# 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.

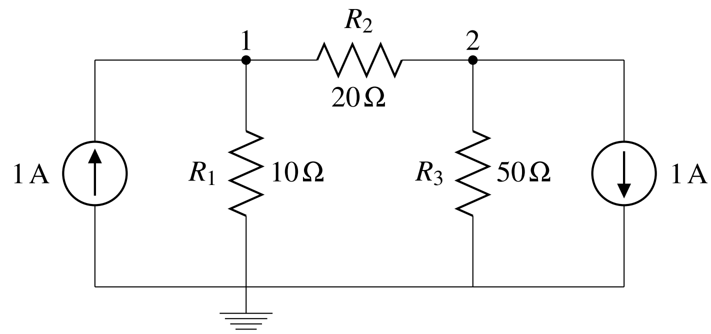
  In order to receive credit for your IPython notebook, you must submit both a "printout" and the code itself.

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.




**Solution:**

Method 1: Nodal Analysis

Applying KCL at Node 1, we get

$$\frac{V_1 - 0}{R_1} + \frac{V_1 - V_2}{R_2} - 1 = 0$$

$$\frac{V_1 - 0}{10} + \frac{V_1 - V_2}{20} - 1 = 0$$

which gives

$$2V_1 + V_1 - V_2 - 20 = 0$$

implying

$$3V_1 - V_2 = 20 \tag{1}$$

Applying KCL at Node 2, we get

$$\frac{V_2 - V_1}{R_2} + \frac{V_2 - 0}{R_3} + 1 = 0$$

$$\frac{V_2 - V_1}{20} + \frac{V_2 - 0}{50} + 1 = 0$$

which gives

$$5V_2 - 5V_1 + 2V_2 + 100 = 0$$

implying

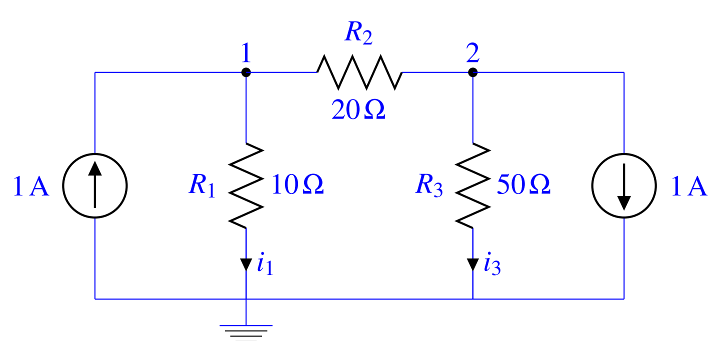
$$-5V_1 + 7V_2 = -100 \tag{2}$$

Writing equations **??** and **??** in matrix form, we get

$$\begin{bmatrix} 3 & -1 \\ -5 & 7 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} 20 \\ -100 \end{bmatrix}$$

Solving the system of equations, we will get $V_1 = 2.5V$ and $V_2 = -12.5V$.

Method 2 (verification): Superposition
We define $i_1$ and $i_3$ as follows:




First, consider the effect of only the left $1A$ current source on. Using current divider rule, we have

$$i_1 = \frac{R_2 + R_3}{R_1 + R_2 + R_3} \times 1A$$

$$i_1 = \frac{70}{10 + 70} \times 1A = 0.875A$$

and

$$i_3 = 1A - 0.875A = 0.125A$$

Therefore,

$$V_1^a = i_1 \times 10\Omega = 0.875A \times 10\Omega = 8.75V$$

and

$$V_2^a = i_3 \times 50\Omega = 0.125A \times 50\Omega = 6.25V$$

Second, consider the effect of only the right $1A$ current source on. Using current divider rule, we have

$$i_1 = \frac{R_3}{R_3 + R_2 + R_1} \times 1A$$

$$i_1 = \frac{50}{50+30} \times 1A = -0.625A$$

and

$$i_3 = -1A + 0.625A = -0.375A$$

Therefore,

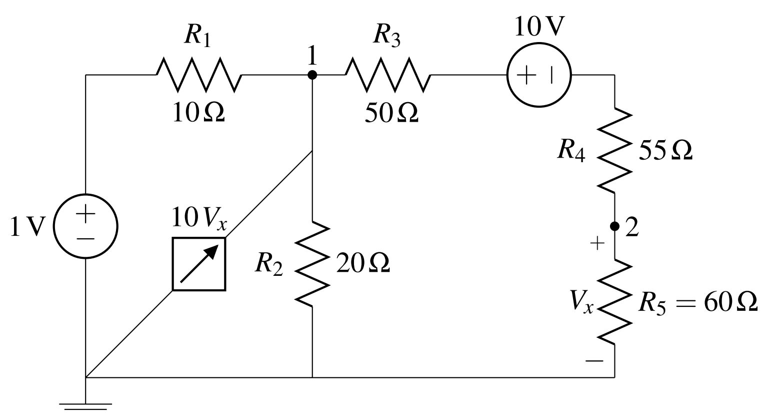
$$V_1^b = i_1 \times 10\Omega = -0.625A \times 10\Omega = -6.25V$$

and

$$V_2^b = i_3 \times 50\Omega = -0.375A \times 50\Omega = -18.75V$$

Since the circuit is linear (i.e. we have linear elements and sources), we can use the *principle of superposition* to get $V_1 = V_1^a + V_1^b$ and $V_2 = V_2^a + V_2^b$. Therefore, we get $V_1 = 8.75V - 6.25V$ and $V_2 = 6.25V - 18.75V$. Finally, $V_1 = 2.5V$ and $V_2 = -12.5V$.
This solution agrees with the solution we obtained using nodal analysis.

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




**Solution:** We assign the two nodes 1 and 2, and note that $V_x = V_2$ (because we have set the bottom wire as ground). Applying KCL at node 1 (ensuring that currents flowing out of node 1 sum to zero), we get

$$\frac{V_1 - 1}{R_1} + \frac{V_1 - 0}{R_2} + \frac{V_1 - (10 + V_2)}{R_3 + R_4} - 10V_2 = 0$$

$$\frac{V_1 - 1}{10} + \frac{V_1 - 0}{20} + \frac{V_1 - (10 + V_2)}{105} - 10V_2 = 0$$

which gives

$$\frac{V_1 - 1}{2} + \frac{V_1 - 0}{4} + \frac{V_1 - V_2 - 10}{21} - 50V_2 = 0$$

implying

$$67V_1 - 4204V_2 = 82$$

Applying KCL to Node 2 (ensuring that currents flowing into node 2 sum to zero), we get

