Exp No.	Title	Date
S3	**Frequency response of series RLC circuit**	**18 Sept 2019**

Objective:

- To identify the resonance frequency of the given series RLC circuit in PSPICE software.

Apparatus/Tool required:

ORCAD / Capture CIS Analog Library – R,
Source Library – Vdc, Idc
Ground (GND) – 0 (zero)

Simulation Settings:

Analysis Type – AC Sweep/Noise
AC Sweep Type – Start Frequency: 1
End Frequency: 100k
Points/Decade : 10

Theory:

Resonance in an AC circuit refers to the state of the RLC circuit in which the inductive reactance of the circuit is equal to capacitive reactance. The angular frequency at resonance state is called resonance frequency.

- Inductive reactance : $X_L = 2\pi f L = \omega L$
- Capacitive reactance : $X_C = \frac{1}{2\pi f C} = \frac{1}{\omega C}$
- When $X_L > X_C$ the circuit is Inductive
- When $X_C > X_L$ the circuit is Capacitive
- Total circuit reactance = $X_T = X_L - X_C$ or $X_C - X_L$
- Total circuit impedance = $Z = \sqrt{R^2 + X_T^2} = R + jX$

At Resonance condition,

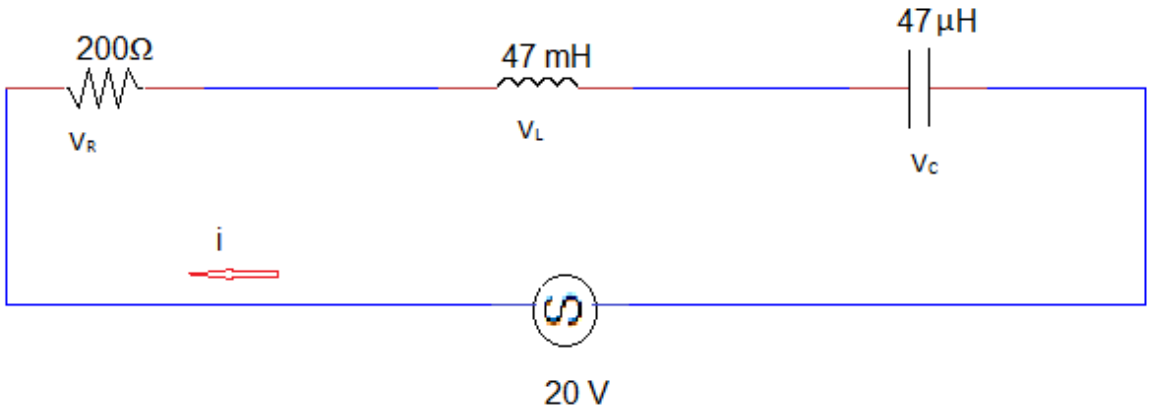
$$X_L = X_C \quad \Rightarrow \quad 2\pi f L = \frac{1}{2\pi f C}$$

$$f^2 = \frac{1}{2\pi L \times 2\pi C} = \frac{1}{4\pi^2 LC}$$

$$f = \sqrt{\frac{1}{4\pi^2 LC}}$$

$$\therefore \ f_r = \frac{1}{2\pi\sqrt{LC}} \text{ (Hz)} \quad \text{or} \quad \omega_r = \frac{1}{\sqrt{LC}} \text{ (rads)}$$

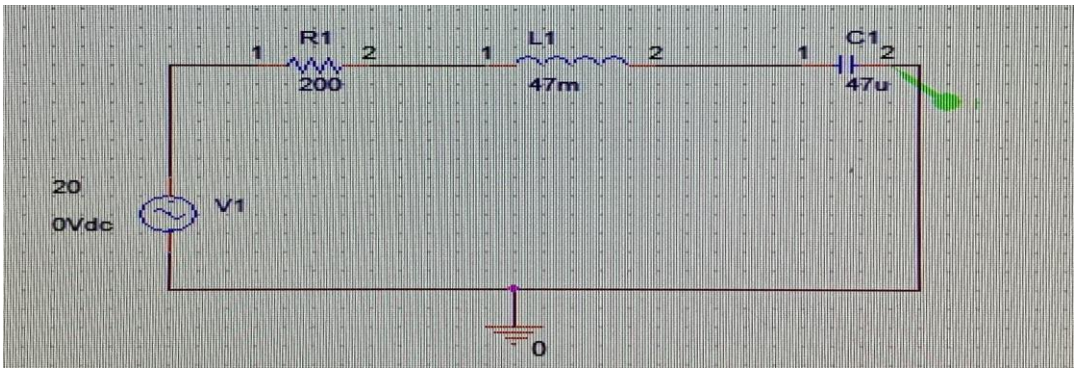
Circuit Diagram:



Procedure:

1. Create the given circuit diagram in new project file using the general procedure.
2. Replace the default component value and source value as per given circuit diagram.
3. Create the New simulation profile and set analysis type as AC Sweep/Noise.
4. Run the simulation and observe the output.
5. Calculate the bandwidth of the circuit from the graph.

Simulation Results:



Bandwidth= $f_H - f_L$
= (691.831 - 16.596) Hz
= 675.235 Hz

Conclusion & Inference:

Thus, the resonance frequency for the given circuit is identified in Pspice simulation.

Reg. No.	Name	Marks
19BCE0811	AKSHAT SRIVASTAV	