This homework is due on Sunday, July 16, 2017, at 23:59.
Self-grades are due on Monday, July 17, 2017, at 23:59.

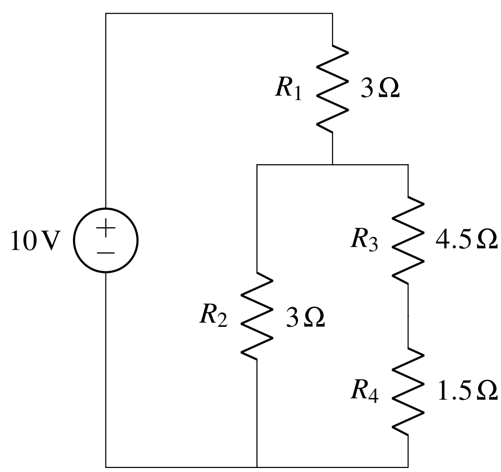
Submission Format
Your homework submission should consist of **one** file.

- `hw4.pdf`: A single PDF file that contains all of your answers (any handwritten answers should be scanned).

Submit the file to the appropriate assignment on Gradescope.

1. Mechanical Circuits

Find the voltages across and currents flowing through all of the resistors.



2. Cell Phone Battery

As great as smartphones are, one of their main drawbacks is that their batteries don't last a very long time. A Google Pixel, under somewhat regular usage conditions (internet, a few cat videos, etc.) uses 0.4 W of power. We will model the battery as a voltage source (which, as you know, will maintain a voltage across its terminals regardless of current through it) with one caveat: they have a limited amount of charge, or capacity. When the battery runs out of charge, it no longer provides a constant voltage, and your phone dies. Typically, engineers specify battery capacity in terms of mAh, which indicates how many mA of current the battery can supply for one hour before it needs to be recharged. The Pixel's battery has a battery capacity of 2770 mAh and operates at a voltage of 3.8 V.

(a) When a battery's capacity is depleted, no longer operates as a voltage source. How long will a Pixel's full battery last under regular usage conditions?

(b) How many coulombs of charge does the battery contain? Recall that $1\,\mathrm{C} = 1\,\mathrm{A} \times 1\,\mathrm{s}$, which implies that $1\,\mathrm{mC} = 1\,\mathrm{mAs}$. An electron has approximately $1.602 \times 10^{-19}\,\mathrm{C}$ of charge. How many usable electrons worth of charge are contained in the battery when it is fully charged?

(c) Suppose the cell phone battery is completely discharged and you want to recharge it completely. How much energy (in J) is this? Recall that a J is equivalent to a W s.

(d) Suppose PG&E charges $0.16 per kWh. Every day, you completely discharge the battery (meaning more than typical usage) and you recharge it every night. How much will recharging cost you for the month of July (31 days)?

(e) The battery has internal circuitry that prevents it from getting overcharged (and possibly exploding!). The circuitry is also used to transfer power into the chemical reactions that store the energy. We will model this internal circuitry as being one resistor with resistance R_{bat}, which is typically a small, non-negative resistance. Furthermore, we'll assume that all the energy dissipated across R_{bat} goes to recharging the battery. Suppose the wall plug and wire can be modeled as a 5 V voltage source and $200\,\mathrm{m\Omega}$ resistor, as pictured in Figure 1. What is the power dissipated across R_{bat} for $R_{\text{bat}} = 1\,\mathrm{m\Omega}$, $1\,\Omega$, and $10\,\mathrm{k\Omega}$? How long will the battery take to charge for each of those values of R_{bat}?

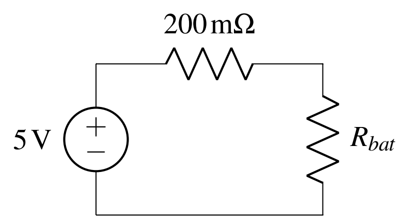
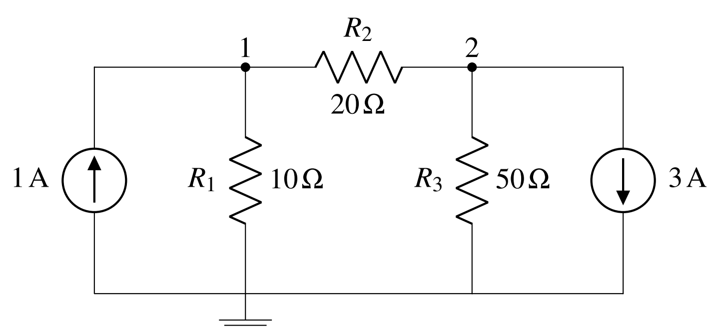


Figure 1: Model of wall plug, wire, and battery.