$$\frac{V_2 - 0}{R_5} + \frac{V_2 - (V_1 - 10)}{R_3 + R_4} = 0$$

$$\frac{V_2 - 0}{60} + \frac{V_2 - (V_1 - 10)}{105} = 0$$

which gives

$$\frac{V_2}{4} + \frac{V_2 + 10 - V_1}{7} = 0$$

implying

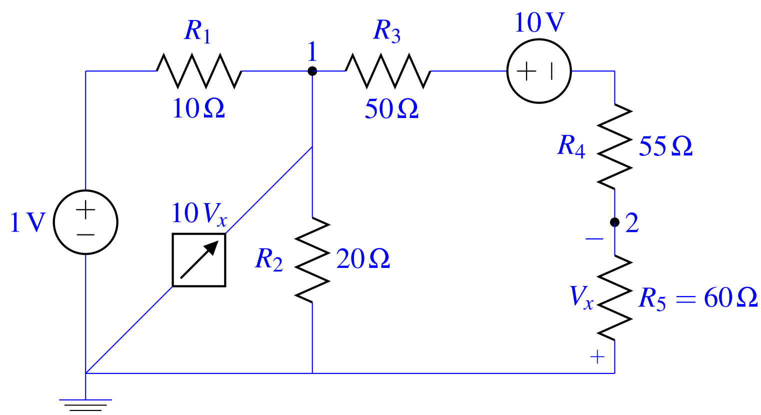
$$-4V_1 + 11V_2 = -40$$

Writing the equations in matrix form, we get

$$\begin{bmatrix} 67 & -4204 \\ -4 & 11 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} 82 \\ -40 \end{bmatrix}$$

Solving the system of equations, we will get $V_1 = 10.4023V$ and $V_2 = 0.1463V$.

If you solved this problem with the old version of the circuit, you will still receive full credit, as long as your answer remains consistent with the incorrect version of the circuit.




To solve this other circuit, note that $V_x = -V_2$ instead, so we can flip the sign of the dependent current source.

After applying KCL at node 1, your new equation will be:

$$\frac{V_1 - 1}{R_1} + \frac{V_1 - 0}{R_2} + \frac{V_1 - (10 + V_2)}{R_3 + R_4} + 10V_2 = 0$$

$$\frac{V_1 - 1}{10} + \frac{V_1 - 0}{20} + \frac{V_1 - (10 + V_2)}{105} + 10V_2 = 0$$

which gives

$$\frac{V_1 - 1}{2} + \frac{V_1 - 0}{4} + \frac{V_1 - V_2 - 10}{21} + 50V_2 = 0$$

implying

$$67V_1 + 4196V_2 = 82$$

The KCL equation for node 2 will remain the same as the first solution, so the final result will be $V_1 = 9.6308, V_2 = -0.13424$.

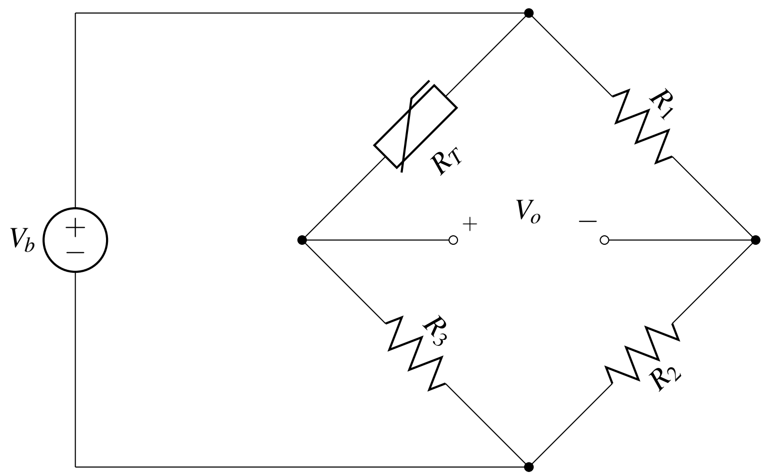
## 2. Thermistor

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:

$$R_T(T) = R(T_0)\exp\left(\beta\left(\frac{1}{T} - \frac{1}{T_0}\right)\right) \tag{3}$$

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




**Solution:** If you solved this problem with the old figure, where $V_b$ was flipped, then you will still receive full credit for accurate solutions, as long as you are consistent with the circuit you used.

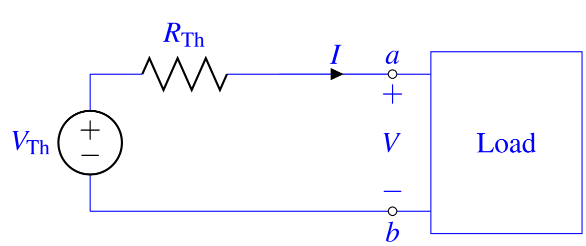
The temperature response of a thermistor is given in the table below:

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.

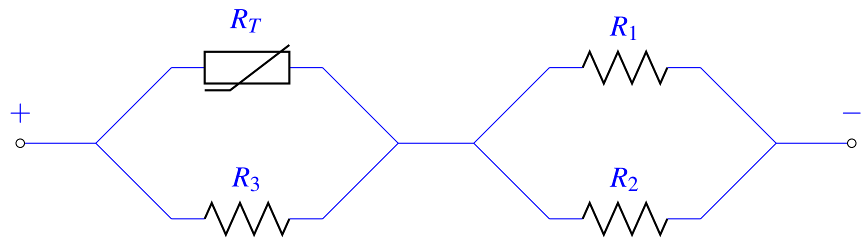
**Solution:** Thevenin equivalent circuit:



Thevenin equivalent voltage, $V_{Th} = V_{oc} = V_o$

$$V_o = V_+ - V_- = V_b \left( \frac{R_3}{R_T + R_3} - \frac{R_2}{R_1 + R_2} \right)$$

Thevenin equivalent resistance, $R_{Th}$:




$$R_{Th} = R_T \parallel R_3 + R_1 \parallel R_2$$

(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.
**Solution:**

$$V_o = V_+ - V_- = V_b \left( \frac{R_3}{R_T + R_3} - \frac{R_2}{R_1 + R_2} \right)$$
$$R_T = R_3 \frac{V_b R_1 - V_o (R_1 + R_2)}{V_b R_2 + V_o (R_1 + R_2)}$$

(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.
**Solution:** If $R_T = R_1 = R_2 = R_3$, the bridge is balanced, therefore $V_+ = V_-$. This results in $V_o = 0$. If the temperature increases $R_T$ will decrease, that means $V_+ > V_-$ because $R_1, R_2, R_3$ do not have temperature dependence. So $V_o > 0V$. On the other hand, if the temperature decreases, $R_T$ will increase resulting in $V_+ < V_-$ and $V_o < 0V$.

(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 **??** and the bridge circuit equation are required for this question.
**Solution:**
From the thermistor relationship,

$$\frac{1}{R_T} \frac{dR_T}{dT} = \frac{-\beta}{T^2}$$

From the bridge relationship,

$$V_o = V_b \left( \frac{R_3}{R_T + R_3} \right) - V_-$$
$$R_T \frac{dV_0}{dR_T} = -\frac{V_b R_3 R_T}{(R_3 + R_T)^2} = -\frac{V_b \alpha}{(1 + \alpha)^2}$$

Combining previous equations,

$$Q = \frac{V_b \alpha \beta}{(1+\alpha)^2 T^2}$$

For maximizing $Q$, we need to set $dQ/d\alpha = 0$ and find that the maximum occurs at $\alpha = 1$.

