EXPERIMENT II

ANALYSIS OF SIMPLE ELECTRICAL CIRCUITS USING KIRCHHOFF's and OHM's LAWS

OBJECTIVE: In this experiment, you will learn:

1. Voltage and current dividers
2. Series-parallel combination of resistors in circuits
3. Application of Ohm's Law, Kirchhoff's Laws, and superposition principle in analyzing and characterizing circuits with multiple sources of AC and DC type.
4. Wheatstone bridge circuit.
5. Basic use of voltage supplies, function generators (FG), multi-meters, and digital oscilloscopesin validating operation of circuits with DC and AC signals.
6. X-Y mode and math operations in digital oscilloscopes.

EQUIPMENT LIST:

Power Supply
Digital Oscilloscope
Function\Arbitrary Waveform Generator
Resistors (100Ω, 6 x 1kΩ, 3.3kΩ, 2 x 1.2kΩ, 6.8kΩ, 10k pot).

PRELIMINARY WORK:

1. **Voltage and Current Dividers**

   A. Given the following circuit with no component connected initially between a and b:

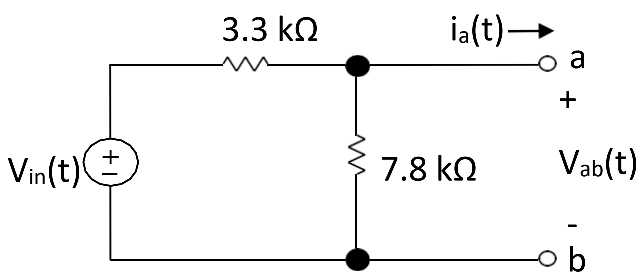

Figure 2.1

   a) How would you obtain **7.8kΩ** using the available components in the equipment list? Sketch and explain.
   b) Use the principle of voltage division to calculate $\boldsymbol{V_{ab}(t)}$ in terms of $\boldsymbol{V_{in}(t)}$. Show your calculation.

   B. Add a 5 kΩ resistance between output nodes a and b as shown below:

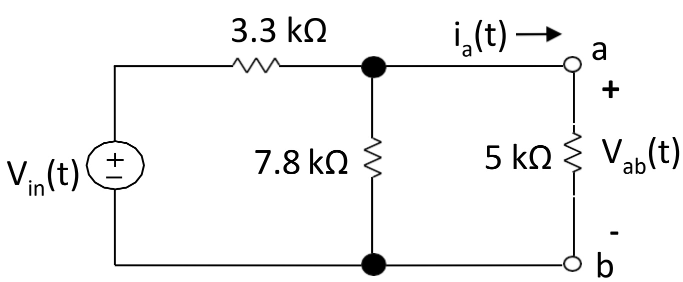

Figure 2.2

a) What is the new equivalent resistance, R_{ab} between nodes a and b? Show calculation.
b) Use the principle of current division to calculate $i_a(t)$ in terms of $V_{in}(t)$. Show your calculation.
c) Also calculate $V_{ab}(t)$ in terms of $V_{in}(t)$ using your result from (b).

C. Based on your observations from Exercise 1 (A) and (B) above, explain if one can use a voltage divider circuit of Figure 2.1 as a voltage source to deliver power to a load connected between nodes a and b. Explain your reasoning carefully. Hint: Think of the properties of an ideal voltage source.

2. Circuit with multiple sources

Given the following circuit with a sinusoidal AC voltage supply and a DC voltage supply:

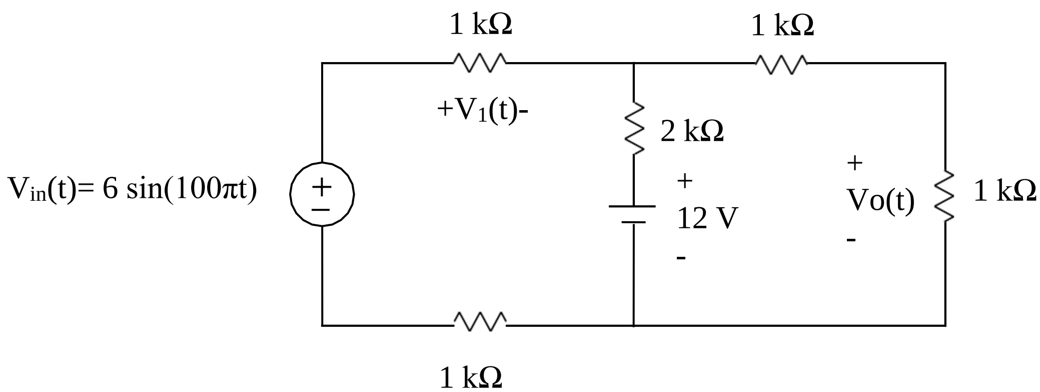

Figure 2.3

A. What is the frequency of the sinusoidal voltage signal in Hertz?

B. Analyze the circuit on paper using the **superposition principle**, derive $V_o(t)$ and $V_1(t)$ voltage waveforms. And sketch them on a graph with ***t*** as the x-axis and ***voltage*** as the y-axis.