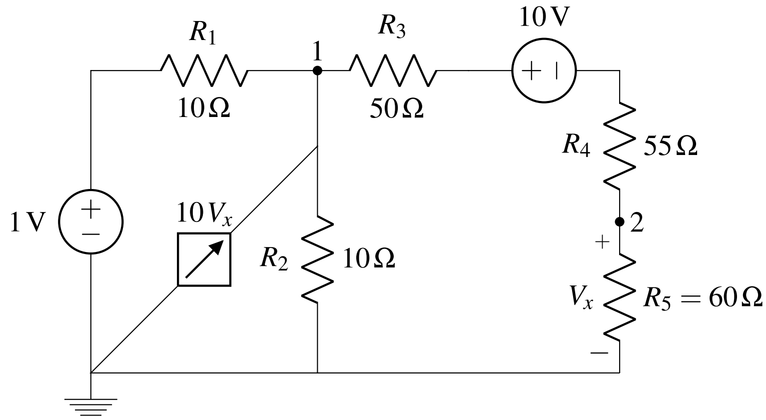
3. Nodal Analysis

Using techniques presented in class, label all unknown node potentials and apply KCL to each node to find all the node potentials.

(a) Solve for all node potentials using nodal analysis. Verify with superposition.

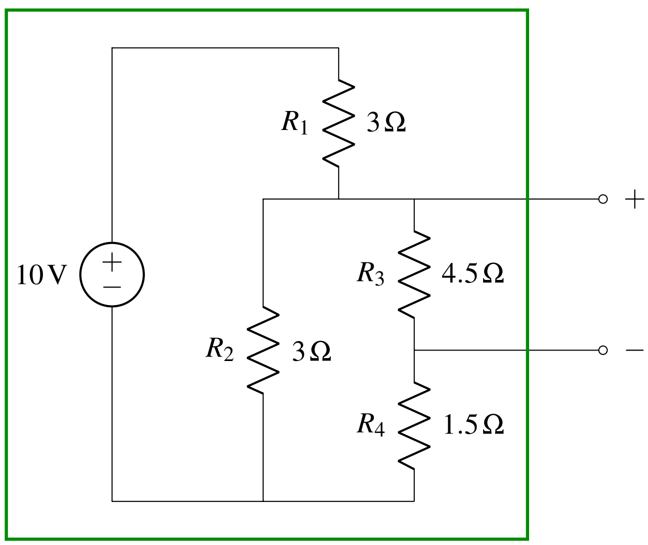


(b) Solve for all node potentials using nodal analysis.

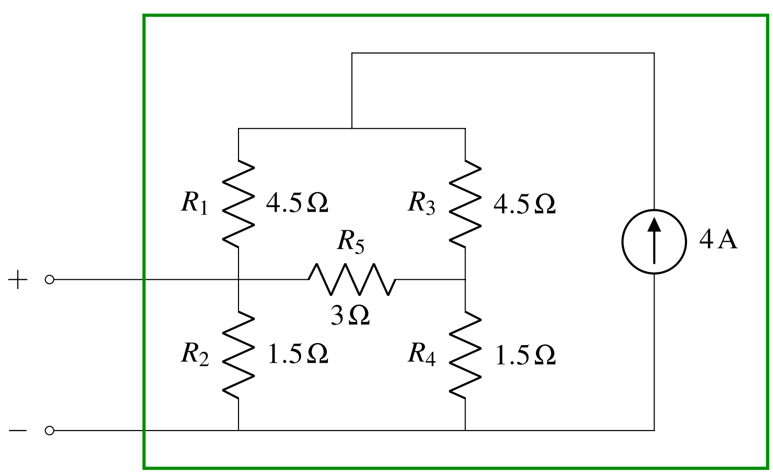


4. Thévenin and Norton Equivalent Circuits

(a) Find the Thévenin and Norton equivalent circuits seen from the outside the box.

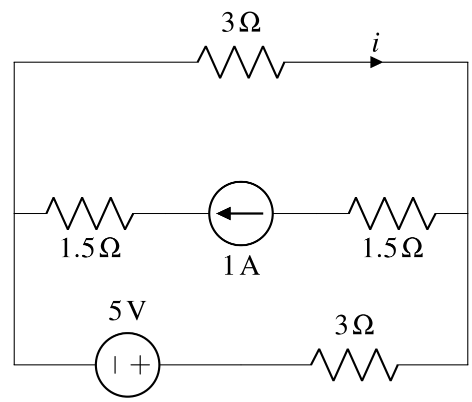


(b) Find the Thévenin and Norton equivalent circuits seen from the outside the box.



5. **Nodal Analysis Or Superposition?**

Solve for the current through the $3\,\Omega$ resistor, marked as i, using superposition. Verify using nodal analysis. You can use IPython to solve the system of equations if you wish. Where did you place your ground, and why?



