# Using MUTISIM software to reinforce superposition analysis technique for electrical circuits

**Dr. Cyrus K Hagigat, The University of Toledo**

Dr. Hagigat is an associate professor in the Engineering Technology department of the College of Engineering of the University of Toledo. Dr. Hagigat has an extensive industrial background, and his teaching technique is based on practical aspects of engineering.

# Using MULTISIM software to reinforce superposition analysis technique for electrical circuits

**Cyrus Hagigat**
**College of Engineering**
**Engineering Technology Department**
**The University of Toledo**
**Toledo, Ohio**

**Keywords:**

MULTISIM, Superposition theory, Hardware Descriptive Language

**Introduction:**

The introductory electrical Circuits course for non-electrical engineering technology majors at the author's institution contains a weekly lab component. However, due to Covid-19 pandemic, the entire class and the lab were converted to an online format. Consequently, the lab could not be held in a traditional laboratory setting.

It was decided to use MULTISIM simulation software in place of the physical lab. It was quickly discovered that not only MULTISIM can replace the traditional labs, but it can also be a valuable teaching aid in enhancing student understanding of circuit analysis techniques.

The traditional textbooks only demonstrate the final result of a circuit analysis using MULTISIM [1, 2]. The approach taken in this article is different. This article demonstrates using the software as an aid to check the intermediate steps of the hand calculation techniques that must be used if no computer software were being used. An example which was a part of a lecture and instructions for a lab assignment is presented where all steps of implementing the superposition technique with the aid of MULTISIM for analyzing a circuit with multiple power sources is presented. This article discusses and demonstrates the use of MULTISIM software to enhance and check the manual superposition techniques when multiple power sources are present.

The example presented in this article replaced an actual lab that verified the superposition theorem. In the physical lab, students used to build a circuit with multiple power sources and make measurements. Then measurements were made with one power source at a time. The measurements with individual power sources were then algebraically added and compared to the measurements made with multiple power sources in order to experimentally prove the validity of the superposition technique.

**Example of a hybrid use of MULTISIM**

Superposition electric circuit analysis techniques can be applied several different ways. When there is more than one power source present in a circuit, one of the easiest techniques that can be understood by students that are just starting to learn the subject is to find the currents (magnitude and direction) through each component from each power source separately, and then adding the currents from different sources algebraically (accounting for current directions as well as

magnitude) to find the total current through each resistor, and then finding the voltages across each resistor by Ohm's law.

A voltage source is a device like a battery where it produces a voltage independent of other factors. A current source is an electronic device that delivers or absorbs an electric current which is independent of the voltage across it. Consequently, while applying the superposition technique, when a current source is not a part of the analysis, the circuit is opened at its location. On the other hand, when a voltage source is not a part of the analysis, the voltage source is shorted at its location. As shown in this article, MULTISIM was used to reinforce these concepts.

MULTISIM software can be used to check and verify every step of implementing the superposition technique. Consider the circuit shown in figure 1.
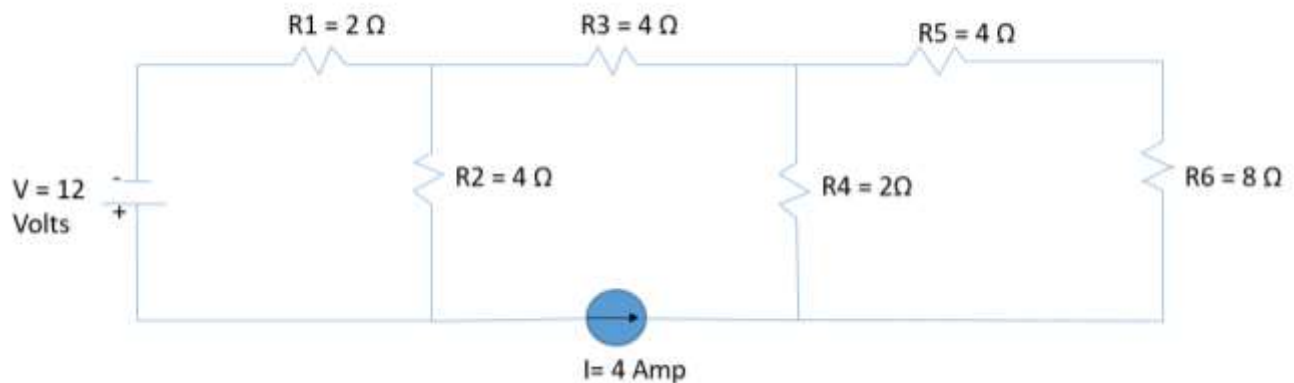


Figure 1: Circuit to be analyzed by superposition technique

The goal is to determine the current through and the voltage across each resistor individually by the superposition technique.

