay sufficiently, the following period will begin with another step response followed by the natural response. Thus, the circuit response will alternate between step and natural responses. To achieve good results, therefore, the period of the excitation must be adjusted to 7 to 10 times the time constant of the circuit.

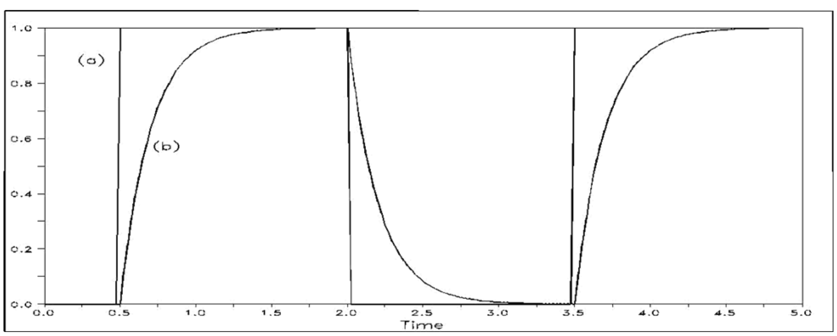


Fig. B. Circuit response to a square pulse excitation with large period. (a) The excitation. (b) The circuit response.

Procedure:

A. The RC Circuit.

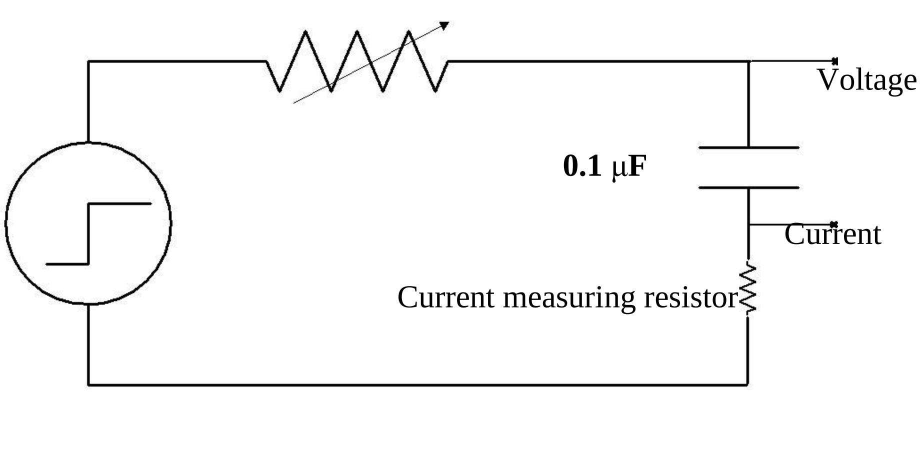


Fig. 1. RC circuit.

1. (a) Construct the R-C circuit of Figure 1. Use a 510 Ω.
   (b) Apply the square pulse generator to produce a periodic response of the circuit that appears clearly on the oscilloscope. The period of the source is adjusted to be at least 20 times higher than the calculated time constant of the circuit. This is necessary, so that the capacitor reaches steady state, in both charging and discharging intervals.
   (c) Observe on the oscilloscope the quantities shown in the figure. Measure the circuit time constant by the 63% decay method described above.
   (d) Plot on a millimeter paper one period of the voltage and current response. Use appropriate scale. Label the axes. Transfer measurements from the oscilloscope to the paper by inspection.
2. Change the resistance to 250 Ω. Repeat Procedure 1. Use the same millimeter paper and the same scale. Align the graphs. Do not change the setting of the source.

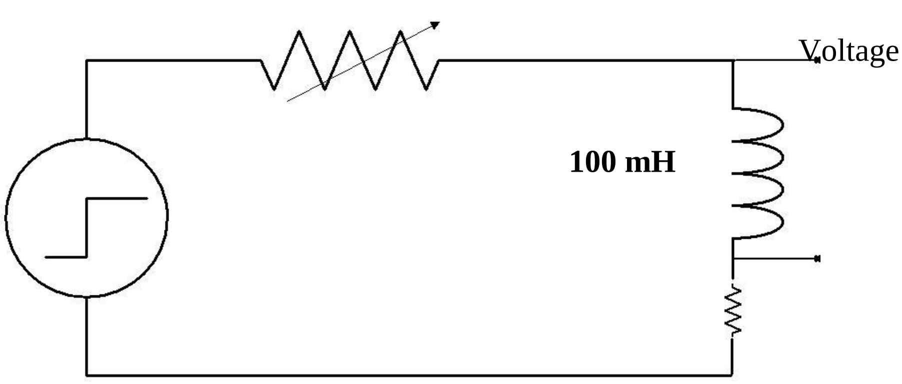
Tabulation

S.No	Resistance (Ω)	Voltage across the capacitor (V)

B. The R-L Circuit.

1. (a) Construct the circuit of Figure 2. Set the resistance box to 500 Ω.
   a. Apply a square pulse voltage. Set the frequency of the source to correspond to 10 times the time constant of the circuit.
   b. Observe on the oscilloscope the quantities shown in the figure. Measure the circuit time constant by the 63% decay method described above.
2. Change the resistance to 250 Ω. Perform again step 1b.

Fig. 2. RL circuit



Tabulation

S.No	Resistance (Ω)	Voltage across the inductor (V)

.

Result: Thus the time constant for the given RC and RL circuit is verified both theoretically and practically.

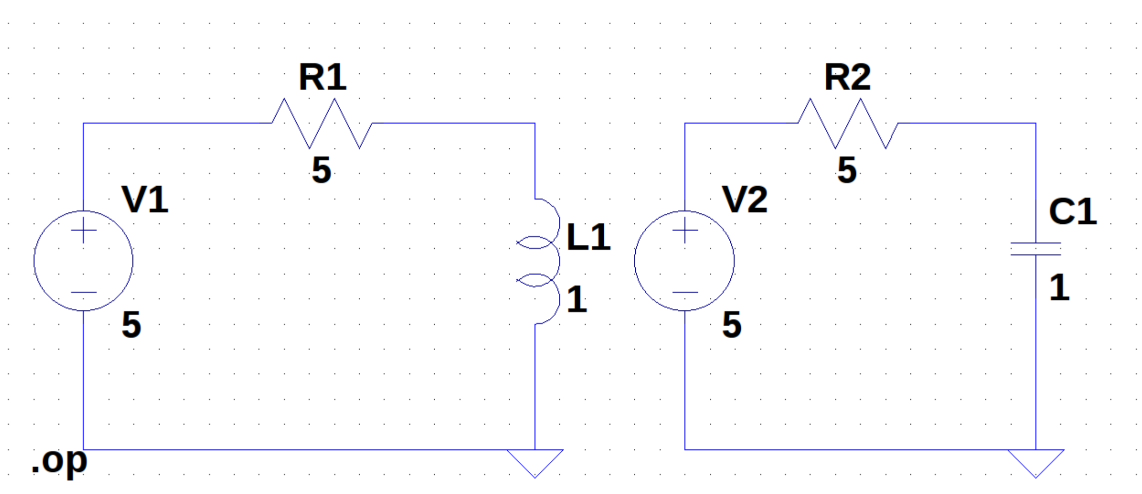
Experiment No. 5b Date :	FIRST ORDER RL & RC TRANSIENT USING LTSPICE

Procedure:

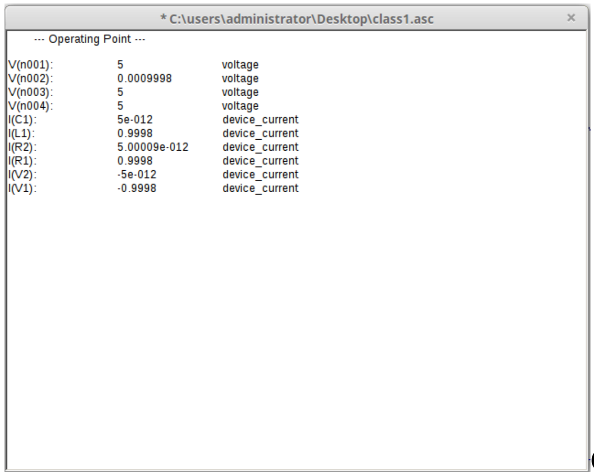
a. Draw the given circuit in the LTSPICE with new schematic.
b. Change the value of source, resistors nad inductors or capacitors.
c. Place the Ground.
d. Set the simulation profile for Operating point analysis.
e. Run the simulation and note the results.
f. Change the circuit accordingly as the switch closes.
g. Set the simulation profile for Transient circuit analysis and set Initial Condition as IC.
h. Run the simulation and plot the graph.

<u>**Operating Point analysis of RL and RC Circuits with Source**</u>

Circuit Diagram 1:

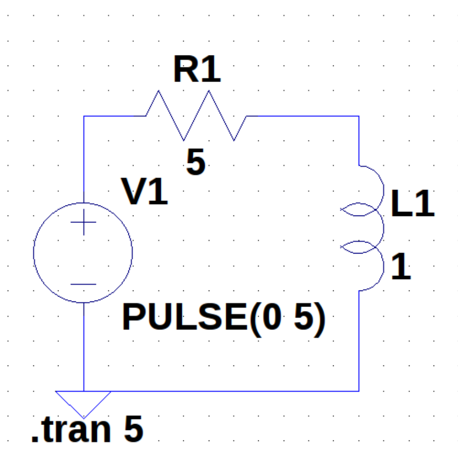


Model Calculations:

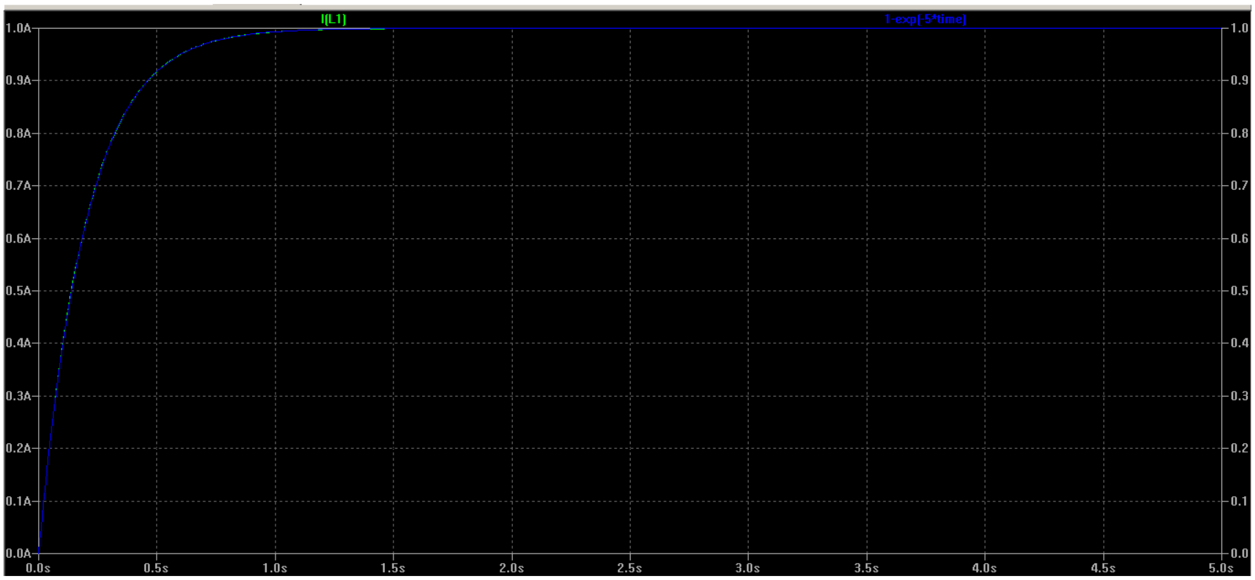


<u>Transient analysis of RL circuit with Source:</u>

Circuit Diagram 2:



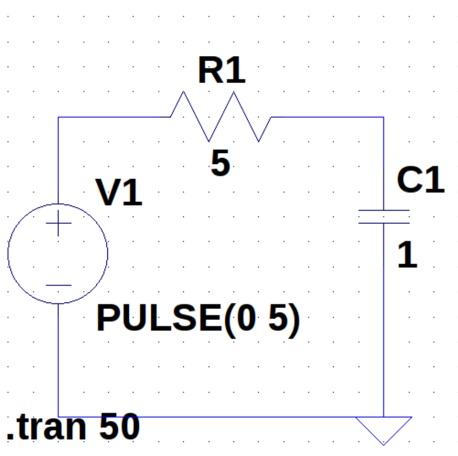
Graph 2:



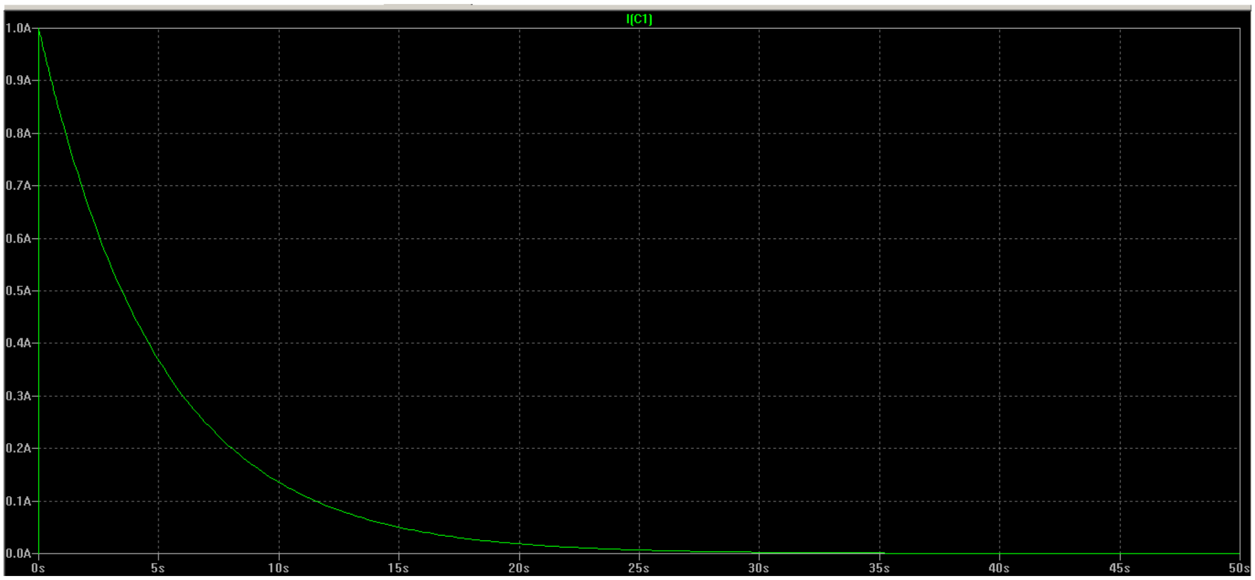
The theoretical calculations match with the LTSpice calculations as the graph of them overlaps.