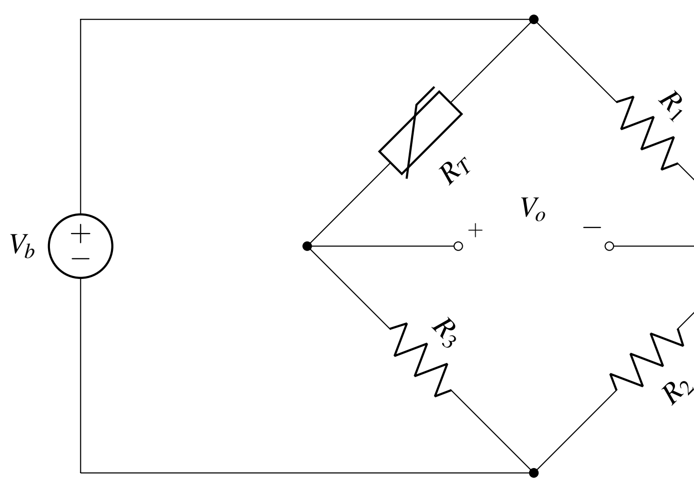
6. **Thermistor**

Thermistors for sensing temperature consist of sintered metal oxide that exhibits an exponential decrease in electrical resistance with increasing temperature. In semiconductors, electrical conductivity is due to the charge carriers in the conduction band. If the temperature is increased, some electrons are promoted from the valence band into the conduction band, and the conductivity also increases.

The relationship between resistance R and temperature T is given by:

$$R_T(T) = R(T_0)\exp\left(\beta\left(\frac{1}{T} - \frac{1}{T_0}\right)\right) \tag{1}$$

Where T is in degrees kelvin, T_0 is the reference temperature, and β is the temperature coefficient of the material. To sense temperature, thermistors are used in a bridge circuit shown below:



The temperature response of a thermistor is given in the table below:

Table 1: Resistance vs. Temperature data for the thermistor

Temperature (°C)	−50	−40	−30	−20	−10	0	10	20	30	40	50
R_T(kΩ)	117.2	65.2	38.8	23.8	15.2	10	6.8	4.7	3.4	2.5	1.8

(a) For the thermistor bridge circuit, find the Thevenin equivalent circuit.

(b) Find V_o and from there derive an equation for R_T as a function of V_o, V_b, and the other resistances.

(c) If $R_T = R_1 = R_2 = R_3$ what will be the output of the bridge circuit? Assuming $R_1 = R_2 = R_3$, then from the Resistance vs. Temperature data for the thermistor, comment of the bridge output if the temperature rises and vice versa.

(d) If $R_2 = R_3$, find what value of $\alpha = \frac{R_3}{R_T}$ (the relation between R_3 and R_T) provides the largest bridge sensitivity to temperature [$\frac{dQ}{d\alpha} = 0$]? The bridge sensitivity, Q is defined as,

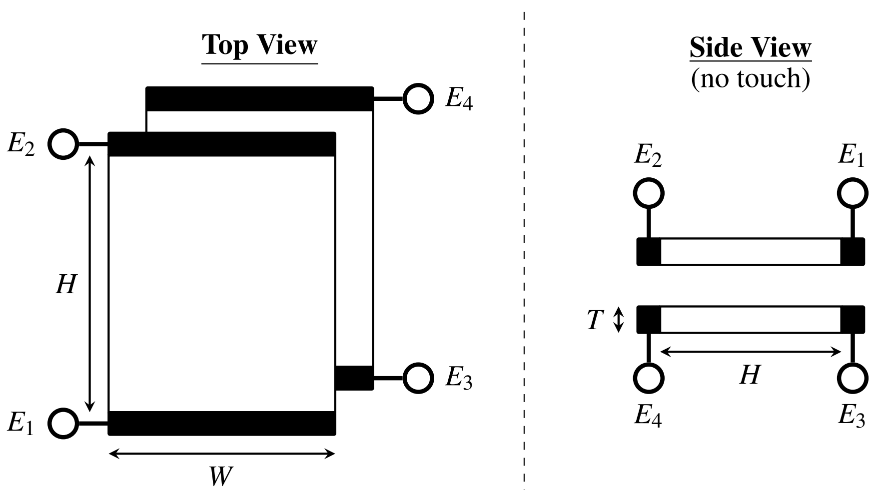
$$Q = \frac{dV_o}{dT} = R_T \frac{dV_o}{dR_T} \frac{1}{R_T} \frac{dR_T}{dT}$$

Hint: Both equation 1 and the bridge circuit equation are required for this question.

(e) Using the the relation between R_3 and R_T design a bridge circuit that will provide highest sensitivity at 30 °C. Draw your circuit, and justify your design choices for R_1, R_2, and R_3 [$V_b = 3.3$ V].