C. **(LTspice)** Find $V_o(t)$ and $V_1(t)$ voltage waveforms in the above circuit using LTspice transient simulation. You can declare the negative terminal of the DC voltage source to be the GND node. You may use the SINE function for a 'voltage' source to declare a sinusoid voltage. You will need to specify Amplitude, Frequency, DC offset of the waveform, and possibly number of cycles. The simulation should last for about 50 ms. Add your LTspice schematics and the simulation result printouts to the prelab report. How do the results compare to your hand calculations in (B)? Comment.

[Note: You can probe the two node voltages on the terminals of the relevant resistors and use the waveform subtraction function in the simulator in order to get $V_o(t)$ and $V_1(t)$. Under 'Plot Settings' click on 'Add a Trace'; choose a node voltage, enter '-' in the expression entry, and choose another node voltage to plot the potential difference between two nodes].

3. Wheatstone Bridge

The circuit given in Figure 2.4 below is called the Wheatstone bridge circuit. The detector in this circuit may be an ammeter or voltmeter. You may use the same AC voltage source as in Exercise 2 above.

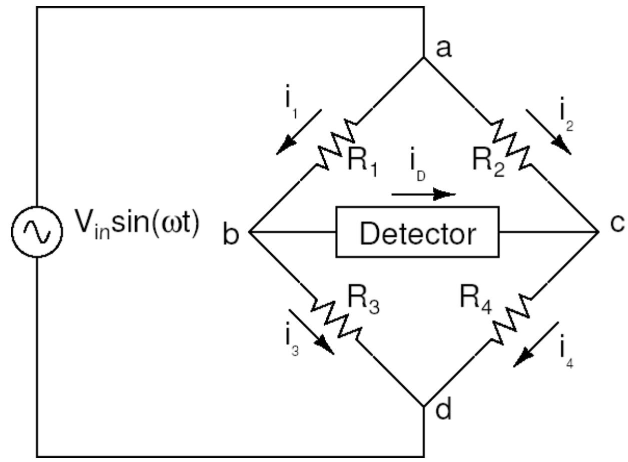

Figure 2.4

A. Derive the relationship among R_1, R_2, R_3, and R_4 such that V_{bc}=0V and i_D=0 A, in other words detector will display zero.

B. **(LTspice)** Construct the circuit in Figure 2.4 and leave the detector portion as open circuit to be measured in simulation. Change the values of the resistors as follows: R_1=2 kΩ, R_3=3.3 kΩ, R_4=1 kΩ and define R_2 as a parameter Rx. You can do this by entering {Rx} (curly brackets are important) instead of a resistor value for R2 component. Define a 20ms transient simulation. You can repeat the same simulation for many values of the declared parameter by using the **.step** spice directive (Edit → spice directive). For example to step the parameter value between 0.2K and 2K with 0.1K resolution, you can enter the following as a spice directive and instantiate the statement somewhere in the schematic window:

.step param Rx 0.2K 2K 0.1K

After running the simulation, can you explain the condition under which the detector will read zero voltage difference between nodes b and c? Check your simulation result against what you obtain with calculation in part 3.A above. Add your LTspice schematics and the simulation results printouts to the prelab report annotating critical observations on the simulation results.

EXPERIMENTAL WORK:

IMPORTANT REMARK: Set the function generator to the "High Z" mode during the experiment, unless otherwise stated.

1. Voltage and current dividers

Setup the circuit of Fig. 2.2, recalling the operation of the function generator and oscilloscope from Laboratory 1. You may use the pot for any of the resistor values you do not have readily available from your equipment list.

a. Measure the equivalent resistance $\boldsymbol{R_{ab}}$. (Remember resistance measurements are done when there is no power in the circuit!) Then set $V_{in}(t)= 6\sin(100\pi t)$ using the function generator as an AC voltage source. Make the necessary probe connections and oscilloscope settings in order to observe $V_{in}(t)$ on Channel 1 and $V_{ab}(t)$ on Channel 2 simultaneously on the oscilloscope screen. After making the appropriate calculations for RMS values of all AC signals, fill in Table 1.