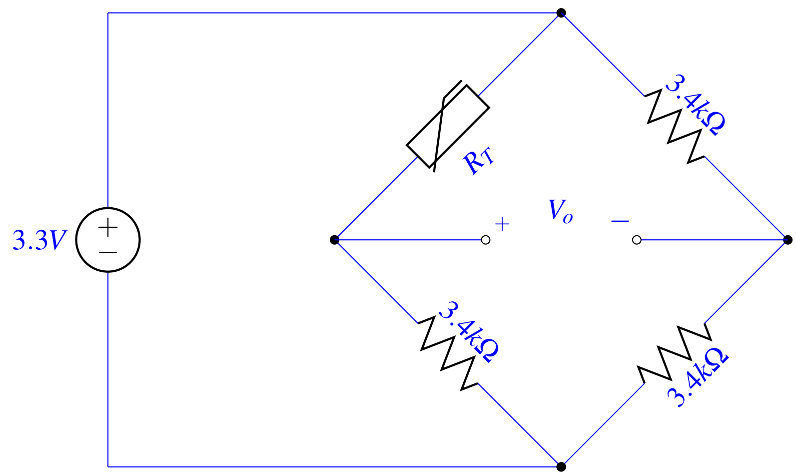
$$\frac{dQ}{d\alpha} = \frac{V_b \beta (1-\alpha)}{T^2 (1+\alpha)^3} = 0$$

Therefore, $R_3 = R_T$ will maximize sensitivity of the bridge.

(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 $R_3$ [$V_b = 3.3V$].

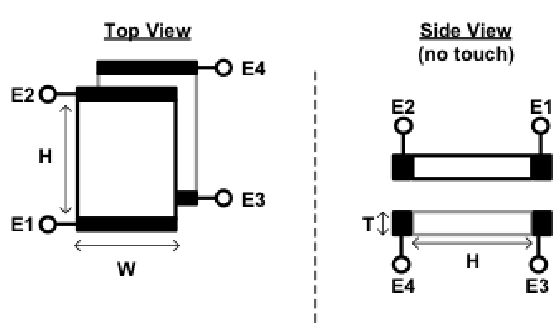
**Solution:** At 30°C, $R_T = 3.4k\Omega$. Therefore, $R_2 = R_3 = 3.4k\Omega$ will provide the highest bridge sensitivity. We choose $R_1 = R_3$ so that the bridge is balanced, and $V_o = 0\,\text{V}$ at 30°C. We prefer having a balanced bridge, so that $V_o$ will wiggle around 0 for varying temperatures.

Any other value of $R_1$ will be accepted, as long as you describe how this will affect $V_0$.




## 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 $E_4$)?

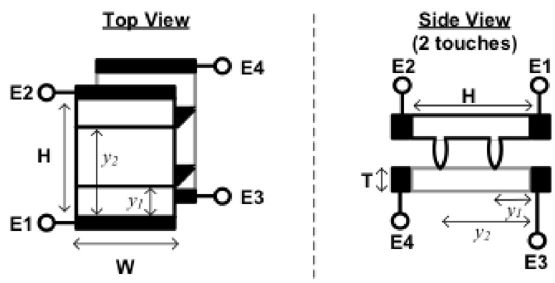
**Solution:**

$$R = \rho \cdot \frac{L}{A} \implies R_{E1-E2} = \rho \left( \frac{H}{W \cdot T} \right)$$

$$R_{E1-E2} = 1\Omega m \left( \frac{12 \times 10^{-2} m}{3 \times 10^{-2} \cdot 0.5 \times 10^{-3} m} \right)$$

$$R_{E1-E2} = 8k\Omega$$

(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.



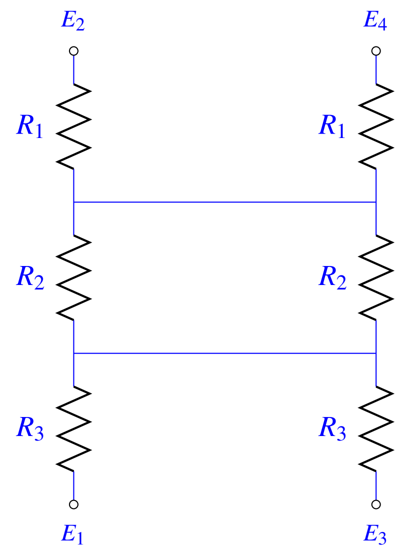

**Solution:**

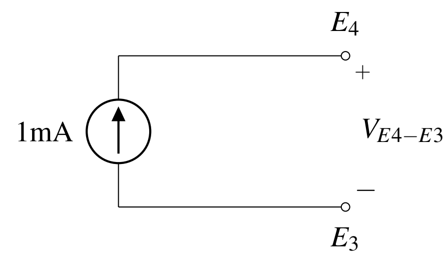
$$R_3 = \frac{3cm}{12cm} \cdot R_{E2-E1} = 1.25k\Omega$$

$$R_2 = \frac{7cm - 3cm}{12cm} \cdot R_{E2-E1} = 1.667k\Omega$$

$$R_1 = \frac{12cm - 7cm}{12cm} \cdot R_{E2-E1} = 2.0833k\Omega$$

(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. $V_{E4-E3}$)?




**Solution:** The equivalent resistance between $E_4 - E_3$ is

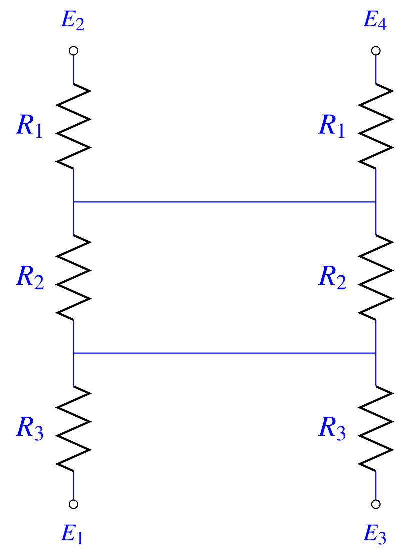
$$R_{E4-E3} = R_1 + R_2 \parallel R_2 + R_3 = R_1 + \frac{R2}{2} + R_3$$
$$= 1.25k\Omega + \frac{1.667k\Omega}{2} + 2.0833k\Omega$$
$$\approx 4.167k\Omega$$
$$V_{E4-E3} = 1mA \cdot R_{E4-E3} \implies V_{E4-E3} = 4.167V$$

(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.