Figure 2 is the circuit of figure 1, modeled using MULTISIM. In figure 2, ammeters and voltmeters are added such that the current through and the voltage across each resistor are calculated and displayed. [3]
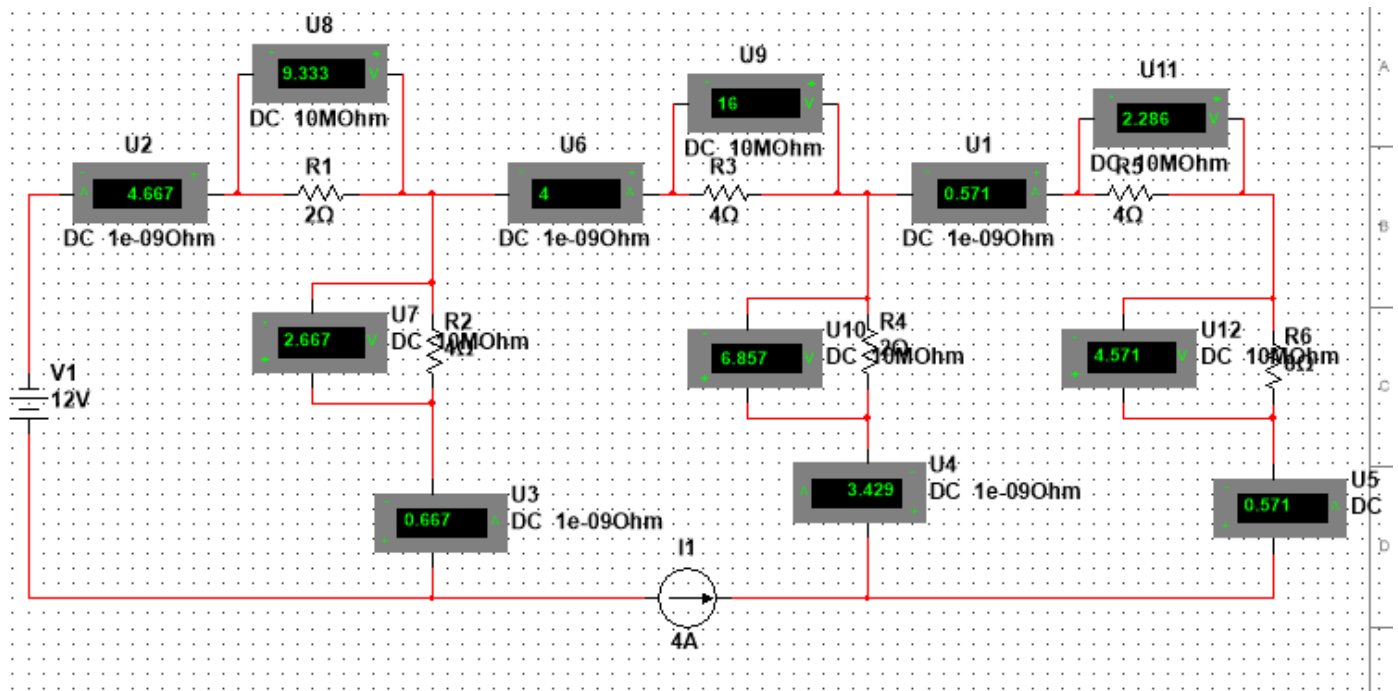
Figure 2: Simulation of circuit of figure 1 in MULTISIM

