ELEC2070 2023
Laboratory Worksheet – Weeks 3 and 4
Introduction to Practical Measurements and LT Spice Simulation

***Abstract*— In this laboratory session of two weeks, practical voltage, current and resistance measurements will be investigated and compared to ideal (or theoretical) measurements. Simple equivalent circuits of the laboratory equipment will be determined for this purpose and in preparation for the upcoming laboratory sessions. LT Spice simulation will be introduced. Complete up to and including Section VI in Week 3.**

I. Preliminary Work

Week 3: Study the lecture notes and the other relevant resources to remind yourself of the methods of measuring voltage and current. Draw a diagram of a multimeter made out of a simple galvanometer in your logbook and show its basic functions.

Week 4: Study the lecture notes and the other relevant resources to become familiar with Thévenin and Norton circuits and power transfer [1-2]. Solve Problem 5.5-11 [1] in your logbook, then calculate the power that would be delivered to a 2 Ω resistor if it is connected between the terminals a and b.

II. Time Management

Look through the whole worksheet before you start and plan your time so that you can complete the work in two weeks. Write your plan in your logbook. At the end of the two weeks compare your actual times to your initial estimates. The aim for this exercise is for you to improve your estimation gradually.

III. Logbooks and Participation Marks

Participation marks are given to each student at each laboratory session, following the marking rubric published on iLearn. The marks cover attendance, punctuality, positive engagement and logbook recording of each session. All students should have their logbooks signed and dated by their tutor before leaving each laboratory session.

IV. Digital Multimeter

1. Study the multimeter/s available in the laboratory. Find its specifications either on the web or on iLearn. Note the specifications may be different for each measurement type. Record the ranges of available measurements (voltage, current, resistance).
2. Draw equivalent circuits for the voltmeter and ammeter functions of the multimeter as in Fig. 1 and Fig. 2. <u>Use the correct resistor value or a correct estimate</u>.

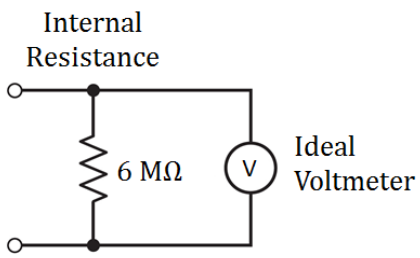

Fig. 1. Equivalent circuit for the practical voltmeter

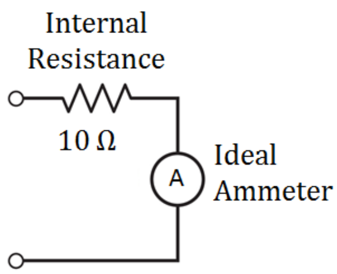

Fig. 2. Equivalent circuit for the practical ammeter

3. The types of equivalent circuits shown in Fig. 1 and Fig. 2 are better for our purposes than the circuit diagram of the multimeter made out of a galvanometer. Why?

V. DC Voltage Source

4. The laboratory power supply PST-3202 is a dc voltage source with an adjustable current limit. But if the current limit is exceeded it acts as a current source. In other words, it is a voltage source for smaller currents only. Test this feature after building your circuit in the next section.
5. Draw an equivalent circuit of the dc voltage supply as shown in Fig. 3. The internal resistance of PST-3202 is less than 1 Ω therefore in most cases you can ignore it. <u>Use the correct resistor value or a correct estimate</u>.

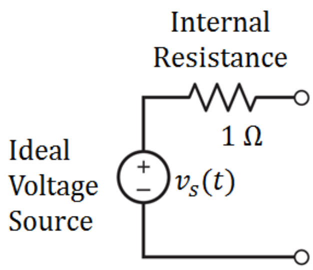

Fig. 3. Equivalent circuit for the practical voltage source

VI. Resistor Circuit

6. Design a circuit with three resistors available in the laboratory and one dc voltage source. Use at least one parallel and one series connection of resistors. Choose resistor values between 1 kΩ and 10 kΩ and a voltage less than 5 volts. Calculate the voltages and currents of your circuit in your logbook.
7. Measure and record the resistance values of your resistors using the multimeter. Build the circuit on a breadboard. (Do a web search and find how the breadboard is connected inside if you don't know this already.)
8. Measure and record the voltages and currents of the circuit. Do not exceed the current limit of your current meter. How do your measurements compare to the calculations? What kind of difference do you expect under the conditions of your experiment? Use equivalent circuits of the meters to answer this.
9. What would you measure if you were using resistors in the range of 1 MΩ to 10 MΩ in your circuit? Try this experimentally if you have enough time. Why does this happen? (Hint: Use the equivalent circuit of the multimeter in addition to your circuit.)