<u>Transient analysis of RC circuit with Source:</u>

Circuit Diagram 3:



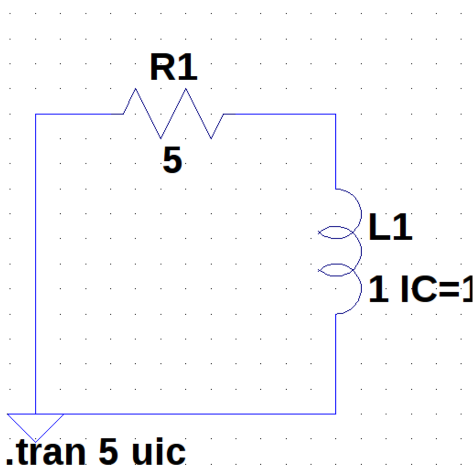
Graph 3:



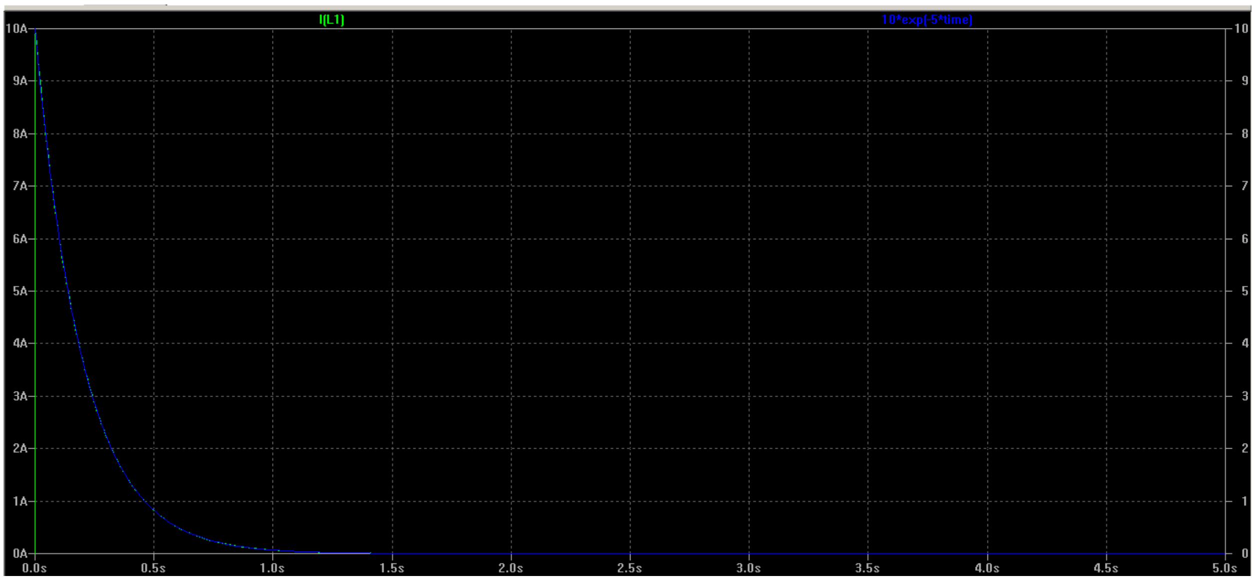
The theoretical calculations match with the LTSpice calculations as the graph of them overlaps.

Source free RL Circuit:

Circuit Diagram 4:



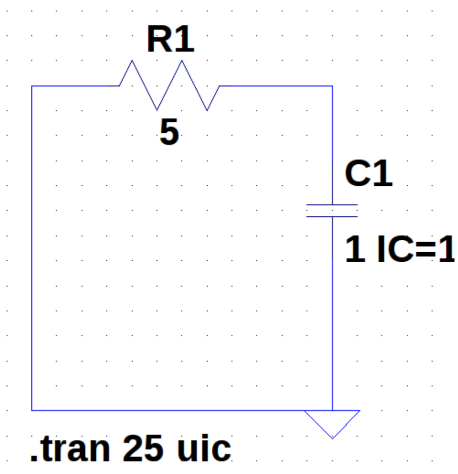
Graph 4:



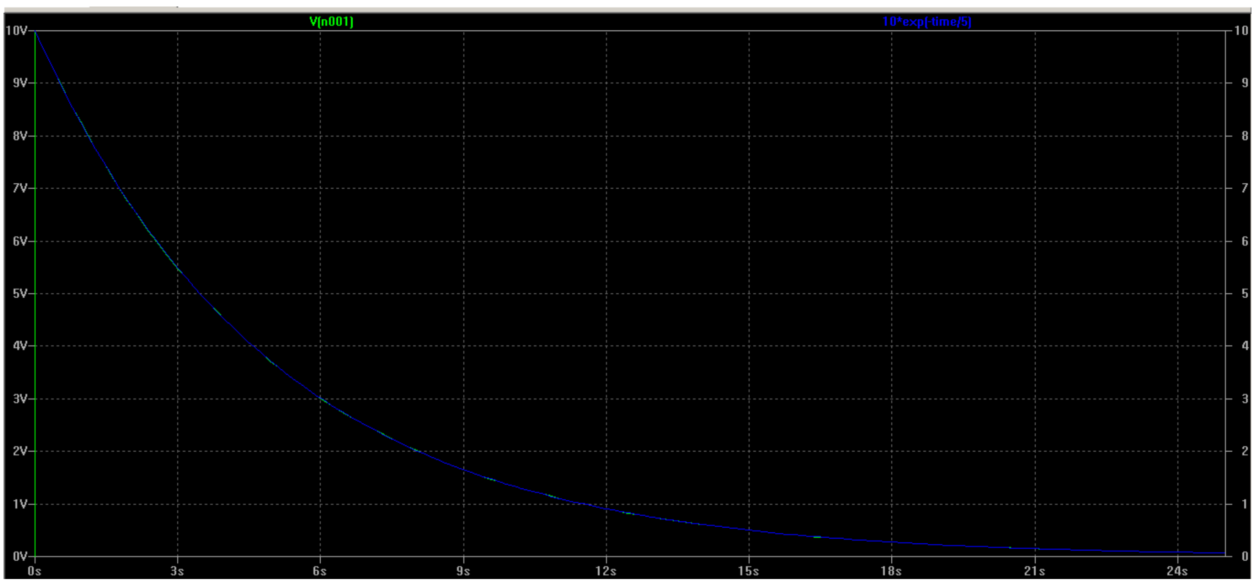
The theoretical calculations match with the LTSpice calculations as the graph of them overlaps.

<u>**Source free RC circuit:**</u>

Circuit Diagram 5:



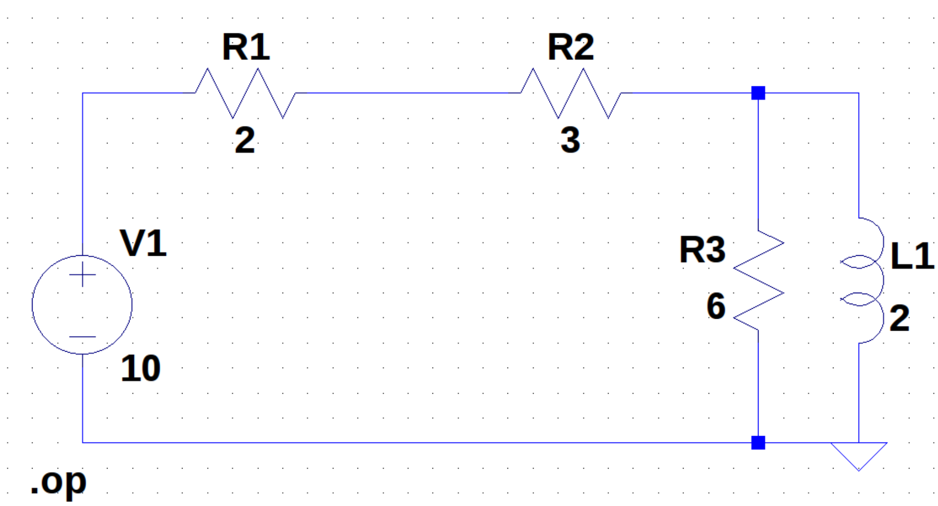
Graph 5:



The theoretical calculations match with the LTSpice calculations as the graph of them overlaps.

<u>Analysis of RL circuit with Source:</u>

Circuit Diagram 6(1):

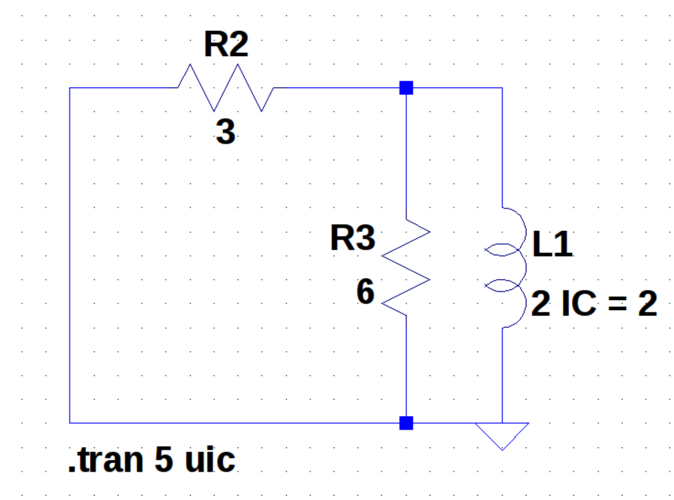


Calculations:

--- Operating Point ---

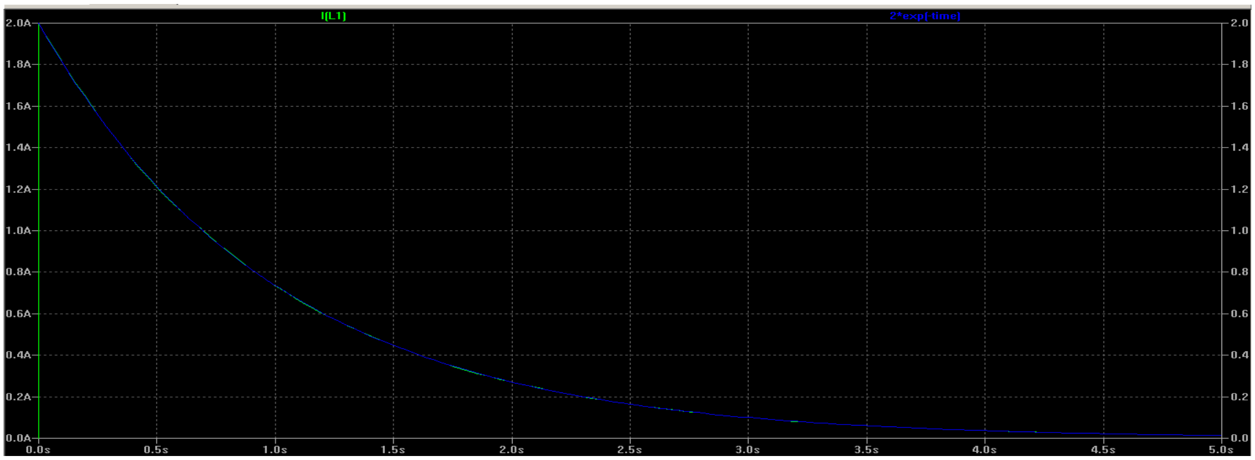
V(n001):	10	voltage
V(n002):	6.0008	voltage
V(n003):	0.00199927	voltage
I(L1):	1.99927	device_current
I(R3):	-0.000333211	device_current
I(R2):	1.9996	device_current
I(R1):	1.9996	device_current
I(V1):	-1.9996	device_current

Circuit Diagram 6(2):



IC is the initial condition which is the initial current through the inductor which is 2A as seen from the calculations above.

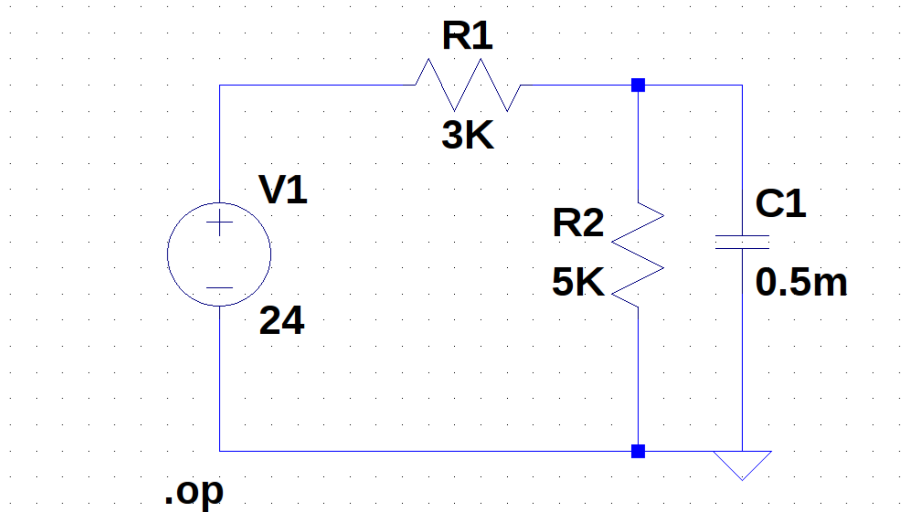
Graph 6:



The theoretical calculations match with the LTSpice calculations as the graph of them overlaps.

<u>Analysis of RC circuit with Source:</u>

Circuit Diagram 7(1):

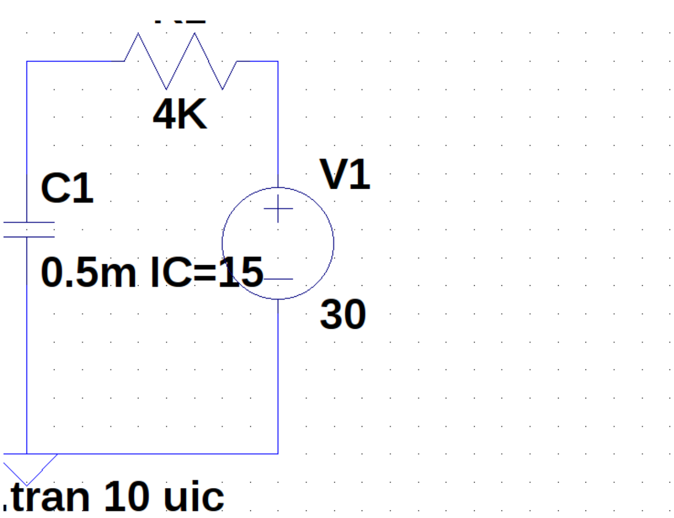


Calculations:

--- Operating Point ---

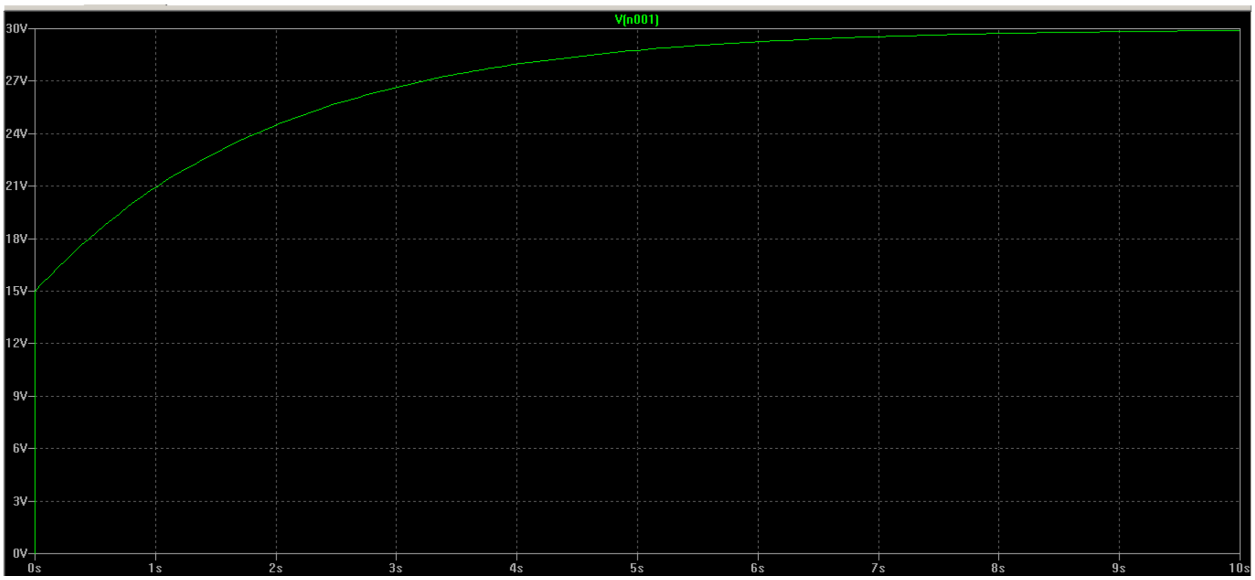
V(n001):	24	voltage
V(n002):	15	voltage
I(C1):	7.5e-015	device_current
I(R2):	-0.003	device_current
I(R1):	0.003	device_current
I(V1):	-0.003	device_current

Circuit Diagram 7(2):



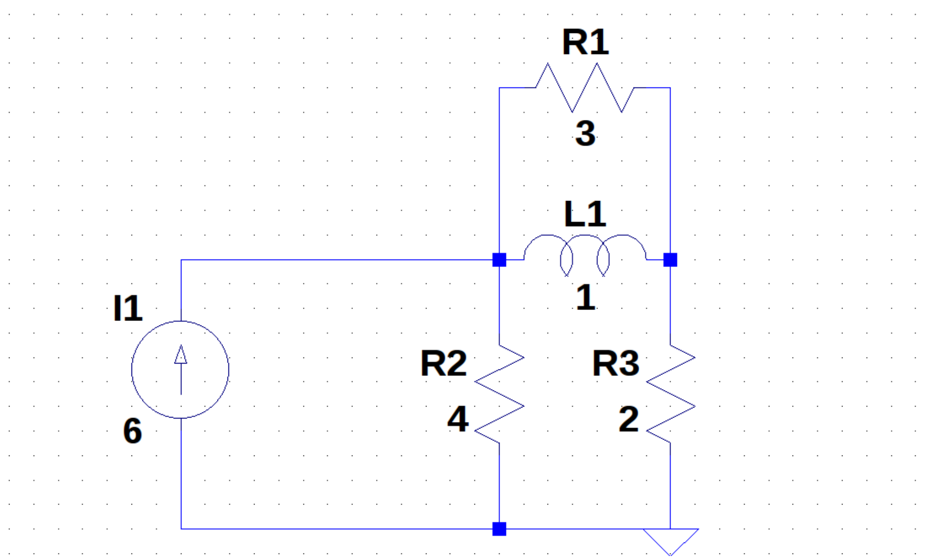
IC is the initial condition which is the initial voltage through the capacitor which is 15V as seen from the calculations above.

