

VIRTUAL CIRCUIT SIMULATION USING LTsprice

Student Name : R.D.P UDAYANGA

Student No : EC/2021/039

Date of Performed : 05/09/2023

Date of Submission : 12/09/2023

Date : 05/09/2023
Experiment No : 08
Experiment Name : Virtual Circuit Simulation Using LTspice

APPARATUS

- A Computer
- LTspice Software

THEORY AND DIAGRAMS

Electronic circuit simulation uses mathematical models to replicate the behavior of an actual electronic device or circuit. Simulation software allows circuit operation modelling and is an invaluable analysis tool. In particular, for integrated circuits, the tooling is expensive, breadboards are impractical, and probing the behavior of internal signals is extremely difficult. Therefore, almost all IC design relies heavily on simulation. The most well-known analogue simulator is SPICE.

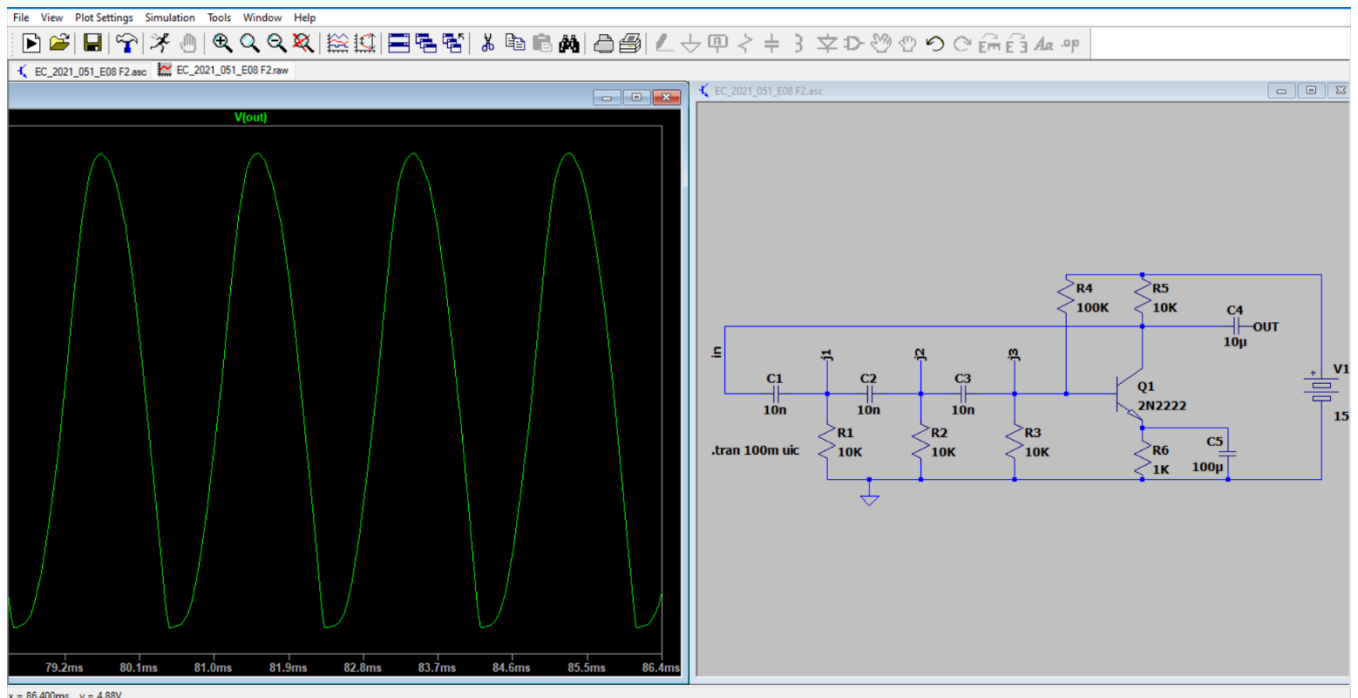


Figure 1: Snapshot of LTspice circuit simulator

Some electronics simulators integrate a schematic editor, a simulation engine, and an on-screen waveform display, as shown in Figure 1, allowing designers to rapidly modify a simulated circuit and see what effect the changes have on the output. They also typically contain extensive model and device libraries. These models typically include IC-specific transistor models, generic components such as resistors, capacitors, inductors and distorters and user-defined models such as controlled current and voltage sources.