Table 1

R_{ab} Calculated from Prework 1.B(a) (kΩ)	$I_{a,rms}$ Calculated from Prework 1.B(b) (A)	R_{ab} Measured (kΩ)	$V_{ab,\,rms}$ Measured (V)	$I_{a,rms}$ Measured (A)

b. Press the "Main/Delayed" button on the front panel of the oscilloscope. From the appearing menu, press the soft key corresponding to 'Time – Base' to switch it from 'Y-T' mode to 'XY'. This mode is known as the "XY mode" and used for plotting the signal in Channel 2 (Y) as a function of the signal in Channel 1 (X). Draw the observed figure.

2. Circuit with multiple sources

Setup the circuit of Fig. 2.3. You will have to use both the function generator and the DC power supply to obtain AC and DC sources, respectively.

a. Make the probe connections of the oscilloscope as shown in Figure 2.5 and observe $V_o(t)$. Plot the waveform and compare it with your result in the preliminary work for three cases: i. When the 12 V DC supply is TURNED OFF, ii. when the function generator is TURNED OFF, iii. when both the DC supply and function generator are TURNED ON.

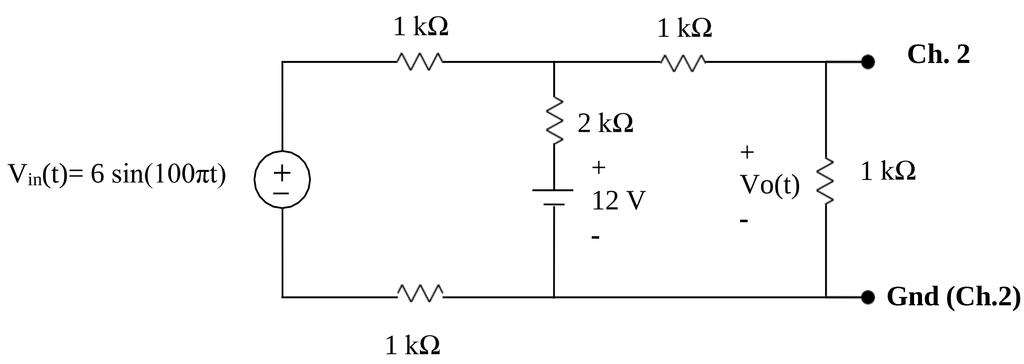

Figure 2.5

Are your plots consistent with the superposition principle? Comment on why.

b. Now, connect channel 1 to the circuit as shown in Figure 2.6.

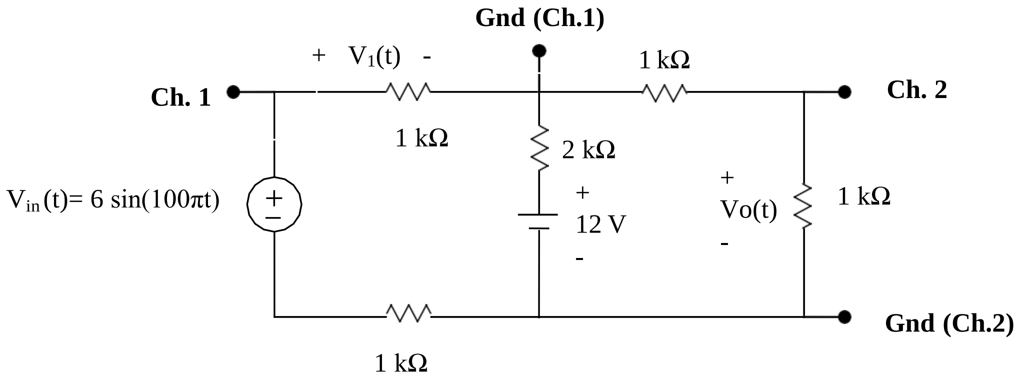

Figure 2.6

Observe the changes in the signal in Channel 2. Also compare the signal you observed in Channel 1 with the one you found in the preliminary work.

The reason of the differences in the practical and theoretical results is the fact that ground leads of both probes are connected to each other through the inside of the oscilloscope. Therefore, when you connect the ground leads of two probes to different nodes in a circuit, these two nodes will be shorted and you will observe a zero voltage signal between these two nodes.