This homework is due March 13, 2017, at 23:59.
Self-grades are due March 16, 2017, at 23:59.

Submission Format
Your homework submission should consist of **two** files.

- `hw7.pdf`: A single pdf file that contains all your answers (any handwritten answers should be scanned) as well as your IPython notebook saved as a pdf.

  If you do not attach a pdf of your IPython notebook, you will not receive credit for problems that involve coding. Make sure your results and plots are showing.

- `hw7.ipynb`: A single IPython notebook with all your code in it.

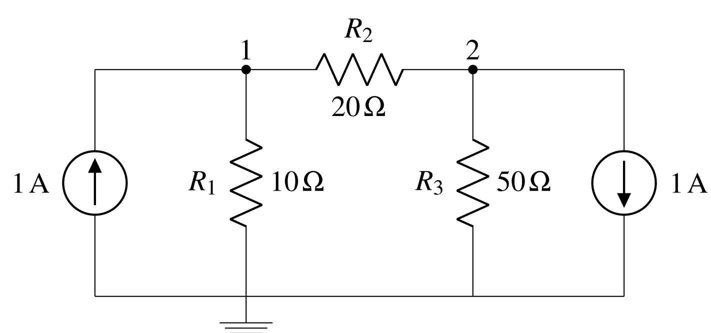
  In order to receive credit for your IPython notebook, you must submit both a "printout" and the code itself.

Submit each file to its respective assignment in Gradescope.

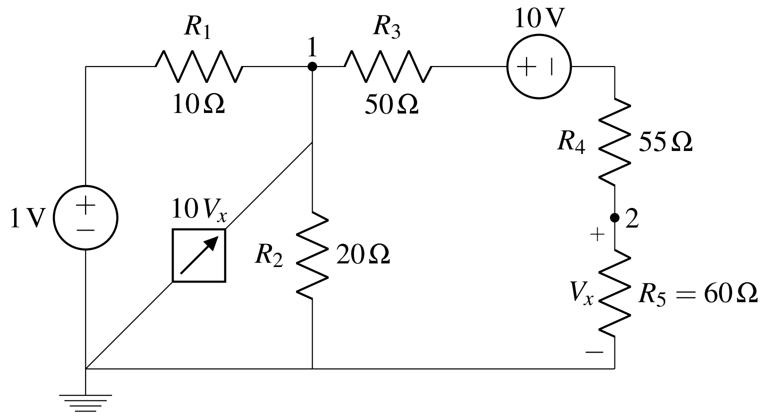
1. Nodal Analysis

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

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



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



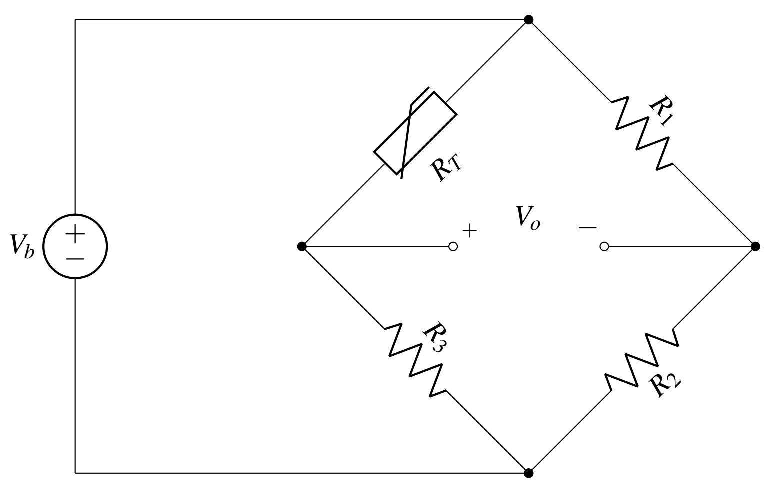
2. Thermistor

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
$R_T (k\Omega)$	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 $\alpha = R_3/R_T$ (the relation between R_3 and R_T) provides the largest bridge sensitivity to temperature $[dQ/d\alpha = 0]$? The bridge sensitivity, Q is defined as,

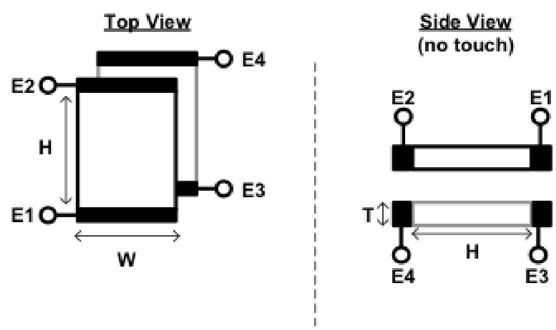
$$Q = \frac{dV_o}{dT} = R_T \frac{dV_o}{dR_T} \frac{1}{R_T} \frac{dR_T}{dT}$$

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 $[V_b = 3.3V]$.