Graph 7:

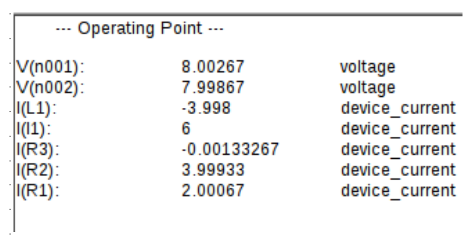


Analysis of RL circuit with Current Source:

Circuit Diagram 8(1):



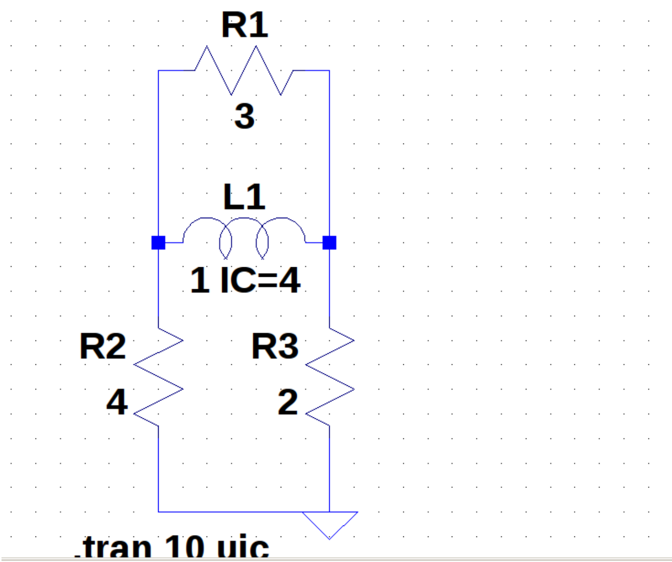
Calculations:



--- Operating Point ---

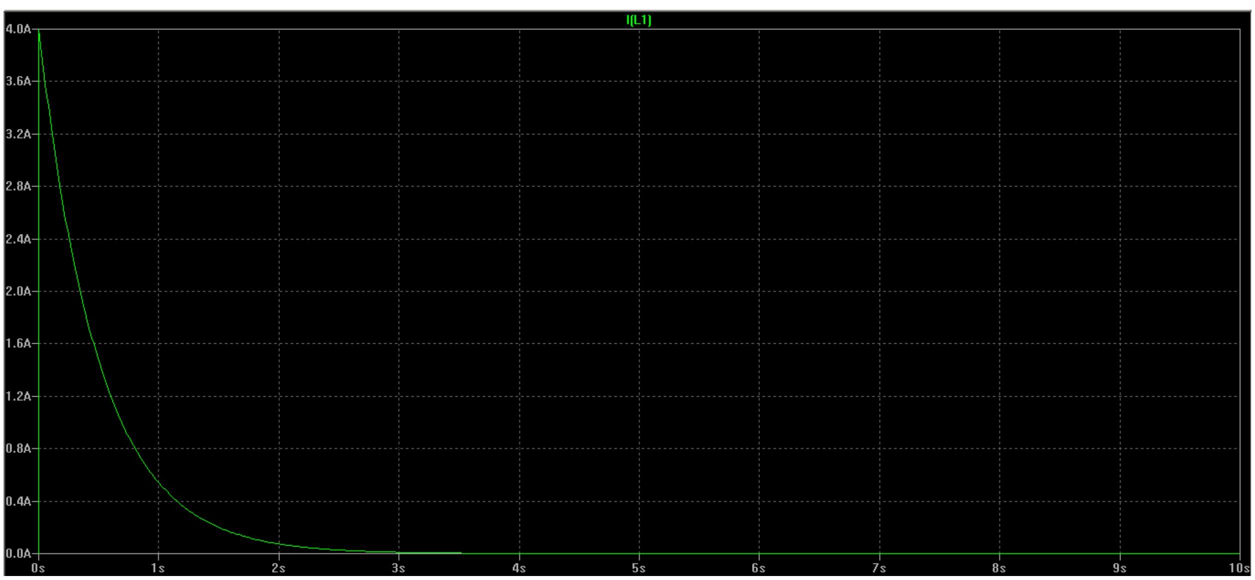
V(n001):	8.00267	voltage
V(n002):	7.99867	voltage
I(L1):	-3.998	device_current
I(I1):	6	device_current
I(R3):	-0.00133267	device_current
I(R2):	3.99933	device_current
I(R1):	2.00067	device_current

Circuit Diagram 8(2):



IC is the initial condition which is the initial current through the inductor which is 4A as seen from the calculations above.

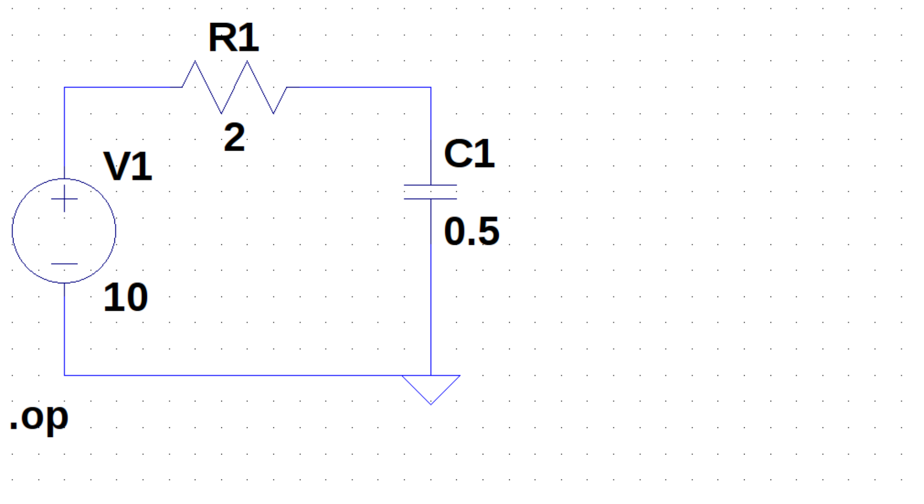
Graph 8:



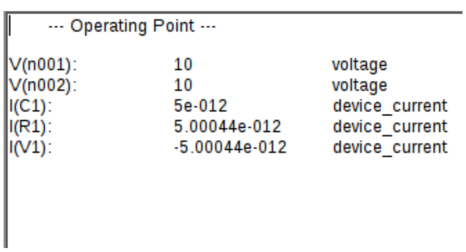
Finally, we get the above graph.

<u>**Analysis of RC circuit with two sources:**</u>

Circuit Diagram 9(1):

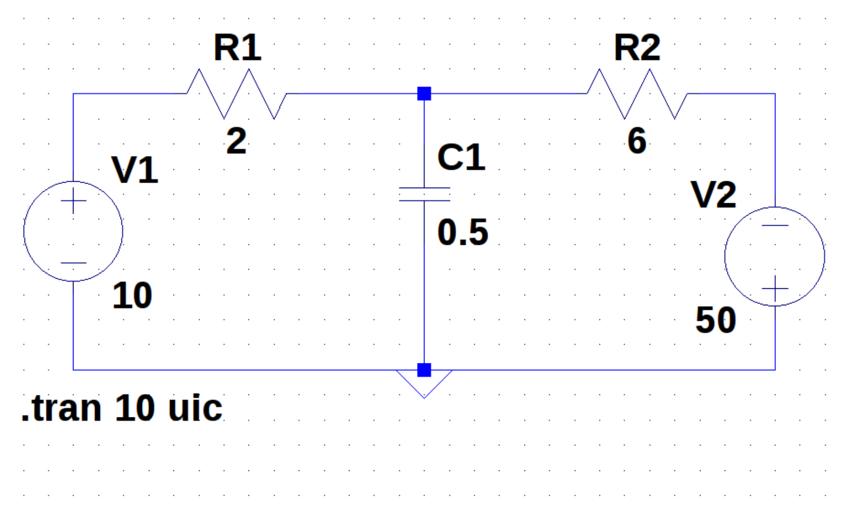


Calculations:


--- Operating Point ---

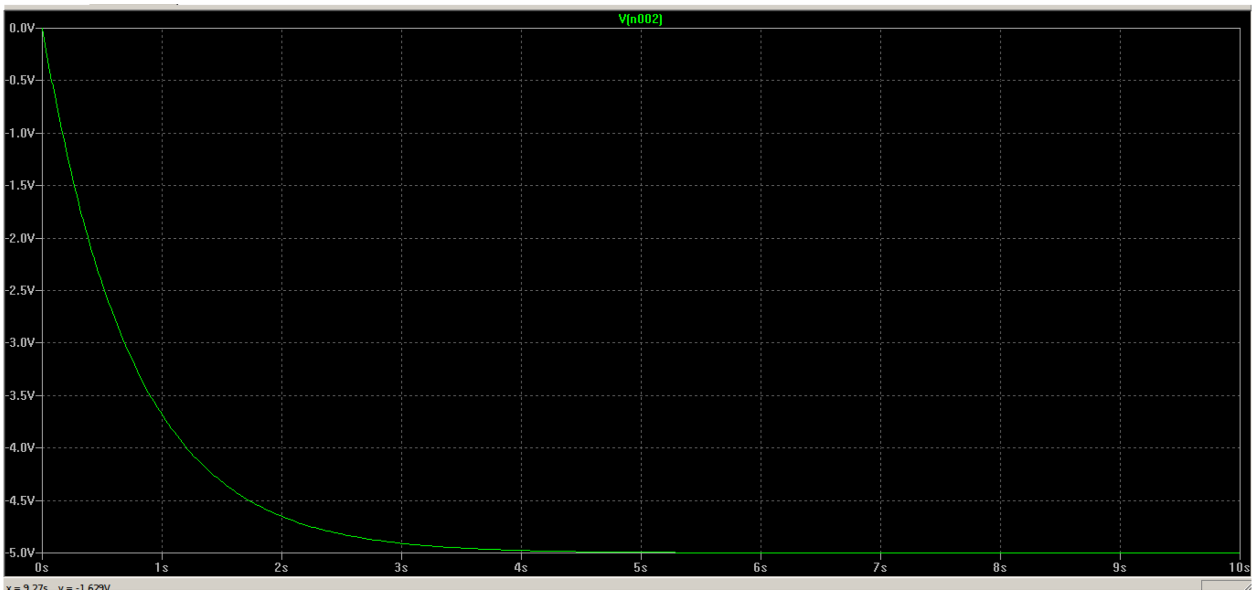
V(n001):	10	voltage
V(n002):	10	voltage
I(C1):	5e-012	device_current
I(R1):	5.00044e-012	device_current
I(V1):	-5.00044e-012	device_current

Circuit Diagram 9(2):



IC is the initial condition which is the initial voltage through the capacitor which is 0.5V as seen from the calculations above.

Graph 9:



The above graph shows the decay of voltage in the capacitor

Experiment No. 6a **Date :**	**AC CIRCUIT ANALYSIS**

Aim:

To familiarize the students with AC circuit analysis and measurements.

Apparatus Required:

Sl.No.	Apparatus	Range	Quantity
1	Function Generator	--	1
2	Multimeter	--	1
3	Resistors	100□	3
4	Resistors	220□	1
5	Resistors	470□	1
6	Inductor	100mH	1
7	Capacitor	0.22uF	1
4	Bread Board	--	--
5	Wires	--	Required

Theory:

While analyzing an electric circuit having reactive components such as inductors and capacitors, the voltages, currents and branch impedance must be treated as complex

Quantities. These quantities may be expressed in rectangular form as a real part and imaginary part, or in polar form as a magnitude and a phase angle. The voltage magnitude is the quantity, which would be measured with ordinary AC voltmeter. Phase relationships may be measured with specialized equipment or may be calculated from

Measurements of DC resistance and of impedance, the impedance being the ratio of the

applied AC voltage to the resulting AC current.

Procedure:

Series AC Circuit:

1. Connect the circuit shown in Figure 1
2. Fix the frequency of the function generator to 500 Hz and its output voltage to 5 V (rms)
3. Measure the AC voltage across each of the elements
4. Increase the frequency by 100 Hz until 1000 Hz is reached and readjusts the output voltage to maintain 5 V and repeat step 3.
5. Tabulate the results in Table 1.

Parallel RLC Circuit:

1. Measure the DC resistance for each resistor.
2. Measure the DC resistance of the inductor.
3. Connect the circuit as shown in Figure 2.
4. Fix the frequency to 500 Hz, and its output voltage magnitude to 5 V (rms).
5. Measure the magnitude of the AC voltage across R_S, R_1, R_2, and R_3 and tabulate the values in Table 2.

CIRCUIT DIAGRAM

Figure 1 Series AC circuit.

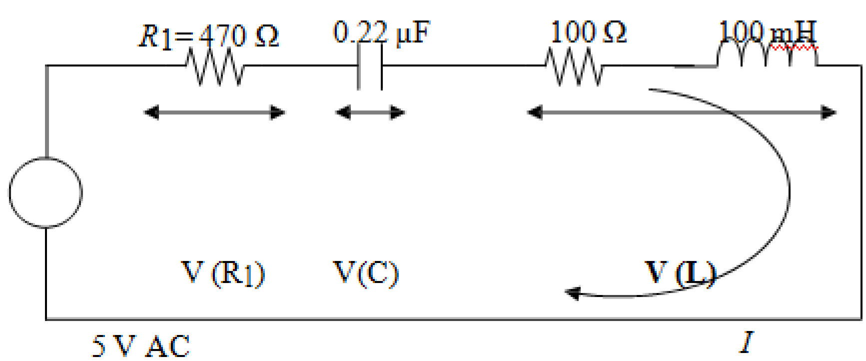


Table 1

Frequency, Hz	V (C)	V (L)	V(R_1)	I
500				
600				
700				
800				
900				
1000				

Figure 2 Parallel AC circuit.

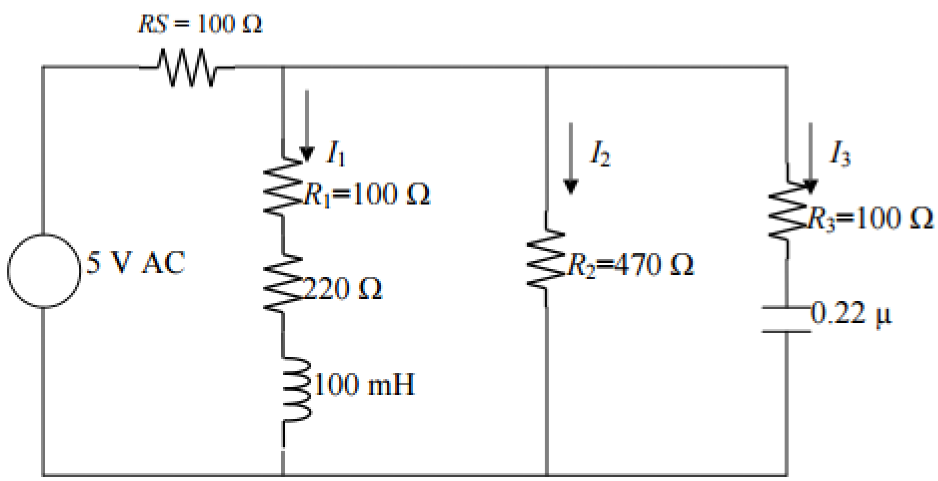


Table 2

Frequency, Hz	I_1	I_2	I_3
500			
600			
700			
800			
900			
1000			

Result

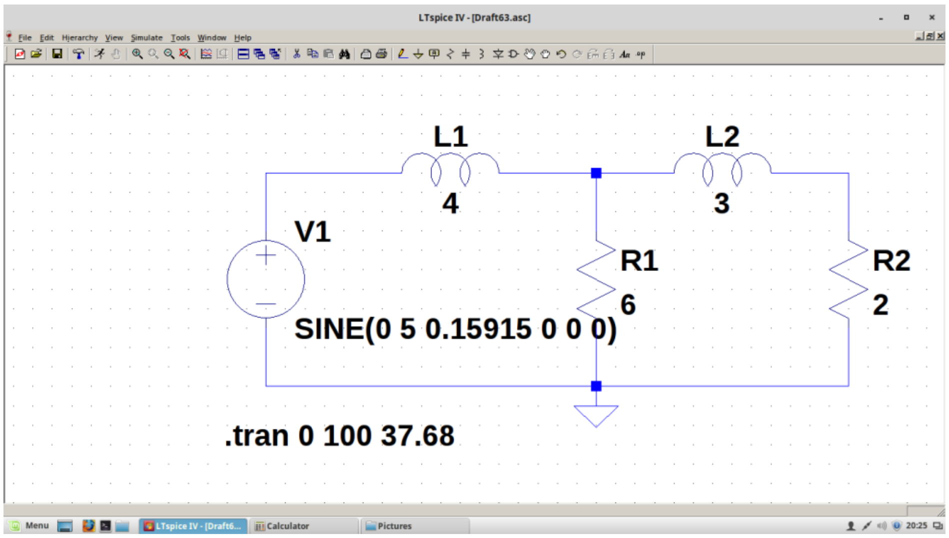
1. Theoretical values (I, V (R_1), V (C), V (L)) at the frequency 1000 Hz for the series *RLC* circuit in Figure 1 and I_1, I_2, and I_3 at the frequency 1000 Hz for the parallel RLC circuit is compared against practical values.

