Exp. No.: 2 **Reg.No. 21BAI1604**

Name: Dhruv Karmokar **Date: 3/10/2021**

Sinusoidal Steady State Response of RLC circuit

(Series Band pass filter using passive components)

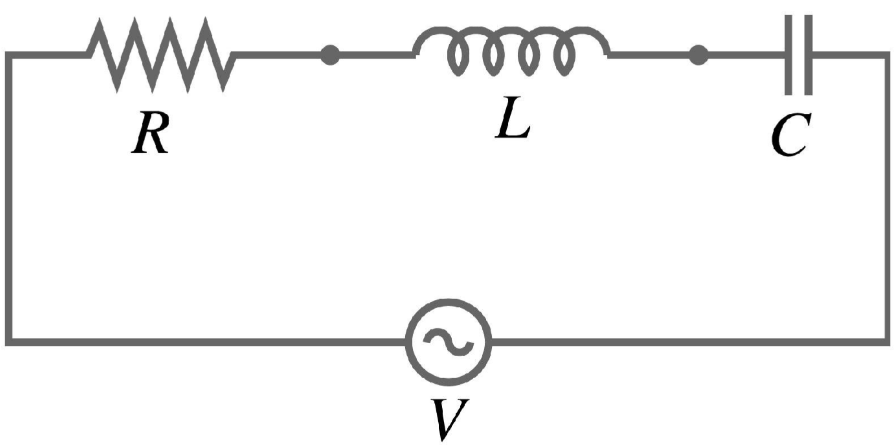
Aim:

To study the sinusoidal steady-state response of the given RLC circuit which is used in a series bandpass filter.

Software Required:

LTspice software

Theory & Circuit Diagram:



The Series RLC circuit is shown in the above figure. The RLC series circuit can be used as a series bandpass filter by placing a series LC circuit in series with the load resistor. In the series RLC circuit, larger reactance determines the net reactance of the circuit.

If $X_L > X_C$ the circuit behaves like an inductive circuit and the current lags the voltage and if $X_C > X_L$, the circuit behaves like a capacitive circuit and the current leads the voltage. The magnitude and Phase angle of the current, I in the series RLC circuit is obtained using the following equation,

$$I = \frac{V}{Z} = \frac{V}{\sqrt{R^2 + (X_L - X_C)^2}}$$

$$\varphi = \frac{X_L - X_C}{R}$$

For RLC circuit with R= 10 Ω, L= 1mH, C= 1 μF and V_m = 100 V, f = 50 Hz, X_L = ωL = 2ΠfL = 0.314 Ω, X_C = (1/ωC) = (1/ 2ΠfC) = 3184 Ω, I_m = 0.0314 A, Φ = 89.82°.

In this example, since $X_C > X_L$, Current leads the voltage.

Procedure:

1. Open LTspice. Go to File New Schematic.
2. On the File Menu, click on Edit Component.
3. Place the voltage sources, resistor, inductor, capacitor and ground on to schematic and make necessary connections as shown in the Figure.
4. Right click on the voltage source V1 and click Advanced option and then Select SINE (Voffset Vamp Freq Td Theta Phi Ncycles) and Set the values as (DC offset = 0, Amplitude =100, Freq = 50).
5. Right click on the resistor, inductor and capacitor and set the values.
6. Go to Edit → SPICE analysis. Set the type of sweep to Linear, Number of points to 100 and start and stop frequency to 50 each in the AC Analysis command and run the simulation. (run symbol on the menu bar).
7. Observe the peak value of the current and phase angle from the obtained

output window and note it in the "Theoretical Value" column of the observation table.

8. **For waveforms:** Go to Edit → SPICE analysis. Set the stop time in Transient command and run the simulation. (run symbol on the menu bar).
9. To view the results, right click → Add Trace → Select V (<<input node>>) and I (L1).
10. Observe the waveforms, change the appropriate colors for proper visibility using color preferences and control panel tool.

Calculation:

1) Capacitive Circuit

$R = 10\ \Omega$

$L = 1\ mH$

$C = 1\ \mu$

$f = 50\ Hz$

$X_L = L\omega = 2\pi f L = 2 \times 3.14 \times 50 \times 0.001$

$X_L = 0.314\ \Omega$

$X_C = \frac{1}{\omega C} = \frac{1}{2\pi f C} = \frac{1}{2 \times 3.14 \times 50 \times 1 \times 10^{-6}} = 3184$

$X_C = 3184\ \Omega$

$\Phi = \tan^{-1}\left(\frac{X_L - X_C}{R}\right)$

$= \tan^{-1}\left(\frac{0.314 - 3184}{10}\right)$

$\Phi = 89.92^\circ$

$X_C > X_L$, The circuit is capacitive in nature.

