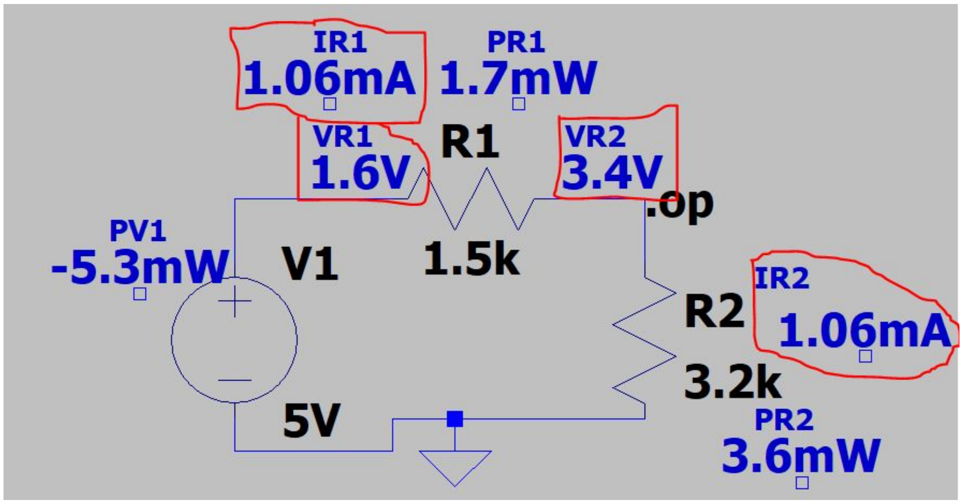
Proof of Concepts Alpha Lab 1: Jack Jiang, Vincet Lin, Saaif Ahmed

A1.Concept name: Proving Ohm's Law

Building Block: Short description and schematic.



Nodes to be Analyzed: VR2, IR2, VR1, IR1

Description: The circuit has 2 resistors (R1 & R2) of values 1.5k Ohms and 3.2k Ohms respectively. The resistors are connected in series to a 5V DC power supply.

Analysis:

Equation and short description. Describe clearly how you are applying the concept

Ohm's Law is stated as follows: $V = R * I$. The Voltage across an Ohmic conductor is equal to its resistance R multiplied by the current I running through it. In the screenshot of the simulation we analyze the two Resistors R1 and R2 and the current and voltage running through them (VR2, IR2, VR1, IR1).

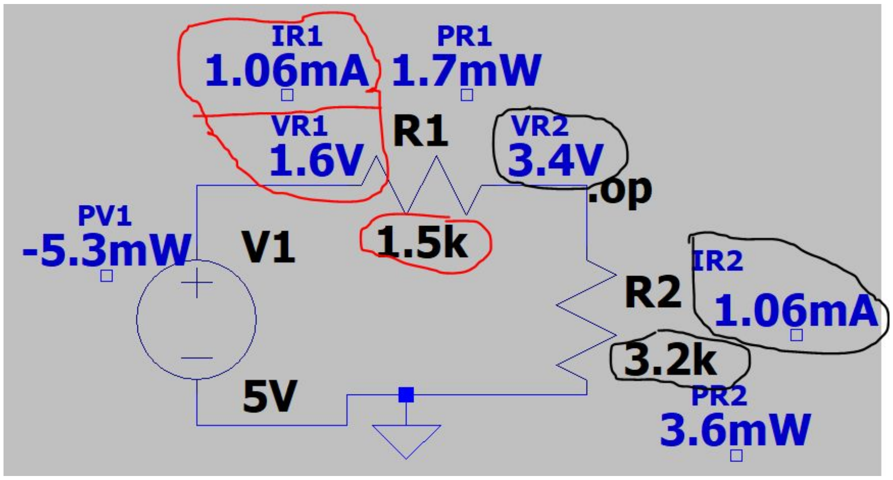
Applying Ohm's Law to R1: $VR1 = IR1 * R1$; $VR1 = 1.06mA * 1500\,\Omega$; $VR1 = 1.6\,V$

This matches what the VR1 node has labeled for the voltage across R1

Applying Ohm's Law to R2: $VR2 = IR2 * R2$; $VR2 = 1.06mA * 3200\,\Omega$; $VR2 = 3.4\,V$

This matches what the VR2 node has labeled for the voltage across R2

Simulation



Specific colored nodes and values/associated computation shown in above analysis section.

Red Colored Nodes and Output: Represent values used for Applying Ohm's law to R1

Black Colored Nodes and Output: Represent values used for Applying Ohm's law to R2

Measurement:

```
        --- Operating Point ---

V(n002):        3.40426         voltage
V(n001):        5               voltage
I(R2):          0.00106383      device_current
I(R1):          -0.00106383     device_current
I(V1):          -0.00106383     device_current
```

Table below added for further clarification

Resistance	Measured Current	Expected Voltage	Measured Voltage
R1: 1500 Ω	0.00106 mA	1.6 V	VR1 = 5-V(n002) = 1.6 V
R2: 3200 Ω	0.00106 mA	3.4 V	VR2 = V(n002) = 3.40 V

Discussion (and answer related questions in Alpha Lab):

Comparison of Analysis, Simulation and Measurement results. Both a simple summary of results (like a numerical chart of values) and a simple description that details if the results are as you expect. Also include any speculations as to why they may be different from one another if they are different. What variation is too much for example…explore this.

Analysis took the circuit schematic and showed how Ohm's Law would be applied to the Resistors to determine the Voltage Across them. The Simulation screenshot showed that the Analysis calculations matched the simulation screenshots. The Measurements were run through LTSpice and these too did match the Analysis and Simulation. Any error that would have been made would be through rounding error/significant figure errors.

Questions from Lab

A2: 1a:

We now change the resistors to R1 = 22k ohms and R2 = 42k ohms. The expected Current running through the circuit should be $V/(R1+R2) = I;\ 5V/(22k + 42k) = 80\mu A$. Our simulation matches this expectation. According to Ohm's Law the Voltage across each resistor can be shown as $VR = IR * R$. Because the components are in series, $IR2 = IR1 = I_{circuit}$. The expected voltages are:

$VR1 = IR1 * R1 = 80\mu A * 22000\Omega = 1.72\ V$

$VR2 = IR2 * R2 = 80\mu A * 42000\Omega = 3.28\ V$.