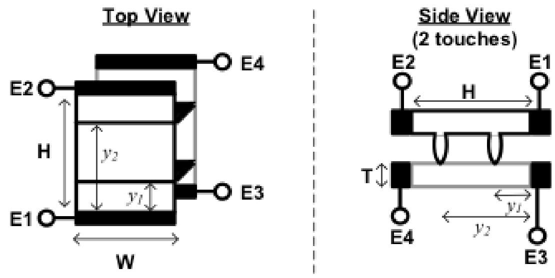
3. Multitouch Resistive Touchscreen

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.

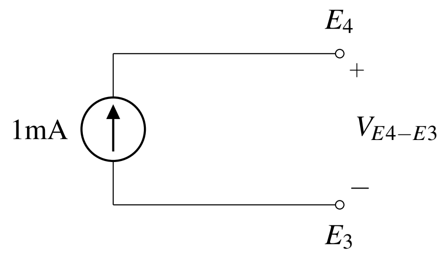


(a) Assuming that both of the plates are made out of a material with $\rho = 1\Omega m$ and that the dimensions of the plates are $W = 3cm$, $H = 12cm$, and $T = 0.5mm$, with no touches at all, what is the resistance between terminals E_1 and E_2 (which would be the same as the resistance between terminals 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 $y = 0cm$ (i.e. a touch at E_1 would be at $y = 0cm$), let's assume that the two touches happen at $y_1 = 3cm$ and $y_2 = 7cm$ and that your answer to part (a) was $5k\Omega$ (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 1mA 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.

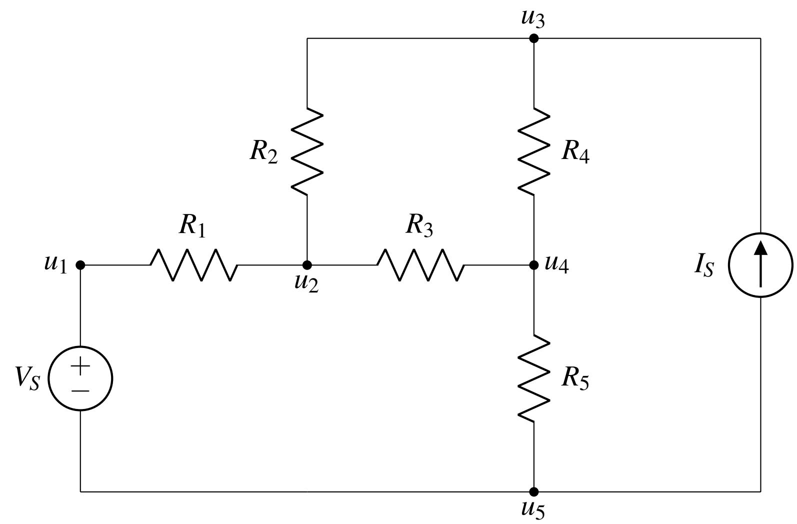
(e) One of your colleagues claims that by measuring the appropriate voltages, not only can they extract what both y_1 and y_2 are in this system, but they can even do so by formulating a system of three independent voltage equations related to y_1 and y_2. As we will see later, this will allow us to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

4. 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. $I_s = 5mA, V_s = 5V, R_1 = 4k\Omega, R_2 = 3k\Omega, R_3 = 5k\Omega, R_4 = 2k\Omega,$ and $R_5 = 6k\Omega$.



(a) Inputs to SPICE would occur in the form of a netlist. SPICE would then translate this netlist into an incidence matrix, ignoring any independent sources such as voltage or current sources. Translate the above circuit into a directed graph, ignoring any edges that belong to a voltage or current source. Write the incidence matrix, **F**, for the graph.

(b) SPICE now has a unique 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 $\mathbf{F}^T\vec{i} = 0$ represents the KCL equations for this circuit (again ignoring independent sources). By ignoring flows corresponding to independent sources, what equations are we missing?

(c) So far, SPICE has ignored independent sources. At this point, it will add independent current sources since it knows the flows through these edges. Redraw the circuit including resistors and the current source, and write KCL at nodes u_5 and u_3.

(d) SPICE now modifies the equation from part (b) to include the KCL constraint from part (c). The new equation is $\mathbf{F}^T\vec{i} = \vec{b}$. Find the vector $\vec{b}$. This product should now represent KCL written at all nodes in the circuit from part (c).

(e) 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 the resistors in the circuit.

(f) Because of independent voltage sources, some nodes have a potential that can be directly found from the potential at another node. Nodes connected by a voltage source are thus combined into one *supernode*. Redraw the graph representing the circuit with nodes u_1 and u_5 combined together into one node. Write the incidence matrix for the new graph. You may ignore the current source for this part.

(g) The equation $\vec{v} = \mathbf{F}\vec{u}$ now has to be modified to take into account the difference between some nodes caused by voltage sources. SPICE now adds a vector of independent sources $\vec{c}$. Find the vector $\vec{c}$ such that $\mathbf{F}\vec{u} + \vec{c}$ represents the voltage across all resisitors in the circuit.