**Solution:**

For general

$$R_3 = \frac{y_1}{12cm} \cdot 5k\Omega$$
$$R_2 = \frac{y_2 - y_1}{12cm} \cdot 5k\Omega$$
$$R_1 = \frac{12cm - y_2}{12cm} \cdot 5k\Omega$$




$$R_{E4-E3} = R_1 + \frac{R_2}{2} + R_3 = \left(12cm - y_2 + \frac{y_2 - y_1}{2} + y_1\right) \cdot \frac{5k\Omega}{12cm}$$
$$= \left(12cm + \frac{y1}{2} - \frac{y2}{2}\right) \cdot \frac{5k\Omega}{12cm}$$

So

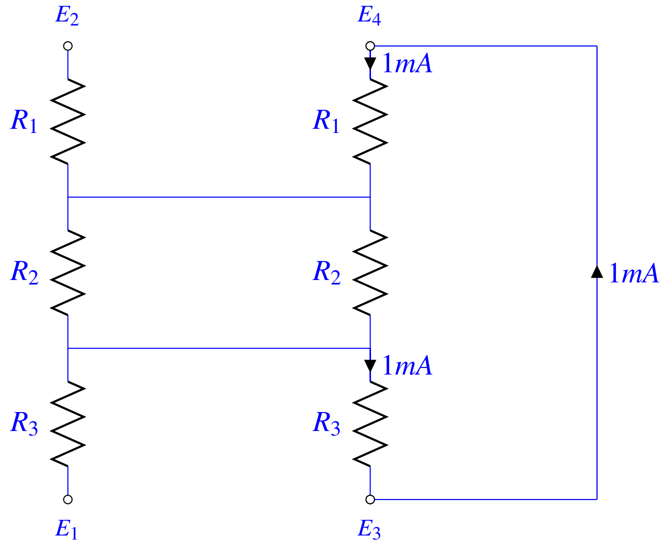
$$V_{E4-E3} = \frac{12cm - (y_2 - y_1)/2}{12cm} \cdot 5V$$

This means that by measuring $V_{E4-E3}$, we can only measure the distance between the two touch points $(y_2 - y_1)$.

(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.)

**Solution:**

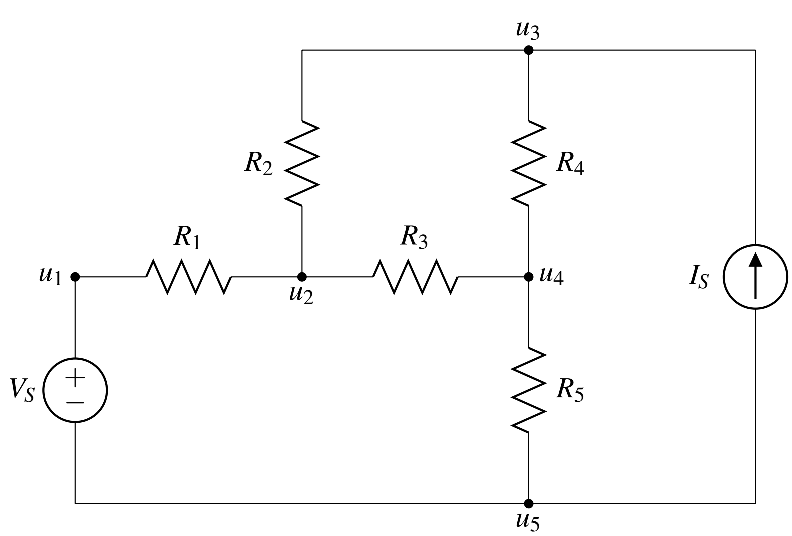



$$V_{E4-E2} = I \cdot R_1 = \frac{12cm - y_2}{12cm} \cdot 5V$$
$$V_{E1-E3} = I \cdot R_3 = \frac{y_1}{12cm} \cdot 5V$$

**4. SPICE-y Circuits**

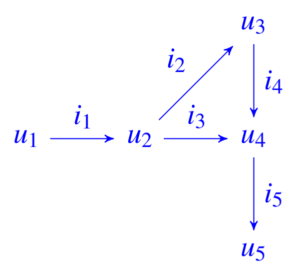
In the 1970s, Laurence Nagel and his advisor Donald Pederson at UC Berkeley created a circuit simulation software called Simulation Program with Circuit Emphasis or SPICE. Today, SPICE is the industry standard for circuit simulation. DC analysis in SPICE is performed using linear algebra tools you are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

**Solution:**




The incidence matrix for the above graph is as follows:

$$\mathbf{F} = \begin{bmatrix} 1 & -1 & 0 & 0 & 0 \\ 0 & 1 & -1 & 0 & 0 \\ 0 & 1 & 0 & -1 & 0 \\ 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 0 & 1 & -1 \end{bmatrix}$$

(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?

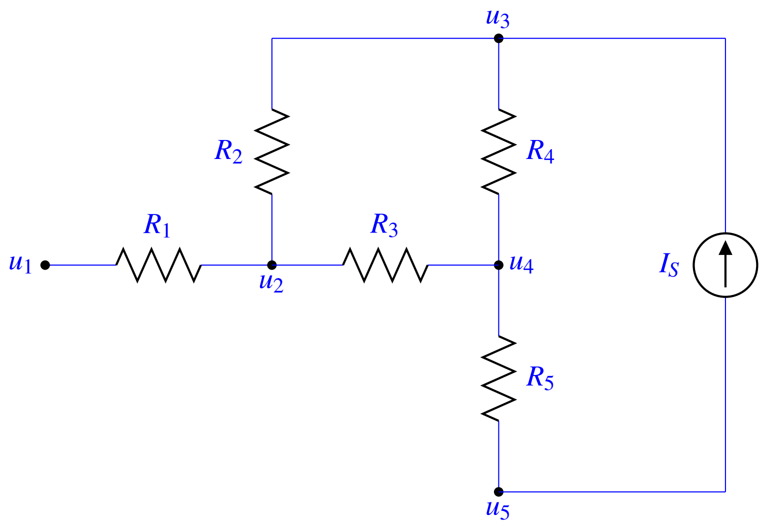
**Solution:** Let the vector $\vec{i}$ represent the flows through each edge. The product $\mathbf{F}^T\vec{i}$ is shown below:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ -1 & 1 & 1 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & -1 & -1 & 1 \\ 0 & 0 & 0 & 0 & -1 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \\ i_4 \\ i_5 \end{bmatrix} = \begin{bmatrix} i_1 \\ -i_1 + i_2 + i_3 \\ -i_2 + i_4 \\ -i_3 - i_4 + i_5 \\ -i_5 \end{bmatrix}$$

The above expression does not include the currents from the independent sources. Specifically, at node $u_1$, the KCL equation would be $-I_{V_s} + i_1 = 0$ and at node $u_5$ the KCL equation would be $I_{V_s} + I_s - i_5 = 0$.

(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 $u_3$.

**Solution:**




At node $u_3$: $-i_2 + i_4 - I_s = 0$
At node $u_5$: $-i_5 + I_s = 0$

(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).

**Solution:** From part (b), we know that the product $\mathbf{F}^T\vec{i} = \begin{bmatrix} i_1 \\ -i_1 + i_2 + i_3 \\ -i_2 + i_4 \\ -i_3 - i_4 - i_5 \\ -i_5 \end{bmatrix}$.

We want the third and fifth element of this vector to have an $I_s$ term in it. Therefore, the vector $\vec{b}$ is as shown below:

$$\vec{b} = \begin{bmatrix} 0 \\ 0 \\ +I_s \\ 0 \\ -I_s \end{bmatrix}$$

(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.