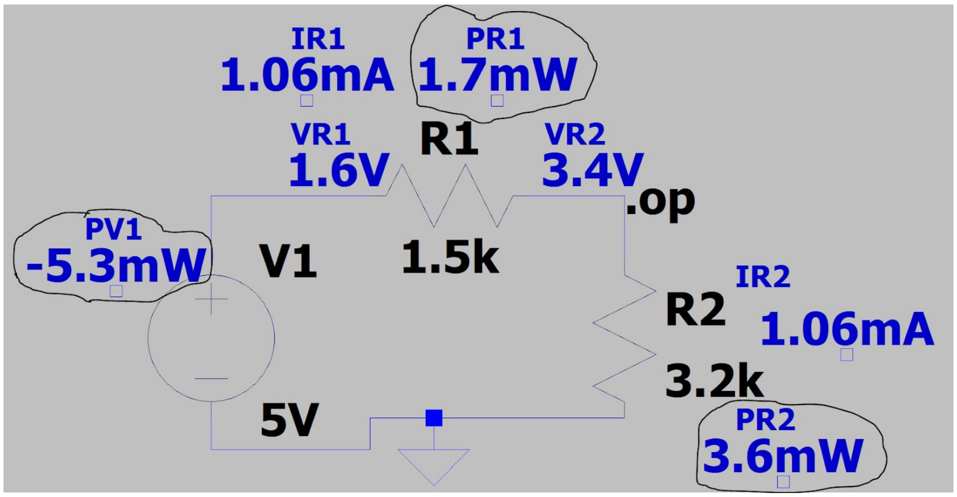
These expected values match our simulated values.

A2: 2a:

We can also confidently say that as a resistor's value increases the current through the circuit decreases because $V/R = I$. This is shown in both analysis and simulation where current changes to $80\mu A$.

A2. Prove that power supplied must equal power absorbed.

Building Block: Short description and schematic.



Nodes to be Analyzed: PV1, PR1, PR2

Description: The circuit has 2 resistors (R1 & R2) of values 1.5k Ohms and 3.2k Ohms respectively. The resistors are connected in series to a 5V DC power supply.

Analysis:

Equation and short description. Describe clearly how you are applying the concept

Power is defined as $P = W/s$ or in the case of electricity $P = J/q * q/s$ which can be restated as $P = V * I$. Thus the power through a component is equal to the Voltage across the component multiplied by the current running through the component. We must define polarity for power supplied and power absorbed. We state that power supplied shall be negative (-) and that power absorbed shall be positive (+). Thus in our circuit the 5V power supply will supply power and our 2 resistors will absorb the power.

$P(source) = V * I$; $P(source) = 5V * -1.06mA$; $P(source) = -5.3mW$

$P(R1) = VR1 * IR1$; $P(R1) = 1.6V * 1.06mA$; $P(R1) = 1.7mW$

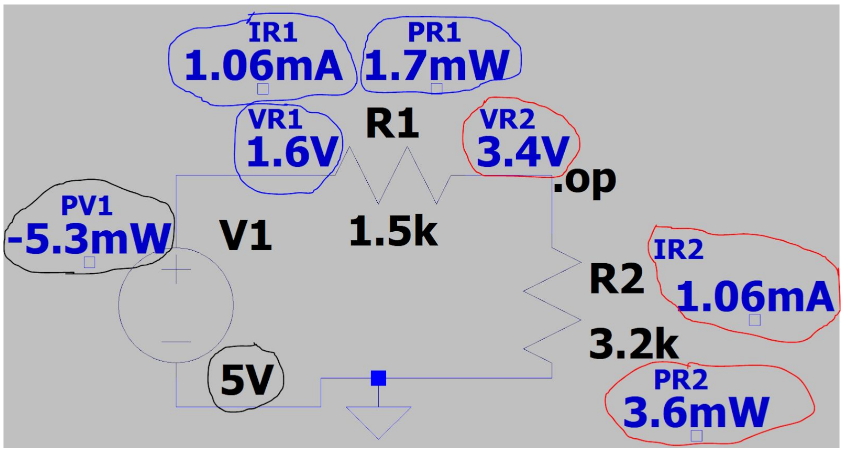
$P(R2) = VR2 * IR2$; $P(R2) = 3.4V * 1.06mA$; $P(R2) = 3.6mW$

If we add the components that are absorbing power (R1 & R2) and add them to the power supplied by the source we should get 0.

$P(source) + P(R1) + P(R2) = 0$; $-5.3mW + 1.6mW + 3.7mW = 0$; $0 = 0$

This is in line with the Law of Conservation of Energy where the energy supplied is the energy absorbed by the system.

Simulation:



Black Circles: Power supplied, Blue Circles: Power absorbed by R1, Red Circles: Power Absorbed by R2

It is shown clearly how each component's respective Voltage and Current multiplied give the power.

$IR1 * VR1 = PR1$; $IR2 * VR2 = PR2$ and the sum of $PR1 + PR2 = PV1$.

Measurement:

There are no measurements in Part A. We do not build the circuit.

Discussion:

Because there was no measurement we compare analysis and simulation. Values were calculated using Ohm's Law. Our Analysis and simulation match strongly. The calculated Power Supplied and Absorbed matches the simulated Power Supplied and Absorbed.

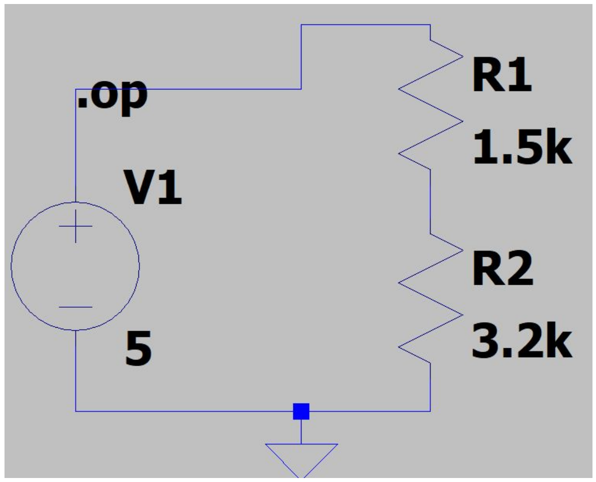
Questions from lab

A2: 1b, A2: 2a, A2: 2b:

We change the R1 and R2 values to 22k and 42k respectively. We calculated the new current to be $80\mu A$. We used Ohm's Law to find the Voltages across the resistors R1 and R2. Thus the power absorbed by the circuit is $P = I * I(R1+R2) = 80\mu A * 80\ \mu A\,(64000\Omega) = 400\ \mu W$. So the power dropped. The reason is because $P = V * I$ and $I = V/(R)$. When you increase R, I decreases and thus P decreases. If you revert the resistance values then the power increases back to what it was initially in simulation and in analysis. This is because if you decrease R, I increases and thus P increases

A3: Prove how Polarity Works in LTSpice

Building Block:



Nodes to be Analyzed: R2, R1

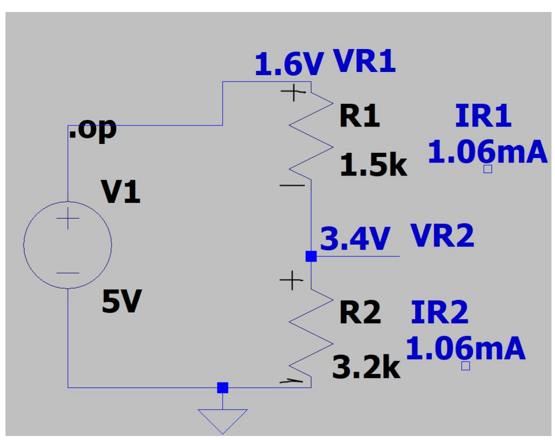
Description: The circuit has 2 resistors (R1 & R2) of values 1.5k Ohms and 3.2k Ohms respectively. The resistors are connected in series to a 5V DC power supply.

