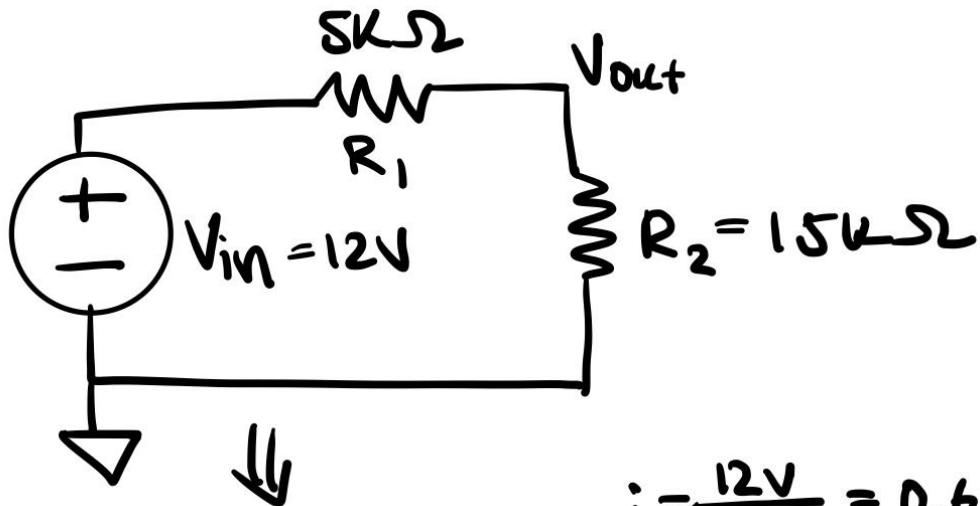


ECE 10BL- Lab 0

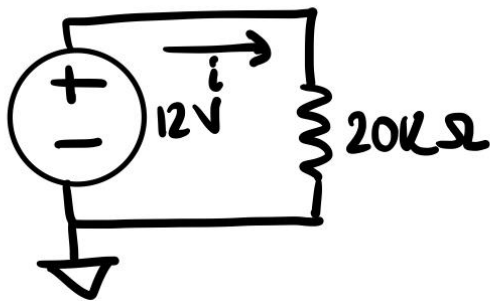
**Introduction:**

The goal of this lab is to get familiar with the LTspice software, which allows users to simulate analog electronic circuits. We will analyze the simple circuit by hand then use the simulator to check if our results match.