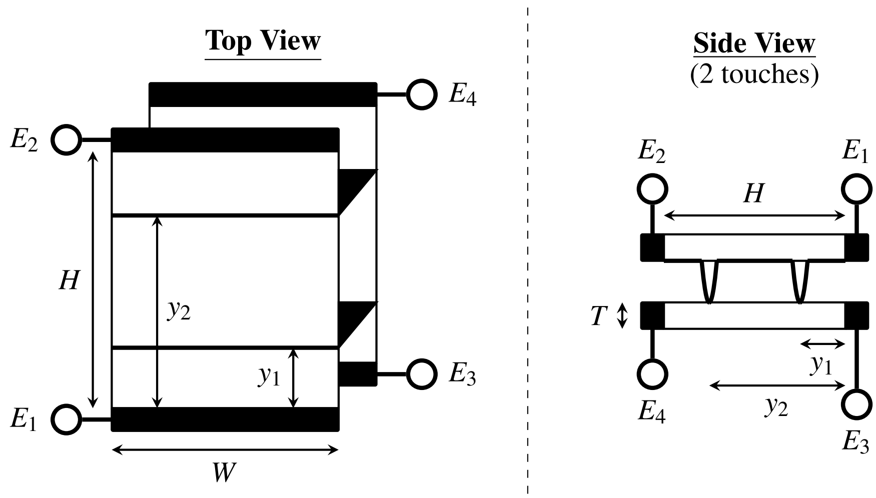
7. Multitouch Resistive Touchscreen

In this problem, we will look at a simplified version of the multitouch resistive touchscreen. In particular, rather than measuring the position of two potential touch points in both dimensions (i.e. a pair of coordinates (x_1, y_1) and (x_2, y_2) corresponding to two touch positions), let's think about a version where we are interested in measuring only the vertical position of the two touch points (i.e. y_1 and y_2). Therefore, unlike the touchscreens we looked at in class, both of the resistive plates (i.e. both the top and the bottom plate) would have conductive strips placed along their top and bottom edges, as shown below.

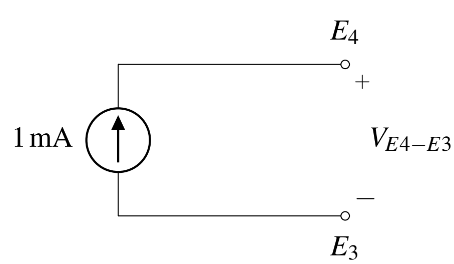


(a) Assuming that both of the plates are made out of a material with $\rho = 1\,\Omega$m and that the dimensions of the plates are $W = 3$ cm, $H = 12$ cm, and $T = 0.5$ mm, with no touches at all, what is the resistance between terminals E_1 and E_2 (which would be the same as the resistance between terminals E_3 and E_4)?

(b) Now let's look at what happens when we have two touch points. Let's assume that at wherever height the touch occurs, a perfect contact is made between the top plate and the bottom plate along the entire width of the plates (i.e. you don't have to worry about any lateral resistors), but that otherwise none of the electrical characteristics of the plates change. Defining the bottom of the plate as being $y = 0\,\text{cm}$ (i.e. a touch at E_1 would be at $y = 0\,\text{cm}$), let's assume that the two touches happen at $y_1 = 3\,\text{cm}$ and $y_2 = 7\,\text{cm}$ and that your answer to part (a) was $5\,\text{k}\Omega$ (which may or may not be the right answer). Draw a model with 6 resistors that captures the electrical connections between E_1, E_2, E_3, and E_4 and calculate their resistances. Note that for clarity, the system has been redrawn below to depict this scenario.



(c) Using the same assumptions as part (b), if you drove terminals E_3 and E_4 with a 1 mA current source (as shown below) but left terminals E_1 and E_2 open-circuited, what is the voltage you would measure across $E_4 - E_3$ (i.e. V_{E4-E3})?



(d) Now let's try to generalize the situation by assuming that the two touches can happen at any two arbitrary points y_1 and y_2, but with y_1 defined to always be less than y_2 (i.e. y_1 is always the bottom touch point). Leaving the setup the same as in part (c) except for the arbitrary y_1 and y_2, by measuring only the voltage between E_4 and E_3, what information can you extract about the two touch positions? Please be sure to provide an equation relating V_{E4-E3} to y_1 and y_2 as a part of your answer, and note that you may want to redraw the model from part (b) to help you with this.