Analysis:

Polarity is crucial when defining a circuit. By convention we establish that the reference point of direction is from positive (+) to negative (-). Which means that when we observe, for example, that current is flowing from negative to positive we state that value as negative. This is useful for things like KVL and KCL. First we use Ohm's Law to find the Voltage across each resistor.

$$VR1 = I * R1 = \frac{5V}{3200 + 1500\,\Omega} * 1500\Omega = 1.6V$$

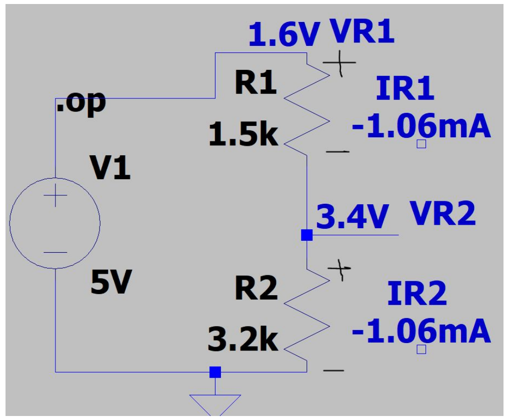
$$VR2 = I * R2 = \frac{5V}{3200 + 1500\,\Omega} * 3200\Omega = 3.4V$$



Now we take KVL around the circuit going clockwise. The equation is as follows:

$$-V1 + VR1 + VR2 = 0\,;\; -5V + 1.6V + 3.4V = 0\,;\; 0 = 0$$

We know that for resistors, polarity is arbitrarily assigned. Which means that if we flip the resistor in LTSpice it should not change our values. We show that this is not the case.

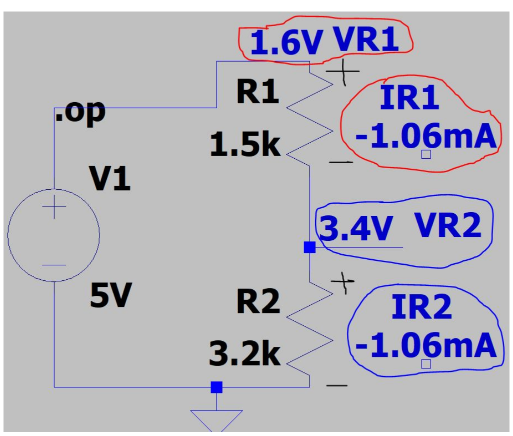


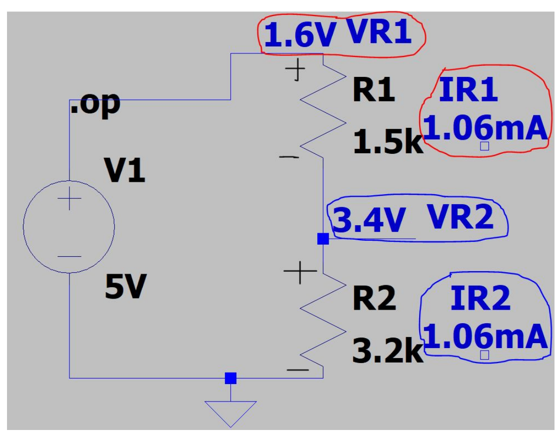
The current values have changed and because our assignment stayed consistent our values for VR1 and VR2 must also change. Thus taking KVL around the circuit going clockwise shows:

$$-V1 + VR1 + VR2 = 0\,;\; -5V - 1.6V - 3.4V = -10V\,;\; -10V \neq 0$$

This shows that polarity in LTSpice is not conventional and is specific to the program. We show this in the next part: simulation.

Simulation:





Here are the 2 circuits side by side. Red Circles refer to R1 and it's values. Blue Circles refer to R2 and it's values. Resistor's resistances and positions are the same, they are just flipped horizontally in LTSpice.

It is clear that the polarity is not the same when we do this, even though in real life resistors polarities are arbitrary. This leads to the conclusion that in LTSpice, they strictly assign, to every component a positive terminal and a negative terminal. This also shows that when a component is selected, rotating it does affect where the negative and positive terminals are.

Measurement:

There are no measurements in Part A. We do not build the circuit.

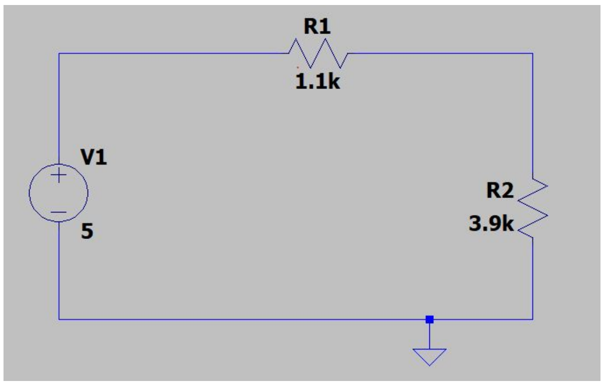
Discussion:

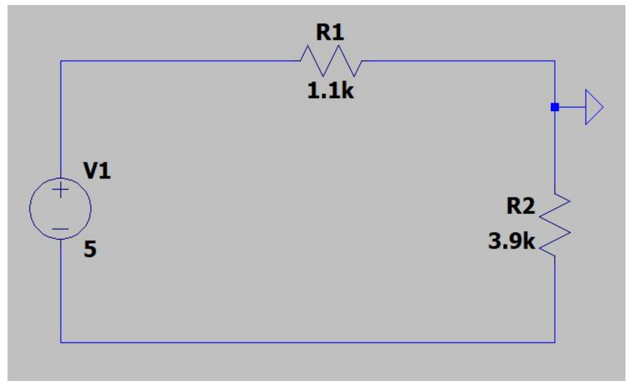
Because there was no measurements we use Analysis and Simulation to prove how polarity works in LTSpice. We discovered that rotating a normally arbitrary component in LTSpice does affect the polarity of the component in the simulation. This is shown through simulation pictures and calculated in the analysis section. Our conclusion was that LTSpice actually hard defines one terminal to be positive and one terminal to be negative in its calculations. This is why the current shown by LTSpice was negative or positive. The results do support this conclusion.

A4. Prove how the placement of ground doesn't affect the circuit

Building Block:

Two circuit with the same component (5V source, 1.1k ohm resistor, 3.9k ohm resistor) and different position of ground.