Ltspice is a SPICE-based analogue electronic circuit simulator computer software produced by semiconductor manufacturer Analog Devices (originally by Linear Technology). It is the most widely distributed and used SPICE software in the industry. Though it is freeware.

Ltspice is not artificially restricted to limit its capabilities (no feature limits, node limits component limits, or subcircuit limits). It comes with a library of SPICE models from Analog.

Devices, Linear Technology, Maxim Integrated, and third-party sources.

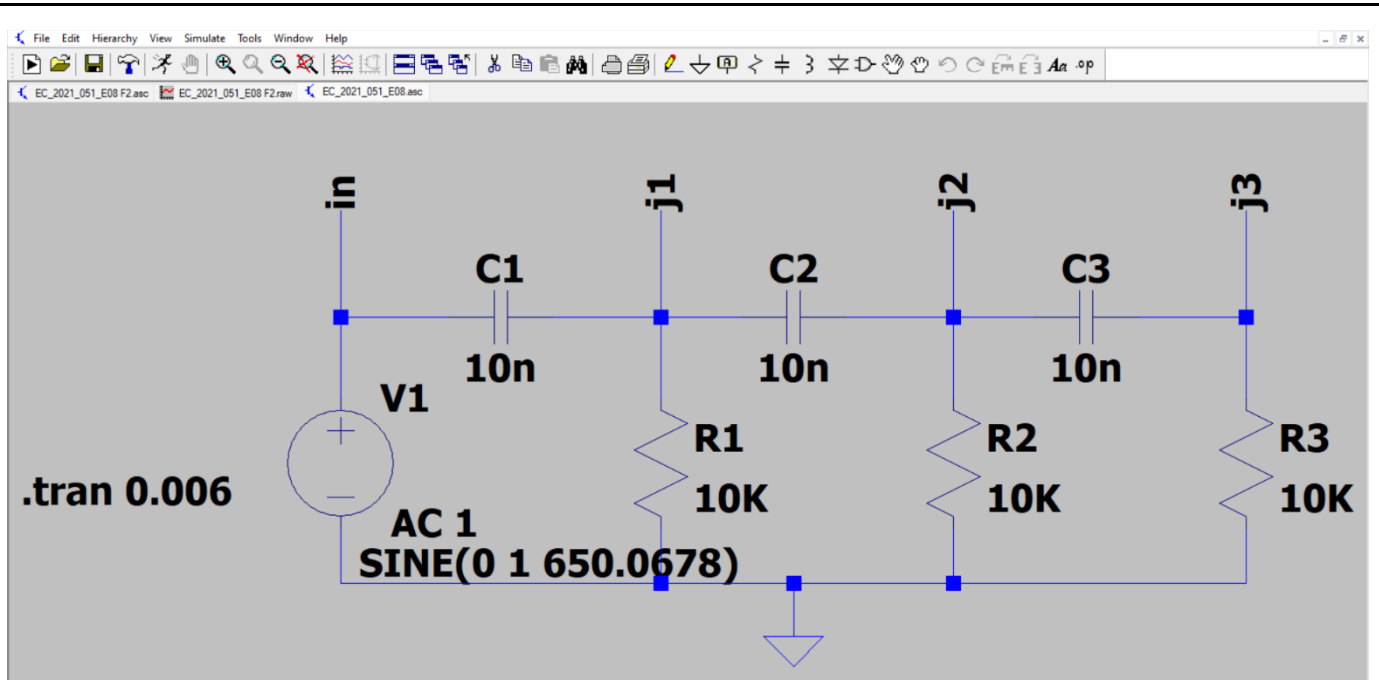


Figure 2: RC Ladder Network

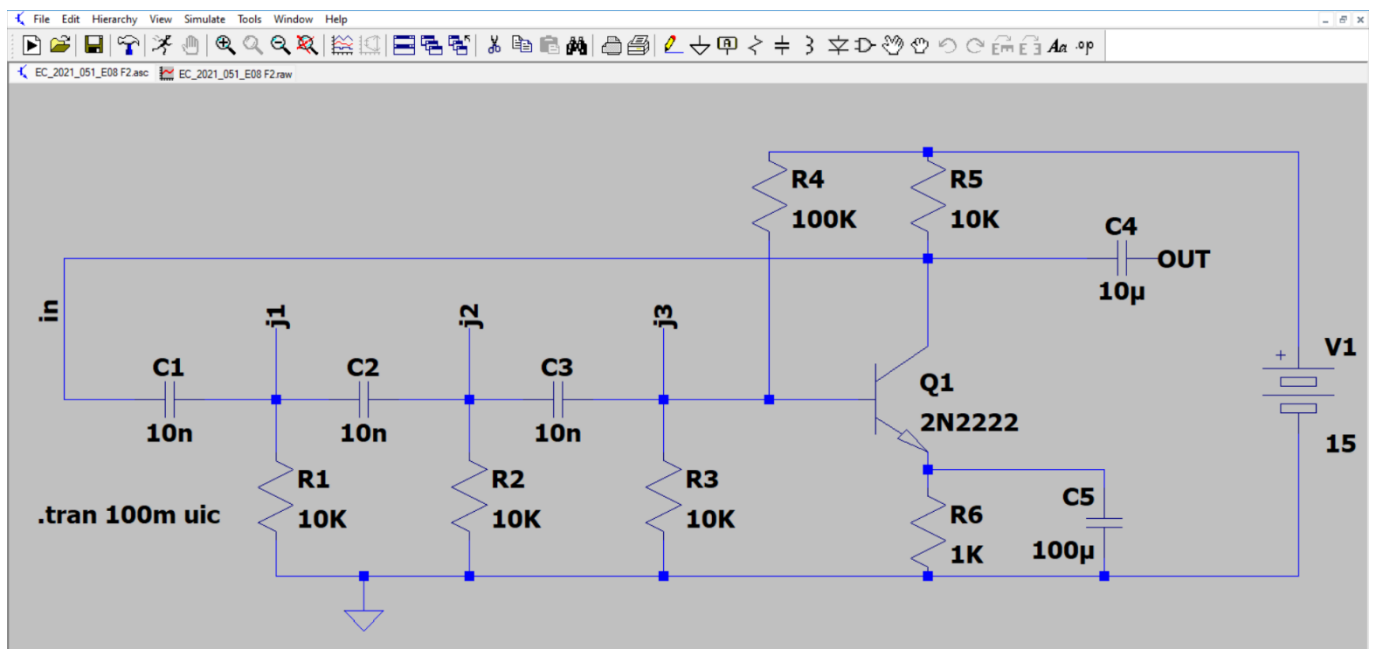
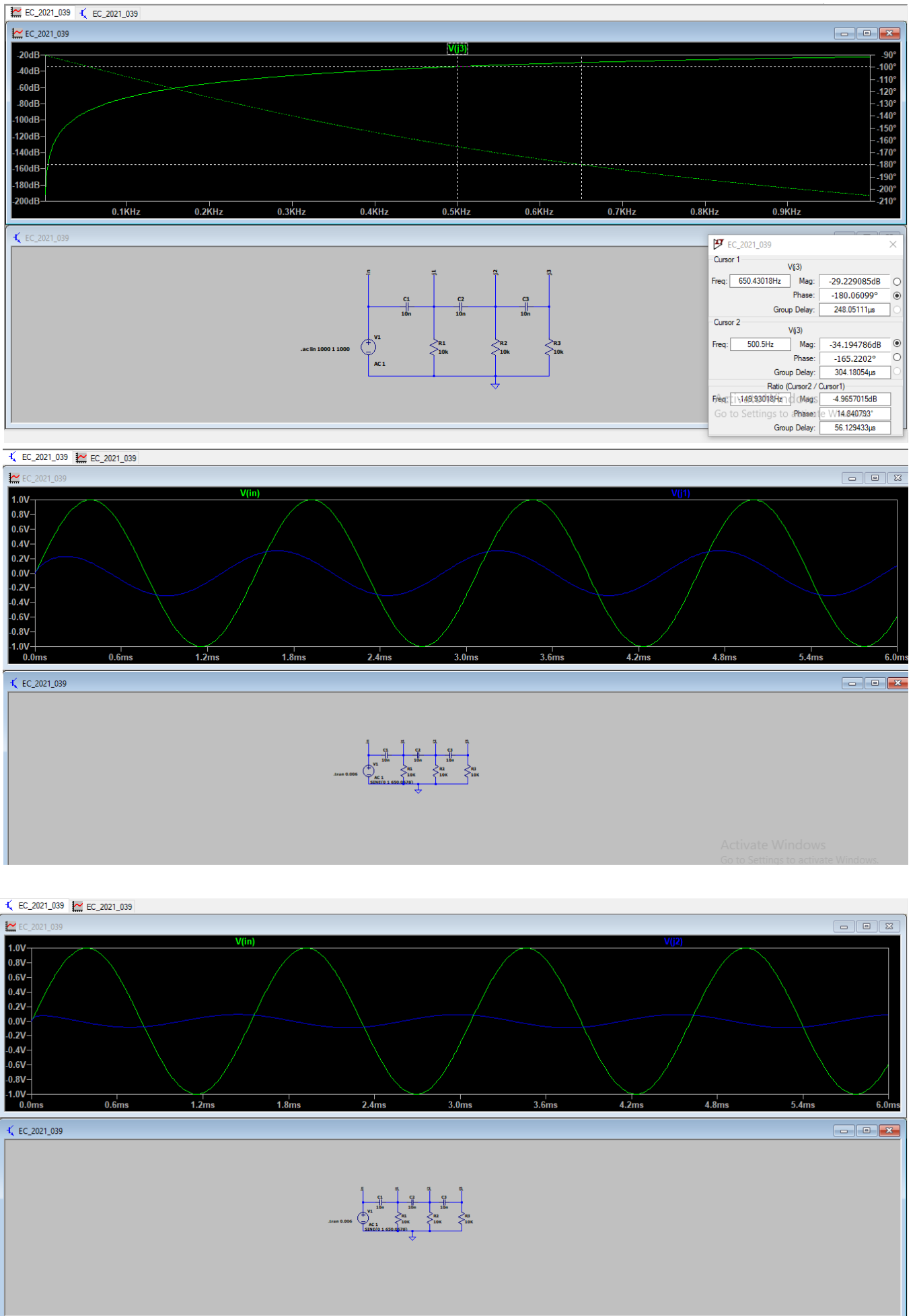