Experiment No. 6b Date :	STEADY STATE AC ANALYSIS USING LTSPICE

AIM : To analyze thecircuit with AC source using LTSPICE.

Procedure:

Consider the following circuit.

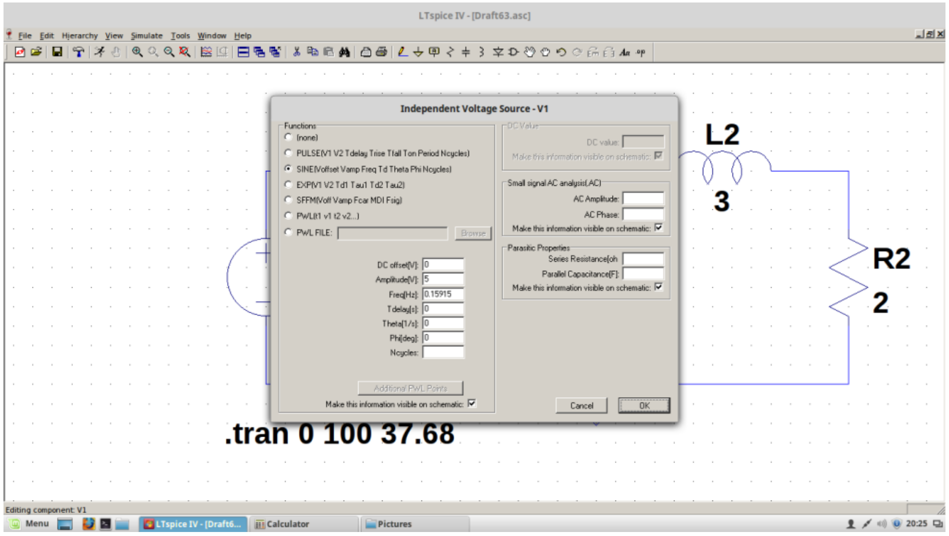


Assume w = 1 rad/s.

Thus f = w/ (2*pi) = 0.15915Hz

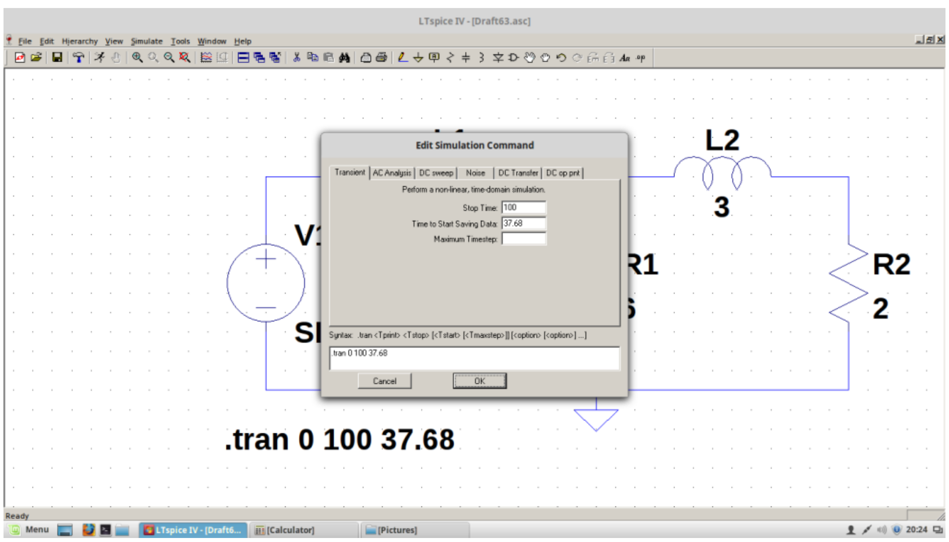
Time period (T): 1/f = 6.28s

We make this circuit using LT spice app. using resistors, inductor, and capacitor. We give them their respective values by right clicking on them. For voltage source we right click then select advanced then select sine and give amplitude 5, frequency 0.15915, and phase 0.



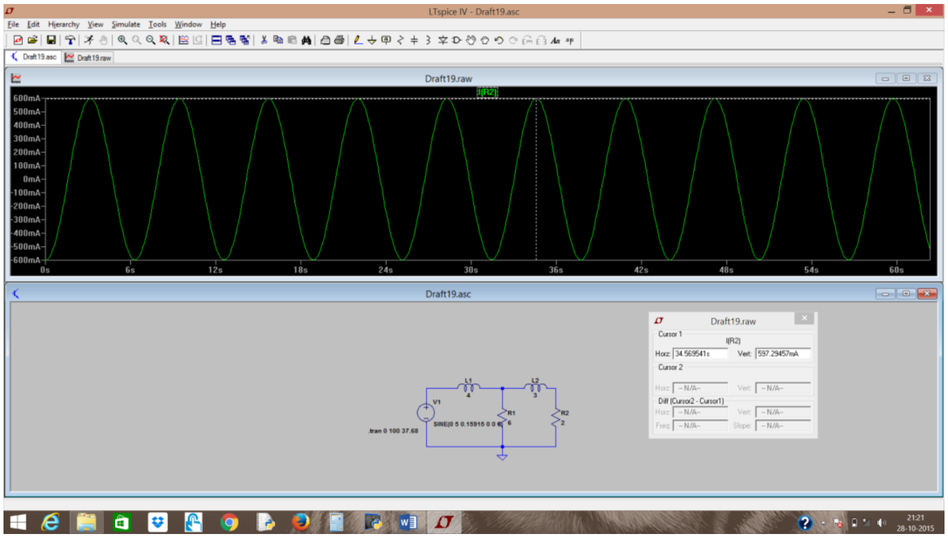
Now go to edit simulation command window and select transient.

Stop time: 100, Time to start saving data: 6T = 37.68s



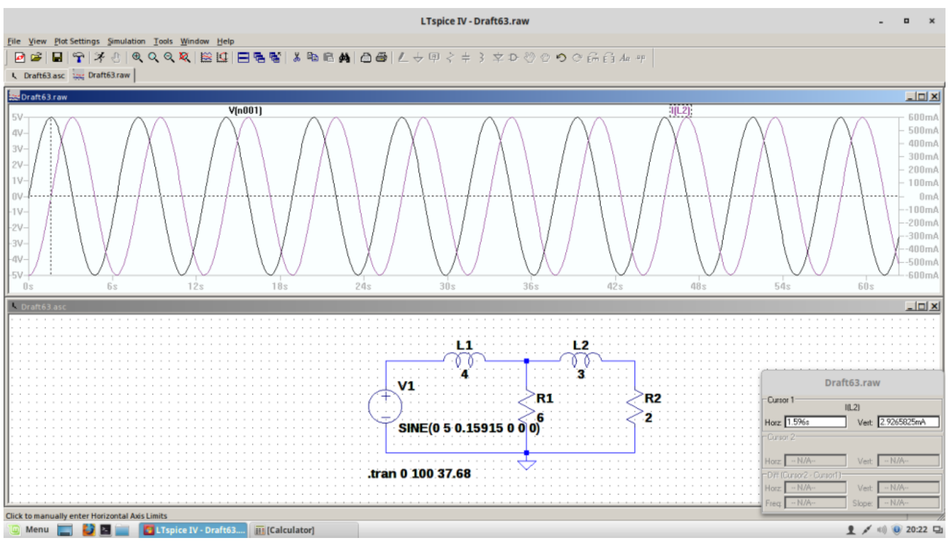
Click OK and run the simulation.

Current through R2



We get the magnitude by placing the cursor at the peak value i.e. 0.59A

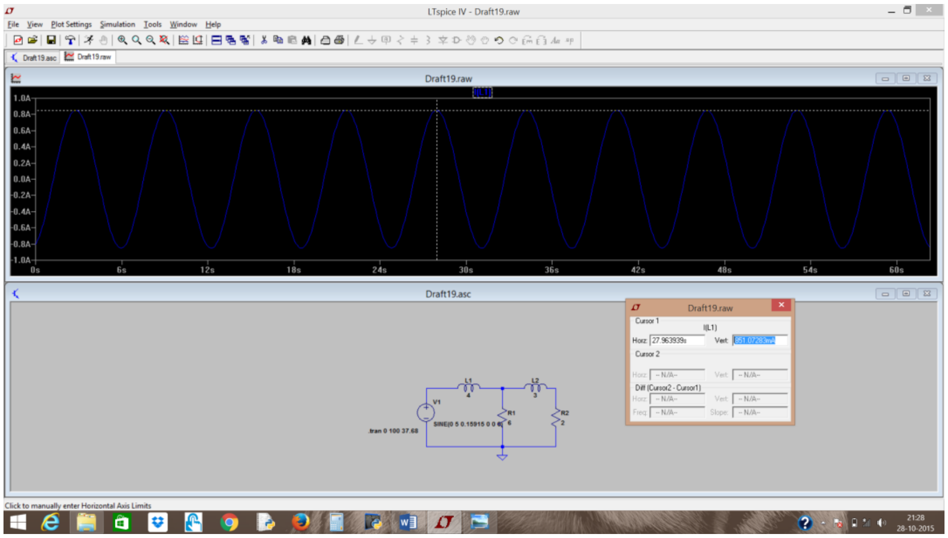
For phase we click on the node above L2.



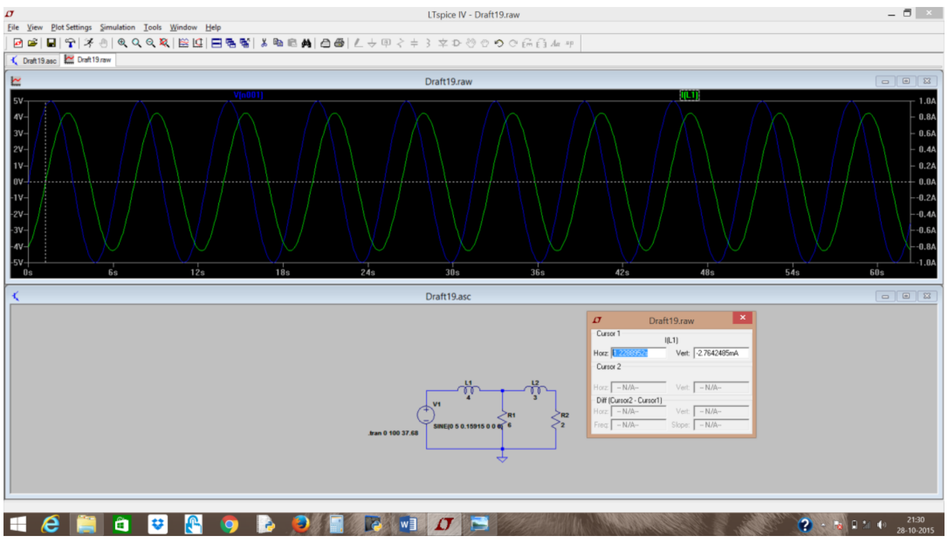
We find current lags voltage by 1.59s

Thus phase = 1.59/6.28 * 360 = 91.14deg

Similarly current through L1



Magnitude: 0.851A. To find phase click on the node and L1.



We find current lags voltage by 1.2288952s

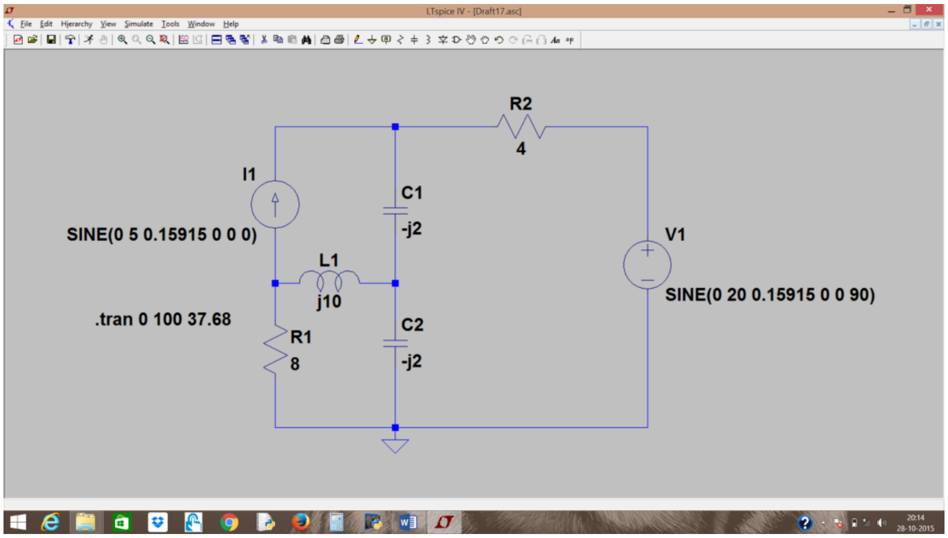
Thus phase 1.22/6.28 * 360 = 69.93deg.

Superposition theorem in AC.

Postulate:

The superposition principle states that the voltage across (or the current through) an element in a linear circuit is the algebraic sum of the voltages across (or current through) that element due to each independent source acting alone.

Consider the following circuit:



Assume the angular frequency = 1rad s^{-1}.

Thus frequency (f) = 1/2(pi)   [since w=2*pi / f]

f=0.15915 Hz

Time period (T) = 1/f = 6.2832 s.