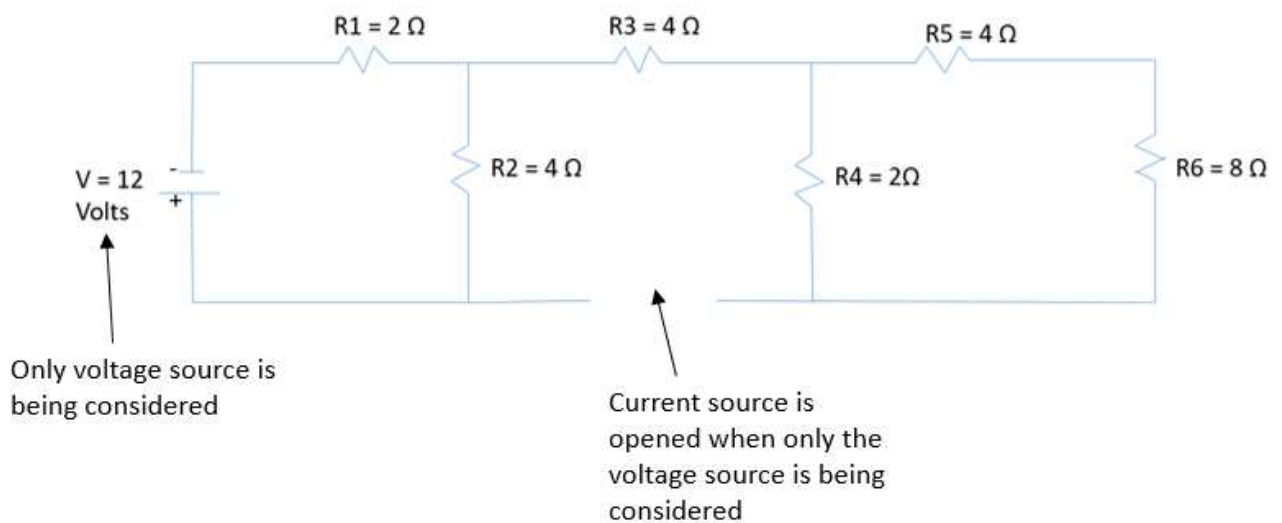The circuit to be analyzed when the voltage is the only power source is shown in figure 3.



R1 = 2 Ω    R3 = 4 Ω    R5 = 4 Ω

V = 12 Volts

R2 = 4 Ω    R4 = 2Ω    R6 = 8 Ω

Only voltage source is being considered

Current source is opened when only the voltage source is being considered

Figure 3: Circuit for analyzing the effect of V only

The circuit of figure 3 can be simplified by taking out the opened segment of the circuit. Figure 4 demonstrates the effective circuit after the current source has been opened
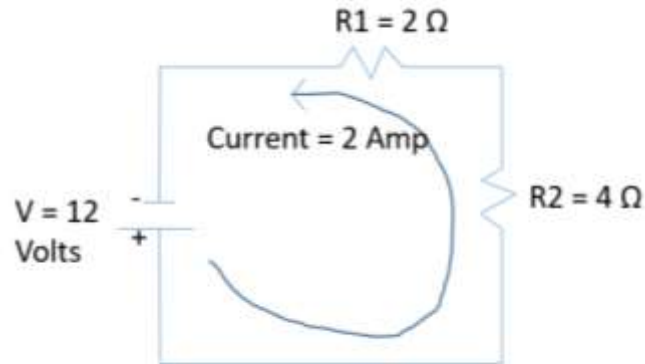
Figure 4: Effective circuit after current source has been opened

In figure 4, the current is assumed to flow from the positive to the negative pole because MULTISIM makes this assumption.