**Solution:**

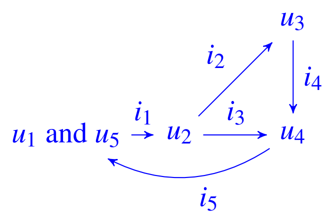
$$\mathbf{F} = \begin{bmatrix} 1 & -1 & 0 & 0 & 0 \\ 0 & 1 & -1 & 0 & 0 \\ 0 & 1 & 0 & -1 & 0 \\ 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 0 & 1 & -1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{bmatrix} = \begin{bmatrix} u_1 - u_2 \\ u_2 - u_3 \\ u_2 - u_4 \\ u_3 - u_4 \\ u_4 - u_5 \end{bmatrix}$$

We see the elements of the resulting vector correspond to the voltages across $R_1$, $R_2$, $R_3$, $R_4$, and $R_5$ respectively.

(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 *supern-*

*ode*. 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.

**Solution:**




The incidence matrix for the graph above is as follows:

$$\mathbf{F} = \begin{bmatrix} 1 & -1 & 0 & 0 \\ 0 & 1 & -1 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 1 & -1 \\ -1 & 0 & 0 & 1 \end{bmatrix}$$

Notice in the above matrix that when SPICE combined two nodes into a supernode, it summed the corresponding columns of the incidence matrix.

(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.

**Solution:** After combining the two nodes, we will assume the potential of the super-node is the potential of $u_5$. Note that it is possible to solve this circuit by assuming the potential at the super-node is the potential of $u_1$. This will lead to different $\vec{c}$ vectors and affect later parts. The final answer between the two should be the same.

Consider the new incidence matrix $\mathbf{F}$. The product $\mathbf{F}\vec{u}$ represents the voltage across all elements, except for elements connected to node 1. To fix this, the vector $\vec{c}$ will be added to $\mathbf{F}\vec{u}$. Since $R_1$ has a $u_5 + V_s$ volts across it, the vector $\vec{c}$ will be:

$$\vec{c} = \begin{bmatrix} V_s \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

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

**Solution:**

$$\mathbf{R} = \begin{bmatrix} R_1 & 0 & 0 & 0 & 0 \\ 0 & R_2 & 0 & 0 & 0 \\ 0 & 0 & R_3 & 0 & 0 \\ 0 & 0 & 0 & R_4 & 0 \\ 0 & 0 & 0 & 0 & R_5 \end{bmatrix}$$

$\mathbf{F}$ is the incidence matrix with the super node. $\vec{v} = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}$, and $\vec{i} = \begin{bmatrix} i_1 \\ i_2 \\ i_3 \\ i_4 \\ i_5 \end{bmatrix}$. From Ohm's law: $\vec{v} = \mathbf{R}\vec{i}$.

Since $\vec{v} = \mathbf{F}\vec{u} + \vec{c}$, we claim that: $\mathbf{F}\vec{u} + \vec{c} = \mathbf{R}\vec{i}$.

(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 $\mathbf{A}$, $\mathbf{B}$, $\mathbf{C}$, and $\mathbf{D}$ 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.

**Solution:** Symbolically the block matrix is shown below:

$$\begin{bmatrix} -\mathbf{R} & \mathbf{F} \\ \mathbf{F}^T & \mathbf{0} \end{bmatrix} \begin{bmatrix} \vec{i} \\ \vec{u} \end{bmatrix} = \begin{bmatrix} -\vec{c} \\ \vec{b} \end{bmatrix}$$

$$\begin{bmatrix} -R_1 & 0 & 0 & 0 & 0 & 1 & -1 & 0 & 0 \\ 0 & -R_2 & 0 & 0 & 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & -R_3 & 0 & 0 & 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & -R_4 & 0 & 0 & 0 & 1 & -1 \\ 0 & 0 & 0 & 0 & -R_5 & -1 & 0 & 0 & 1 \\ 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ -1 & 1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & -1 & 1 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \\ i_4 \\ i_5 \\ u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} -V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ -I_s \\ 0 \\ I_s \\ 0 \end{bmatrix}$$

Note: An equivalent representation of the above matrix is shown below:

$$\begin{bmatrix} \mathbf{0} & \mathbf{F}^T \\ \mathbf{F} & \textbf{-R} \end{bmatrix} \begin{bmatrix} \vec{u} \\ \vec{i} \end{bmatrix} = \begin{bmatrix} \vec{b} \\ -\vec{c} \end{bmatrix}$$

(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 $\mathbf{F}$ has full column rank. Verify that the matrix $\mathbf{F}$ does not have full column rank, and show that $\vec{1}$ is in the null space of *any* incidence matrix.

**Solution:** The matrix $\mathbf{R}$ is a diagonal matrix. A diagonal matrix will always be invertible as long as none of the entries are zero. The only way to get a zero entry in $\mathbf{R}$ would be to have a resistance of value $0\,\Omega$.

For the block matrix to be invertible, the matrix $\mathbf{F}$ must have full column rank, this would then fulfill the pivot in every column requirement. Similarly, $\mathbf{F}$ having a pivot in every column is equivalent to $\mathbf{F}^T$ having a pivot in every row, which again satisfies the pivot in every row requirement for an inveritble matrix.

Row reducing the matrix $\mathbf{F}$:

$$\begin{bmatrix} 1 & -1 & 0 & 0 \\ 0 & 1 & -1 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 1 & -1 \\ -1 & 0 & 0 & 1 \end{bmatrix} \sim \begin{bmatrix} 1 & -1 & 0 & 0 \\ 0 & 1 & -1 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 1 & -1 \\ 0 & -1 & 0 & 1 \end{bmatrix} \sim \begin{bmatrix} 1 & -1 & 0 & 0 \\ 0 & 1 & -1 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 1 & -1 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

This is in fact true for any incidence matrix represinting a connected graph. Consider one row of such an incidence matrix. Since each row corresponds to an edge in the graph, it must only have two non-zero entries, a $+1$ and a $-1$. If this matrix is applied on a uniform vector, each row of the resulting vector would be zero. Any uniform vector is then in the nullspace, and the space of uniform vectors is spanned by the one vector: $\vec{1}$.

(k) Because $\vec{1}$ is in the null space of $\mathbf{F}$, 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 $\mathbf{F}$. Write the new block matrix representing the system.

**Solution:** The matrix below creates $u_4$ as the ground node, not something a human would do while solving the circuit. Selecting a different ground node leads to different answers but all voltages(difference in potentials) should be equal regardless of choice of ground.

$$\begin{bmatrix} -R_1 & 0 & 0 & 0 & 0 & 1 & -1 & 0 \\ 0 & -R_2 & 0 & 0 & 0 & 0 & 1 & -1 \\ 0 & 0 & -R_3 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -R_4 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & -R_5 & -1 & 0 & 0 \\ 1 & 0 & 0 & 0 & -1 & 0 & 0 & 0 \\ -1 & 1 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \\ i_4 \\ i_5 \\ u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} -V_s \\ 0 \\ 0 \\ 0 \\ 0 \\ -I_s \\ 0 \\ I_s \end{bmatrix}$$

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

**Solution:** Again, with $u_4$ chosen as the ground node, $i_1 = -2.4mA$, $i_2 = -2.2mA$, $i_3 = 0.2mA$, $i_4 = 2.8mA$, $i_5 = 2.6mA$, $u_{1,5} = -15.6V$, $u_2 = -1V$, $u_3 = 5.6V$. The potential at the supernode is $u_{1,5}$. Since the supernode was taken to have the same potential as $u_5$, the potential at $u_1$ is $u_{1,5} + 5V = -10.6V$.

