Homework 1 **ECE 302H: Introduction to Electrical Engineering**

I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

Problem 1.1 – LTSpice Tutorial *(1h)*

LTspice is a circuit simulator built by Linear Technology which was acquired by Analog Devices. Visit the Analog Devices website to download the latest version of LTspice: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html . Run LTspice and follow the steps below:

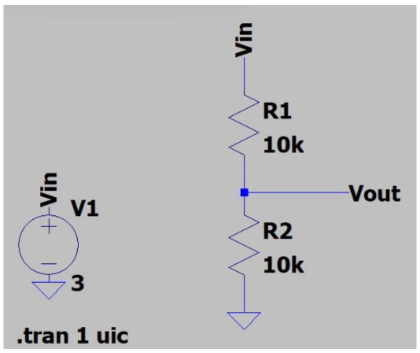
For MAC users: Note that the ECE department does not support the MAC operating system for undergraduates. https://www.ece.utexas.edu/academics/undergraduate/laptop . Nevertheless, you can do everything in this class on a MAC. The MAC version of LTSpice is not nearly as good – the best thing you can do is to Boot Camp into Windows and use the Windows version of LTSpice. Alternatively, the program known as "Wine" can make windows software run on Linux machines – since MAC is unix-based, it usually works there too. For those who wish to use the MAC version of LTSpice, you will need to adapt these instructions on your own.

a) Create a new file by clicking the button in the top-left corner of the window or under File > New Schematic.

b) To place a new component in general, press F2 to bring up the Select Component Symbol window. However, certain components are so common that they have their own hotkeys. Use the "r" key to get a resistor.

c) You can rotate the resistor if you wish using "ctrl+r." Click anywhere in your schematic to place the resistor symbol. If you do nothing at this point, you will be able to place more resistors. Place one more resistor and then press ESC to exit the symbol placement tool.

d) Press F3 to enter wire-drawing mode. Click to begin drawing; click again to create a new point (usually for changing directions); click on a component terminal to end the path (but remain in wire-drawing mode); press ESC to exit wire-drawing mode. Wire up the circuit as follows (we will add the labels in the next step):



I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

If you make a mistake, use F9 to undo or press DEL to enter the cutting mode. If you want to move components around, use F7 to move the component independent of any wires. Use F8 to move a component along with any wires already connected to it.

e) The wires connecting components together are also known as "nodes" or "nets." One important rule in circuit design is: *Two nodes/nets with the same name are actually a single node/net and are treated as being connected, even if the connection is not visible*. This helps clean up schematics with lots of components. Add a label by pressing F4, call the label "Vin" and place the label above R1 as in the Figure. Add another label for Vout as in the Figure.

f) One very special net name is called "ground." Ground rarely has any special meaning – it's just a name given to a net. A lot of beginners assign a special significance to ground which makes it seem more mysterious and important than it is. For almost all circuit-analysis purposes, just think of ground as a commonly-used net name. Ground does get a special symbol, however. Access the ground symbol in LTspice by pressing "g" and place the ground symbol below R2 as in the Figure.

g) Now, below off to the side, use F2 and find a voltage source symbol. Place a GND symbol on the negative terminal of the voltage source and a "Vin" label on the positive terminal. This will connect the voltage source to the resistor divider, even though there are no visible connecting wires.

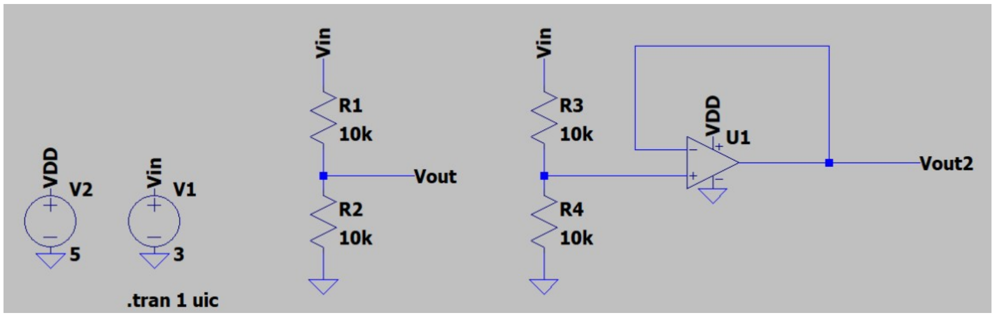
h) We're not ready to simulate the circuit until we have values for all of the components, including the input voltage. Right-click on each component value (not the component reference designator like "R3" or "V2") to change its value – make your circuit match the Figure above. Note that units are implied, so when you type "10k" for the resistor, it is implied that you mean 10 killo-ohms, where "ohms" is the unit of resistance.