The simplified circuit of figure 4 can be verified by setting the current value to zero (0) in the simulation shown in figure 2. This simulation is shown in figure 5. Figure 5 contains the direction of the currents calculated by MULTISIM based on the positive readings of the ammeters. In MULTISIM, when positive current values are displayed, it means that the current is going from the positive side of the meter to the negative side of the meter [3].
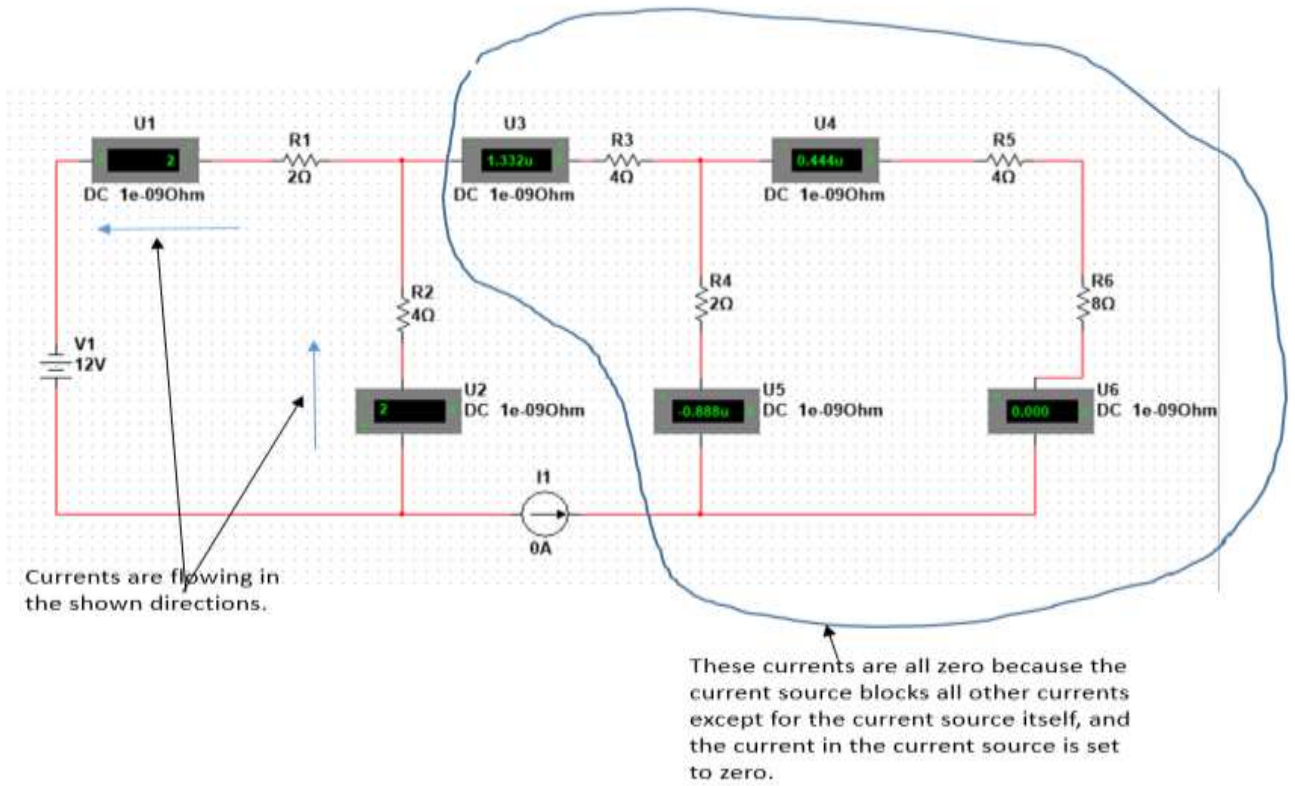
Figure 5: Simulation showing the current values and directions when there is
a non-zero voltage source and the current source is set to zero (0)

Figure 6 is the circuit that must be analyzed when the voltage source is set to zero (0) and the currents from the current source are to be analyzed.
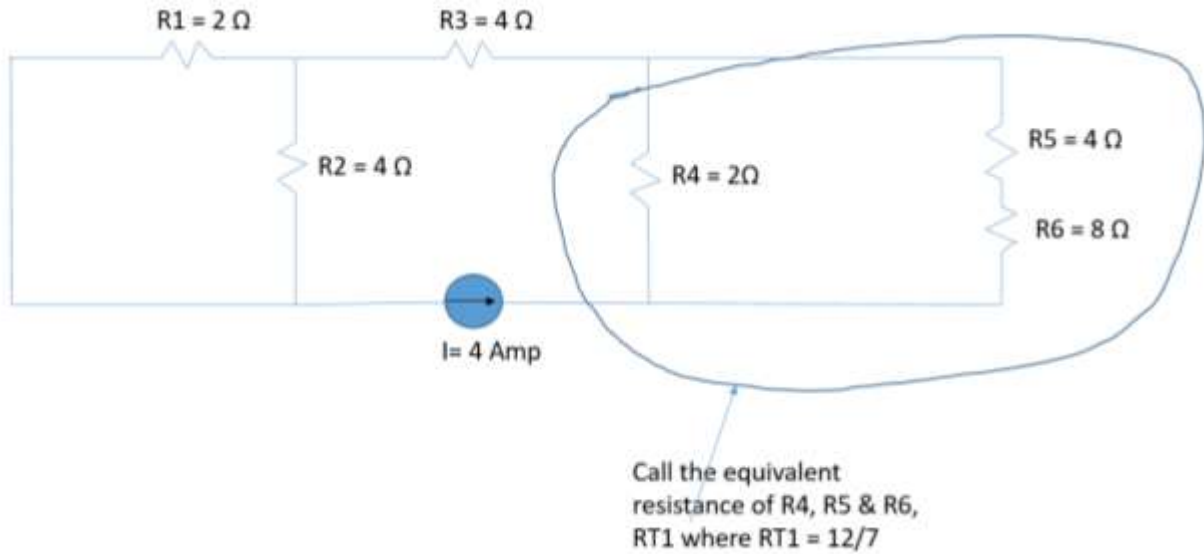
R1 = 2 Ω   R3 = 4 Ω