## 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.

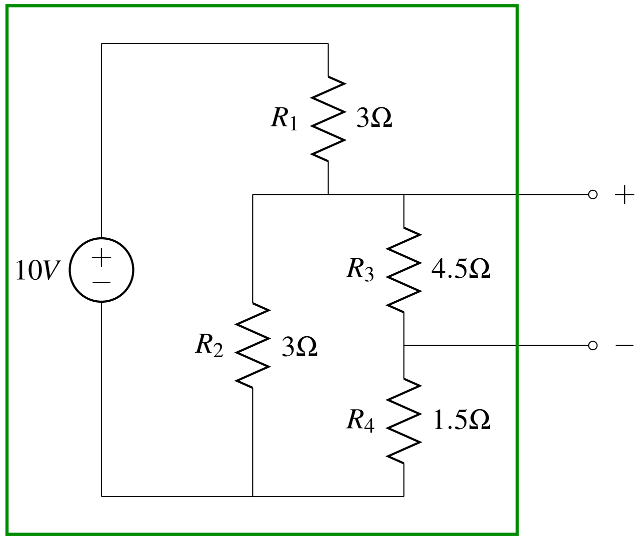
**Solution:** I worked on this homework with...

I first worked by myself for 2 hours, but got stuck on Problem 5 so I went to office hours on...

Then I went to homework party for a few hours, where I finished the homework.

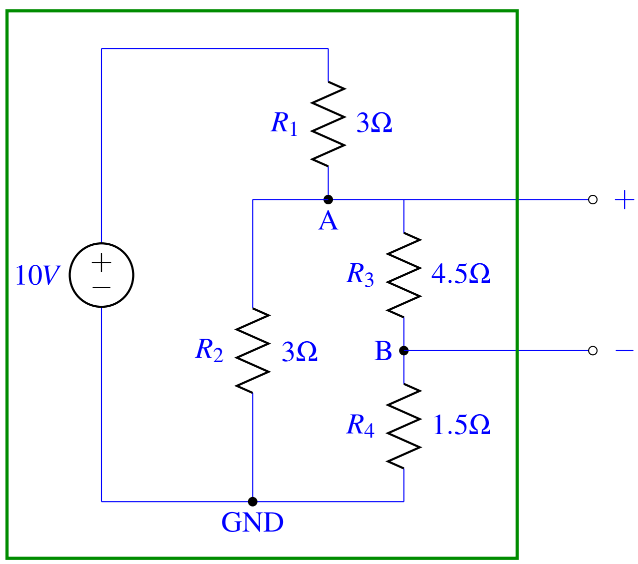
## 6. (PRACTICE) Thévenin and Norton equivalent circuits

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




**Solution:** To find the Thévenin and Norton equivalent circuits, we are going to find the open circuit voltage between the output ports and the current flowing through the output ports when the ports are shorted.

For finding the open circuit voltage between the output ports, let us label the nodes as shown in the figure below.




First, let us begin by calculating the effective resistance between nodes A and GND. We have $3\Omega$ resistor in parallel to $4.5\Omega + 1.5\Omega$ resistance. This gives an equivalent resistance of

$$\frac{1}{\frac{1}{3\Omega} + \frac{1}{4.5\Omega+1.5\Omega}} = 2\Omega$$

Then we see that we have a voltage divider from the positive terminal of 10V supply. Voltage divider is made up of two resistances in series, where the resistances are $3\Omega$ and $2\Omega$. This gives the voltage at node A equal to

$$V_{\text{A}} = 10V \frac{2\Omega}{3\Omega + 2\Omega} = 4V$$

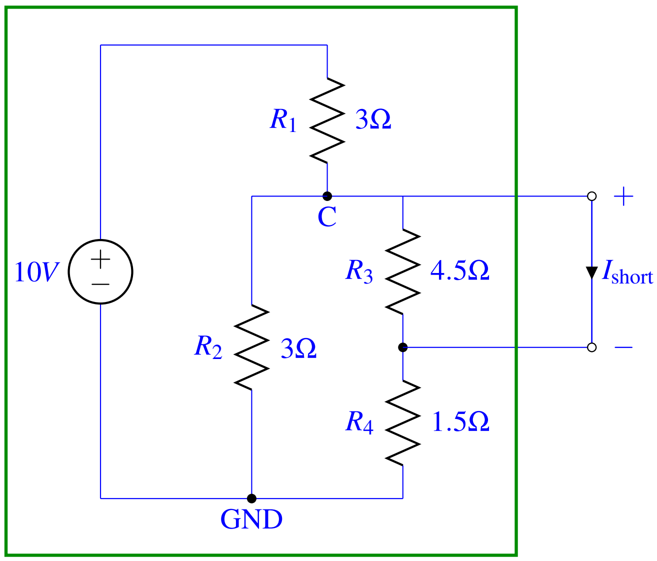
To find the voltage at node B, note that we have another voltage divider between nodes A and GND. Hence, we can find the voltage at node B as

$$V_{\text{A}} \frac{1.5\Omega}{4.5\Omega + 1.5\Omega} = 4V \frac{1}{4} = 1V$$

Hence the open circuit voltage seen between the output ports is equal to

$$\begin{aligned} V_{\text{open}} &= V_A - V_B \\ &= 4V - 1V \\ &= 3V \end{aligned}$$

Now let us find the short circuit current flowing through the output ports. When doing this, we get the following circuit.




Note that when we short the output terminals, the voltages at the nodes change, this is why we changed the label of the node below the resistor $R_1$. Since there is a short circuit parallel to the resistor $R_3$, there will be no current flowing through it, hence we have

$$I_{\text{short}} = I_{R_4}$$

To find this current, let us find the equivalent resistance due to $R_2$ being connected parallel to $R_4$ when we short the output ports. We have $3\Omega$ parellel to $1.5\Omega$, which gives an equivalent resistance

$$\frac{1}{\frac{1}{3\Omega} + \frac{1}{1.5\Omega}} = 1\Omega$$

We again have a voltage divider between the positive side of the 10V supply and the ground. Using this voltage divider, we calculate the voltage at node C as

$$V_C = 10V \frac{1\Omega}{3\Omega + 1\Omega} = 2.5V$$

Hence, we see that the voltage across the resistor $R_4$ is equal to $2.5V$. Using Ohm's law, we get