**Step 1:**



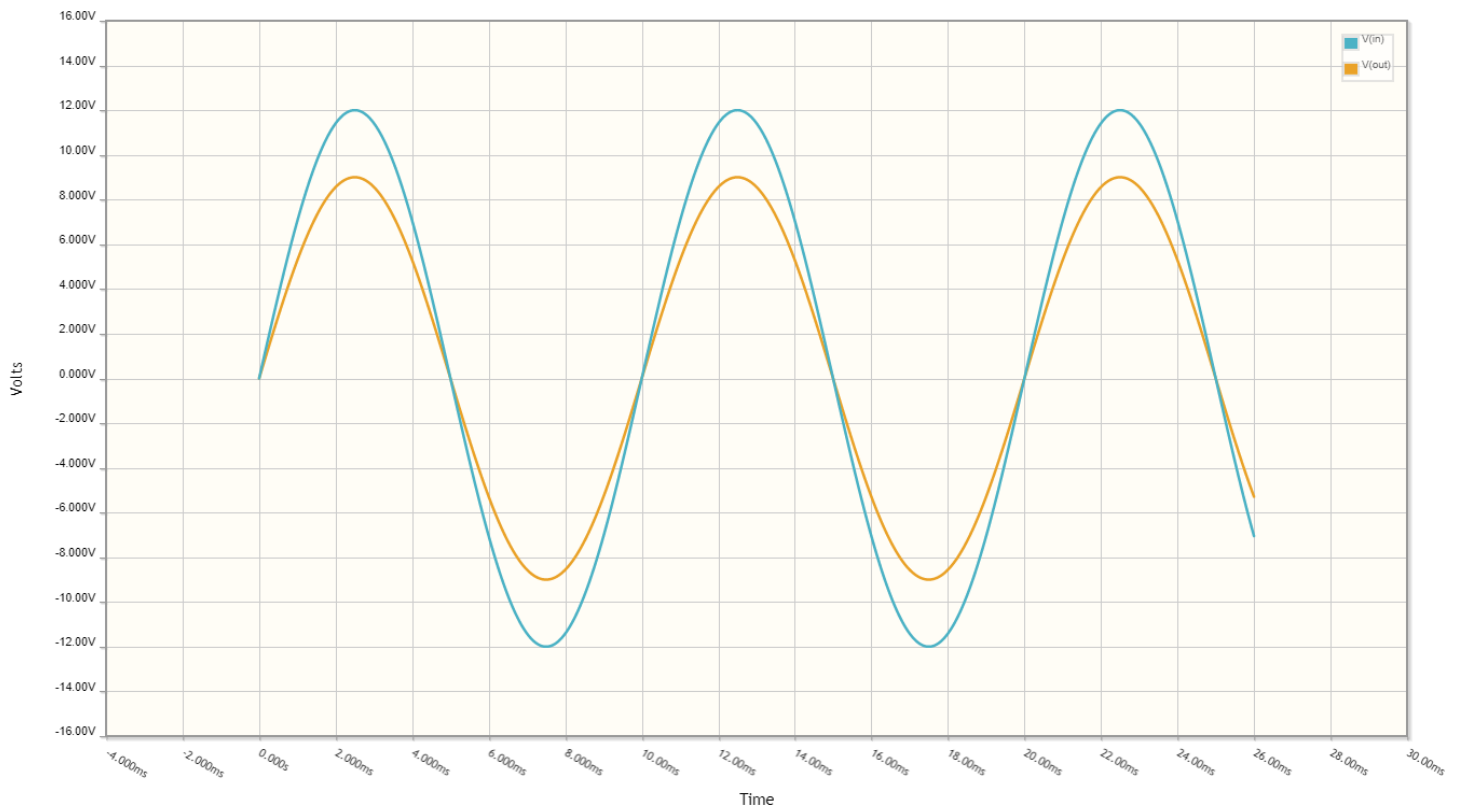
$$i = \frac{12V}{20000\Omega} = 0.6mA$$



$$\begin{aligned} V_1 &= R_1 \times i \\ &= 5k\Omega \times 0.6mA \\ &= 3V \end{aligned}$$

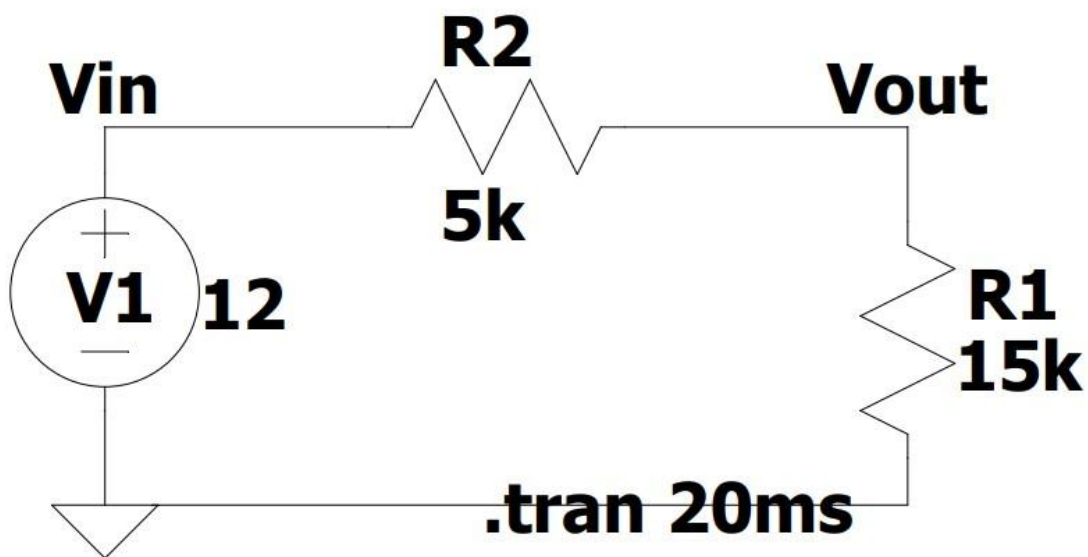
$$\begin{aligned} V_{out} &= V_{in} - V_1 \\ &= 12 - 3 \\ &= 9V \end{aligned}$$

[RP1]

