Homework 3   **ECE 302H: Introduction to Electrical Engineering**   **Fall 2023**
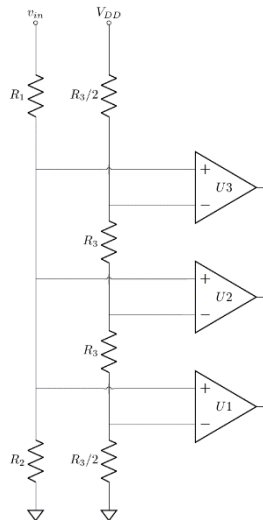
I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

Problem 3.1 – Analog to Digital Converter (1h)

Consider the circuit below which is composed of resistors (which you know) and a new component called a comparator. Comparators are quite easy to understand – they simply compare the voltages at their inputs. If the positive input is greater than the negative input, then the comparator outputs a digital "1". If the negative input is greater, then the comparator outputs a digital "0". Comparators do not draw any current from their inputs, i.e. they do not disturb the circuit they are measuring. If the outputs of the comparators are (from top down) [0001] then we interpret the digital output D as "D=1"; if the output of the comparators is [0011] then we interpret the digital output D as "2", and so forth. This is "thermometer code" because the 1's at the output rise like mercury in an old thermometer.

a) Calculate the voltage at the positive input to each comparator assuming v_i is set by a voltage source. Call it vin,low (note that this node is not connected to the other resistor string; the wire lines "hop over" the other resistor string). You expect the maximum value of vin to be 19.8 V and each pin of the comparator is rated to handle up to 3.3 V; what must the ratio R_1/R_2 be so that you can measure the full input voltage range while using the full pin rating? Please use that value for the rest of the problem.

b) Calculate the voltage at the negative input to each comparator assuming V_{DD} is set by a voltage source. Call them Vref1, Vref2, and Vref3. Let VDD = 3.3V.

c) Sketch a graph of the digital output D ("1", "2", etc.) as a function of vin. How many different levels can D represent? How many bits would you need to represent this number of levels in binary code (i.e., how many bits of resolution are there)?

d) How many more resistors and comparators would you need to add one more bit of precision?

e) Discrete resistors are typically rated to dissipate about 1/4 W of power each. Using this rating and the ratio you found in (a), calculate the minimum allowable values for R_1, R_2, and R_3.
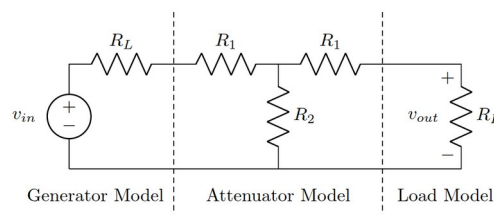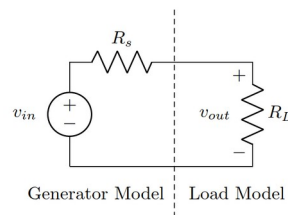
I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

Problem 3.2 – RF Attenuator (1h)

In radio-frequency (rf) systems, it is common to have a high-power source of energy and a sensitive detector or load that can't absorb very much power. In these cases, you can use an attenuator to reduce the amount of power absorbed by the load by a known amount. Let the energy source be modeled by a voltage vin with a resistor Rs in series; let the load be modeled by a resistor RL. Refer to the Figure.

a) Calculate the power delivered to the load when no attenuator is used.
b) Consider the attenuator in the Figure, which consists of a T-network of two resistors of values R1 and one resistor of value R2. Calculate what the new network "looks like" from the perspective of the generator. In other words, calculate the equivalent resistance Rnet of everything to the right of the generator.
c) Generators of rf energy often behave best when they are loaded with the same resistance as the generator's internal resistance (i.e., when $R_L = R_S$), but when we add the attenuator we risk changing the resistance that the source "sees" (i.e., Rnet). Let RL = Rs = 50 Ω. Normally our design task would be to find values for R1 and R2 that give a target attenuation while making the load+attenuator resistance look like 50 Ω. This is pretty messy to calculate in the general case so let's do a specific case instead. Suppose that R2 = 25.975 Ω. Calculate the required value of R1 to make the equivalent resistance of the load+attenuator combination equal to 50 Ω.
d) Calculate the attenuation of the resulting attenuator circuit, i.e., the ratio of power delivered to the actual load when the attenuator is inserted in the circuit divided by the power delivered to the load with no attenuator.
e) Simulate the circuit you designed in LTspice. What measurements would you take, and what calculations would you make based on those measurements, to evaluate if your circuit does what you want it to do? Submit a screenshot of your LTSpice graphs and circuit and, separately, any calculations you need to do to prove that the circuit works. Be sure to label any nodes whose voltage you will be measuring so that the graph is self-explanatory. (Currents and powers don't need to be labeled since they're named after their component)