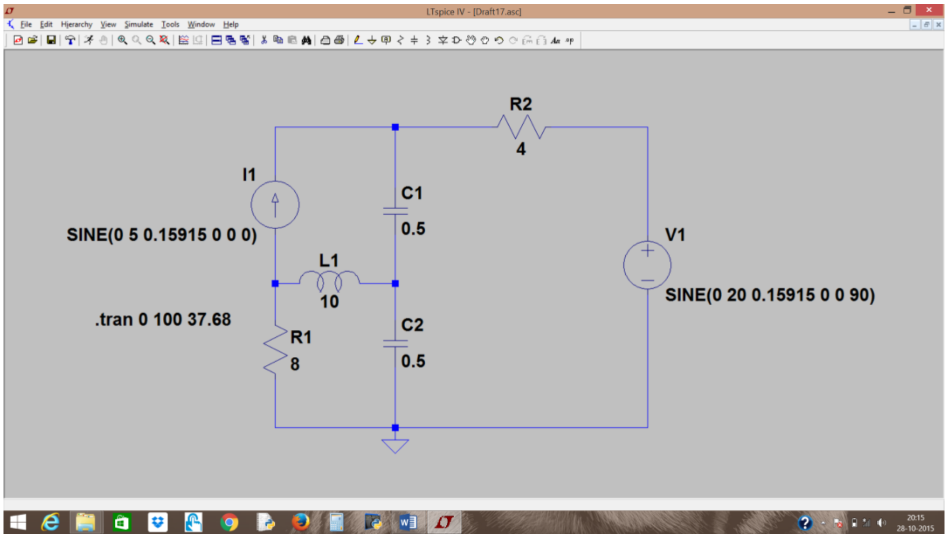
Now X_L = jwL = 12j

Thus L = 10H

X_C = -j/wC = -j2

Thus C = 0.5F

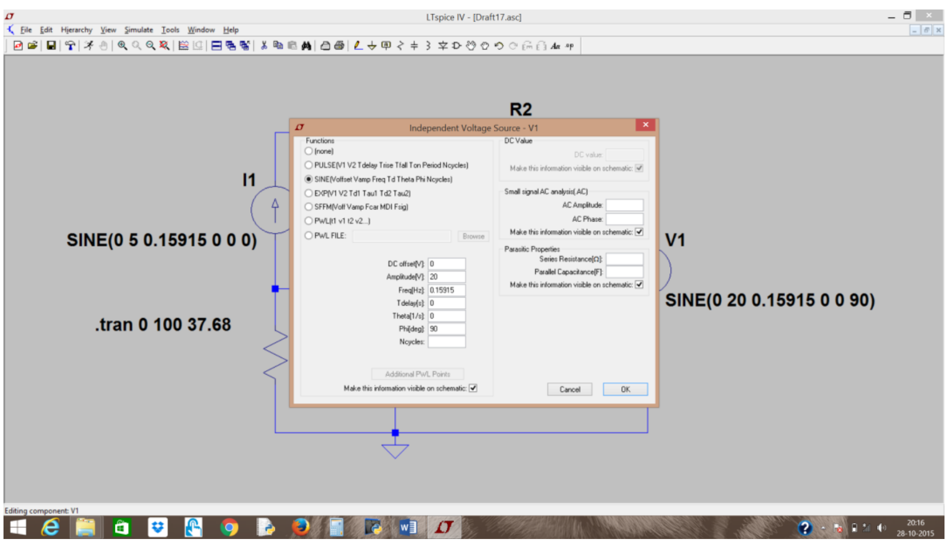
Thus our circuit becomes as given below



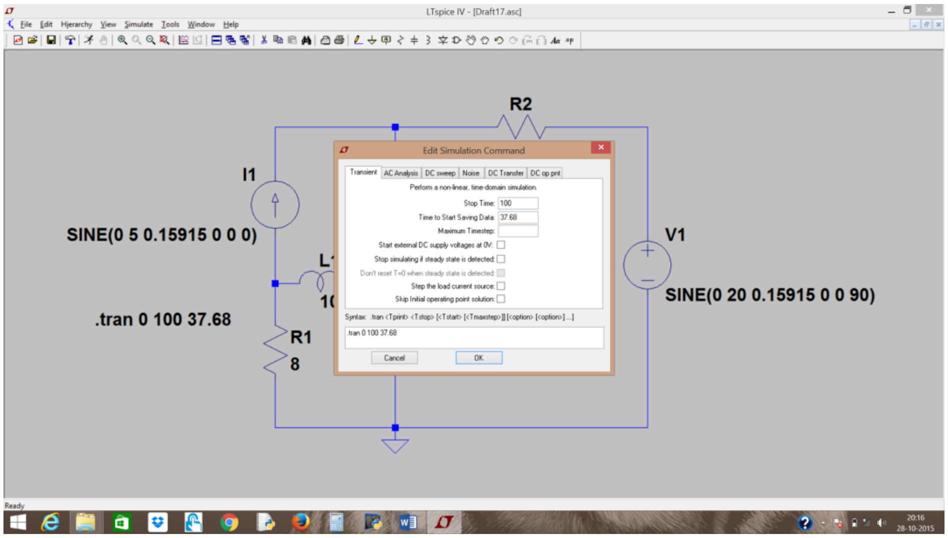
We draw the given circuit using voltage source, resistors, capacitors, inductors and current source. We give the components their values by right clicking on them.

For current and the voltage source after right clicking in them select ‘advanced’ and then select sine function and then give it amplitude 10 and 20 for current and voltage source respectively.

Since the angular frequency = 1rad/s we give frequency 0.15915Hz to both of them and phase angles 0 and 90^{o} to current and voltage source respectively.



Now click on edit simulation command window and select transient.

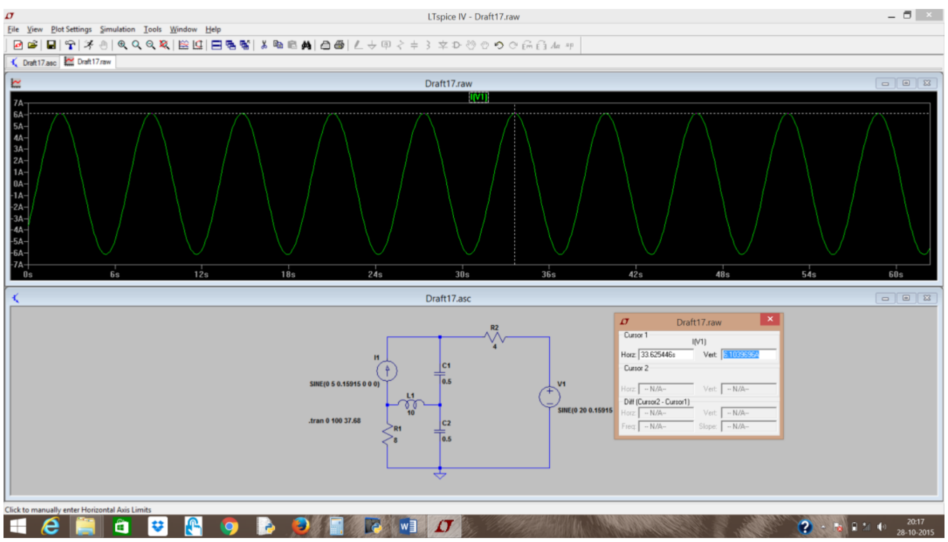


Time to start saving data: 6*T = 37.68s

Stop time: 100

Click OK and run the simulation.

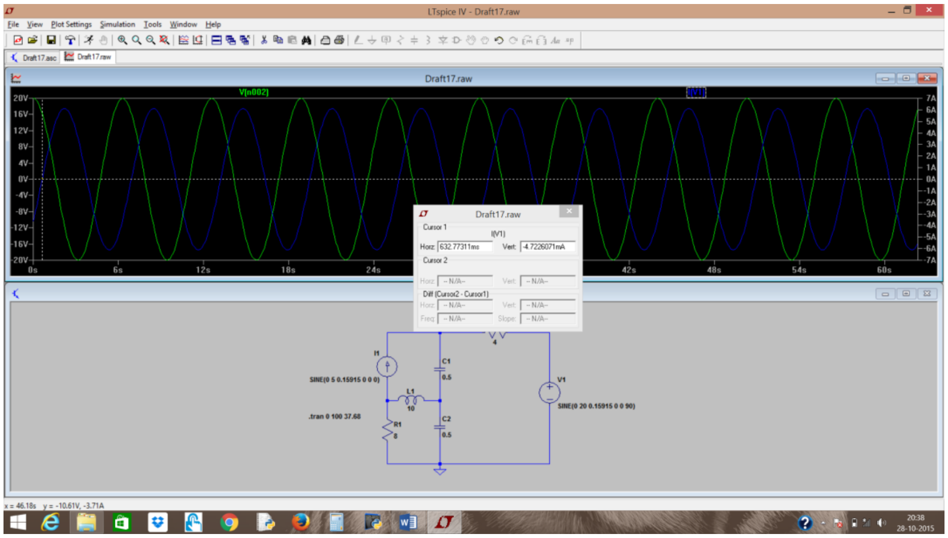
Now let current through V1 = I_o



Its magnitude is given by placing the cursor at the peak of the curve.

We find the magnitude to be: 6.10A

Now to find phase we click on the element to get current and at node to get voltage as

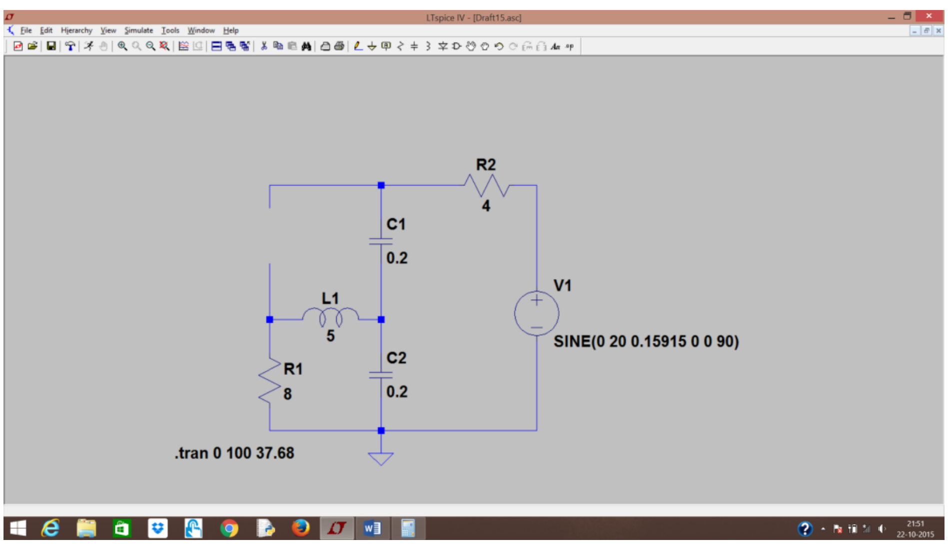


We observe current is lags the current by 0.632 s.

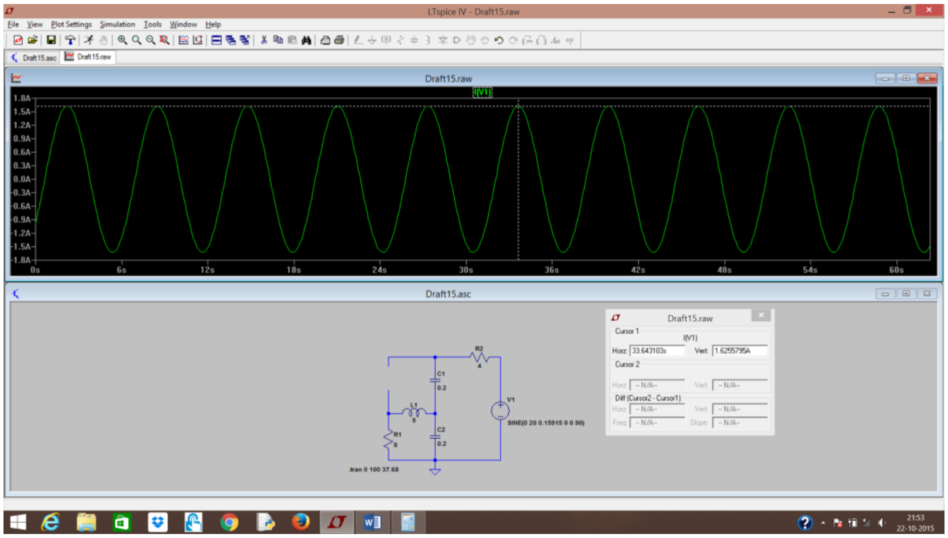
Thus phase = $0.632/6.28 * 360^0 = 36.22$deg

Thus we have $I_o = 6.10\angle -36.22$

Now we de-activate one source say current source thus our circuit becomes:

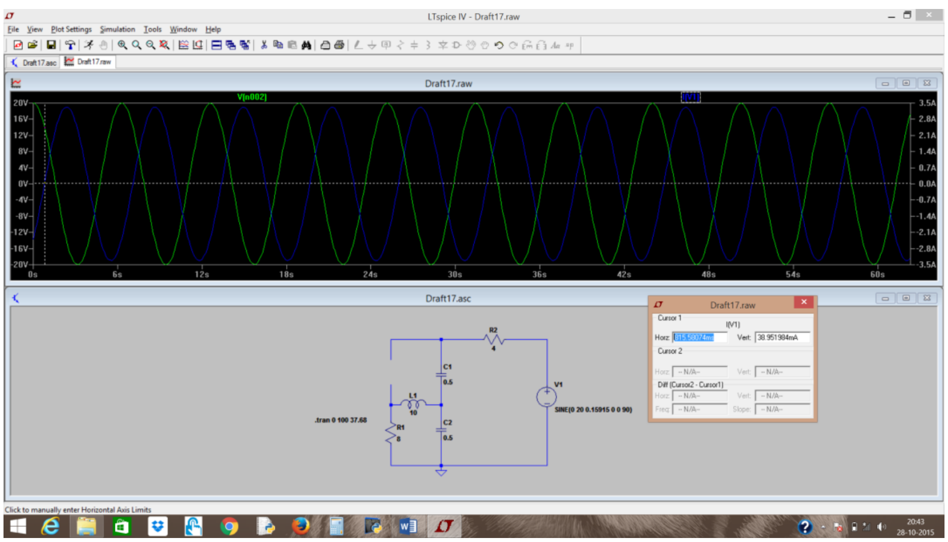


Again we find the current and the phase difference using the above method



We find the magnitude to be: 3.32A

Now for phase

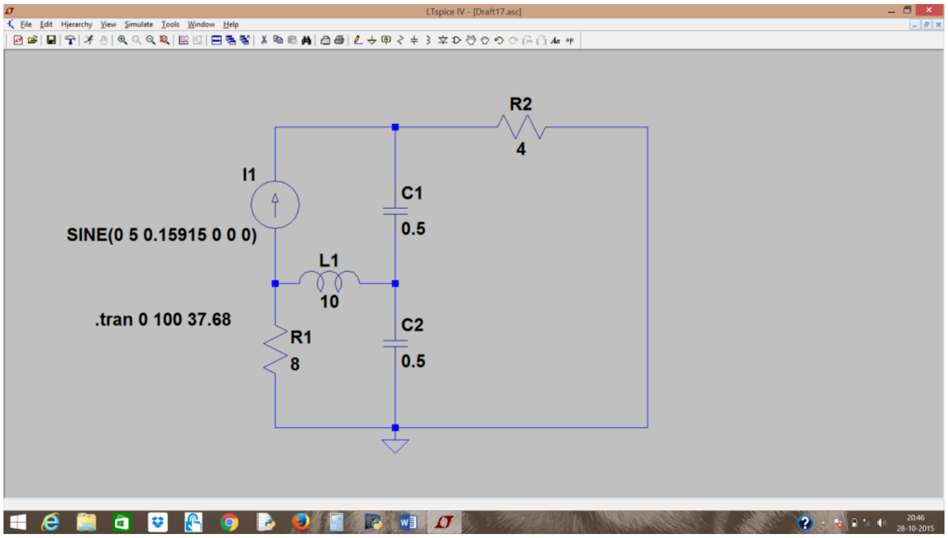


We find that current lags the voltage by 815.58074ms

Thus the phase difference 0.815/6.28 *360 = 46.71deg

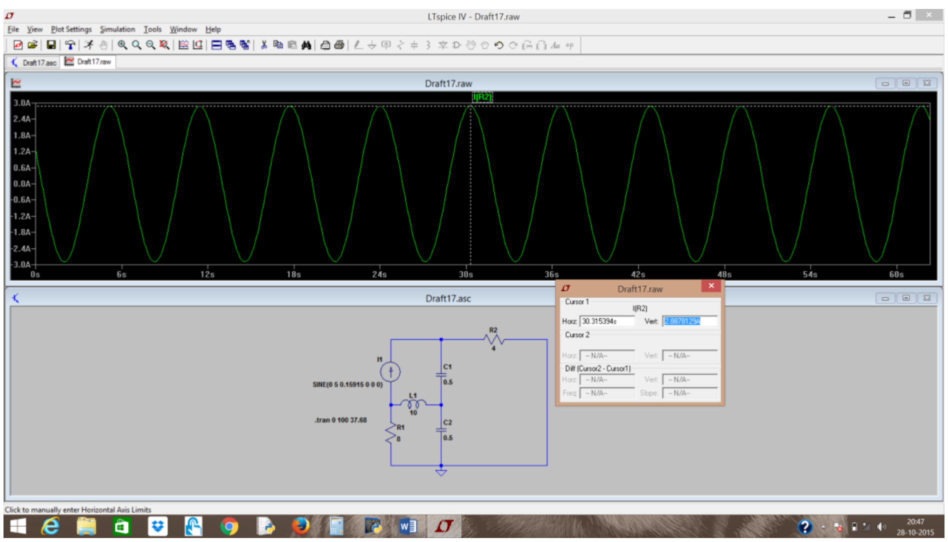
Thus I_o^1 = 3.32<-46.71

Now we de-activate the other source, thus we have



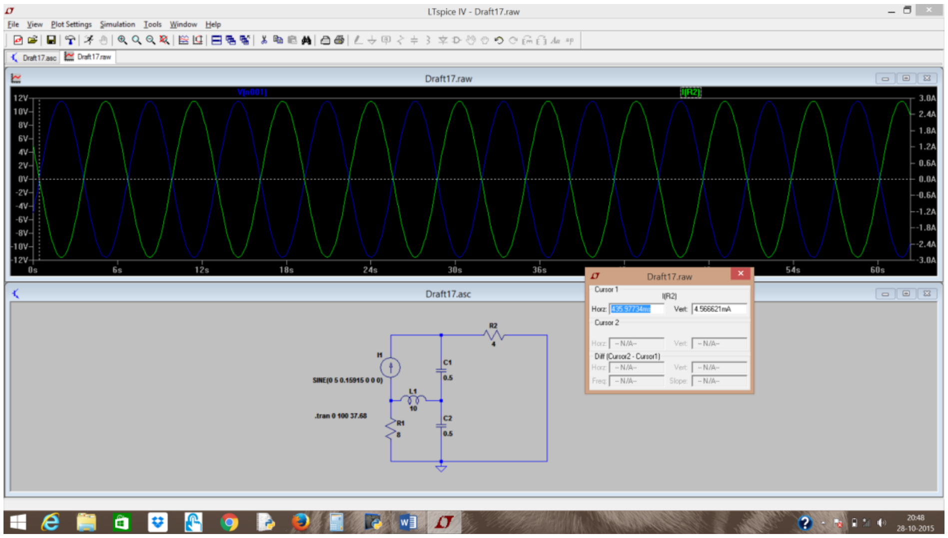
Again we find current and voltage due to current source using the above method

Thus we have



Thus we find the magnitude to be 2.89A

Now to find phase



Current lags the voltage by 435.97734ms

Thus the phase= 0.435/6.28 *360 = 24.93 deg.