R5 = 4 Ω

R2 = 4 Ω   R4 = 2Ω

R6 = 8 Ω

I= 4 Amp

Call the equivalent
resistance of R4, R5 & R6,
RT1 where RT1 = 12/7

Figure 6: Relabeled circuit with only current source as the active power source

As it can be seen in figure 6, R5 & R6 are in series, and the combination of "R5, R6" and R4 are in parallel. Using these relationships, the combination of R4, R5 and R6 from the point of view of the current source has an equivalent resistance value of 12/7 Ω. This value can be checked by MULTISIM by adding an Ohmmeter as shown in figure 7.
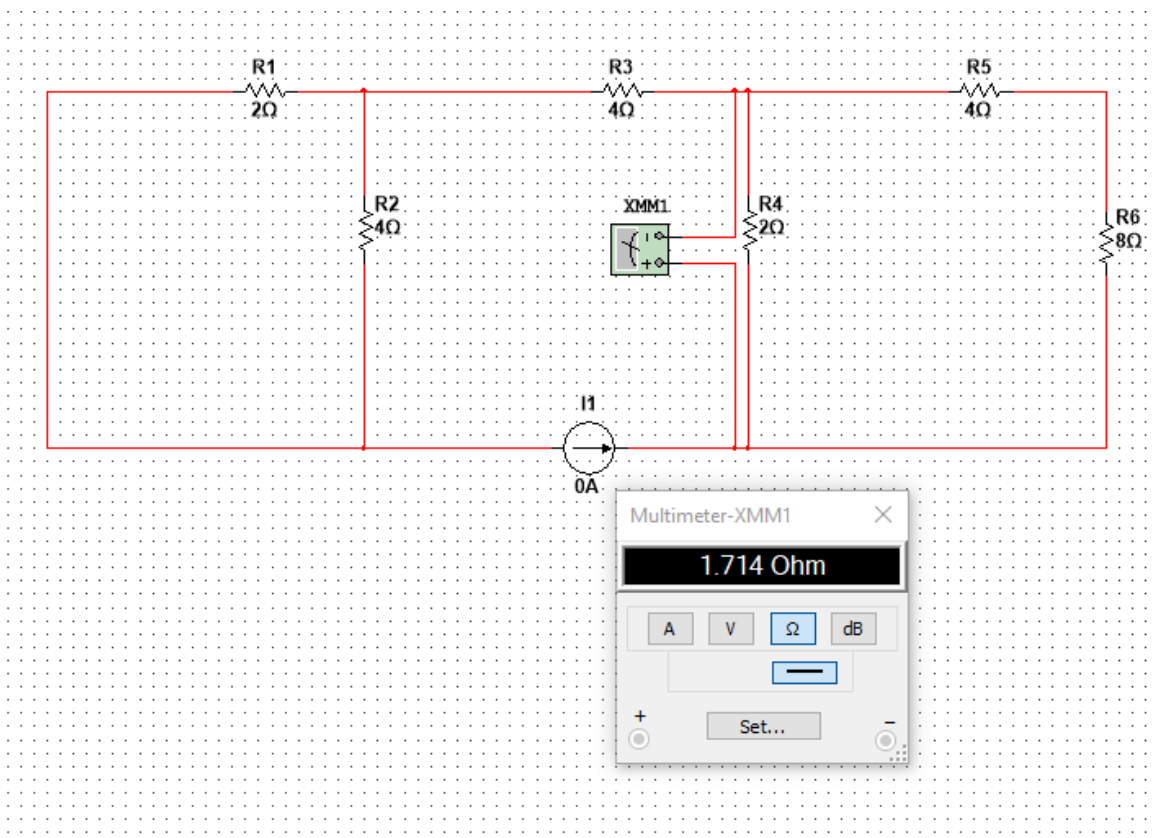


Figure 7: determining the equivalent resistance of combination of R4, R5 and R6 by MULTISIM

The following relationships determine the currents through R5, R6 and R4.