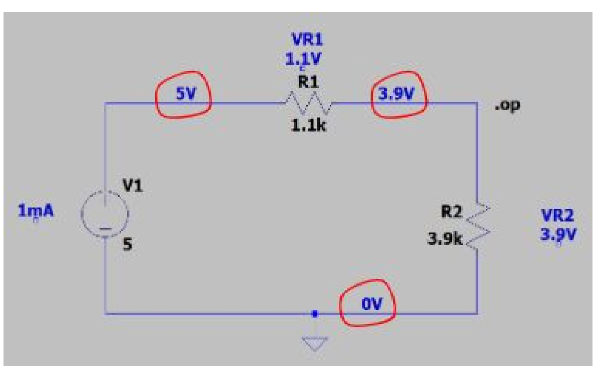


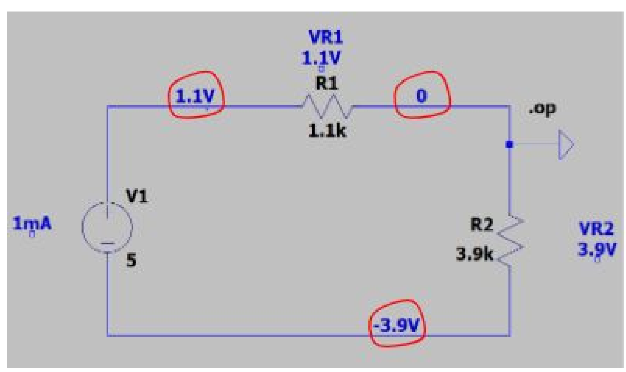


Analysis:

Ground is only the point we define 0V in a circuit, so the placement of ground doesn't affect circuit. It could only influence the relative voltage shown on the circuit.

Simulation:





--- Operating Point ---

V(n001):	5	voltage
V(n002):	3.9	voltage
I(R2):	-0.001	device_current
I(R1):	-0.001	device_current
I(V1):	-0.001	device_current

--- Operating Point ---

V(n001):	1.1	voltage
V(n002):	-3.9	voltage
I(R2):	-0.001	device_current
I(R1):	-0.001	device_current
I(V1):	-0.001	device_current

Measurement:

No measurement for Part A.

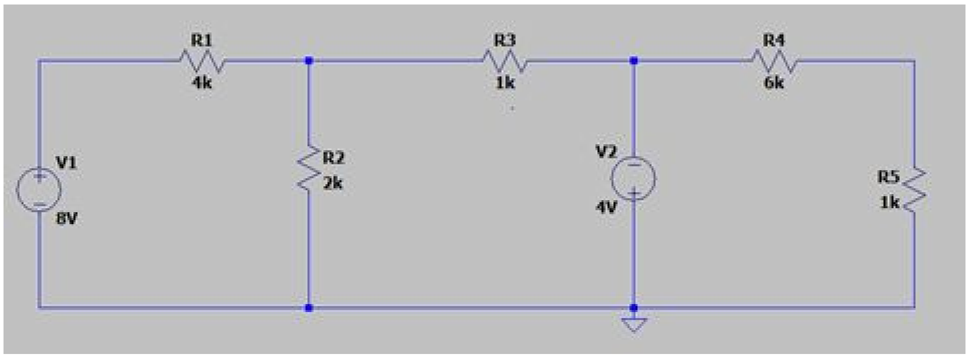
Discussion:

According to the simulation, since ground moves from the bottom node to the right side node, the voltage on the bottom node no longer be 0V but -3.9V instead. But if we calculate the voltage across each resistor, it is the same for both the position of the ground. Besides, we can see that all the current flow through each resistor keeps the same and the resistor value also keeps the same, then the voltage across each device should keep the same. Then we prove that the placement of ground does not affect the circuit.

A5. Prove the calculation of IR5 in HW1 using KCL/KVL

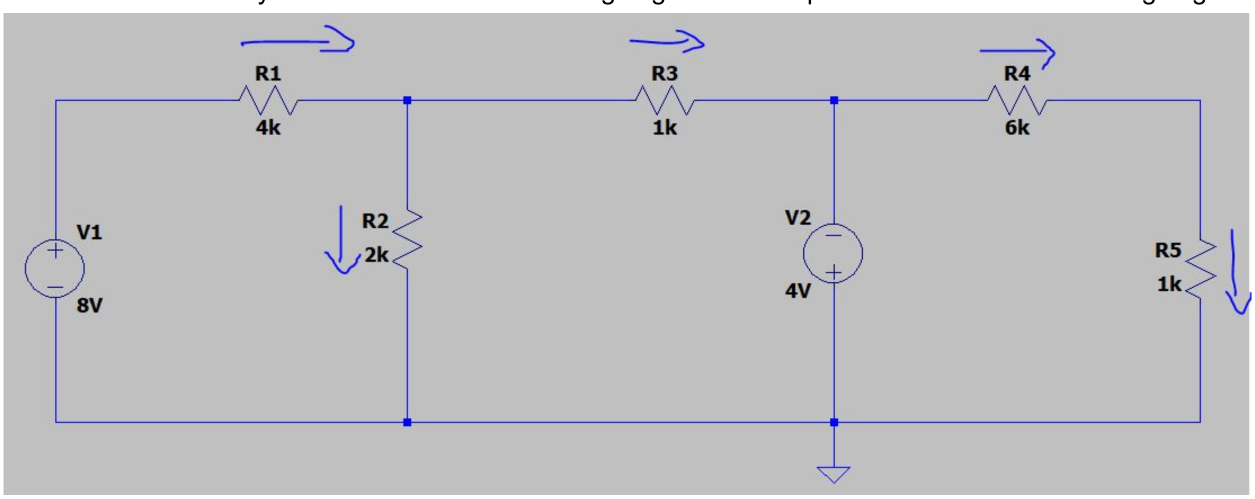
Building Block:

A ladder circuit with a 2 DC voltage of 8 and 4 volts with 5 resistors (4k,2k,1k,6k and 1k)



Analysis:

KVL states that the algebraic sum of all the voltages around any closed loop in a circuit is equal to zero.
KCL states that for any node sum of all the current going out must equal to sum of all the current going in.