VII. LT Spice Simulation

10. Start LT Spice program by following the instructions given in iLearn.
11. Make a circuit representing your resistive circuit from the previous section.
12. Do a bias point analysis to simulate the voltages and currents of the circuit and compare to calculations and measurements.
13. LT Spice is free to download for your home computer and you may use it for checking your calculations at home or for drawing circuit diagrams for the laboratory reports. Investigate a method for transferring your circuit diagram into a text file, for example by printing it into a PDF file.

VIII. Oscilloscope

14. An oscilloscope measures and displays voltages continuously with respect to time. Therefore, it can be considered as a voltmeter with an equivalent circuit similar to Fig. 4. Note that it also has an internal capacitance (in parallel with the internal resistance). The values of the internal resistance and capacitance are written on the front panel (also in the datasheet). Draw the equivalent circuit of the oscilloscope and record its limitations (frequency, voltage amplitudes) by checking the front panel and the datasheet.

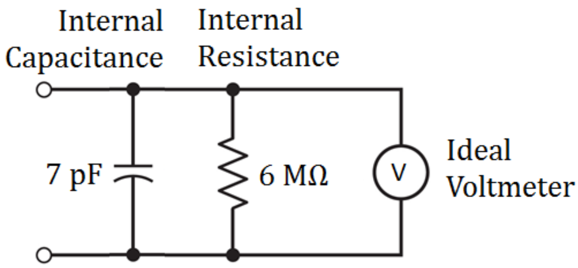

Fig. 4. Equivalent circuit for the oscilloscope

15. Note the oscilloscope (and most other ac equipment) has coaxial inputs. Draw a diagram of a coaxial connector and mark the signal and ground connections.
16. The oscilloscope connections can be made by either standard coaxial cables or special oscilloscope probes. Investigate the oscilloscope probes available in the laboratory. They have their own internal resistance and capacitances. If you use them you need to modify your equivalent circuit for the oscilloscope accordingly. Some probes have gain and require you to change the setting of the oscilloscope. Note that the oscilloscope probes are one directional and they cannot be used as source cables to bring the source signal to your circuit.

IX. Ac Signal Generator

17. Read the specifications of the waveform generator either on the web or on iLearn and draw its equivalent circuit as in Fig. 3. The term "impedance" is similar to resistance for ac signals. Record the limitations of waveform generator for its sinewave function (frequency and amplitude).
18. Set the waveform generator for a 1kHz sinewave with 2V peak-to-peak amplitude.
19. Measure its output with the oscilloscope using a coaxial cable. It is easier to measure peak-to-peak voltages of sinewave signals with an oscilloscope, why? How would you calculate the amplitude of this sinewave?
20. Compare the peak-to-peak value measured by the oscilloscope and the amplitude setting of the signal generator. Try to explain this discrepancy yourself first and ask tutors for the correct explanation. (Hint: Use the equivalent circuits.) Record the correct explanation in your logbook and highlight it for easier finding in the future. This is important!
21. You can test the validity of this explanation by connecting a 50Ω resistor between the signal and ground connections of the signal generator (use the breadboard) and measuring the voltage waveform with the oscilloscope.

X. LT Spice Simulation for Finding Thévenin Equivalent Circuit with Dependent Source

22. Study the ORCAD solution to Example 5.8-1 [1] (a scanned copy is posted on iLearn) on how to simulate the dependent sources with PSpice. Solve the preliminary work question (Week 4) with LT Spice.

23. Solve one your own questions from Assignment 1 or Test 1 with LT Spice.

XI. Preparation for Next Laboratory Session on Capacitors and Inductors

24. Investigate the type of capacitors and inductors available in the laboratory, and the range of their values. They are labeled in different ways. Search the internet on how to read them. Measure the capacitors with the multimeter.

References

[1] J. A. Svoboda, R. C. Dorf, "Introduction to Electric Circuits 9th edition," Wiley, 2014.
[2] A. R. Hambley, "Electrical Engineering, Principles and Applications, International Sixth Edition," Pearson, 2014.