

Lab 1: Introduction to TCO laboratory practices: measurement of voltages and currents in direct current (DC) circuits with resistors.

Objectives:

At the end of this lab, the student should:

- Know and understand the basic operation of a digital multimeter for the measurement of DC voltages and currents, and resistors.
- Know and understand the basic operation of the DC power supply.
- Become familiar with the rest of the lab instruments, particularly with the connection board, and the different elements of connection (wires and probes).
- Know a CAD (Computer aided design) tool for electronic circuit simulation.
- Learn the basics of circuit simulation with PSpice.
- Analyse simple circuits with resistors using PSpice.
- Compare the actual behaviour of the circuits with its corresponding simulated behaviour.

Material and equipment

Components	Instrumentation
3 Resistors of 100 Ω 2 Resistors of 220 Ω	Digital multimeter Power Supply Connection board PC with simulation software Orcad PSpice for Windows,. There is a student version in PoliformaT.

Previous work

Before this practice, you should read the document "Introduction to PSpice", available in the folder "**Recursos / Documentación Prácticas**" of TCO subject in PoliformaT.

- Moreover, some videos with information and instructions about the instruments to use during the lab are also available within the folder "**Recursos / Objetos de Aprendizaje Polimedia**" of TCO subject in PoliformaT.

Development of the Lab

1) Recognition of passive components

a) Identify the resistors used in this practice and learn, from the colour code marked on the component, the nominal value and tolerance of each one. As an alternative to manual measurement of resistance values, you can use the following applet:

(<http://www.ee.upenn.edu/rca/calcjs.html>).

b) Determine the real value of resistance using the digital multimeter in the ohmmeter position and compare it to the nominal value plus the tolerance.

2) Measure of DC voltage and current in circuits with resistors

a) Assemble the circuit in Figure 1 on the connection board; connecting the power supply according to the polarity marked, that is, the ground terminal of the circuit must be

attached to the negative terminal of the supply. Set the short circuit current of the supply (I.LIMIT) to about 100 mA. The switch can be implemented with a simple wire that closes the circuit or leaves it open.

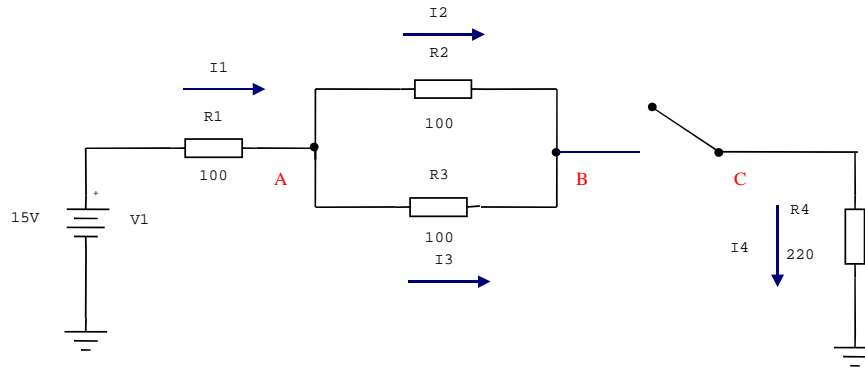


Figure 1: DC circuit with resistors.

b) Considering the switch open:

1. Measure with the multimeter (set up it to measure DC voltages) the voltage at points A, B and C. To do so, you must connect the negative terminal of the multimeter to the circuit ground, and the positive terminal to the point whose voltage you want to know. When measuring voltages, the instrument should be placed in **PARALLEL** with the circuit under test.
2. Verify that the current supplied by the power supply is zero. To do so, insert the multimeter in a position to measure DC currents, between the positive terminal of power supply and R1 resistor, as in current measurements, the instrument should be located in **SERIES**. Please also note that, in current measurement, the positive terminal can be **different** to the used to measure voltages.

c) Considering the switch closed:

1. Repeat the above voltage measurements.
2. Measure the current flowing through each of the resistors. Check that the current through R1 is identical to the current through R4 and currents of resistors R2 and R3 are very similar, as their respective resistances are.

d) Repeat the above paragraph changing the value of the resistor R3 to 220Ω. Notice how the currents through R2 and R3 are no longer equal.

e) Observe whether the currents vary when resistors R1 and R4 are exchanged. What happens to the voltages in the points A and B?

3) Simulation of a circuit with resistors

In this section we will start using SPICE (Simulation Program with Integrated Circuit Emphasis), a simulation program that allows computer-aided electronic circuit design and simulate their operation, with a high degree of accuracy and without having to build it physically. SPICE was developed at the University of Berkeley during the 70s and has become the standard in electronic simulation programs. With the development of operating systems with graphical environment and the subsequent development IDEs (Integrated Development Environment), many software companies have developed simulation programs with graphical editors, but with the same simulation engine, the SPICE program.

The process of circuit analysis consists firstly on drawing the circuit and translating this scheme to a connection file (.CIR. and .NET). These files can be read by SPICE

simulation engine. Once the circuit has been simulated, an output file (.DAT) is generated. This file can be understood by many graphics packages. In the case of OrCAD SPICE, this whole process is centralized in the environment named OrCAD Design Manager.

For more information, we recommend reading the manual of introduction to SPICE that we have uploaded in the PoliformaT platform.

a) Start the computer with Windows. The simulator runs on the command **Start/Programs /Pspice Student/Schematics**.

Edit the circuit in Figure 2. Use the command **Draw/Get New Part** to get the various components: a voltage generator (VDC), resistors (R) and GROUND (EGND). Store the file by creating a new folder called **"My Documents\Prac1"**. This folder **MUST BE DELETED** at the end of the Lab.