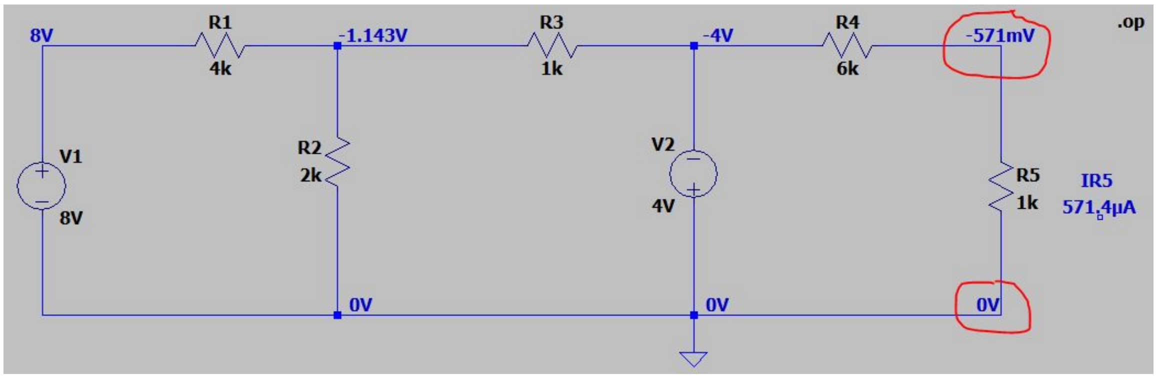


Blue arrow indicate current flow direction
From KVL and KCL it can derive 5 equations for calculation of circuit (for 5 current pass through each resistor)
V1 = IR1*R1 + IR2*R2
V1 + V2 = IR1*R1 + IR3*R3
V1 = IR1*R1 + IR3*R3 + IR4*R4+ IR5*R5
IR1 = IR2 + IR3
IR4 = IR5

Result in:
IR1 = 16/7mA
IR2 = -4/7mA
IR3 = 20/7mA

IR4 = IR5 = -4/7mA = -0.5714mA
According to the calculation IR5 equals to 0.571mA flow from bottom to top (since we assume the current flow from top to bottom and the sign is negative).

Simulation:




```
        --- Operating Point ---

V(n001):        8               voltage
V(n002):        -1.14286        voltage
V(n003):        -4              voltage
V(n004):        -0.571429       voltage
I(R5):          -0.000571429    device_current
I(R4):          0.000571429     device_current
I(R3):          -0.00285714     device_current
I(R2):          0.000571429     device_current
I(R1):          -0.00228571     device_current
I(V2):          -0.00342857     device_current
I(V1):          -0.00228571     device_current
```

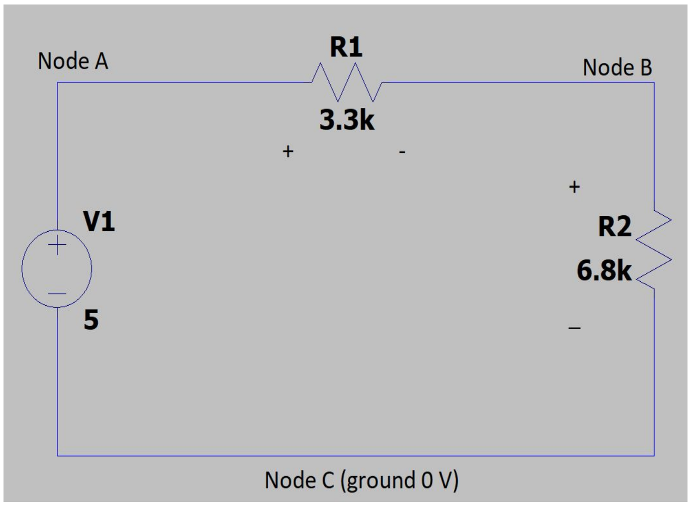
Discussion:

According to the simulation, we know that the absolute value of current is 571.4uA which is the same as what we calculate. Then we can determine the direction using the voltage of two node across R5. Since bottom have higher voltage (0V) than top (-571mV) we know the current flow from bottom to top which also meet what we got from the calculation.

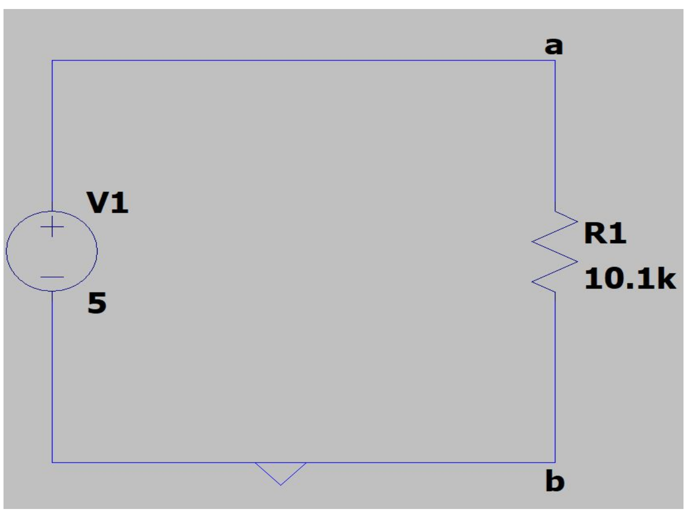
Part B2: Prove circuit reduction (equivalent resistance)

Building Block:

A voltage divider with DC source of 5V and two resistors in series R1=3.3k and R2= 6.8k with the following reference marks.



The reduced circuit derived by combining the 3.3k and 6.8k resistor:



Analysis:

Series circuit analysis as is:

Proof of voltage divider from the proved series equivalent resistor below in Reduced circuit analysis part. Since it is proved that equivalent resistor in a series is $R_1 + R_2$ for two resistor series, thus V=I($R_1 + R_2$) where V is the source of the circuit. Dividing both sides by $R_1 + R_2$ yield $\frac{V}{R_1+R_2} = I$ find the voltage across any resistor is found using ohm's law V=IR and since I is the same in a series circuit, If we multiply the equation by R1 or R2, we get the voltage drop across the that particular resistor hence we proved the voltage divider formula that I will use below.

Using voltage divider equation where the voltage drop across a resistor in series is the ratio of that particular resistor with total resistance in the circuit multiplied by the supplied voltage.

$V_{3.3k} =$ - 5v($\frac{3.3kv}{10.1kv}$)= 1.63366336634V
$V_{6.8k} =$ - 5v($\frac{6.8kv}{10.1kv}$) = 3.36633663366V

Using ohm’s law V=IR to calculate the current across each resistor:

1.63366336634V=I(3.3k)=0.000495049504952A ~= 495.05uA across 3.3k resistor

3.36633663366V=I(6.8k)=0.00049504950495 ~= 495.05 uA 6.8k resistor

Reduced circuit analysis:

$Series\ Resistance_T = R_1 + R_2 + R_3 + R_4 + ... + R_n$ Where n is the total number of resistors in series. Equivalent resistance of resistor in series is the sum of all the resistance.

Proof: at node B, using KCL, the current entering equals the current leaving the node so current i1 equals to current i2 equals to current in the two resistor series circuit.

Using KVL: -5V+R1I+R2I=0 (signs from the assigned polarities)

R1I+R2I=V1

Since I are equals we can factor that out and get

I(R1+R2)=V1

And thus R1 and R2 is the series equivalent resistance since we get back ohm's law.