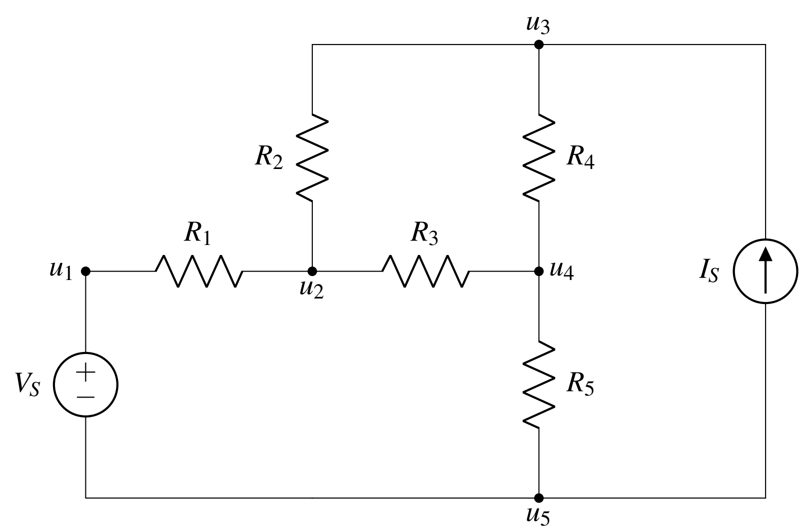
(e) One of your colleagues claims that by measuring the appropriate voltages, not only can they extract what both y_1 and y_2 are in this system, but they can even do so by formulating a system of three independent voltage equations related to y_1 and y_2. As we will see later, this will allow us to gain some robustness to noise in the voltage measurements.

In order to facilitate this, write equations relating V_{E4-E2} and V_{E1-E3} to y_1 and y_2. (The third voltage we'll use is V_{E4-E3}, which you should have already derived an equation for in the previous part of the problem.)

8. SPICE-y Circuits

In the 1970s, Laurence Nagel and his advisor Donald Pederson at UC Berkeley created a circuit simulation software called Simulation Program with Circuit Emphasis or SPICE. Today, SPICE is the industry standard for circuit simulation. DC analysis in SPICE is performed using linear algebra tools you are familiar with. In this problem, we will explore the linear algebra behind SPICE.

Throughout the problem, we will be referring to the circuit below.



(a) A circuit designer inputs a circuit to SPICE in the form of a netlist. SPICE then translates the netlist into an incidence matrix. Translate the circuit above into a directed graph and write the edge-node incidence matrix $\mathbf{F}$ for the graph. $\mathbf{F}$ should have dimensions $M \times N$, where M is the number of branch currents (which is equal to the number of components) and N is the number of node potentials.

(b) SPICE now has a representation for the circuit, in the form of an incidence matrix. SPICE represents the current through each of the elements as one vector $\vec{i}$, whose entries correspond to the currents in all branches. Find the product $\mathbf{F}^T\vec{i}$ and show that $\mathbf{F}^T\vec{i} = \vec{0}$ represents the KCL equations for this circuit.

(c) Now let's include our knowledge about the current source in our KCL equation. Rewrite $\mathbf{F}^T\vec{i} = \vec{0}$ as $\widetilde{\mathbf{F}}^T\widetilde{i} = -I_s\vec{c}$. A quick note on the notation here: $\widetilde{\mathbf{F}}^T$ is $\mathbf{F}^T$ with column $\vec{c}$ removed. Column $\vec{c}$ is the column of $\mathbf{F}^T$ that corresponds to the branch current that has the current source. $\widetilde{i}$ is the same as the current vector $\vec{i}$ but has the branch current corresponding to the current source removed.

(d) SPICE then assigns each node a potential and represents these potentials in a vector $\vec{u}$. Show that the multiplication $\vec{v} = \mathbf{F}\vec{u}$ represents the voltages across all components in the circuit.

(e) Now let's look at our vector of voltages $\vec{v}$. First of all, there is a voltage source in our circuit with its voltage corresponding to $u_1 - u_5 = V_s$. Second of all, there are resistors in our circuit. By Ohm's law, the voltage across a resistor is equal to the current through the resistor multiplied by the resistance, i.e. $v_R = i_R R$. Third of all, we have a current source in our circuit, which means that we cannot immediately say anything about the voltage between u_3 and u_5 (we'll see later on that this is a-ok). Write a vector

$\vec{d}$ that satisfies the equality $F\vec{u} = \vec{v} = \vec{d}$. Where $\vec{d}$ is a vector containing source voltages, products of resistor currents and resistances and v_{cs} for the voltage across the current source.

(f) Now, write the vector $\vec{d}$ as a sum $\vec{d} = \mathbf{R}\widetilde{i} + \vec{v}_s$. In this expression, $\widetilde{i}$ is the vector of all currents in the circuit with the current source current removed, same as before. $\vec{v}_s$ is a vector that only contains voltages across sources (and zeroes everywhere else), and $\mathbf{R}$ is a diagonal matrix with resistances along the diagonal. Rearrange this equation to $\mathbf{F}\vec{u} - \mathbf{R}\widetilde{i} = \vec{v}_s$ and write all the matrices and vectors explicitly.