Thus I_o^{11}= 2.89<-24.93

Thus we have

I_o= 6.10<-36.22

I_o^1= 3.32<-46.71

I_o^{11}= 2.89<-24.93

Thus we observe $I_o = I_0^1 + I_o^{11}$

Hence superposition theorem for AC circuit is verified.

Experiment No. 7a Date :	**Series and Parallel Resonance**

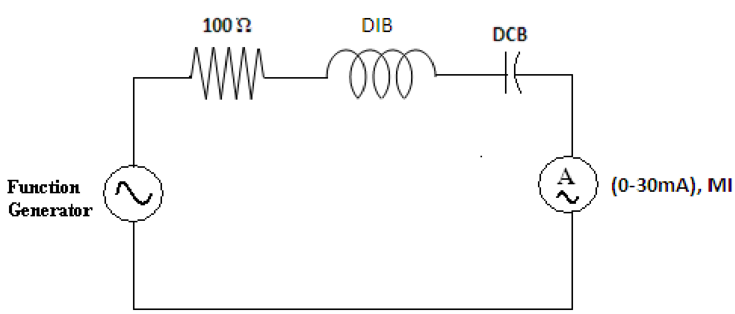
Series Resonance

Aim: To plot the current Vs frequency graph of series and parallel resonant circuits and hence measure their bandwidth, resonant frequency and Q factor.

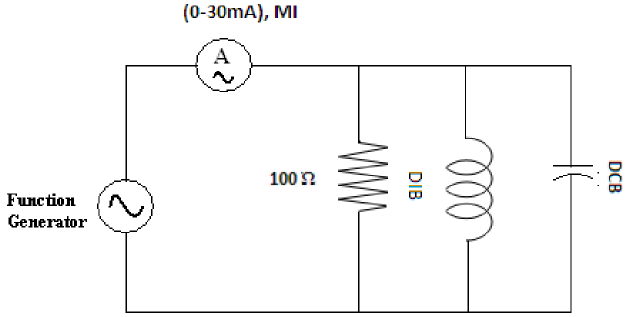
Apparatus Required:

S.No.	Name of the Components/Equipment	Type	Range	Quantity required
1	Function Generator			1
2	Resistor		100 Ω	1
3	Decade Inductance Box			1
4	Decade Capacitance Box			1
5	Ammeter	MI	(0-30) mA	1
6	Connecting Wires	Single Strand		Few nos

CIRCUIT DIAGRAM FOR SERIES RESONANCE



CIRCUIT DIAGRAM FOR PARALLEL RESONANCE



THEORY:

A circuit is said to be in resonance when applied voltage V and current I are in phase with each other. Thus at resonance condition, the equivalent complex impedance of the circuit consists of only resistance (R) and hence current is maximum. Since V and I are in phase, the power factor is unity.

The complex impedance

$Z = R + j (XL – XC)$

Where $XL = \omega L$

$XC = 1/\omega C$

At resonance, XL= XC and hence Z= R

Bandwidth of a Resonance Circuit:

Bandwidth of a circuit is given by the band of frequencies which lies between two points on either side of resonance frequency, where current falls through 1/1.414 of the maximum value of resonance. Narrow is the bandwidth, higher the selectivity of the circuit.As shown in the model graph, the bandwidth AB is given by f2 – f1. f1 is the lower cut off frequency and f2 is the upper cut off frequency.

Q - Factor:

In the case of a RLC series circuit, Q-factor is defined as the voltage magnification in the circuit at resonance. At resonance, current is maximum. Io= V/R.

The applied voltage V = IoR

Voltage magnification = VL/V = IoXL

In the case of resonance, high Q factor means not only high voltage, but also higher sensitivity of tuning circuit. Q factor can be increased by having a coil of large inductance, not of smaller ohmic resistance.

$Q = \omega L / R$

FORMULAE USED:

Resonant frequency fr =1/2π LC Hz

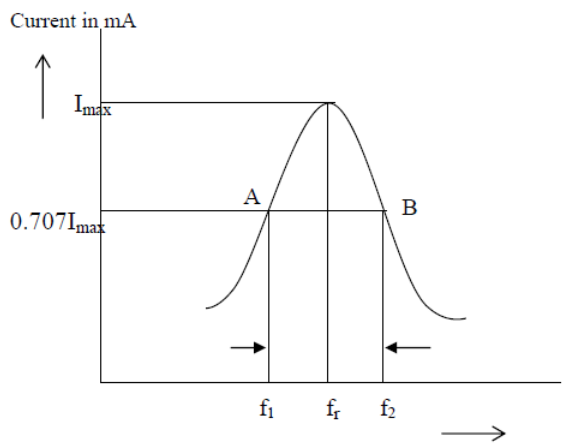
Bandwidth BW = f2 – f1

Quality Factor =fr /BW

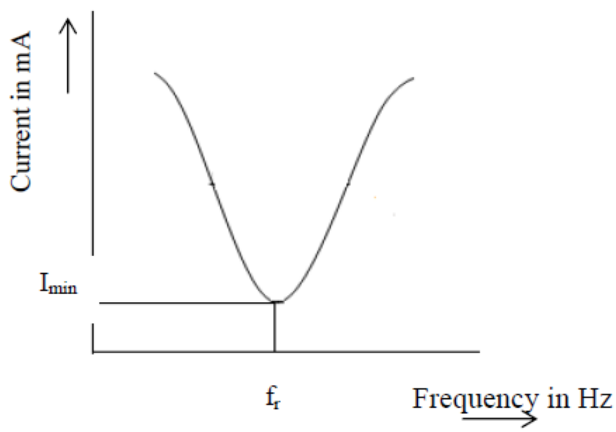
PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Vary the frequency and note down the corresponding meter reading.
3. Draw the current Vs frequency curve and measure the bandwidth, resonant frequency and Q factor.

MODEL GRAPH FOR SERIES RESONANCE:



MODEL GRAPH FOR PARALLEL RESONANCE:



OBSERVATION TABLE:

(a) Series resonance

S.No.	Frequency in Hz	Output Current in mA

(b) Parallel resonance

S.No.	Frequency in Hz	Output Current in mA

<u>MODEL CALCULATIONS:</u>

<u>RESULT:</u>

Thus the current Vs frequency graphs of series and parallel resonant circuits were plotted and the bandwidth, resonant frequency and Q factor were measured.

They were found to be

(a) Series resonance

Resonant frequency = _____________
Bandwidth = _____________
Q- Factor = _____________

(b) Parallel Resonance

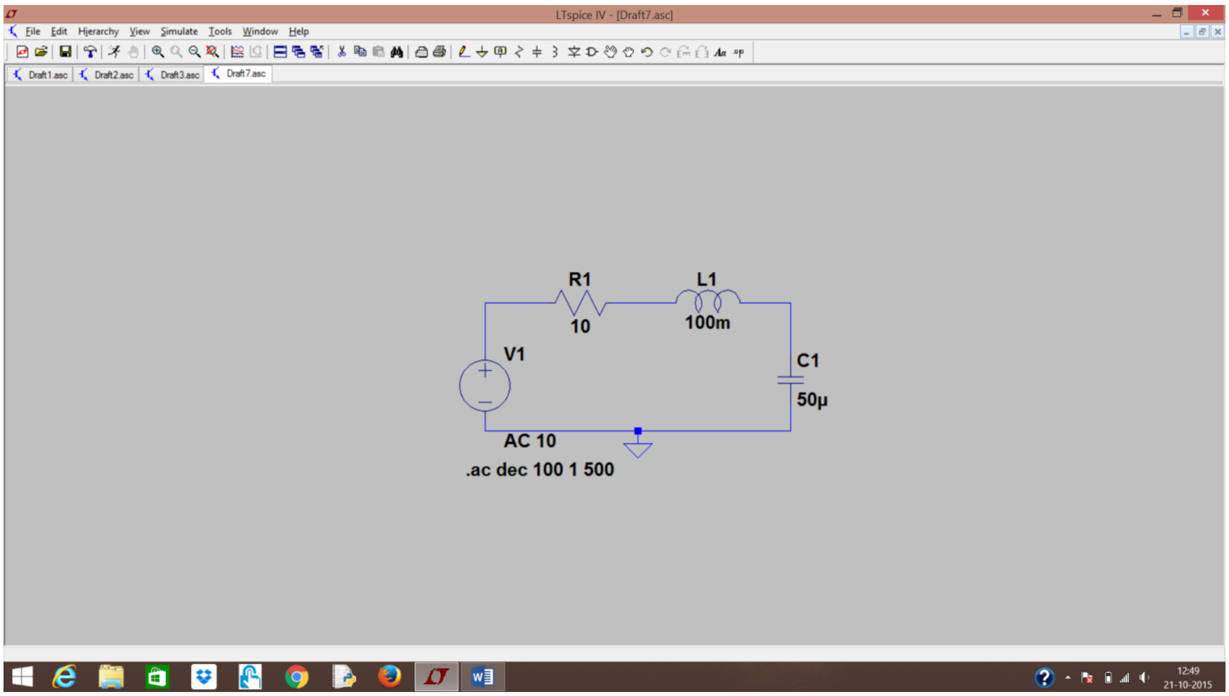
Resonant frequency = _____________
Bandwidth = _____________
Q- Factor = _____________

Experiment No. 7b **Date :**	**Series and Parallel Resonance using LTSPICE**

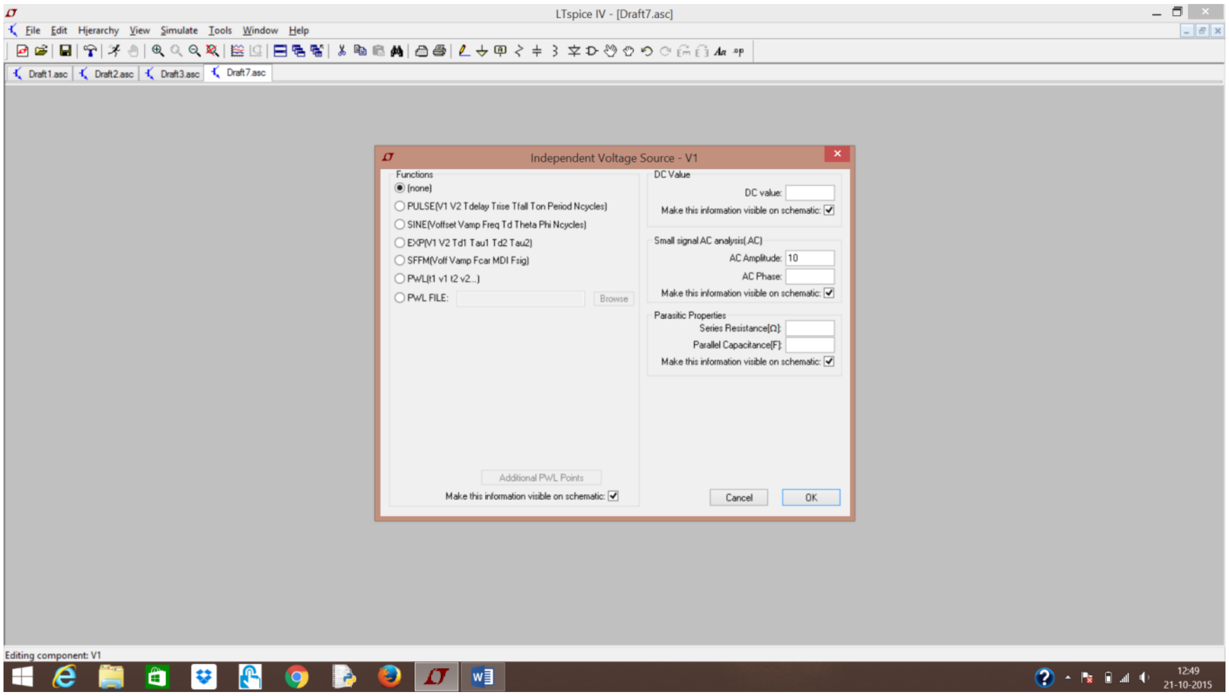
Series Resonance

Aim: To obtain the characteristics of series and parallel resonant circuits.

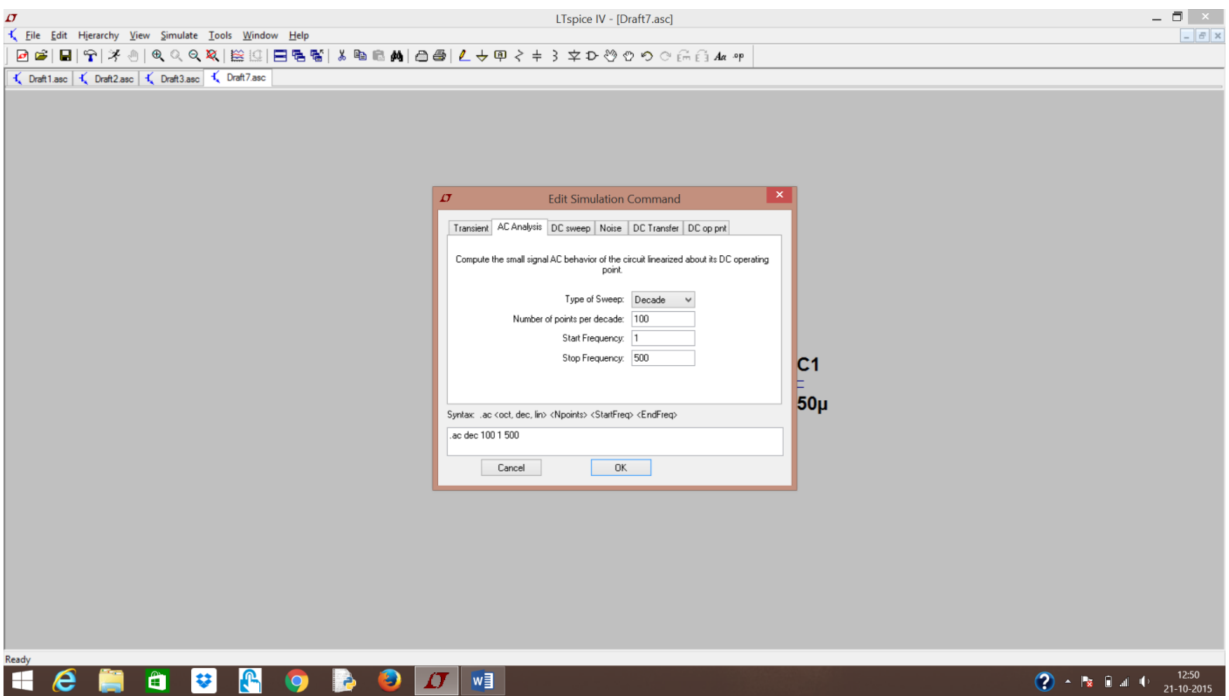
Circuit Diagram:



Procedure:



- Go to edit simulation command window and select AC analysis.