Thus using what is proven we use series reduction formula and get:

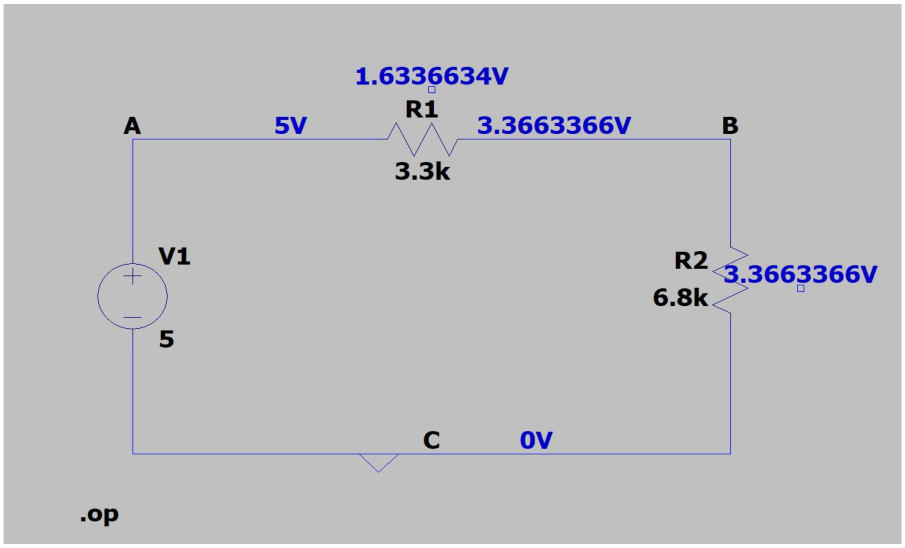
$Series\ Resistance_T$ =3.3K+6.8K=10.1K ohms

Using ohm’s law V=IR to calculate the current across the reduced circuit:

$I_T = \frac{5V}{10.1K\ ohms}$ = 0.00049504950495 A ~= 495.05 uA

Simulation:

Screenshot of simulation



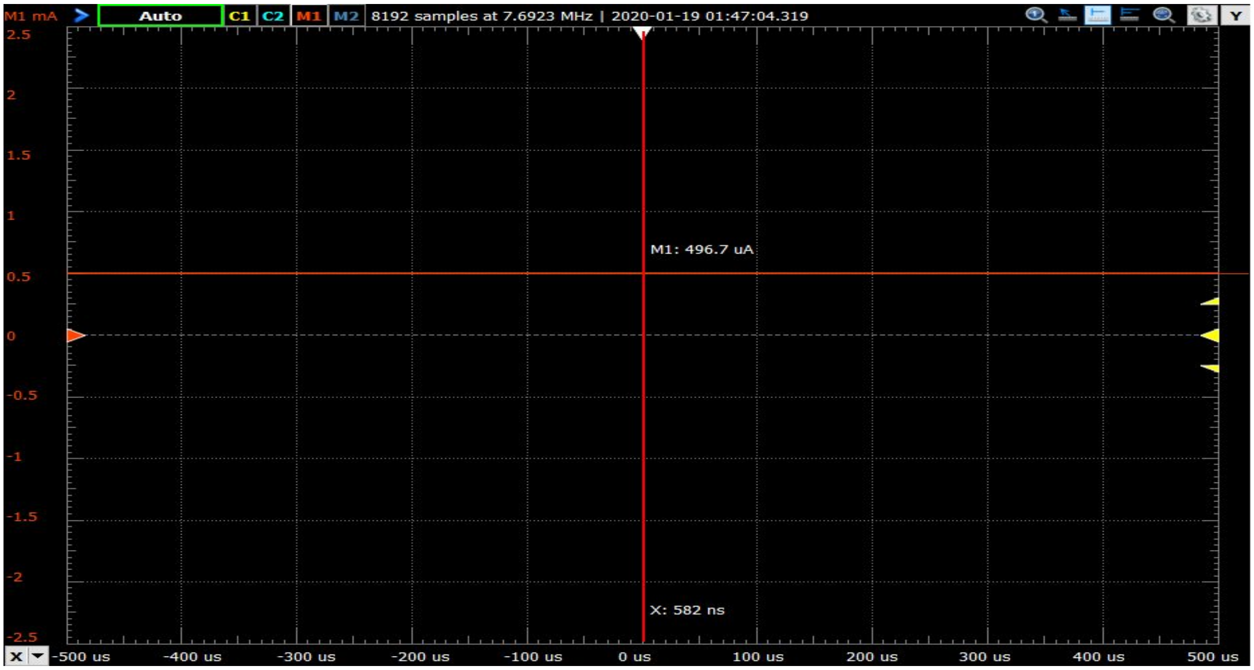
At node A there is 5V and at node B it is 3.3663366V, so the voltage drop across R1 is 5V-3.3663366V which is 1.6336634V. R2 has the remaining voltage which is the same as the voltage a node B, 3.3663366V.

Measurement:

Measurement of voltage drop across 3.3K

	Channel 1
DC	1.637 V
True RMS	1.637 Ṽ
AC RMS	2 mṼ

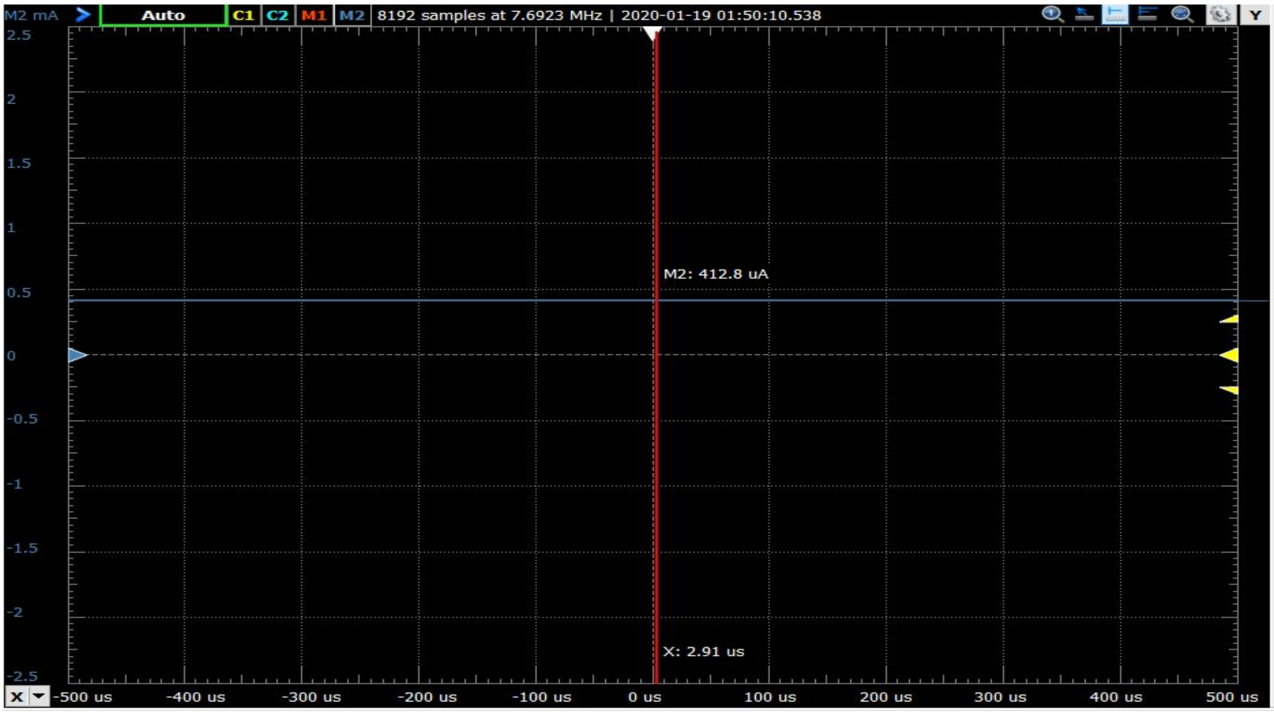
Measurement of current across 3.3k



Measurement of voltage drop 6.8K in schematic

Channel 2
3.351 V
3.351 Ṽ
1 mṼ

Measurement of current across 6.8k



Discussion

The calculated current in the circuit and the simplified circuit are the same which both yeild ~= 495.05uA. The simulation current results agrees with the calculated result since the voltage drop across each resistor is the same as the calculated voltage drop (calculated drop across R1=1.63366336634V is the same as simulated drop across R1= 1.6336634 when rounded to the 8 significant value and R2=3.36633663366V and the simulated drop across R2= 3.3663366 when rounded to the 8 significant value) thus using ohm's law of V=IR, since V and R are the same, I will be the same as the calculated current. The measured current yelid a current of 496.7uA across 3.3k resistor and 412.8 uA across the 6.8k resistor. The current value are different because the calculation done in waveform is done by measuring the voltage drop across a resistor divided by the resistor value. Resistor have 5% tolerance and that can throw off the current value which explain why even though the resistors are in series the measured current are a bit off from each other. Even so, the measured current in agree with the calculated. The calculated current ~=495.05 uA and 5% tolerance range for this value is [470.2975 uA, 519.8025 uA]. The measured current is 496.7uA for 3.3k resistor which is within the range and 412.8 uA for 6.8k resistor and that is not within the range because the resistance inherent in the probes itself.

The calculated voltage drop across 3.3k is 1.63366336634V and the 5% range is [1.55198019802v,1.71534653466v] and the measured drop across 3.3k is 1.637v which is within the range. The calculated voltage drop across 6.8k is 3.36633663366V and the 5% range is [3.19801980198v,3.53465346534v] and the measure drop across 6.8k is 3.351 which is within the range. The measured voltage drop and the simulated voltage drop agree with each other and within tolerance of the simulated voltage drops, with all sum up to a drop of 5V like in the unreduced circuit.

Since the measured current (*mostly*) and the voltage drop across each resistor are within tolerance and agree in both the reduced and unreduced circuit, the key signals V,I and P in the reduced and unreduced circuit agree, circuit reduction yield equivalent circuit.

B2 Discussion Questions:

Voltage across R1= 1.637 V

Voltage across R2= 3.351 V

a) 1.637+3.351= 4.998V which is close to 5 V. This is consistent with expectations of KVL since the voltage drops sum very close to 5V supplied.

Voltage drop across 100k ohms

	Channel 1
DC	4.826 V
True RMS	4.826 Ṽ
AC RMS	0 Ṽ

Symbolically VR2=5V

b) REQ= R1+R2 when R2>> R1

REQ=R2

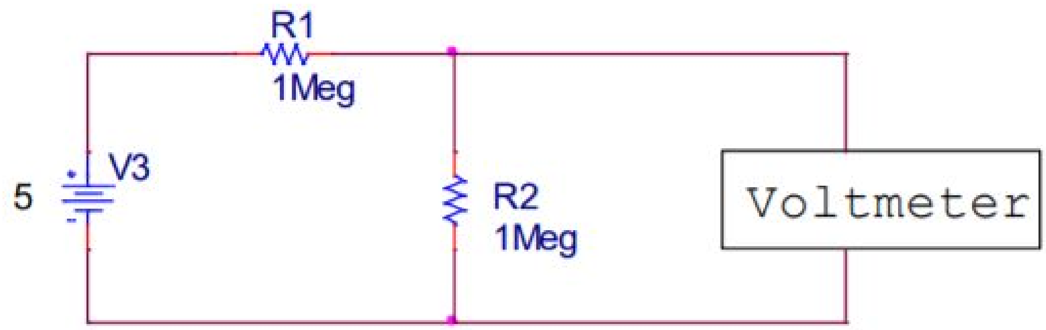
The screen shot for B1 for the expected 5V (just in case this is needed)

	Channel 1
DC	5.003 V
True RMS	5.003 Ṽ
AC RMS	1 mṼ

Part B3: Voltage Divider/Series Resistors

Building Block:

A voltage divider with DC source of 5V with two 1 Meg resistors in series and with a voltmeter in parallel with one 1 Meg that ideally does not affect the circuit..



Analysis:

Using voltage divider equation where the voltage drop across a resistor in series is the ratio of that particular resistor with total resistance in the circuit multiplied by the supplied voltage.

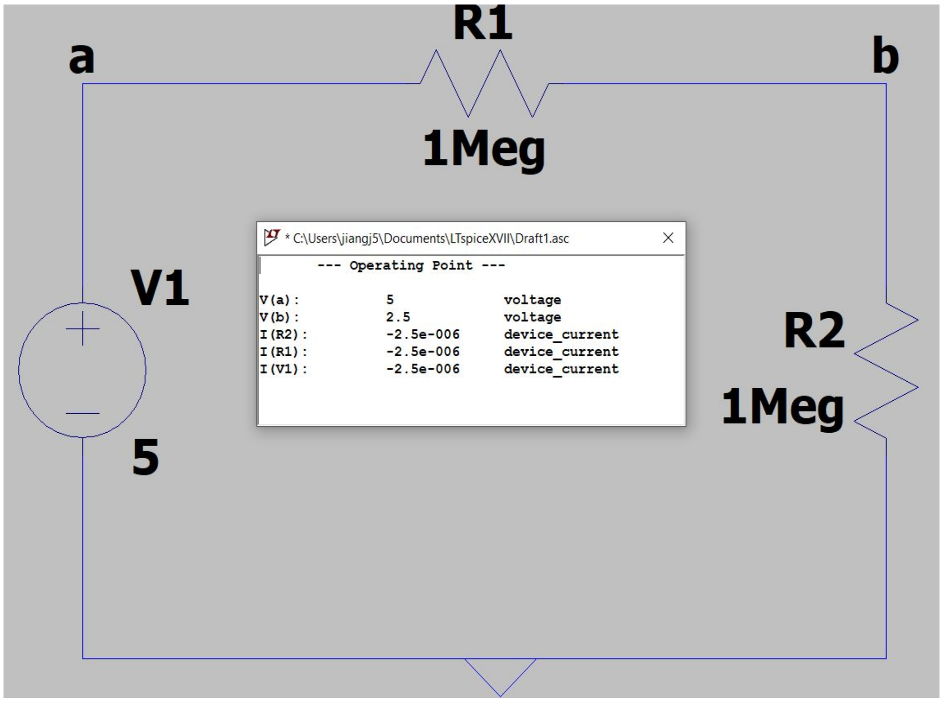
$V_{3.3k} =$ - 5v($\frac{1Meg}{2\,Meg}$)= 2.5 V
$V_{6.8k} =$ - 5v($\frac{1\,Meg}{2\,Meg}$) = 2.5 V