$$I_{R_4} = \frac{2.5V}{1.5\Omega} = \frac{5}{3}A$$

Since we have $I_{\text{short}} = I_{R_4}$, we have

$$I_{\text{short}} = I_{R_4}$$

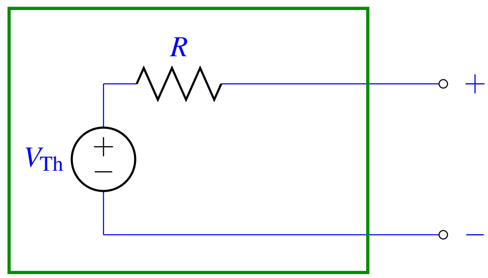
Summarizing the results, we have

$$V_{\text{open}} = 3V$$
$$I_{\text{short}} = \frac{5}{3}A$$

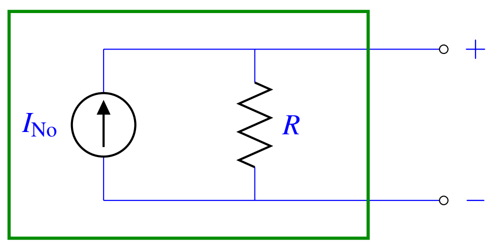
This gives

$$R_{\text{Th}} = \frac{V_{\text{open}}}{I_{\text{short}}} = \frac{9}{5}\Omega$$

Hence the Thévenin equivalent circuit is given by

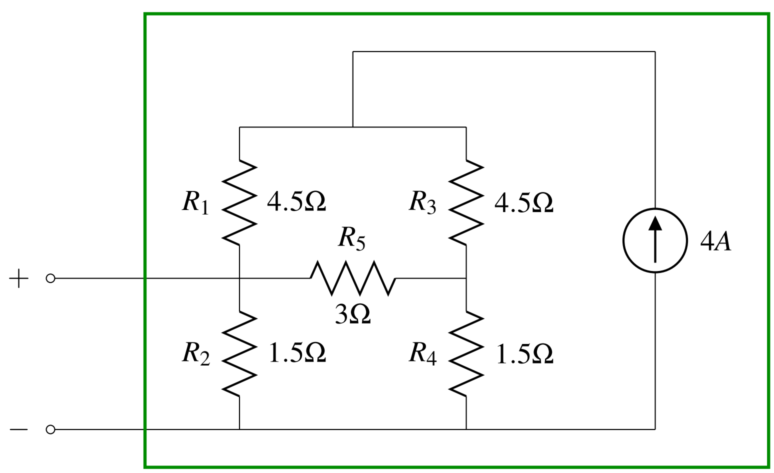



where $R = R_{\text{Th}}$ and $V_{\text{Th}} = V_{\text{open}}$, and the Norton equivalent circuit is given by




where $R = R_{\text{Th}}$ and $I_{\text{No}} = I_{\text{short}}$.

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



**Solution:** As with the previous part of this question, to find the Thévenin and Norton equivalent circuits we are going to find the open circuit voltage between the output ports and the current flowing through the output ports when the ports are shorted.

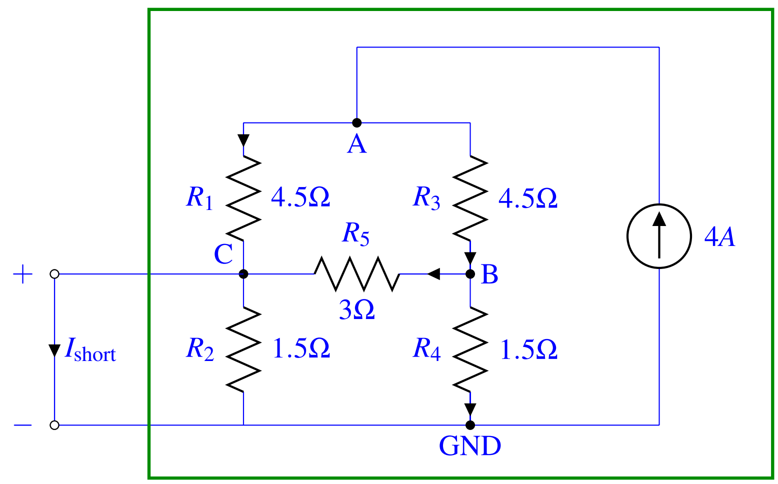
Let us first find the open circuit voltage between the output ports. In the solutions of homework 6, we have seen 3 different approaches for finding the voltages at each node of this same circuit. One of those approaches was to use the symmetry in the circuit to see there will be no current flowing through $R_5$. Then, we have a current divider where each branch has the same resistance, hence the current $4A$ divides equally between the left and right branches. Hence we have

$$I_{R_1} = I_{R_2} = I_{R_3} = I_{R_4} = 2A$$

This gives us the voltage across $R_2$, equivalently the open circuit voltage between the output terminals, equal to

$$V_{\text{open}} = 2A \times 1.5\Omega = 3V$$

Now let us find the short circuit current across the output terminals. Let us find this using nodal analysis on the resulting circuit when we short the output ports. To help do the analysis, let us label the nodes as shown in the figure below.




Now what are the unkown node voltages? We do not know the voltage at node A and B. On the other hand, because node C is connected by a short circuit to GND, we know its voltage is equal to the GND which we set as 0; hence voltage at node C is not an unkown. Next, because there is a short circuit across resistor $R_2$, there will be no current flowing through it.

Let us write KCL at the nodes

$$4A = I_{R_1} + I_{R_3} \qquad \text{(Node A)}$$
$$I_{R_3} = I_{R_4} + I_{R_5} \qquad \text{(Node B)}$$
$$I_{\text{short}} = I_{R_1} + I_{R_5} \qquad \text{(Node C)}$$

Now let us relate the currents $I_{R_1}$, $I_{R_2}$, $I_{R_3}$, $I_{R_4}$ and $I_{R_5}$ to node voltages using Ohm's law. We have

$$I_{R_1} = \frac{V_A - V_C}{R_1} = \frac{V_A}{4.5\Omega}$$

since $V_C = 0$ because it is connected to the ground by short circuit. Furthermore, we have

$$\begin{aligned} I_{R_2} &= 0, \\ I_{R_3} &= \frac{V_A - V_B}{R_3} = \frac{V_A - V_B}{4.5\Omega} \\ I_{R_4} &= \frac{V_B}{R_4} = \frac{V_B}{1.5\Omega}, \\ I_{R_5} &= \frac{V_B - V_C}{R_5} = \frac{V_B}{3\Omega} \end{aligned}$$

Plugging these into the first two KCL equations, we get

$$\begin{aligned} 4A &= \frac{V_A}{4.5\Omega} + \frac{V_A - V_B}{4.5\Omega} \\ \frac{V_A - V_B}{4.5\Omega} &= \frac{V_B}{1.5\Omega} + \frac{V_B}{3\Omega} \end{aligned}$$

These equations are solved by

$$\begin{aligned} V_A &= 9.9V, \\ V_B &= 1.8V \end{aligned}$$

Using the KCL at node C, we get

$$\begin{aligned} I_{\text{short}} &= I_{R_1} + I_{R_5} \\ &= \frac{V_A}{4.5\Omega} + \frac{V_B}{3\Omega} \\ &= \frac{9.9}{4.5\Omega} + \frac{1.8}{3\Omega} \\ &= 2.8A \end{aligned}$$