Type of sweep: Decade

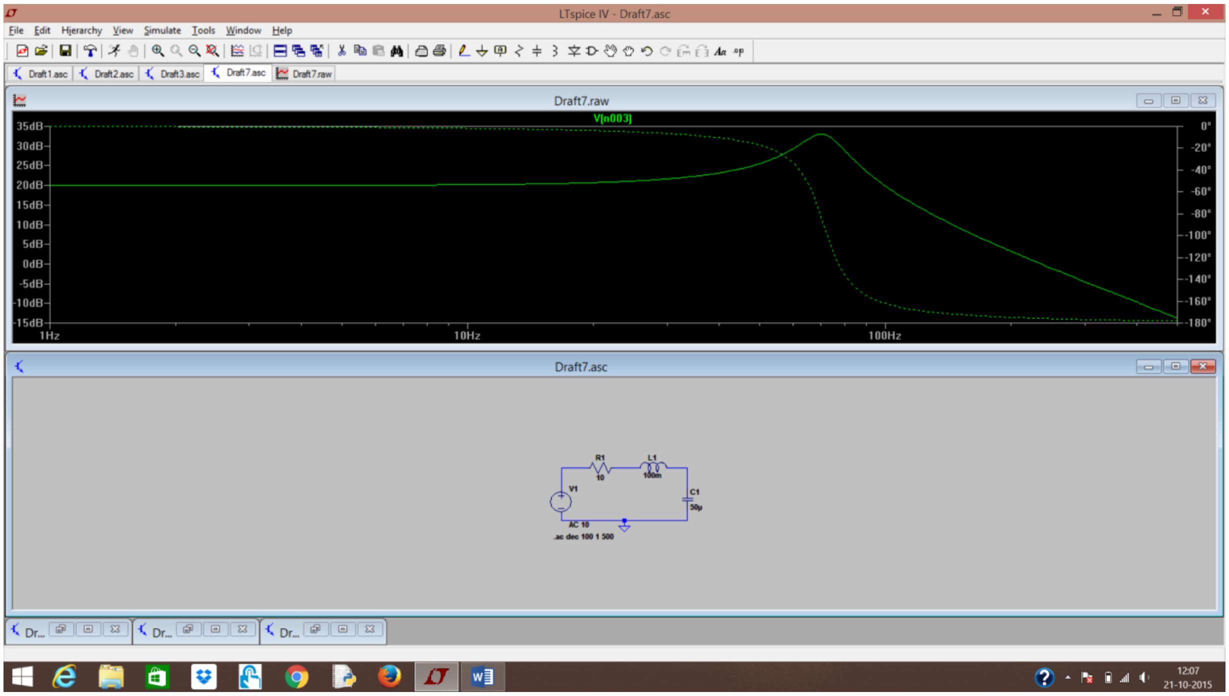
Number of points per decade: 100

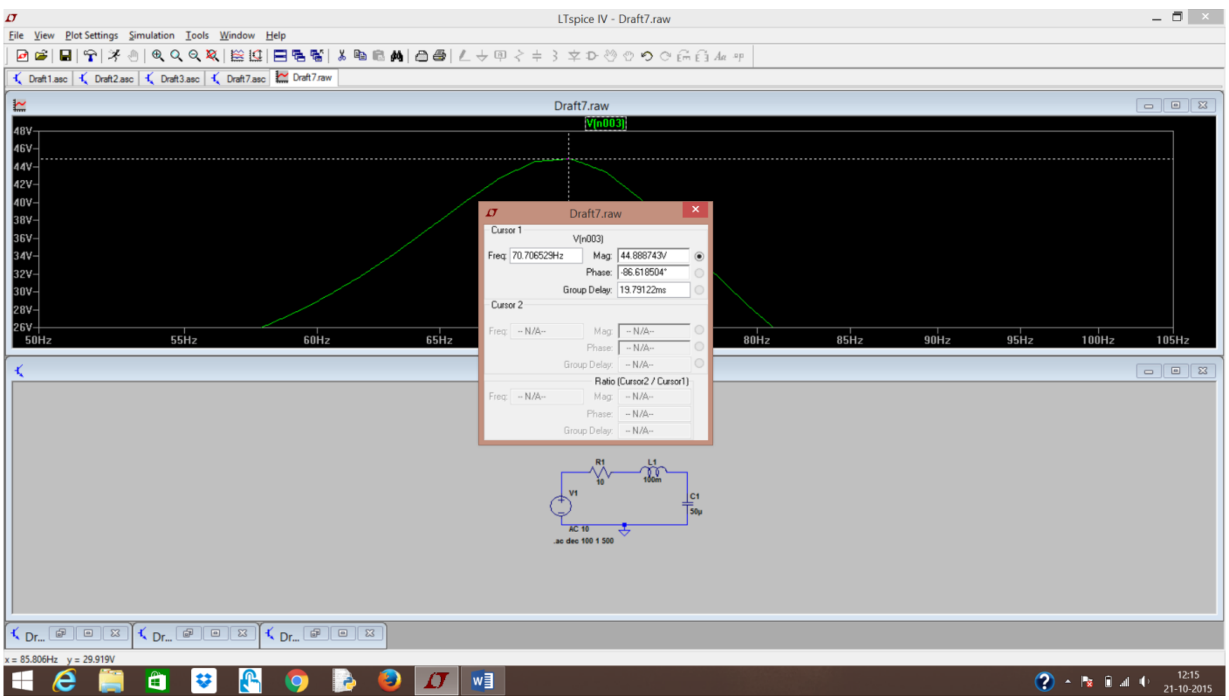
Start frequency: 1

Stop frequency: 500

Then click OK. Now run the simulation.

Output:





The frequency at the peak of the curve is the resonant frequency (f_r).

From the table

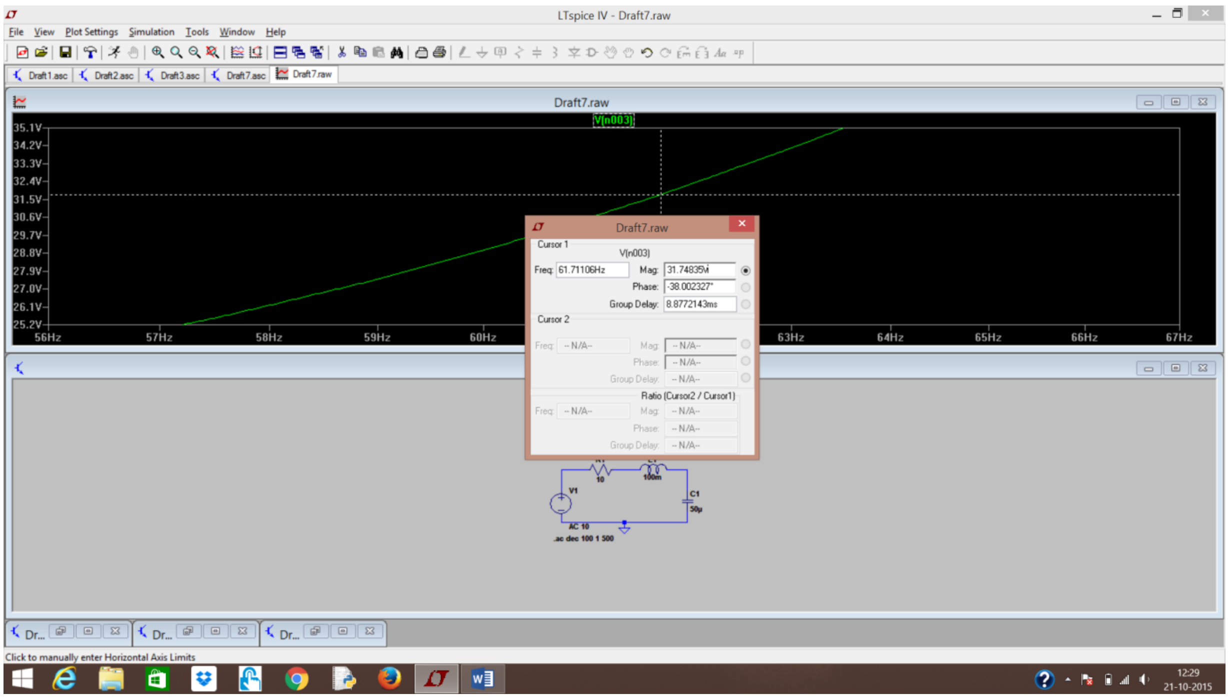
f_r = 70.706529Hz.

Magnitude of the peak= 44.888743V

To find f_l and f_h

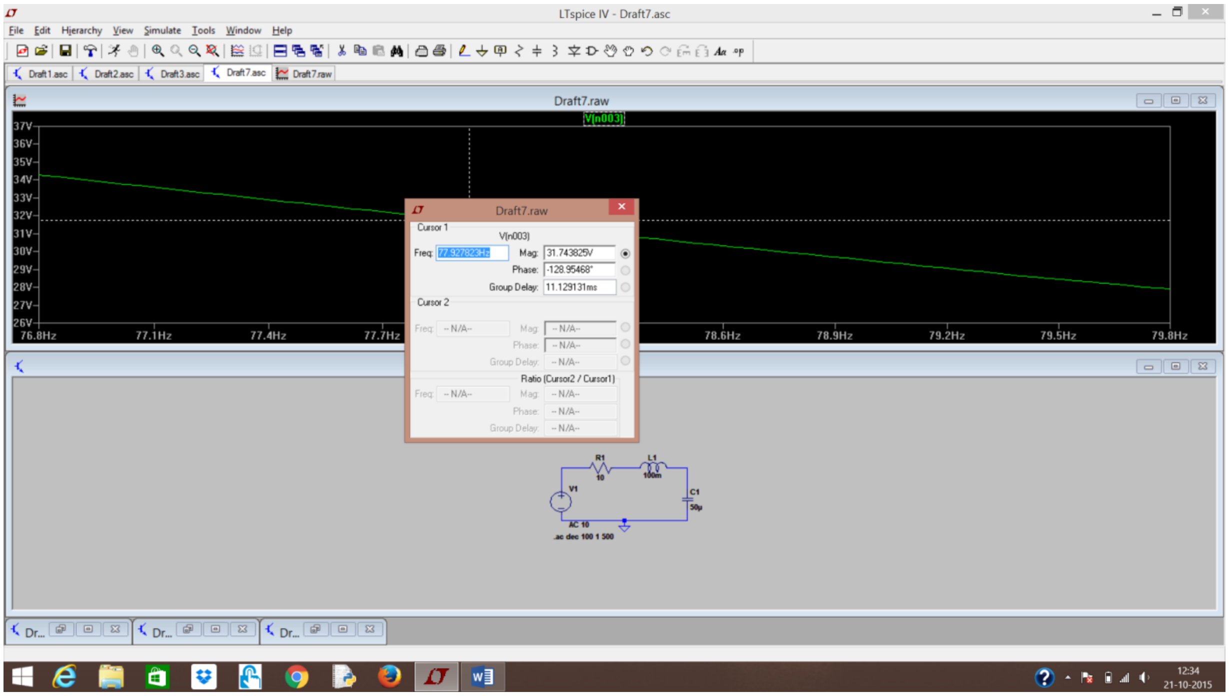
0.707*(peak value) = 31.74

f_l and f_h = (frequency, at magnitude = 31.74)



f_l= 61.71106Hz

On right side of the curve



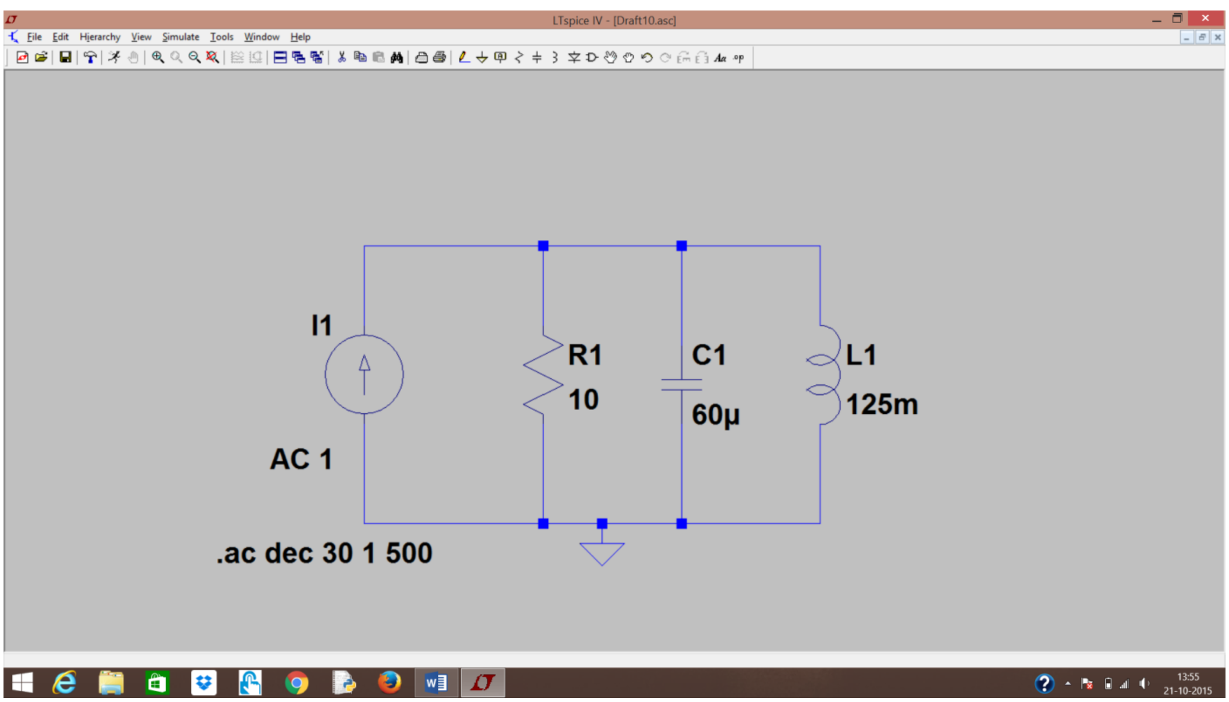
f_h= 77.927823Hz

Thus band width = f_h- f_l = (77.927823 - 61.71106) Hz= 16.216763Hz

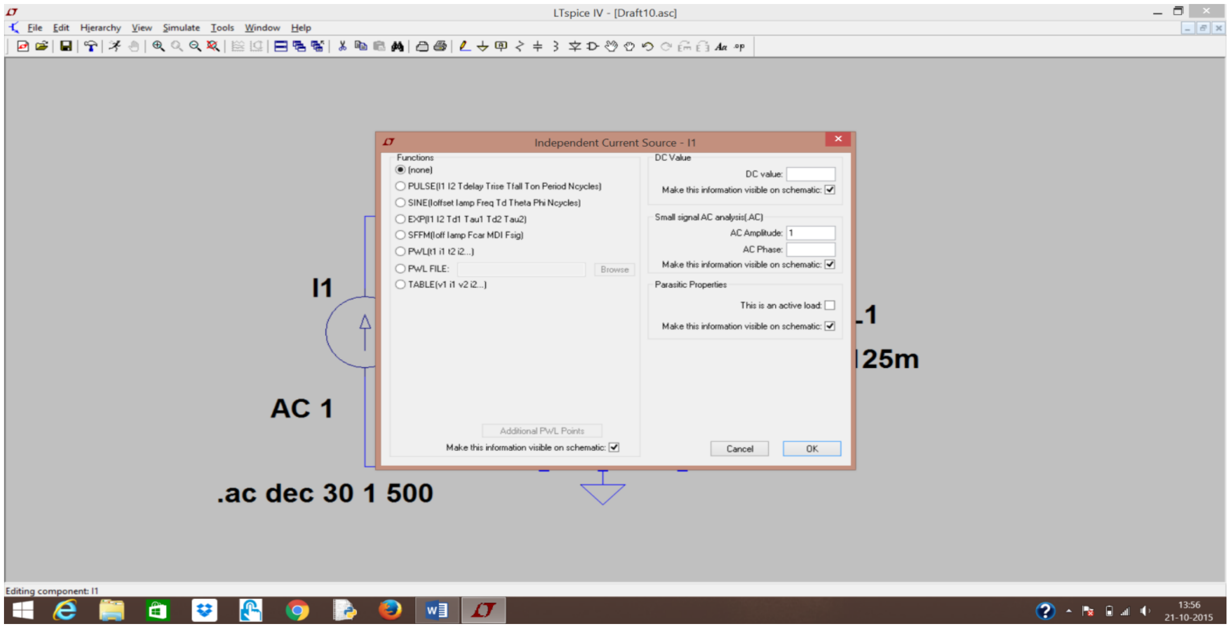
Quality Factor = [2*(3.14)* f_r *L]/R=4.43