(h) SPICE represents all resistances in a diagonal matrix **R**. Show that the equation $\mathbf{F}\vec{u} + \vec{c} = \mathbf{R}\vec{i}$ represents Ohm's law for the circuit.

(i) At this point, we have two matrix equations in terms of two unknown vectors $\vec{i}$ and $\vec{u}$. We will combine the matrices into a larger matrix, and represent this as a *block matrix*. A block matrix is a matrix made up of smaller matrices and is a useful tool for solving systems of matrix equations. For example, consider two equations $\mathbf{A}\vec{x}+\mathbf{B}\vec{y}=\vec{c}$ and $\mathbf{C}\vec{x}+\mathbf{D}\vec{y}=\vec{d}$, the block matrix representation of the system is shown below:

$$\begin{bmatrix}\mathbf{A} & \mathbf{B}\\ \mathbf{C} & \mathbf{D}\end{bmatrix}\begin{bmatrix}\vec{x}\\ \vec{y}\end{bmatrix}=\begin{bmatrix}\vec{c}\\ \vec{d}\end{bmatrix}$$

where **A**, **B**, **C**, and **D** are matrices of appropriate dimension. Represent our system so far as a block matrix. Be sure to consider how the vector of independent current sources has changed.

(j) We now have to verify that our system has a solution. Argue why the block matrix will be invertible as long as the matrix **F** has full column rank. Verify that the matrix **F** does not have full column rank, and show that $\vec{1}$ is in the null space of *any* incidence matrix.

(k) Because $\vec{1}$ is in the null space of **F**, the potentials $\vec{u}$ can all have a steady offset. However, in the circuit, SPICE cares about voltages or the differences in potential. For this reason, SPICE creates a *ground node* (In real SPICE programs, the ground node is specified by the designer). This is a node whose potential is assigned to zero. SPICE grounds a node by removing it from the vector $\vec{u}$ and its corresponding column from the matrix **F**. Write the new block matrix representing the system.

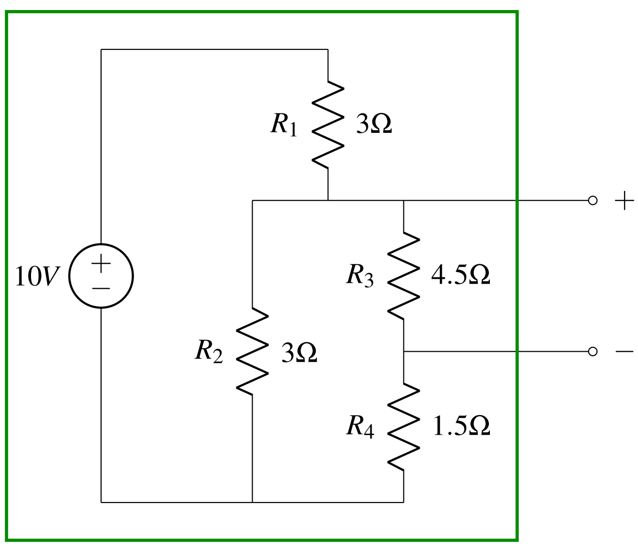
(l) Using IPython or any numerical tool, solve the system for the node potentials $\vec{u}$ and branch currents $\vec{i}$.

5. Homework process and study group

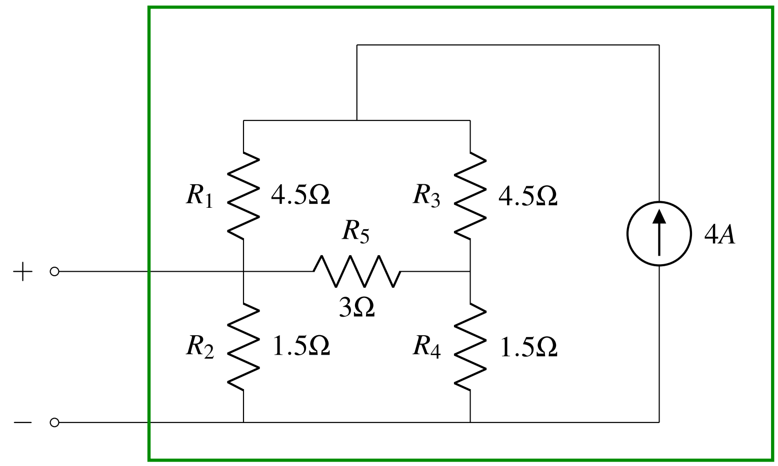
Who else did you work with on this homework? List names and student ID's. (In case of hw party, you can also just describe the group.) How did you work on this homework?
Working in groups of 3-5 will earn credit for your participation grade.

6. (PRACTICE) 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.



7. (PRACTICE) Nodal Analysis Or Superposition?

Solve for the current through the 3Ω 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?