

PHY338K Electronic Techniques


Fall 2025

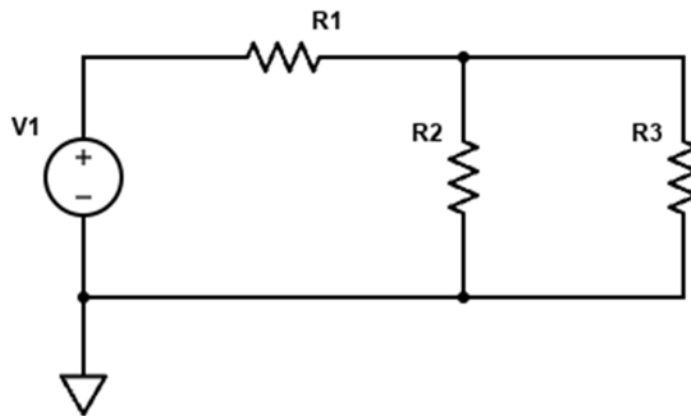
Homework 1

Due: Sept. 3, 2025

1. (30 points) DC Circuit, analytical calculation

In the following circuit, assume that $V_1 = 5.40 \text{ V}$, $R_1 = 215 \text{ } \Omega$, $R_2 = 740 \text{ } \Omega$, and $R_3 = 510 \text{ } \Omega$.

(The symbol  denotes “circuit ground.” The electric potential (voltage) is defined to be zero at this point. The voltage at any other point is equal to the potential difference between that point and the circuit ground.)



Analytically calculate the following quantities:

- The currents through R_1 , R_2 , and R_3 .
- The voltages across R_1 , R_2 , and R_3 .
- The power dissipated in the resistors R_1 , R_2 , and R_3 .
- Check that the sum of the powers dissipated in the resistors is equal to the power supplied by the voltage source.

2. (30 points) DC circuit, LTSpice calculation

Spice is a powerful circuit simulation program that is widely used in electronic design. It comes in different versions. LTSpice is the most popular version. It is available for free this website:

<https://www.analog.com/en/resources/design-tools-and-calculators.html>

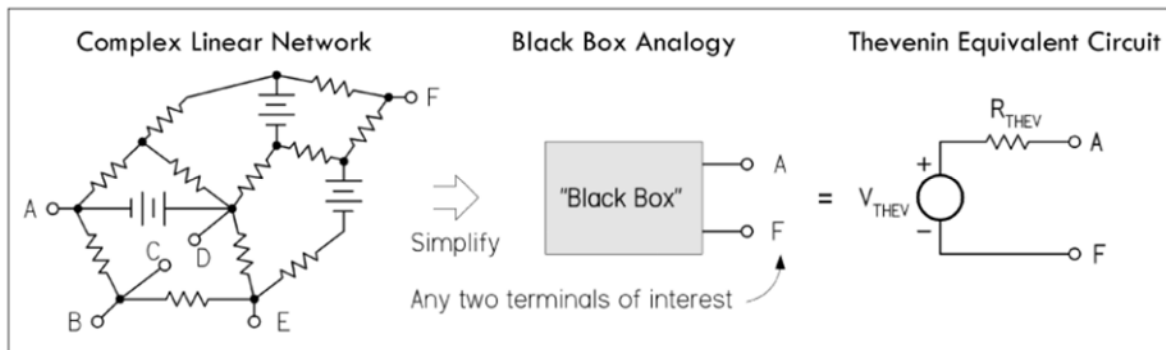
- If you do not already have it, download LTSpice and install it on your computer. If you are not already familiar with LTSpice, take some time to learn how it works. If you would like to start with a YouTube video, I thought this one seemed like a reasonable introduction:

<https://www.youtube.com/watch?v=JRcyHuyb1V0>

b) Build the circuit of problem 1 in LTSpice. Use LTSpice to find the currents through the resistors and the voltages across the resistors. Check that you obtained the same results as in Problem 1.

3. (40 points) Thevenin equivalent

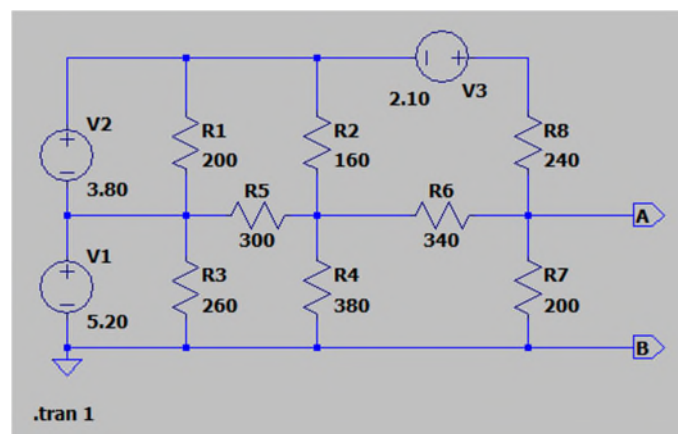
Thevenin's theorem states that two distinct points on any network of resistors and voltage sources has a simple *Thevenin equivalent*, as illustrated in the figure below (reproduced from *Practical electronics for inventors*).



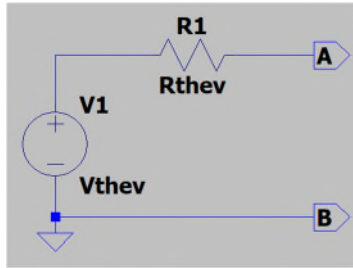
That is, the electrical behavior of the network for any load connected to the two points is equivalent to an ideal voltage source V_{th} in series with a resistance R_{th} . (If you are not already familiar with this theorem from Phys. 316, it is covered in your textbook starting on page 76.)

In the general case, calculating the values of V_{th} and R_{th} for a particular network can turn into a relatively complicated linear algebra problem. (See the example starting on page 71 of your textbook for an example.) In such cases a simulator can provide a solution much more quickly.

a) To see how this works, enter the following circuit into LTSpice:



b) According to Thevenin's theorem, from the perspective of points A and B this circuit is equivalent to



Using your LTSpice simulation, determine the Thevenin equivalent voltage $V_{\text{thев}}$ and resistance $R_{\text{thев}}$ for the circuit of part (a). (Hint: measure the open circuit voltage between points A and B, and then measure the short circuit current between points A and B. You can measure the short circuit current by adding a very low resistance resistor (*e.g.* $0.0001 \, \Omega$) between those two points and measuring the current through that. From those two quantities you can work out what $V_{\text{thев}}$ and $R_{\text{thев}}$ are.)

c) To check that the circuit does in fact behave as its Thevenin equivalent, add a resistor R_L between points A and B of your circuit of part (a). Measure the voltage difference between A and B for the values $R_L = 0.5 R_{\text{thев}}$, $R_L = R_{\text{thев}}$, and $R_L = 2.0 R_{\text{thев}}$. Verify that the voltages are the same that you would obtain for the same loads applied to the Thevenin equivalent circuit in part (b).