Current through R5 = current through R6 = {(I)(RT1)} / (R5 + R6)= {(4)(12/7)} / (4 + 8) = 4/7 Amp

Current through R4 = {(I)(RT1)} / R4 = {(4)(12/7)} / 2 = 24/7 Amp

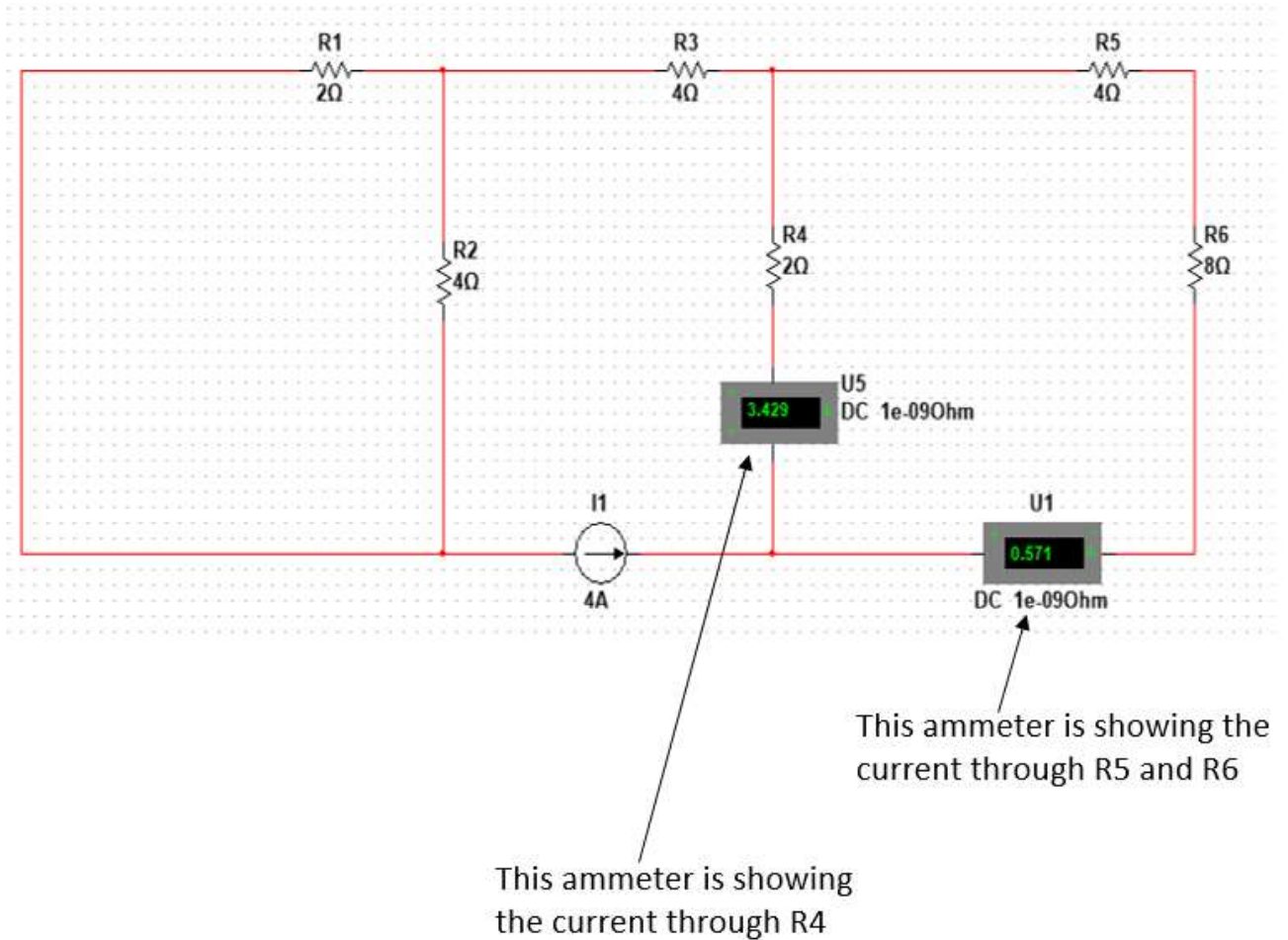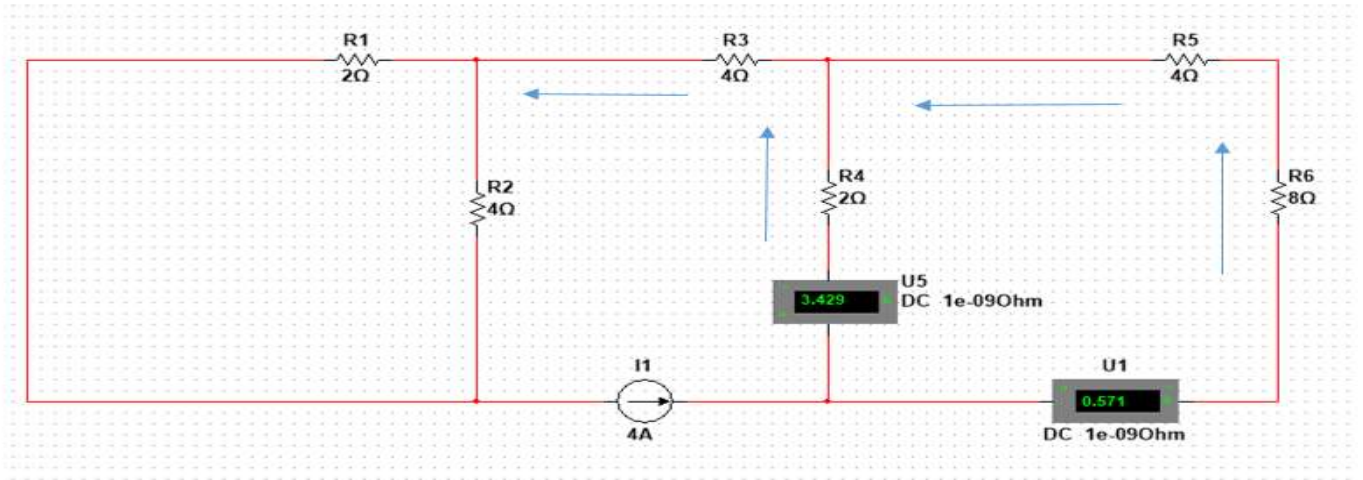Figure 8 shows the MULTISIM simulation that verifies these results.