2) Inductive Circuit

$R = 10\ \Omega$, $L = 1\ H$, $C = 1000\ \mu$

$f = 50\ Hz$

$X_L = \omega L = 2\pi f L = 2 \times 3.14 \times 5 \times 1$

$X_L = 314.15\ \Omega$

$X_C = \frac{1}{\omega C} = \frac{1}{2\pi f C} = \frac{1}{2 \times 3.14 \times 50 \times 1000 \times 10^{-6}} = 314.8\ \Omega$

$X_C = 3.184\ \Omega$

$\Phi = \tan^{-1}\frac{(X_L - X_C)}{R} = \tan^{-1}\left(\frac{314.15 \times 3.148}{10}\right)$

$\Phi = 88.158^\circ$

$X_L > X_C$, The circuit is inductive in nature.

3) Resistive Circuit

$R = 10\ \Omega$, $L = 10\ mH$, $C = 1000\ \mu F$

$f = 50\ Hz$,

$X_L = \omega L = 2\pi f L = 2 \times 3.14 \times 50 \times 10\ mH$

$X_L = 3.1425\ \Omega$

$X_C = \frac{1}{\omega C} = \frac{1}{2\pi f C} = \frac{1}{2 \times 3.14 \times 50 \times 1000 \times 10^{-6}} = 3.184$

$X_C = 3.184\ \Omega$

$\Phi \cong 0^\circ$

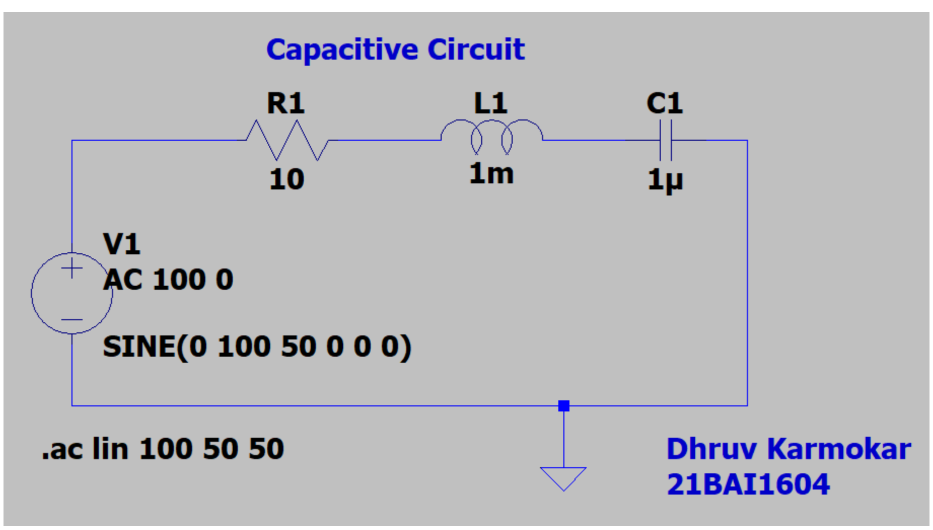
$X_C \cong X_L$, the circuit is resistive in nature.

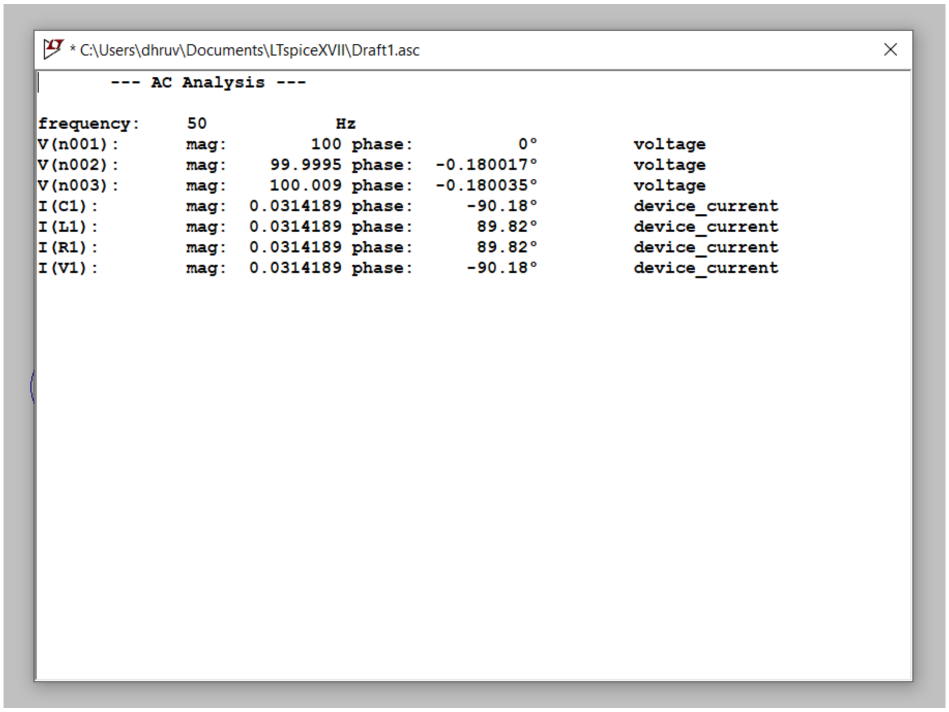
Observation table:

Parameter	Circuit behavior	Theoretical Value	Observed Value
I (Peak value)	**capacitive**	**0.0314**	**0.0314**
Phase angle		**89.82**	**89.92**
I (Peak value)	**Inductive**	**0.3214**	**0.3214**
Phase angle		**88.158**	**88.158**
I (Peak value)	**Resistive**	**9.99**	**10**
Phase angle		**0.237**	**0**

Circuits and Waveforms:

Case 1

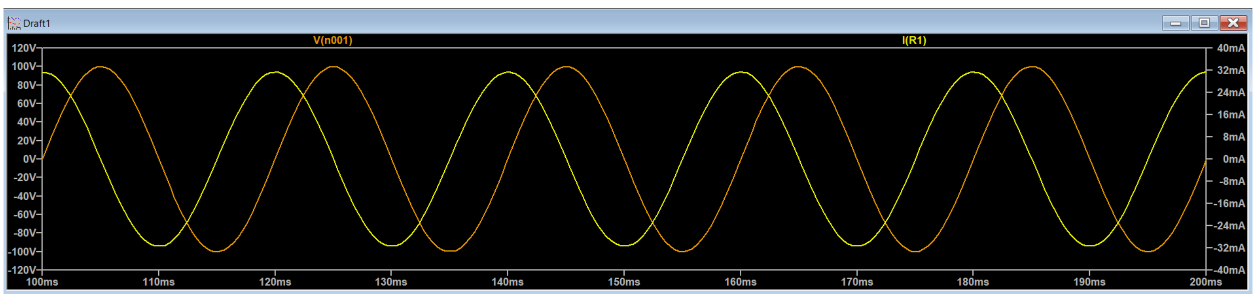




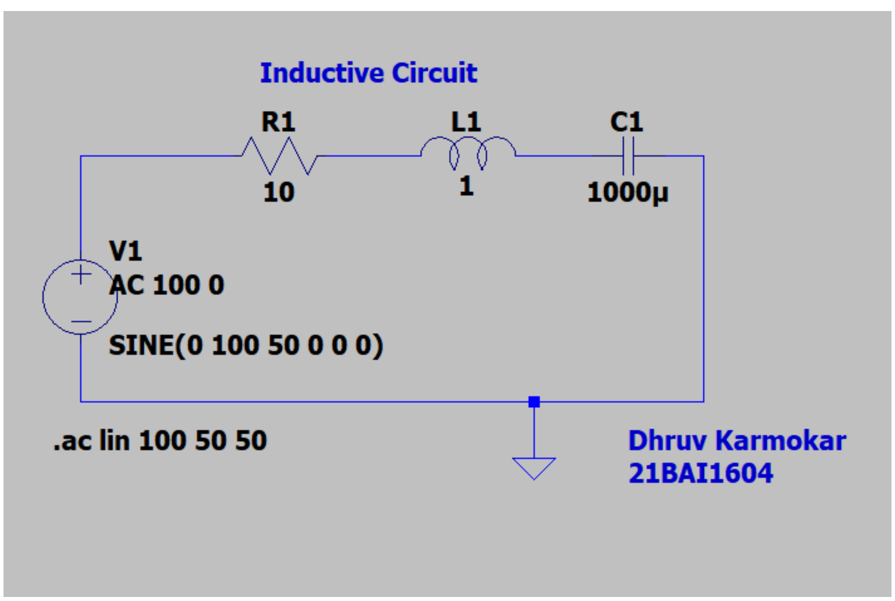
* C:\Users\dhruv\Documents\LTspiceXVII\Draft1.asc

```
       --- AC Analysis ---

frequency:    50            Hz
V(n001):      mag:      100 phase:          0°        voltage
V(n002):      mag:  99.9995 phase:  -0.180017°        voltage
V(n003):      mag:  100.009 phase:  -0.180035°        voltage
I(C1):        mag: 0.0314189 phase:    -90.18°        device_current
I(L1):        mag: 0.0314189 phase:     89.82°        device_current
I(R1):        mag: 0.0314189 phase:     89.82°        device_current
I(V1):        mag: 0.0314189 phase:    -90.18°        device_current
```