Note: Be careful with the entries of $\mathbf{R}$ that correspond to the voltage source and the current source. Recall that a voltage source supplies a specified voltage between its terminals regardless of the current through it, and that in $\vec{d}$ we have the entry for the voltage across the current source set to v_{cs}.

(g) At this point, you have two matrix equations (from parts (f) and (c)) in terms of two unknown vectors: the potentials $\vec{u}$ and the remaining currents not defined by the current source $\widetilde{i}$. Combine these two matrix equations into a single, giant system of equations $\mathbf{A}\vec{x} = \vec{b}$. $\vec{b}$ should be a vector whose only non-zero entries are the source voltages (V_s and v_{cs}) and the current of the current source I_s (pay attention to the signs). $\vec{x}$ should be a vector that combines $\vec{u}$ and $\widetilde{i}$. The formats of the different elements are:

$$\mathbf{A} = \left[\begin{array}{c}\text{Left-hand-side from part (f)} \\ \hline \text{Left-hand-side from part (c)}\end{array}\right], \vec{x} = \begin{bmatrix} | \\ \vec{u} \\ | \\ | \\ \widetilde{i} \\ | \end{bmatrix} \text{ and } \vec{b} = \left[\begin{array}{c}\text{Right-hand-side from part (f)} \\ \hline \text{Right-hand-side from part (c)}\end{array}\right]$$

(h) In this and the next part, we will use linear dependence to our advantage. First, in this part, we note that the columns of $\mathbf{A}$ are linearly dependent ($\mathbf{A}$ is not full rank). This is because if you sum the columns of $\mathbf{A}$ corresponding to the unknowns $u_1, \ldots, u_5$ you will get $\vec{0}$. This will always be the case because we know from lecture that each current leaves one node u_i and enters another node u_j ($i \neq j$), so that in each row in the incidence matrix $\mathbf{F}$, there will be a single entry of $+1$, a single entry of -1, and the rest will be 0 (therefore they will always sum to 0). Because this is the case, we are guaranteed a degree of freedom in the variables $u_1, \ldots, u_5$. This is, algebraically speaking, precisely why we are allowed to set the ground on any node of our choosing (sometimes called reference node or reference potential). Setting a ground is equivalent to selecting the value 0 for the free variable (potential) that we are always guaranteed in a circuit. This reference potential, or ground, is typically chosen by the circuit designer to be any node if her or his choosing (that's you). In this circuit, you are told to use u_5 as the reference potential in order to standardize the problem statement (set it to 0). Setting this variable to 0 means two things. First, it is no longer unknown. Second, since $\mathbf{A}\vec{x}$ is simply a linear combination of the columns of $\mathbf{A}$, weighted by the components of $\vec{x}$, the column corresponding to u_5 now will not contribute to this linear combination (multiplied by 0).[1] This means we can take this column out of $\mathbf{A}$ and remove u_5 from the vector $\vec{x}$. Do so and write a new matrix equation $\hat{\mathbf{A}}\hat{x} = \vec{b}$. Write the matrix $\hat{\mathbf{A}}$ and the vector $\hat{x}$ explicitly.

(i) At this point we should have 10 unknowns and 12 equations. That is, we have 2 more equations than unknowns. Therefore, we have the freedom to get rid of two equations that are redundant to create a new system of linear equations $\hat{\hat{\mathbf{A}}}\hat{x} = \hat{\hat{b}}$. This is good news because we remember having one equation from part (f) that was particularly hard to solve. That is the equation corresponding to the voltage

[1] If you need to convince yourself of this fact: compare $\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \\ a_{41} & a_{42} & a_{43} \end{bmatrix} \begin{bmatrix} x \\ 0 \\ y \end{bmatrix}$ with $\begin{bmatrix} a_{11} & a_{13} \\ a_{21} & a_{23} \\ a_{31} & a_{33} \\ a_{41} & a_{43} \end{bmatrix} \begin{bmatrix} x \\ y \end{bmatrix}$.

across the current source v_{cs}. We will eliminate this equation by taking its corresponding row from $\hat{\mathbf{A}}$ and its corresponding right-hand-side (v_{cs}) from $\vec{b}$. We are allowed to eliminate another equation.[2] We note that the sum of all equations from part (c) sum to $0 = 0$, so that one of them can be taken out (this will always be the case because the rows of $\mathbf{A}^T$ are the columns of $\mathbf{A}$ and they always sum to $\vec{0}$, as discussed earlier). Choose an equation from these to eliminate and remove its corresponding row from $\hat{\mathbf{A}}$ and its corresponding right-hand-side from $\vec{b}$. After these edits, write the new system of linear equations $\hat{\hat{\mathbf{A}}}\hat{x} = \hat{\hat{b}}$ explicitly.