Figure 3: RC Oscillator Circuit

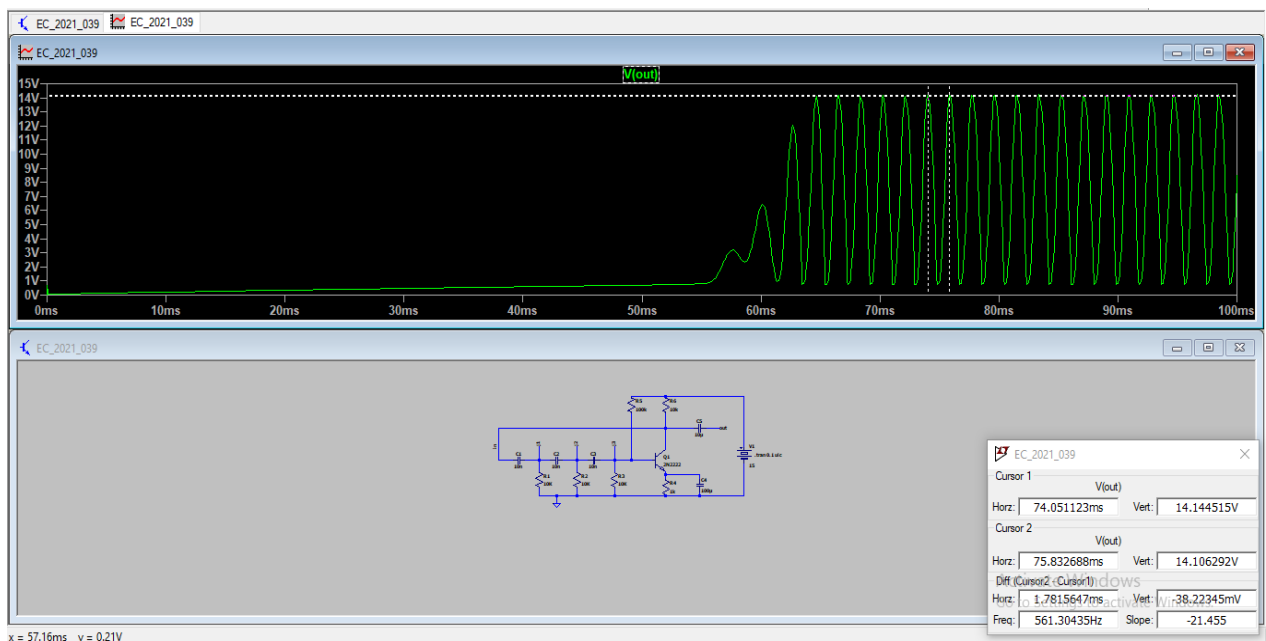