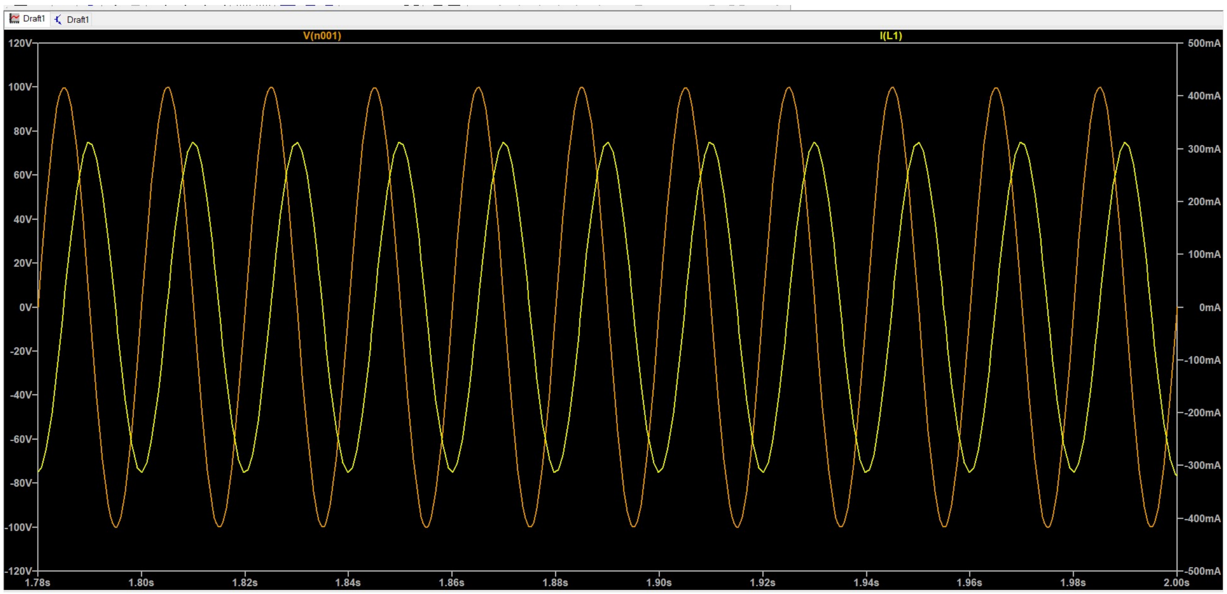
Case 2



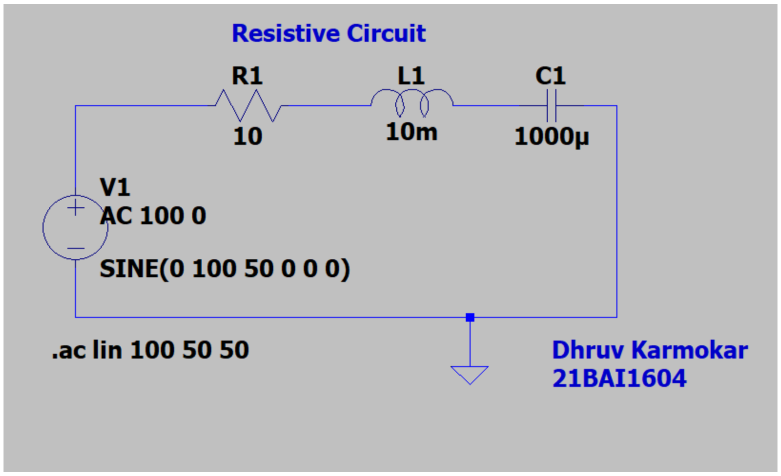
* C:\Users\dhruv\Documents\LTspiceXVII\Draft1.asc

```
       --- AC Analysis ---

frequency:     50            Hz
V(n001):       mag:        100 phase: -2.54444e-016°      voltage
V(n002):       mag:    99.9483 phase:     1.84181°        voltage
V(n003):       mag:    1.02305 phase:    -178.158°        voltage
I(C1):         mag:   0.321402 phase:     -88.158°        device_current
I(L1):         mag:   0.321402 phase:     -88.158°        device_current
I(R1):         mag:   0.321402 phase:     -88.158°        device_current
I(V1):         mag:   0.321402 phase:      91.842°        device_current
```