i) To simulate the circuit, press the little running man button on the toolbar or go to Simulate > Run. You will be met with a dialog box. Keep yourself on the Transient tab. Enter a Stop Time of 1 ("seconds" is implied). Further, click the option that says "skip initial operating point solution". When you click OK, a new command ".tran 1 uic" will automatically be added to your schematic and a simulation results window will appear. If you want to see a voltage, click on a node in the circuit schematic – for example, click on Vin and Vout. You should see that Vin is 3V and Vout is half of that value. Since the circuit itself is static, we observe that nothing changes with time.

j) Let's create a modified copy of this circuit as shown below. Either place all new components or use F6 to copy existing components. Press F2 and find the "universalopamp2" component and place it on the schematic. We will learn about operational amplifiers or "op amps" later in the semester. For now, it suffices to understand the the op amp is an *active* circuit element, meaning it is powered and can do active things in the circuit (as opposed to

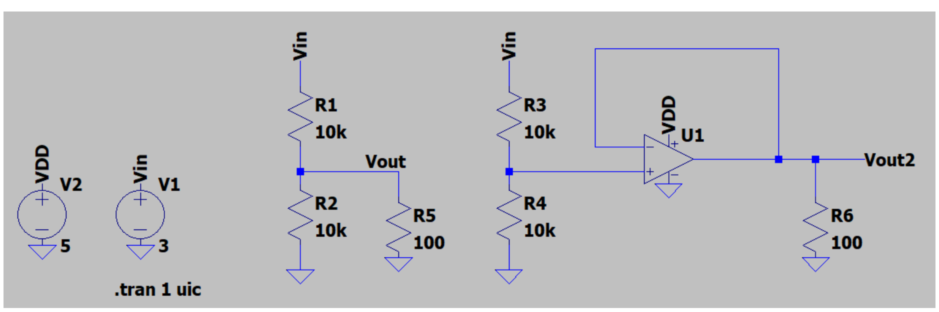
I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

resistors which are called "passive" components). Therefore we need a voltage source to serve as the power supply for the op amp. Make your circuit look like the circuit below.



k) Run the simulation – what are Vout and Vout2?

l) Both of these circuits are intended to take Vin and reduce its value by a factor of 2. This may be desirable if you want to measure Vin with a microcontroller with a limited safe input voltage range, for example. However, when the microcontroller is connected to the circuit, it's possible for the presence of the microcontroller itself to affect the "output" voltage, throwing off the measurement. To examine this possibility, attach a resistor from Vout to GND to represent a "load," meaning the next circuit in the signal chain (like a microcontroller). Do the same for Vout2, as shown in the Figure. What are Vout and iR5 and Vout2 and iR6 if the load resistance is 100k? 10k? 1k? Plot your results on an output voltage vs output current graph. Also submit a screenshot of your circuit and an example simulation output.



m) What advantage does the second circuit provide over the first circuit?

[By the way, you may already be equipped to analyze the first circuit by hand, and you will build these circuits in your first lab. We will study op-amps later in the semester. And the voltage "droop" that you observe in this problem can be described by something called the Thevenin Equivalent Circuit, which we will also study. By the first or second midterm, this will all be second nature to you!]

I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

Problem 1.2 – Systems of Equations By Hand *(1h)*

Solve the following systems of algebraic equations by hand, showing all your work. In this course, you will be making extensive use of your ability to solve systems of linear equations, so developing good proficiency in this task is important.

a) Solve for *x* and *y*:

$$2x+6y=367x-3y=6$$

b) Solve for *x*, *y* and *z*:

$$\frac{1}{8}x+\frac{1}{2}y-\frac{1}{4}z=\frac{-1}{2}\frac{2}{3}x-\frac{1}{4}y+\frac{1}{6}z=\frac{7}{2}\frac{1}{3}x+y-\frac{1}{3}z=\frac{2}{3}$$

c) Solve for *x*, *y* and *z* in terms of *a*:

$$4x-2y+6z=10+18a$$
$$6x+4y-z=222x+8y-5z=14-16a$$

Problem 1.3 – Charge carriers and current *(0.25h)*

While current into and out of batteries is mediated by electrons, internally the flow of electricity is mediated by ions. Traditional batteries rely on a single species of ion (for example, Lithium ions Li+) traveling between the cathode (+) and anode (-). Novel "dual-ion" batteries have two species of ion flowing simultaneously, such as batteries based on sodium hexafluorophosphate (NaPF6). Imagine a battery oriented in the z direction with the cathode at the top (positive z) and the anode at the bottom (negative z). If 3×10^{20} hexafluorophosphate ions ($PF_6^{-¿¿}$) go through the interior of the battery in the positive z direction in 4 seconds, and simultaneously 1.5×10^{20} sodium ions (Na+) go through the same cross section in the negative z direction, what is the magnitude and direction of the current flowing through the cross section? Is the battery being charged or discharged?

Problem 1.4 – Power and Energy *(0.25h)*

I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

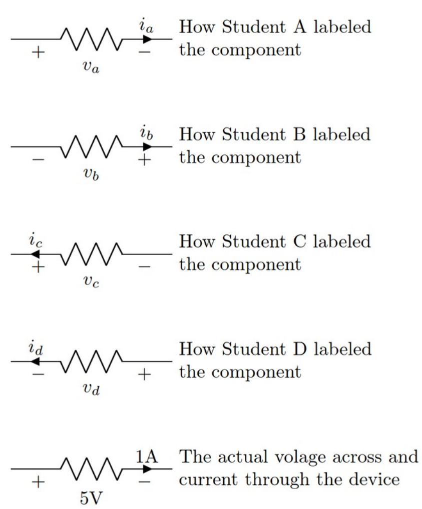
I would like to bake some cookies in my electric oven which operates at 120 V. During pre-heating, the oven operates at its rated power of 0.6 kW. However, after pre-heating, the oven only needs to consume 0.25 kW to maintain temperature. How much current does the oven draw in each case? How much energy does it consume during the 20-minute pre-heat and the 12-minute baking time? How many batches of cookies would I need to bake to use the same amount of energy in baking as I did in pre-heating?

Problem 1.5 – Passive Sign Convention *(0.25h)*

Before solving a problem, we usually label the voltage across and current through each component with variable names and reference directions. Four students labeled a component's voltage and current before solving a problem, each in a different way, as shown in the Figure (the Figure shows a resistor, but it could be any component). That actual polarity and magnitude of the voltage and current are also in the Figure. The students all solved the problem correctly. For each student, please answer the following:

a) How would the student write down their numerical answer for the voltage across and current through the resistor? In other words, what are ia, va, ib, and vb?
b) Does the student's labeling conform to the passive sign convention?
c) If the student calculated power as v × i, what numerical answer would they get?
d) Would the student interpret their power number as power entering or leaving the resistor?

Finally, write a few sentences answering whether it is necessary to follow the passive sign convention when assigning reference directions, and what advantages or disadvantages it may provide to follow the convention.



Problem 1.6 – Solving a Circuit *(1h)*

I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

In power systems, distributed energy resources (DERs) all supply power to the same grid. It is desirable to maintain the grid voltage at a target value and for the DERs to share power as equally as possible. This is illustrated in the Figure by two DERs powering the same bus, with a single load represented by a resistor R.

a) Suppose each DER is programmed to act like a pure voltage source. Suppose the DER voltages are different from each other. Explain why this would lead to disastrous results, even for tiny voltage differences.

b) Now suppose each DER is programmed to act like a voltage source in series with a resistance (there's no actual resistance; the DERs are programmed to act like this equivalent circuit). This is known as "droop control." Let both DER voltages be arbitrary but their droop resistances be the same. Calculate the grid voltage V_R and the ratio in currents supplied by the two DERs, i_{RD1}/i_{RD2}.
[Hint – as always, write down KVL for every loop, KCL for every note, and every component law; the rest is just math. Another hint – you can use LTSpice to check your answer!]

c) Suppose that the objective is to generate a grid voltage of 110V. Let V1 = 105V, V2 = 115V, and R = 10Ω. Use any program you wish to plot the actual grid voltage normalized to its target value, i.e. $V_R/110$, as a function of R_D letting R_D vary from $1\,\Omega$ to $100\,\Omega$. On the same graph, plot the ratio of currents supplied by the two DERs as functions of Rd.

d) Use your results to explain the tradeoff associated with choosing R_D. What bad thing happens if you let R_D be too small? What bad thing happens if you let R_D be too big? What value of R_D seems good to you and why?