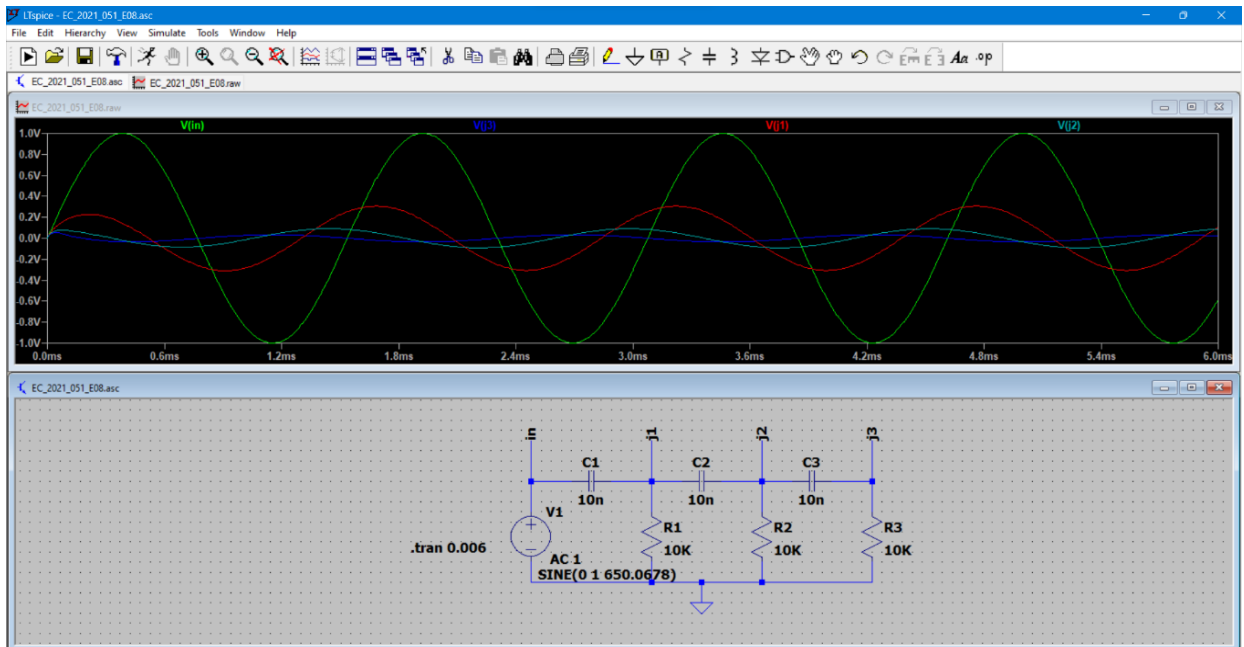
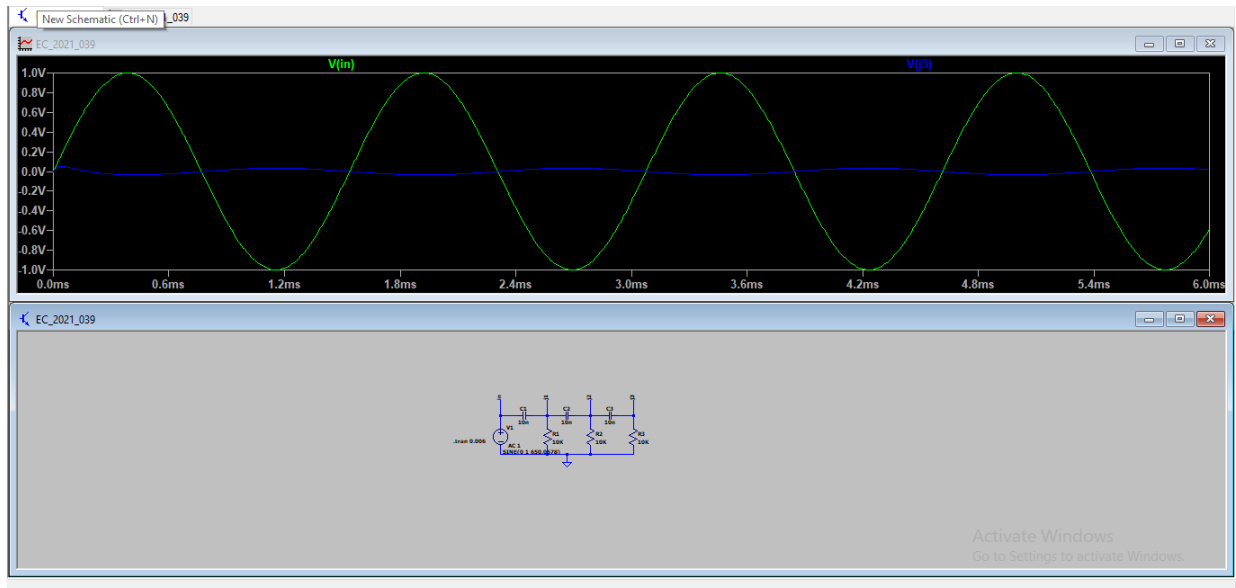
PROCEDURE