Using ohm's law V=IR to calculate the current across each resistor:

2.5 V=I(1 Meg)=0.0000025A = 2.5 uA

2.5 V=I(1 Meg)=0.0000025A = 2.5 uA

Simulation:



The expected voltage drop across R1 and R2 is 2.5 voltage each in the ideal world where the measuring equipment does not interfere and add a load change the value of R2 when trying to measure the voltage across R2.

Measurement:

Voltage across first 1 Meg

	Channel 1
DC	3.296 V
True RMS	3.296 Ṽ
AC RMS	2 mṼ

Voltage across second 1 Meg

Channel 2
1.694 V
1.694 Ṽ
2 mṼ

Voltage drop across each resistor when measured simultaneously

	Channel 1	Channel 2
DC	3.709 V	1.28 V
True RMS	3.709 Ṽ	1.28 Ṽ
AC RMS	2 mṼ	2 mṼ

Discussion

The ideal voltage drop is 2.5 V (5% tolerance is [2.375v,2.625v]) for both R1 and R2 (1 MEG ohm) which the analysis and the LT splice simulation both shows. However, when measured in waveform individually, the voltage drop across R1 and R2 are 3.296V and 1.694V respectively and when measured together, the voltage drop is 3.709 V and 1.28 V. The differences in the measured result compared to the ideal result is greater than 5% tolerance difference and cannot be explained by resistor value differences from expected value This means that there is a load resistance when measuring voltage across the resistor. The difference in voltage drop when measuring resistors simultaneously and individually further suggests that load resistance from the probes are not the same.

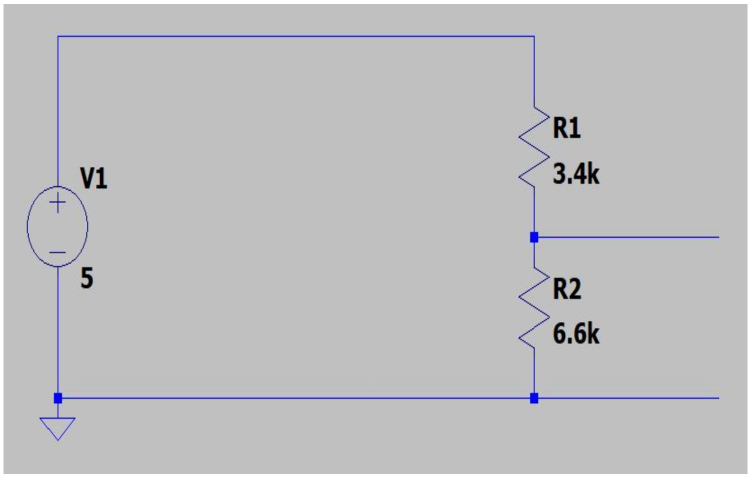
B3 Discussion Questions:

No the measurement across the 1 MEG ohm does not agree with expectation of ~2.5 V. Channel 1 probe has internal resistance (about 1 MEG ohm) that add a load to R2 which caused the voltage across R2 to change drastically from 1 MEG (about 500,000 ohm when combined in parallel) which cause the voltage drop across it to decrease.

Part C: Prove (demonstrate) an interesting application for a voltage divider circuit

Building Block:

A voltage divider with a DC supply of 5V and two resistors in series. R1=3.4k ohm and R2=6.6k ohm

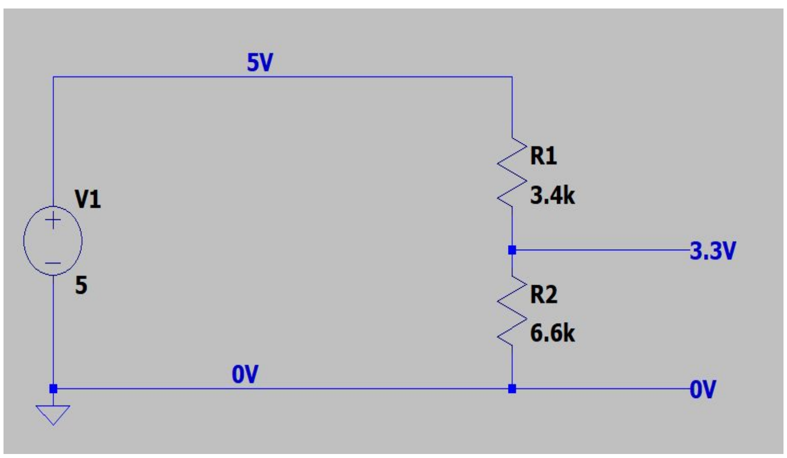


Analysis:

Voltage divider is commonly used as a logic level shifting tool in the big circuit. This circuit create 3.3V output from normal 5V input so that component can operate on two different voltage level. Use two resistor 3.4k ohm and 6.6k ohm it create a 3.3/5 ratio to the output voltage.-

Vout = 5V * 6.6k / (3.4k + 6.6k) = 3.3V

Simulation:



The voltage drop across R1 is 5-3.3 = 1.7 V and we get the 3.3 voltage we can hook up to a load that use analog 3.3V as a logic high to achieve logic level shifting .

Measurement:

	Channel 1	Channel 2
DC	-5.011 V	3.262 V
True RMS	5.011 V	3.262 V
AC RMS	1 mV	2 mV

Channel 2 measure the voltage at the node between R1 and R2 which is around 3.3V and Channel one measure the voltage across the entire circuit.

Discussion

The aim of the logic level shifting is to decrease 5V to 3.3V which is logic high for many chip devices. There is a discrepancy between the calculated 3.3V and the measured 3.262 V that is due to the resistor not being the exact value that it claims to be due to 5% tolerance. $\frac{3.3-3.262}{3.3}$ *100= 1.15151515152 which is less than 5% which means this difference is within expected error range from -5% to 5% of the gold band resistors

B3 Discussion Questions:

1)Voltage divider can be used via sensor measurement, tachometer, and logic level shifter in circuits.

4)Erika and Alex (omega level lab). They are using voltage divider through wheatstone bridge application to find the resistance of an unknown resistor using the equation $\frac{R1}{R2} = \frac{R3}{R4}$.