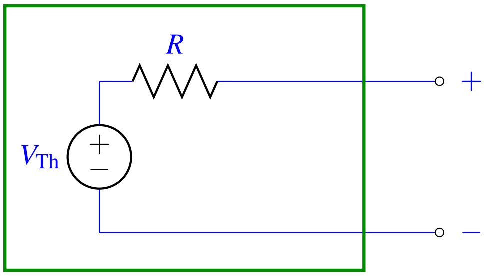
Summarizing the results, we have

$$\begin{aligned} V_{\text{open}} &= 3V \\ I_{\text{short}} &= 2.8A \end{aligned}$$

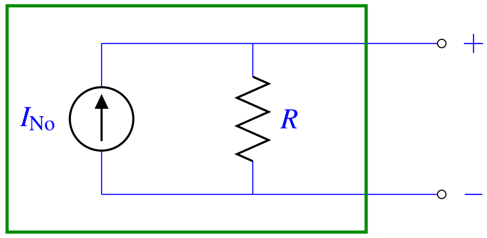
This gives

$$\begin{aligned} R_{\text{Th}} &= \frac{V_{\text{open}}}{I_{\text{short}}} \\ &= \frac{15}{14}\Omega \end{aligned}$$

Hence the Thévenin equivalent circuit is given by



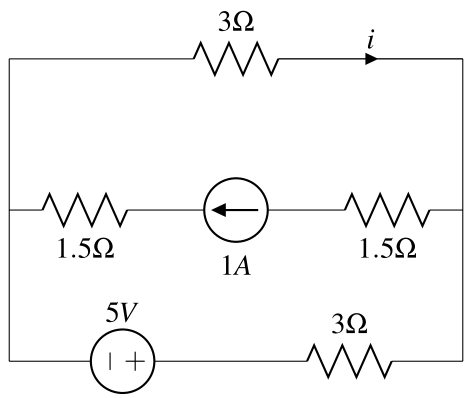
where $R = R_{\text{Th}}$ and $V_{\text{Th}} = V_{\text{open}}$, and the Norton equivalent circuit is given by




where $R = R_{\text{Th}}$ and $I_{\text{No}} = I_{\text{short}}$.

**7. (PRACTICE) 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?

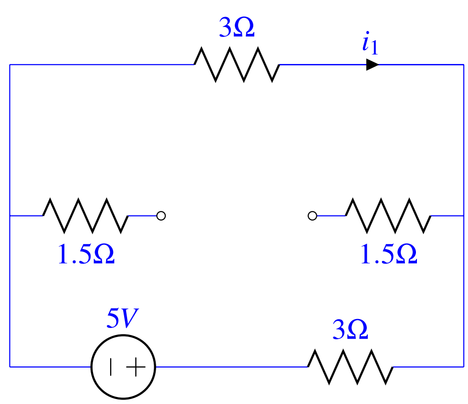



**Solution:** $i = -\frac{1}{3}A$.

**Method 1: Superposition**

Consider the circuits obtained by:

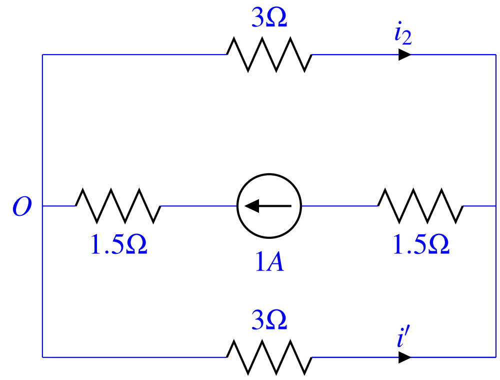
(a) Turning off the $1A$ current source:



In the above circuit, no current is going to flow through the middle branch, as it is an open circuit. Thus this is just a $5V$ voltage source connected to two $3\Omega$ resistors in series so

$$i_1 = -\frac{5}{6}A$$

(b) Turning off the $5V$ voltage source:




In the above circuit, notice that the $3\Omega$ resistors are in parallel and therefore form a current divider. Since the values of the resistances are equal, the current flowing through them will also be equal, that is $i_2 = i'$. Applying KCL to node $O$, we get

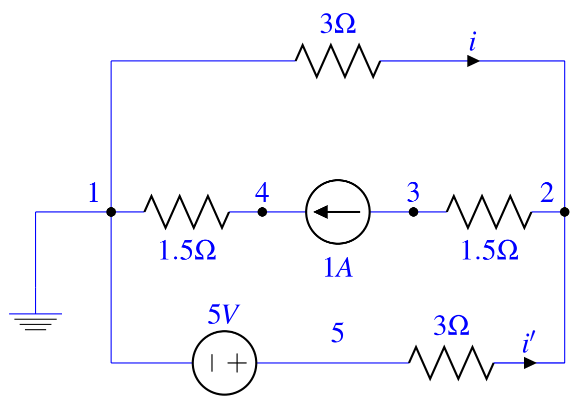
$$1 - i_2 - i' = 0$$

which gives us

$$i_2 = \frac{1}{2}A$$

Now, applying the principle of superposition, we have $i = i_1 + i_2 = -\frac{5}{6} + \frac{1}{2} = -\frac{1}{3}A$.

**Method 2: Nodal Analysis** First, let's identify and label the nodes on the circuit. (Note that the numbers are arbitrary.) We also ground node 1. (Our choice of ground is arbitrary (whatever you chose is fine) but node 1 is a convenient choice because node 5 will turn out to be 5V from the voltage source, the voltage at node 4 can be calculated quickly from the current source and the $1.5\Omega$ resistor using Ohm's law, and $i$ can be calculated quickly once we know the voltage at node 2. )



First, we write KCL at each node. At nodes 1 and 2, we get the same equation.

$$i + i' = 1A$$

At nodes 3 and 4, we get the trivial equation

$$1A = 1A.$$

We now write the voltage drops across the circuit elements in terms of the currents using Ohm's law or in terms of known voltages

$$\begin{aligned}
V_5 - V_1 &= 5 \\
V_5 - V_2 &= i'(3\Omega) \\
V_2 - V_3 &= 1A(1.5\Omega) \\
V_4 - V_1 &= 1A(1.5\Omega) \\
V_2 - V_1 &= -i(3\Omega)
\end{aligned}$$

Since we've chosen node 1 as ground ($V_1 = 0$), we can rewrite the equations involving $V_1$ which gives us values for $V_5$ and $V_4$.

$$\begin{aligned}
V_5 &= 5 \\
V_4 &= 1A(1.5\Omega) \\
V_2 &= -i(3\Omega)
\end{aligned}$$

We next combine the KCL equations and the Ohm's Law equations to solve for $V_2$ and $V_3$. (We don't actually need to solve for $V_3$ once we know $V_2$ but the calculation is easy.)

$$\begin{aligned}
-\frac{V_2}{3\Omega} + \frac{5V - V_2}{3\Omega} &= 1A \\
\implies V_2 &= 1V
\end{aligned}$$

and

$$\begin{aligned}
V_3 &= V_2 - 1A(1.5\Omega) \\
V_3 &= -0.5V
\end{aligned}$$

Finally, we use $V_2$ to solve for $i$:

$$i = -\frac{V_2}{3\Omega} = -\frac{1}{3}A$$