Generator Model ┊ Load Model

Generator Model ┊ Attenuator Model ┊ Load Model

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 3.3 – Linearity (0.5h)

Use the definition of linearity to show whether the following SISO functions are linear:
- a) $f(x)=mx$, where m is a constant
- b) $f(x)=mx$, where $m=1/x^2$
- c) $f(x)=mx+b$, where m and b are constants
- d) $f(x)=e^x$

Problem 3.4 – Linearization (0.5h)

In a previous homework, we analyzed a MOSFET-based circuit. We said that the MOSFET could be modeled by a current source of value $g_m v_{gs}$, where gm was called the transconductance and vgs was the voltage between the MOSFET gate and source. This model is actually a simplification from a more complete model for the current that flows from drain to source, given by

$$i=\frac{1}{2}\mu_n C_{ox}\frac{W}{L}\left(v_{gs}-V_{TH}\right)^2\left(1+\lambda v_{ds}\right)$$

where μ_n is the mobility of electrons, Cox is the gate capacitance per unit area, Vth is called the "threshold voltage" of the MOSFET, and W and L are the MOSFET width and length. The parameter λ represents channel-length modulation (if λ = 0, then the term 1 + λvds falls out of the expression). We'll learn more about all of these parameters later in the course.

- a) Please identify two features of the above equation that make it nonlinear. (Hint – identify which terms are circuit variables and which ones are constants)
- b) We have surprisingly few tools for dealing with nonlinear systems. The most common thing we can do is "linearize" the system, which means that we need to find a linear equation that is approximately equal to the real equation for small deviations away from an operating point. This is also called "small-signal modeling." Replace the circuit quantities i, vgs, and vds with an operating point value plus a small deviation away from that operating point. Thus i → I + Δi, vgs → Vgs + Δvgs, and vds → Vds + Δvds. Show that the resulting equation reduces to the following:

$$\Delta i=\left(\mu_n C_{ox}\frac{W}{L}\left(1+\lambda V_{ds}\right)\left(V_{gs}-V_{TH}\right)\right)\Delta v_{gs}+\left(\frac{1}{2}\mu_n C_{ox}\frac{W}{L}\left(V_{gs}-V_{TH}\right)^2\lambda\right)\Delta v_{ds}$$

    Be sure to not confuse the operating point values (indicated by capital letter) from the small perturbation values (indicated by Δ). You will need to cancel an object on the left hand side with an equal object on the right hand side. Further, you will need to make an approximation that any product of small quantities can be approximated as zero (a small number times a small number is a really small number). So, any terms in your answer with two or more Δ quantities multiplied together can be approximated as zero. These are the nonlinear terms – approximating them as zero is the "linearization" part of this process.
- c) Using sentences and a circuit diagram, show how this equation can be converted to a dependent current source in parallel with a resistor. What is the current source voltage-to-current constant gm? What is the resistance value ro? You can see now that the previous homework lied to you just a little bit. The model was correct, but it only models the behavior of the *deviations* of the
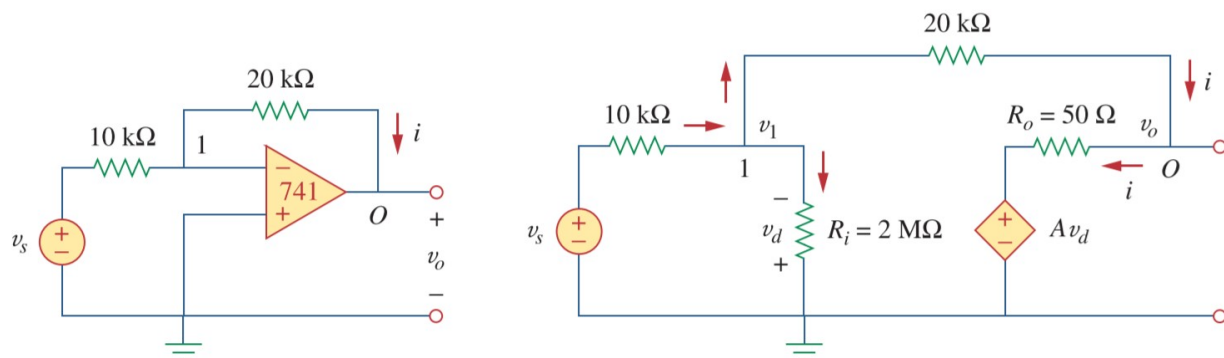
I include time estimates for each problem. A problem may take you more or less time, but if you spend more than half of the allotted time on a problem and aren't making progress, it's time to ask for help!