(j) Now given the following values: $I_s = 5\,\text{mA}, V_s = 5\,\text{V}, R_1 = 4\,\text{k}\Omega, R_2 = 3\,\text{k}\Omega, R_3 = 5\,\text{k}\Omega, R_4 = 2\,\text{k}\Omega$, and $R_5 = 6\,\text{k}\Omega$, solve for all non-reference potentials and all currents (you may use python for this part).

(k) **PRACTICE:** We showed in this problem, so far, that the SPICE method works for the given circuit. Starting from this part, we will show that the methods followed in this problem will always work (assuming the circuit doesn't have contradictions such as a voltage source in a short circuit, a current source in an open circuit, voltage sources in parallel or current sources in series). From this point onward in this problem, we will assume that we have a circuit with a total of M components (resistors, independent voltage sources and independent current sources), M_c of which are current sources. Moreover, we will assume that we have N nodes in the circuit. What are the dimensions of the incidence matrix $\mathbf{F}$?

(l) **PRACTICE:** If we were to eliminate the variables in $\vec{i}$ for current sources and move their corresponding columns from $\mathbf{F}^T$ in $\mathbf{F}^T\vec{i} = \vec{0}$ to the right-hand side to get $\widetilde{\mathbf{F}}^T\widetilde{i} = -\left(I_{s_1}\vec{c}_1 + \cdots + I_{s_{M_c}}\vec{c}_{M_c}\right) \triangleq \vec{c}^*$ (where $\vec{c}_i$ is the column of $\mathbf{F}^T$ corresponding to the current sources I_{s_i}), similar to part (c), what would be the dimensions of the matrix $\widetilde{\mathbf{F}}^T$?

(m) **PRACTICE:** If we were to combine the two systems of linear equations $\mathbf{F}\vec{u} - \mathbf{R}\widetilde{i} = \vec{v}_s$ (similar to part (b)) and the system $\widetilde{\mathbf{F}}^T\widetilde{i} = \vec{c}^*$ (from last part) into one big system of linear equations $\mathbf{A}\vec{x} = \vec{b}$ (similar to part (g)), what would be the dimensions of the matrix $\mathbf{A}$?

(n) **PRACTICE:** If we were to select a reference node (which we are always guaranteed to be able to do) and, by that, remove the corresponding column from matrix $\mathbf{A}$ and the corresponding variable from the vector $\vec{x}$ to create the system $\hat{\mathbf{A}}\hat{x} = \vec{b}$ (similar to part (h)), what would be the dimension of the matrix $\hat{\mathbf{A}}$?

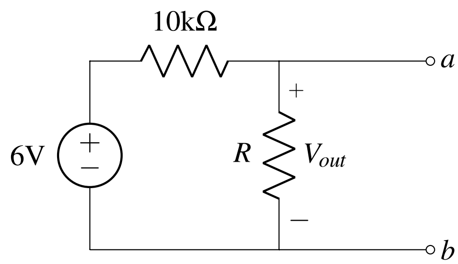
(o) **PRACTICE:** Given the new dimensions of $\hat{\mathbf{A}}$, how many equations can we eliminate without risking removing linearly independent equations? Which equations would you remove (similar to part (i))? After removing these equations, would the resultant system $\hat{\hat{\mathbf{A}}}\hat{x} = \hat{\hat{b}}$ have a square matrix $\hat{\hat{\mathbf{A}}}$?

9. Resistive Voltage "Regulator"

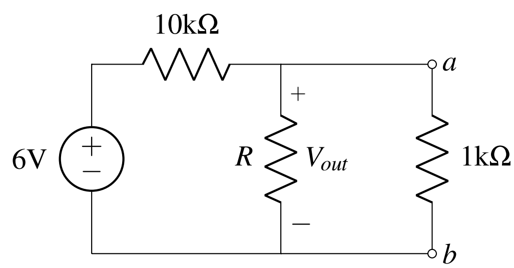
In this problem, we will design a circuit that provides an approximately constant voltage divider across a range of loads. We will use a resistive voltage divider circuit. The goal is to design a circuit that from a source voltage of 6V would yield an output voltage within 5% of 4V for loads in the range of 1kΩ to 100kΩ.

(a) First, consider the resistive voltage divider in the following circuit. What resistance R would achieve a voltage V_{out} of 4V?