1. LTspice software was installed and opened.
2. The guidelines were followed to design the circuits by LTspice.
3. Figure circuit was designed according to the instructions.
4. The voltage source was designed to supply a sinusoidal signal with and 1V amplitude.
5. After running the simulation “j3” node was clicked on and the graph window was opened.
6. The frequency was found the designed RC ladder circuit with 180° phase shift of the output and a screenshot was took.
7. The properties were adjusted to provide a sinusoidal signal and the simulation was run.
8. Both “in” and “j3” labels were clicked on, and input output waveforms were observed. (180° Phase shift)
9. Screenshots were taken of the waveforms of an “in”, “j2” and “j3” labels.
10. Now to check the validity of the measured frequency value, a sinusoidal signal was provided by us with the measured frequency and the output waveforms were observed.
11. A new schematic was created, and the RC oscillator was constructed shown in figure 3.
12. The RC oscillator circuits was simulated.
13. The voltage at the node labeled was observed as out in figure 3.
14. The period of output waveform was measured when it reached steady state and the frequency was calculated.

OBSERVATION



OBSERVATION



CALCULATIONS

The theoretical value of frequency

$$\begin{aligned}F_r &= \frac{1}{2\pi RC\sqrt{2N}} \\&= \frac{1}{2 \times \frac{22}{7} \times 10 \times 10^3 \times 10 \times 10^{-9} \times \sqrt{2 \times 3}} \\&= 0.065 \times 10^4 \\&= 650\text{Hz}\end{aligned}$$

CONCLUSION

	Theoretical Value		Experiment Value	
	Time period (s)	Frequency (Hz)	Time period (s)	Frequency (Hz)
RC Oscillator Circuit				

DISCUSSION

DISCUSSION