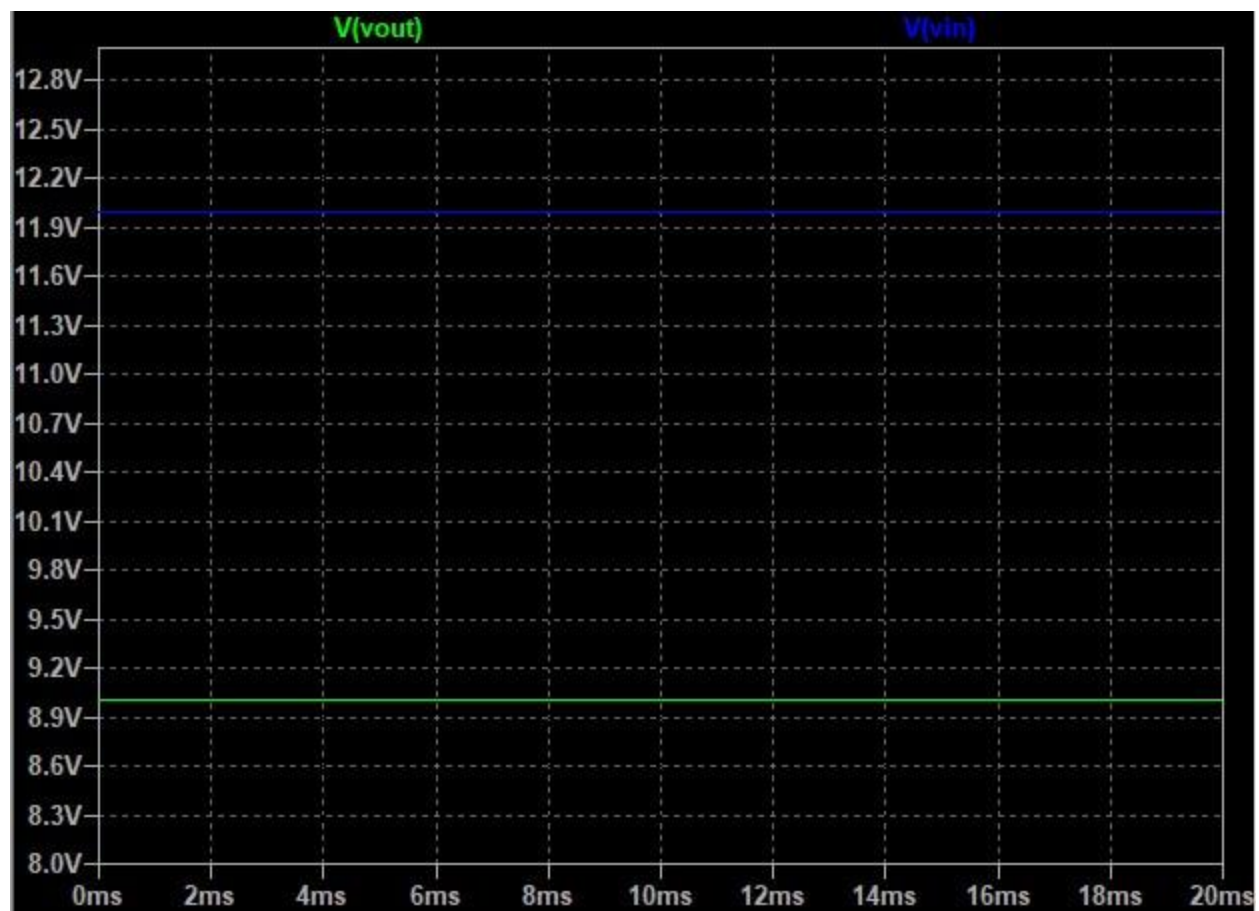


[RP2] Blue-  $V_{in}$  , Orange- $V_{out}$

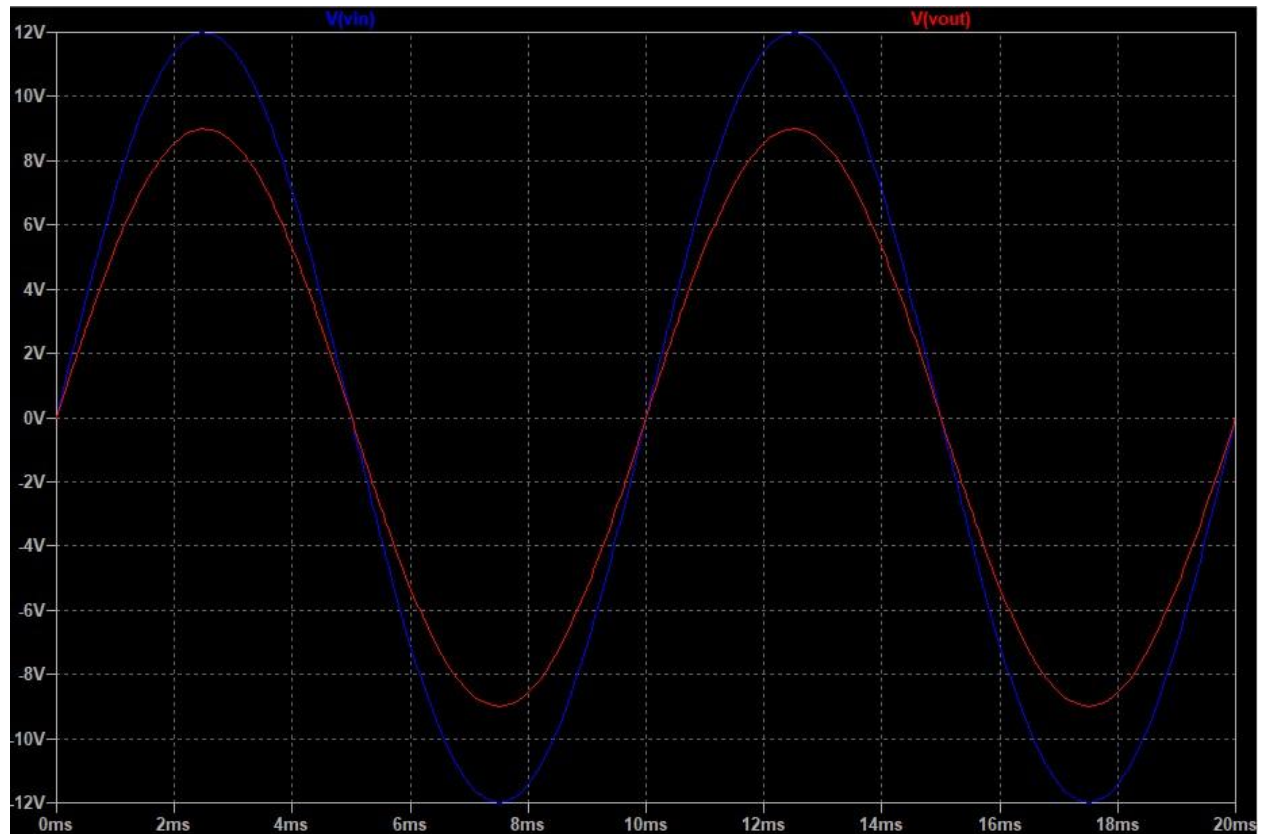
Step 5:



[RP3]

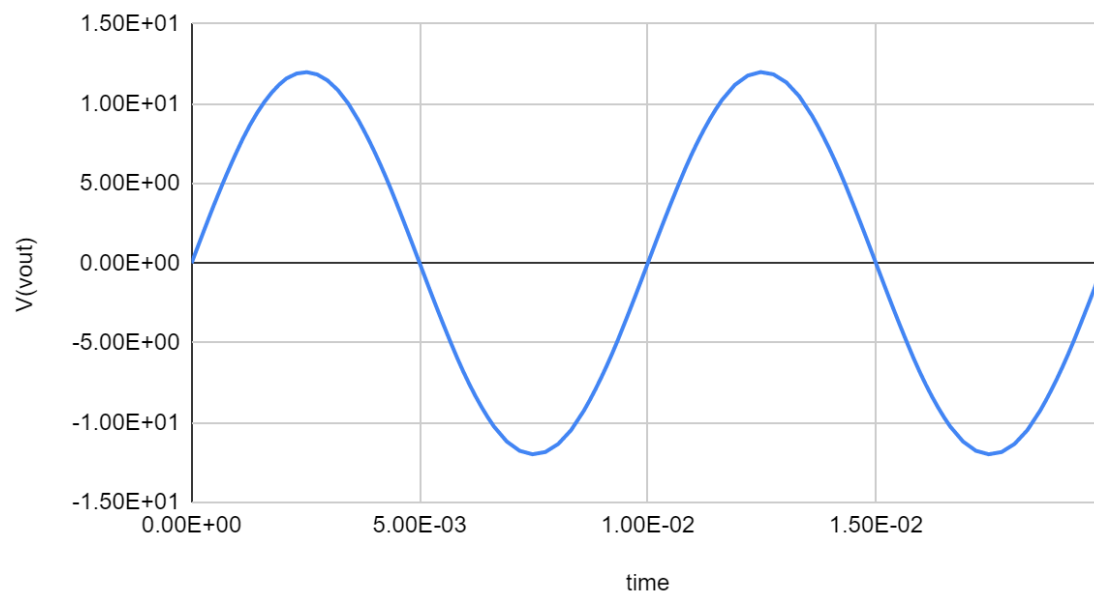


[RP4]



[RP5] Blue-  $V_{in}$  , Red- $V_{out}$

$V(out)$  vs. time



[RP6]

**Conclusion:**

This lab familiarized us with the LTspice software by guiding us to create a schematic and use the run functionality to plot a waveform graph. I was able to check if my analysis for  $V_{out}$  being 9V was correct by drawing the schematic and utilizing its graph to confirm my answer. I was also able to check my graph for the sinusoidal voltage source graph with the graph the simulator gave after I updated the schematic to incorporate the sinusoidal voltage source.