Parallel Resonance

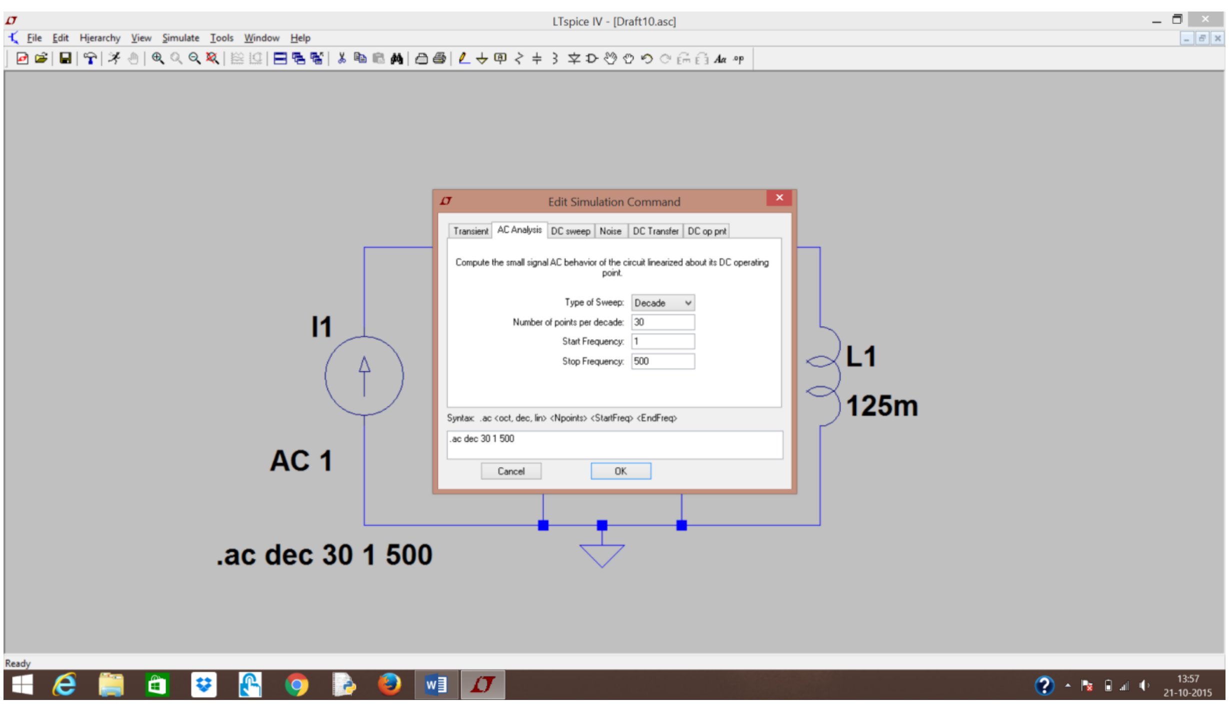
Circuit Diagram:



Procedure:

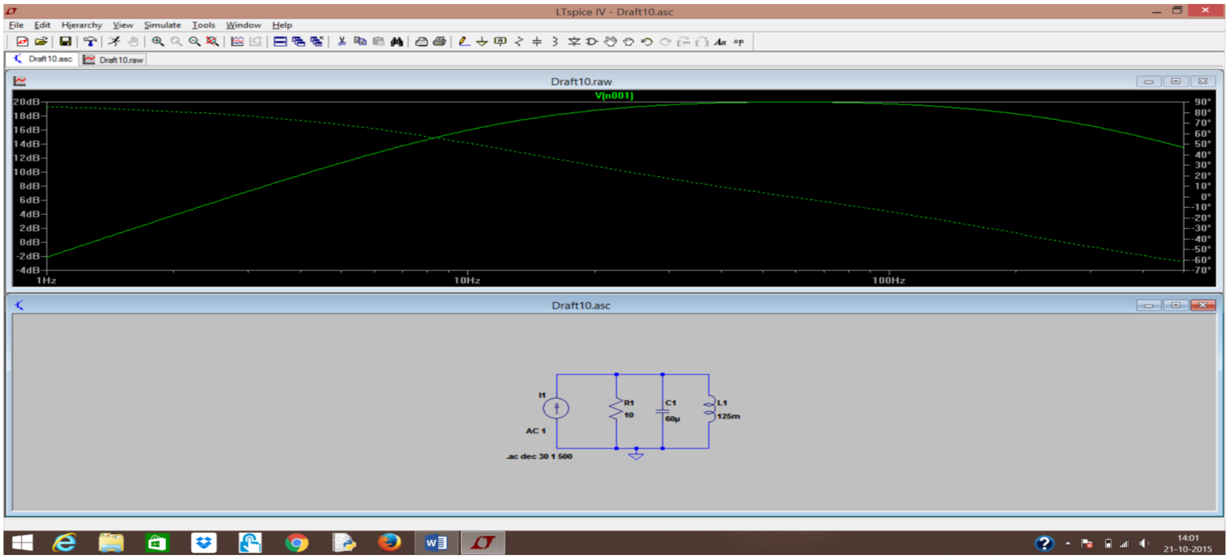


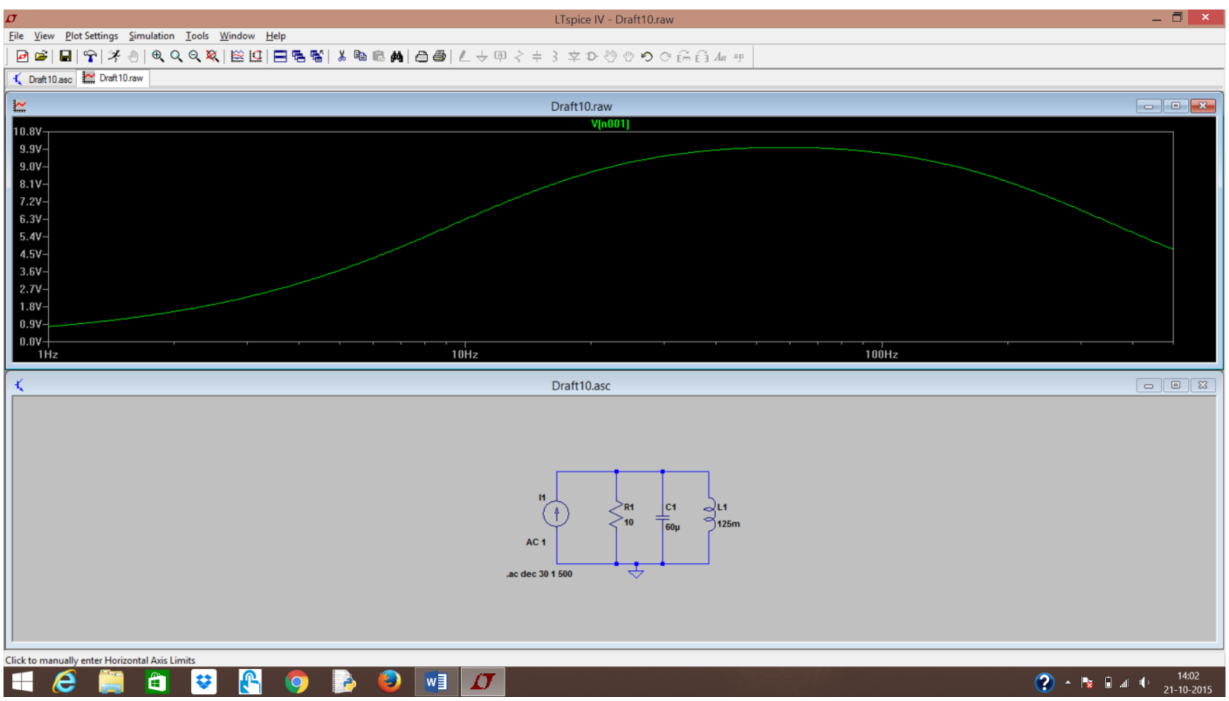
- Go to edit simulation command window and select AC analysis.

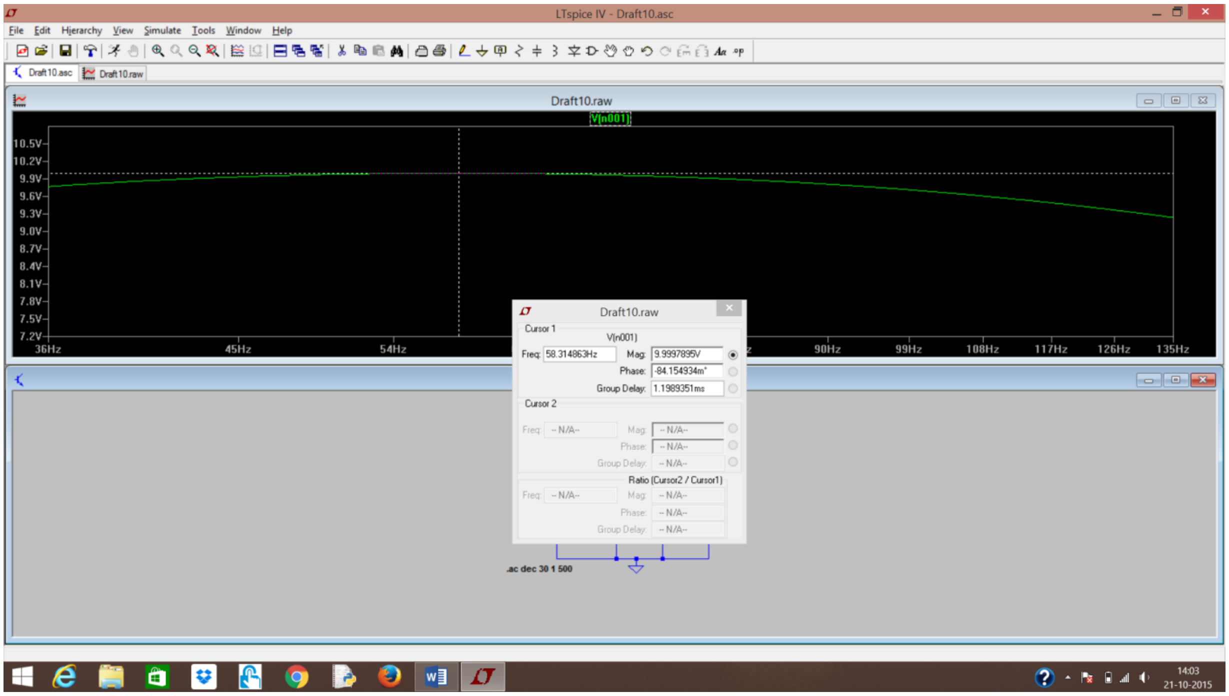


- Click OK. Then run the simulation.

<u>Output:</u>







The frequency at the peak of the curve is the resonant frequency (f_r).

From the table

f_r = 58.314863Hz

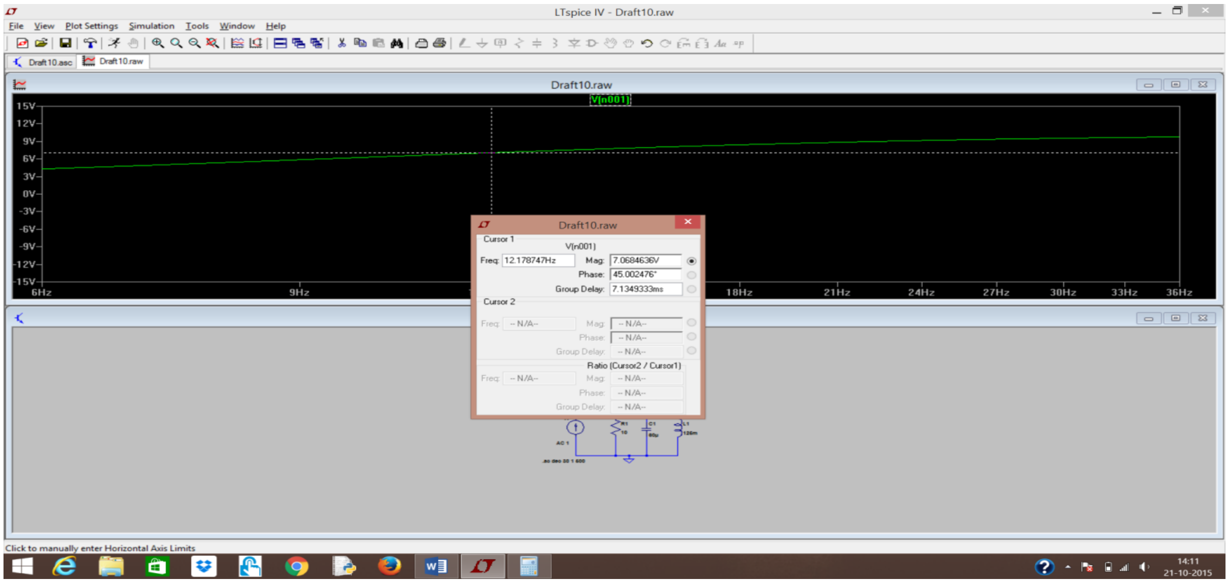
Magnitude of the peak= 9.9997895V

Now to find f_l and f_h

0.707*(peak value) =7.0698

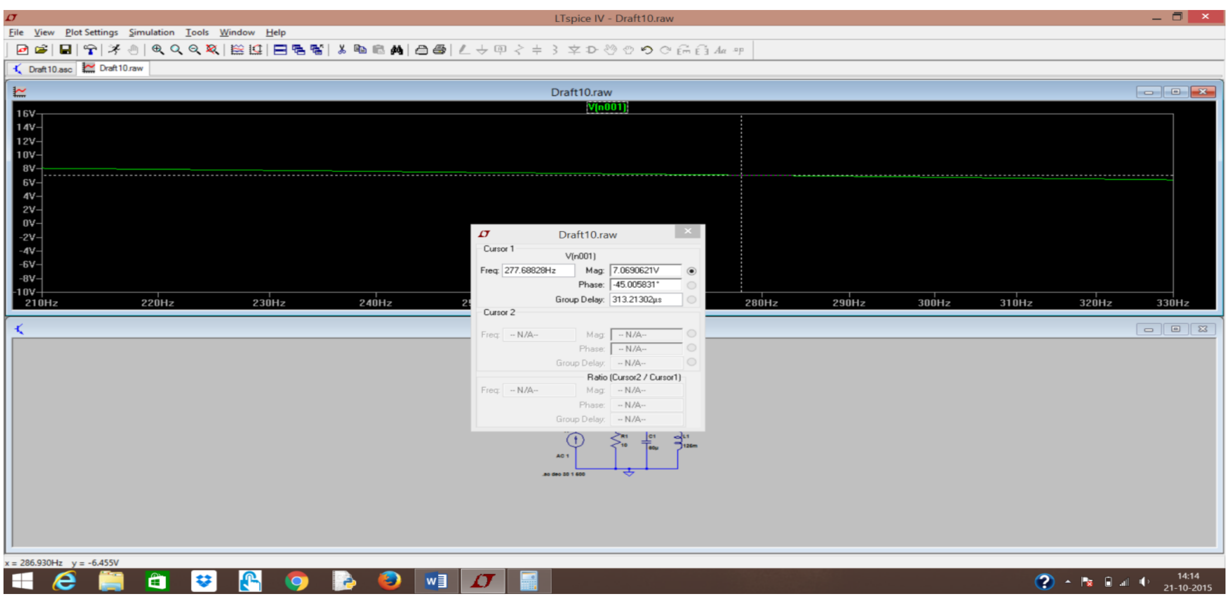
Now f_l and f_h = (frequency, at magnitude = 7.0698)

On left of peak



f_l= 12.178747Hz

On right side of the peak



f_h= 277.68828Hz

Thus band width = f_h- f_l = (277.68828 - 12.178747) Hz = 265.509533

Quality Factor = [2*(3.14)* f_r *L]/R = 4.577

Result:

	Resonant Frequency	Band-Width	Quality Factor
Series Resonance	70.706529Hz.	16.216763Hz	4.43
Parallel Resonance	58.314863Hz	265.509533	4.577