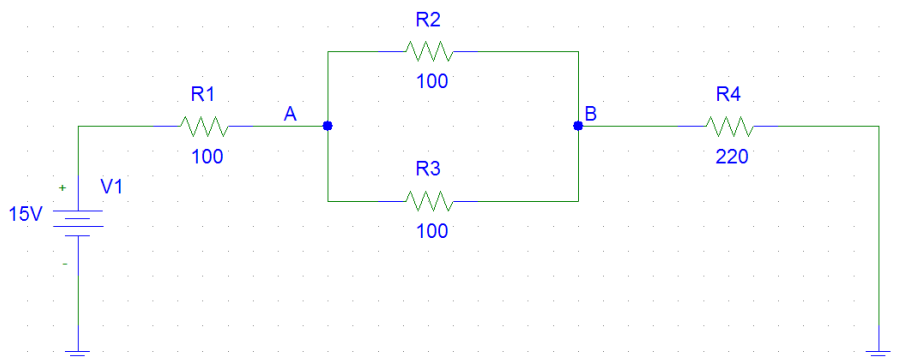


Figure 2: DC circuit with resistors.

NOTE: Be sure that the connecting cables are right at the terminal end of the component. To verify that the components are connected correctly, click first on them, select them and then try to move them. If the connections are correct, the wires will remain connected to their terminal ends when moving the components.

NOTE: You can change the value associated to a component by double clicking on it, or by clicking on the component and then editing the VALUE field.

b) Select the type of simulation to be performed in **Analysis/Setup**, marking the **Bias Point Detail** option. Finally, launch the simulation with **Analysis/Simulate**. Display the value of the voltages and currents in the circuit by pressing **V** and **I** buttons.

c) Repeat the above paragraph changing the value of the resistor R3 to 220Ω. Notice how the currents through R2 and R3 are no longer equal.

d) Observe whether the currents vary when resistors R1 and R4 are exchanged. What happens to the voltages in the points A and B?

e) Now, we will simulate the circuit in Figure 3, i.e. a circuit with an open switch. To do this, modify the previous circuit to add a switch that opens the circuit (**Draw/Get New Part** and take the component **Sw_tClose**).

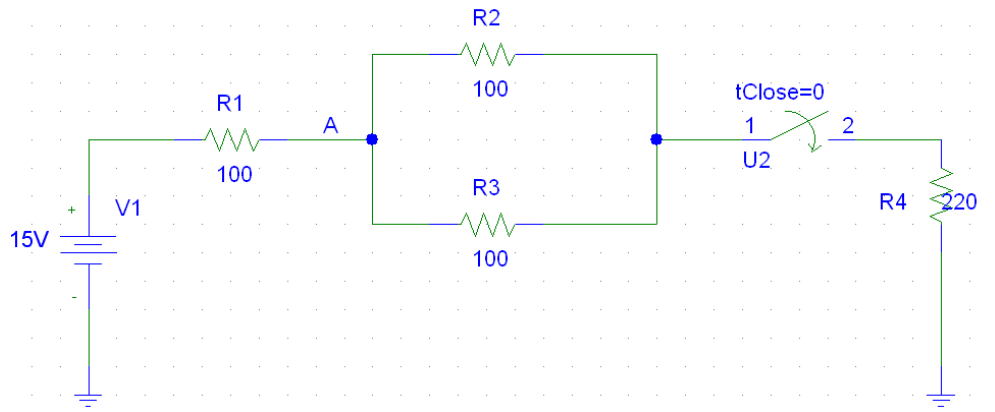


Figura 3: DC circuit with resistors and a switch.

f) Simulate again and compare the voltages and currents with those obtained in paragraph b) of Exercise 2. Justify the differences.

NOTE: When the switch is open, it has a resistance that although it is quite high, it yet allows driving a small current. In an ideal switch, this current would be zero. By double-clicking the component, you can see the value of the open-circuit resistance (R_{open}) and change it. For example, it can be increased from 1Meg (Meg = M Ω) to 1000Meg. On this way, we simulate a more ideal open switch with a negligible current.

g) Obviously, PSpice allows the simulation of more complex circuits, such as the one shown in Figure 4. Simulate this circuit ($V1 = 7\text{ V}$ and $V2 = 5\text{ V}$) and obtain the following output data:

V_A , V_B , I_1 , I_2 , I_3 , I_4 and I_5 .

NOTE: Check in the simulator the real direction of each current.

h) Repeat the above paragraph for $V1 = 7\text{ V}$ and $V2 = -5\text{ V}$.

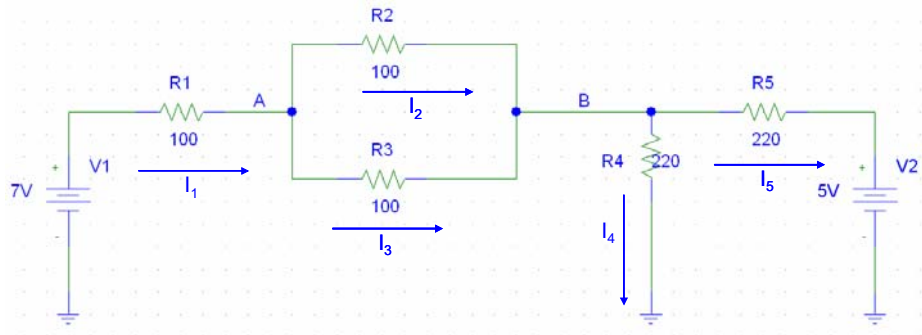


Figura 4: DC circuit with resistors.

Bibliografía

- "Introduction to PSpice", document available in the folder "**Recursos / Documentacion Practicas**" of TCO subject in PoliformaT.
- "Instrumentación de laboratorio" document available in the folder "**Recursos / Objetos de Aprendizaje Polimedia**" of TCO subject in PoliformaT.