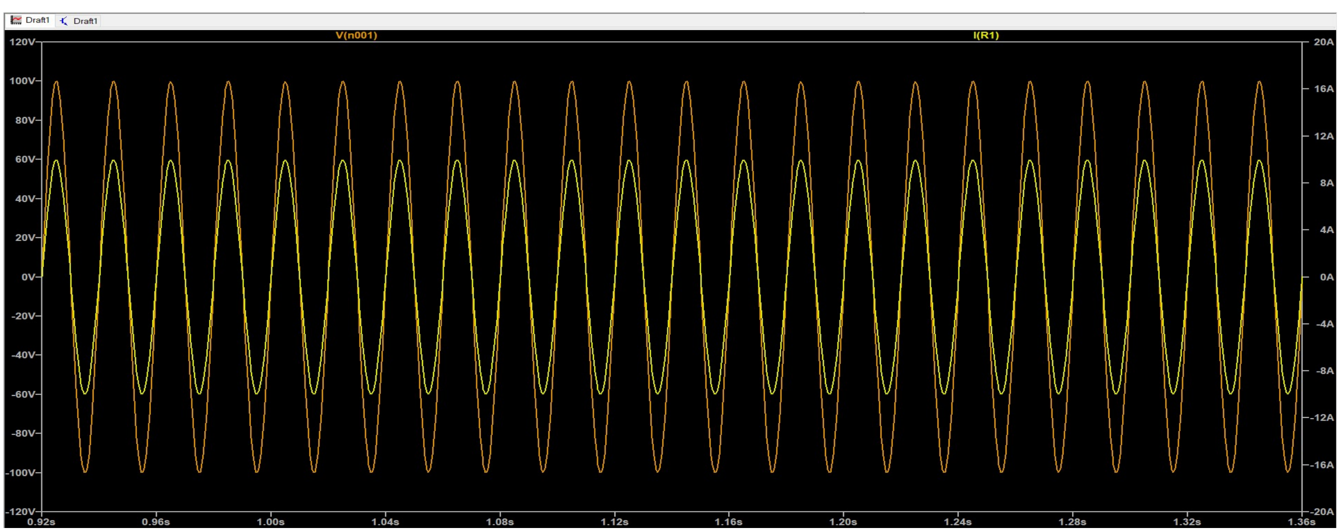
Case 3



* C:\Users\dhruv\Documents\LTspiceXVII\Draft1.asc

```
       --- AC Analysis ---

frequency:     50             Hz
V(n001):       mag:        100 phase: 3.18055e-017°       voltage
V(n002):       mag:   0.415137 phase:    -88.3821°        voltage
V(n003):       mag:    31.8275 phase:    -89.7622°        voltage
I(C1):         mag:    9.99891 phase:    0.237788°        device_current
I(L1):         mag:    9.99891 phase:    0.237788°        device_current
I(R1):         mag:    9.99891 phase:    0.237788°        device_current
I(V1):         mag:    9.99891 phase:    -179.762°        device_current
```



Result & Inferences:

Thus, a series RLC circuit has been designed and implemented in LTspice software.

Case 1: The current amplitude and phase angle is observed as 0.0314 A and 89.82° respectively which is matching with Theoretical Values. From the waveforms, it is also observed that the current leads the voltage.

Case 2: The current amplitude and phase angle is observed as 0.3214 A and 88.158° respectively which is matching with Theoretical Values. From the waveforms, it is also observed that the voltage leads the current.

Case 3: The current amplitude and phase angle is observed as 9.99 A and 0.23° respectively which is matching with Theoretical Values. From the waveforms, it is also observed that the current is equal to the voltage.

Practical Applications:

The three circuit elements, R, L, and C can be combined in several different topologies by connecting them in series or parallel. RLC circuits have many applications as follows:

- Variable tuned circuit
- Filters - Band-pass filter, Band-stop filter, Low-pass filter or High-pass filter
- Oscillator
- Voltage multiplier
- Pulse discharge circuit

Course Outcome:

CO2. Analyze AC power circuits and networks, its measurement and safety concerns

CO6. Design and conduct experiments to analyze and interpret data

Student Learning Outcomes (SLO):

SLO1. Having an ability to apply mathematics and science in engineering applications

SLO9. Having problem-solving ability- solving social issues and engineering problems