> variables from their operating points (the previous homework also assumed $\lambda=0$). Still, that's often all we care about, so the result is useful.

Problem 3.5 – Node Analysis on Op Amp Circuit (0.5h)

We most commonly encounter dependent voltage sources when we use circuit elements called operational amplifiers or "op-amps." An ideal op-amp is nothing more than a voltage-dependent voltage source; nonideal op-amps also have some input resistance R_i and output resistance R_o. Therefore, even though we have not yet covered op-amps explicitly in class, you are nevertheless prepared to analyze op-amp circuits. A circuit involving an op amp is shown in the figure, with its equivalent circuit model. For all parts, let the parameter A be a variable and then take $A \rightarrow \infty$ in your final answers.

   a) Using the equivalent circuit model, analyze the circuit for an ideal op-amp, i.e. let $R_i=\infty$ (open circuit) and let $R_o=0$ (short circuit). What is the output voltage for a given input voltage? What is the voltage at the negative input of the op-amp, v_1?

   b) Using the equivalent circuit model, analyze the circuit again for a non-ideal op-amp ($R_i=2M\Omega$, $R_o=50\Omega$ as shown), calculating v_o and v_1.

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 3.6 – Node Analysis on BJT Circuit (1h)

The first transistor was not a MOSFET, but rather a Bipolar Junction Transistor (BJT, or bipolar transistor). The BJT has three terminals like the MOSFET, but they are called the Collector (instead of Drain), the Emitter (instead of Source), and Base (instead of Gate). BJTs have largely been supplanted by MOSFETs in modern times, but still find use in applications that demand very high speed or high current.

A BJT is shown in the figure below along with its equivalent circuit when in its active mode, which consists of a fixed voltage source V_{BE} and a current-dependent current source. The parameters V_{BE} and β are properties of the transistor. Use Node Analysis to solve for I_B and v_o in the circuit in the figure for a BJT with $\beta = 200$ and $V_{BE} = 0.7\,V$.
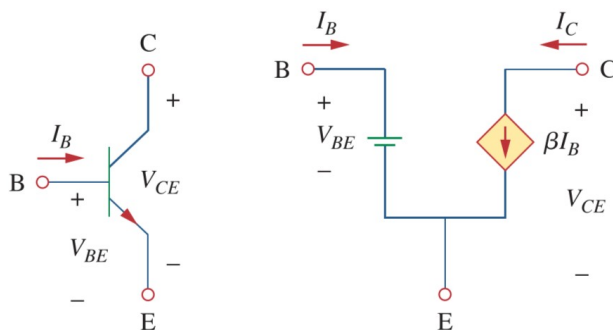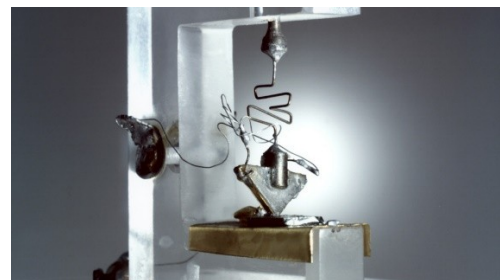

The bipolar transistor circuit diagram and its equivalent circuit model
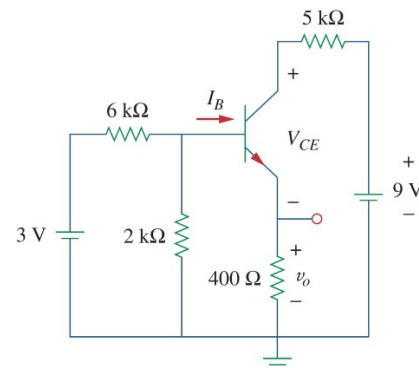
The circuit to be solved

The developers of the first working transistor, a BJT: John Bardeen, Willliam Shockley, and Walter Brattain

The first working transistor was created at Bell Labs in 1947 out of Germanium