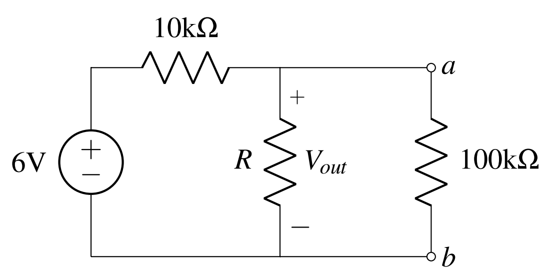
[2] We actually don't have to take a second equation out at this point. Instead, we can solve using Gaussian elimination. However, we will take a second equation out, so that we are able to invert the resultant matrix $\hat{\hat{\mathbf{A}}}$ to find the unique solution in our circuit (unique because we selected a reference node).



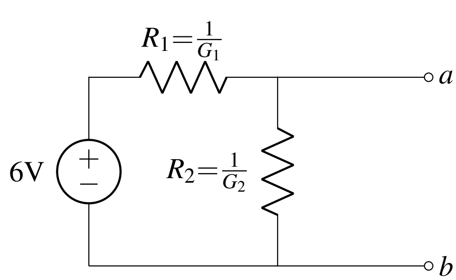
(b) Now using the same resistor R as calculated in part (a), consider loading the circuit with a resistor of 1kΩ as depicted in the following circuit. What is the voltage V_{out} now?

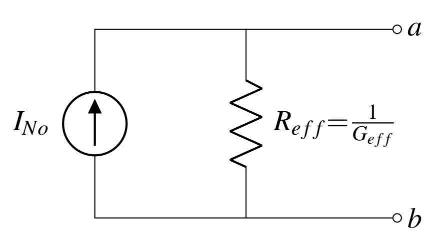


(c) Now using the same resistor R as calculated in part (a), consider loading the circuit with a resistor of 100kΩ, instead, as depicted in the following circuit. What is the voltage V_{out} now?

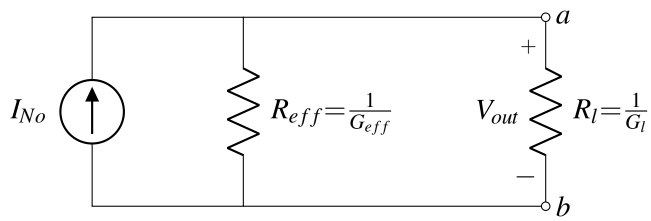


(d) Now we would like to design a divider that would keep the voltage V_{out} regulated for loads for a range of loads R_l. By that, we would like the voltage to remain within 5% window of 4V. That is, we would like to design the following circuit such that $3.80\text{V} \leq V_{out} \leq 4.20\text{V}$ for a range of loads R_l. As a first step, what is the Norton equivalent of the circuit on the left? Write I_{No} and G_{eff} it in terms of conductance values $G_1 = \frac{1}{R_1}$ and $G_2 = \frac{1}{R_2}$.





(e) For the second step, using the Norton equivalent circuit you found in part (d), what is the range of G_{eff} that achieves $3.80\text{V} \leq V_{out} \leq 4.20\text{V}$ in terms of I_{No} and G_l?



(f) Translate the range of G_{eff} in terms of I_{No} and G_l (that you found in part (e)) into a range on G_2 in terms of G_1 and G_l.

(g) Say we want to support loads in the range $1\text{k}\Omega \leq R_l \leq 100\text{k}\Omega$ with approximately constant voltage as described above (that is, $3.80\text{V} \leq V_{out} \leq 4.20\text{V}$). What is the range of G_2 in terms of G_1 now? Translate the range of G_2 in terms of G_1 into a range of R_2 in terms of R_1.

(h) Note that conductance is always non-negative. From the bounds on G_2 you found in the previous part, derive a bound on G_1 that ensures that G_2 is always non-negative and non-empty (that is, the whole range of possible G_2 values is non-negative and is not empty). Translate this range into a range of possible R_1 values.

Hint: In addition to the conductance being non-negative, also make sure that the range for G_2 is non-empty.

(i) Pick the values of R_1 and R_2 that achieve $3.80\text{V} \leq V_{out} \leq 4.20\text{V}$ for $1\text{k}\Omega \leq R_l \leq 100\text{k}\Omega$ while minimizing the power consumed by the voltage divider circuit in open circuit (when there is no load attached to the output). What are these values R_1 and R_2? How much power is consumed in this case? Calculate and report this power consumption using both the original circuit and the Norton equivalent circuit. Are the power you calculated using the original circuit and the power you calculated using the Norton equivalent circuit equal?

(j) **PRACTICE:** Now using the same values R_1 and R_2 from the previous part, load the circuit with a load of $51\text{k}\Omega$. How much power is consumed by each of the three resistors, R_1, R_2 and R_l (use the original circuit to compute the power)?

10. Homework Process and Study Group

Who else did you work with on this homework? List names and student ID's. (In case of homework party, you can also just describe the group.) How did you work on this homework?
Working in groups of 3-5 will earn you credit for your participation grade.