Figure 8: Multisim simulation with only the 4 amp power source
showing the currents through R5 & R6 and the current through R4

Figure 9 shows the directions of the currents of figure 8 based on MULTISIM displaying convention, and the fact that they add up and become the current through R3.

The arrows where shown indicate current directions.
Meter U1 shows the current through R5 and R6.
Meter U5 shows the current through R4.
The sum of the currents shown by meters U1 and U5 show the current through R3.

Figure 9: Clarification of MULTISIM display of figure 8

According to the display of figure 9, the current through R3 when only considering the current source of 4 Amperes is 4 Amps. Figure 10 is the MULTISIM simulation that confirms this fact.
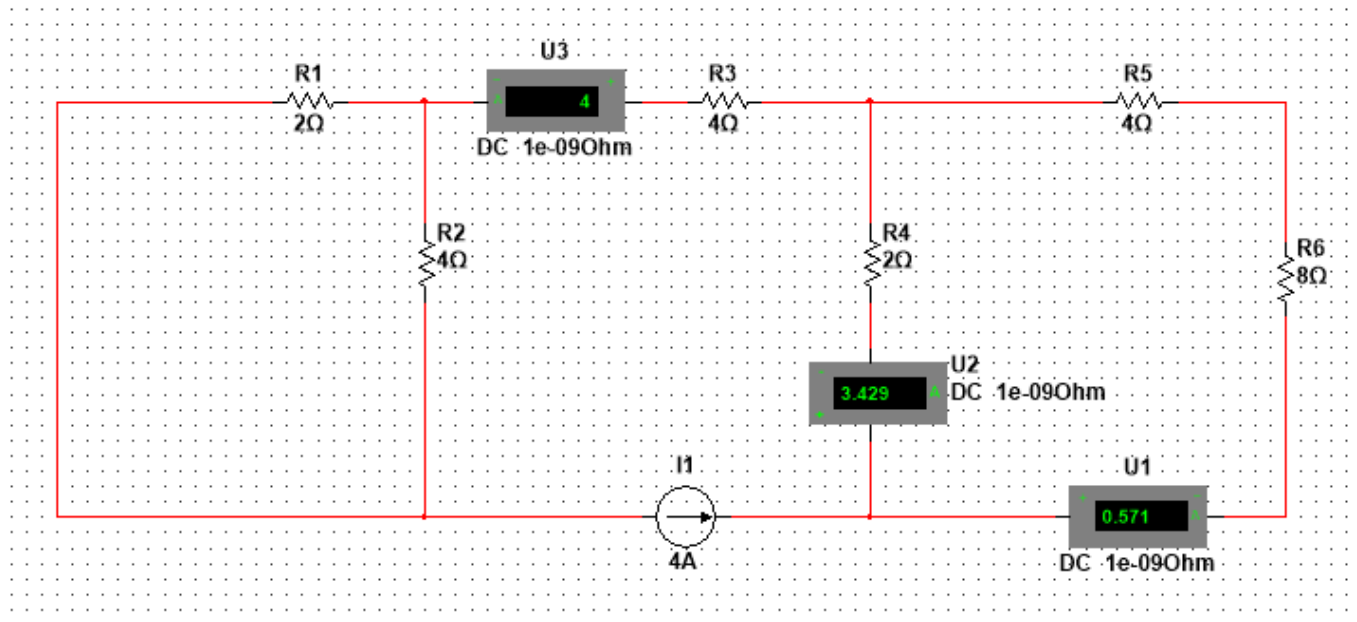


Figure 10: Demonstrating by MULTISIM simulation that the current through R3
is the same as the 4 Amp current source

The 4 Amp current is being divided between R1 & R2 as shown in figure 11.
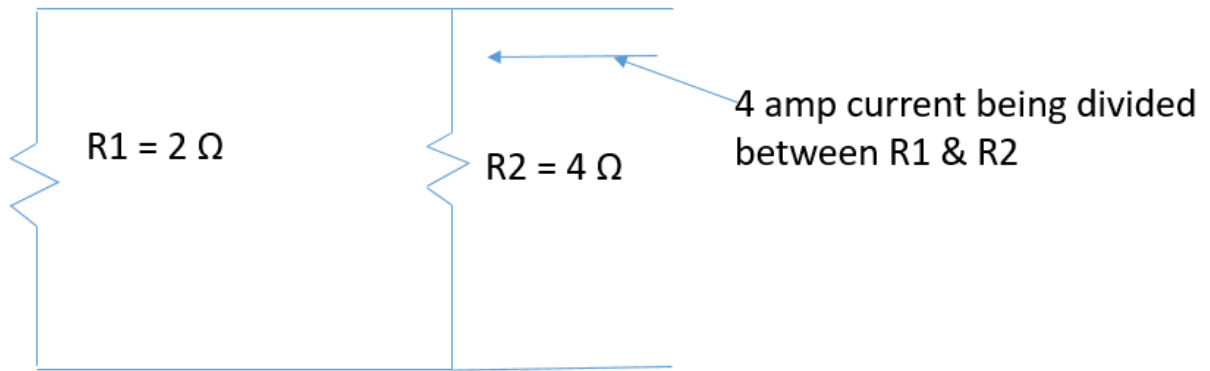


Figure 11: Sketch showing R1 & R2 are parallel circuits

The following calculations determine the value of the currents through R1 & R2.

IR1 = {(I)(R2)} / (R1 + R2) = {(4)(4)} / (2 + 4) = 8/3 Amp
IR2 = {(I)(R1)} / (R1 + R2) = {(4)(2)} / (2 + 4) = 4/3 Amp

Figure 12 shows a MULTISIM simulation verifying the currents through R1 & R2.
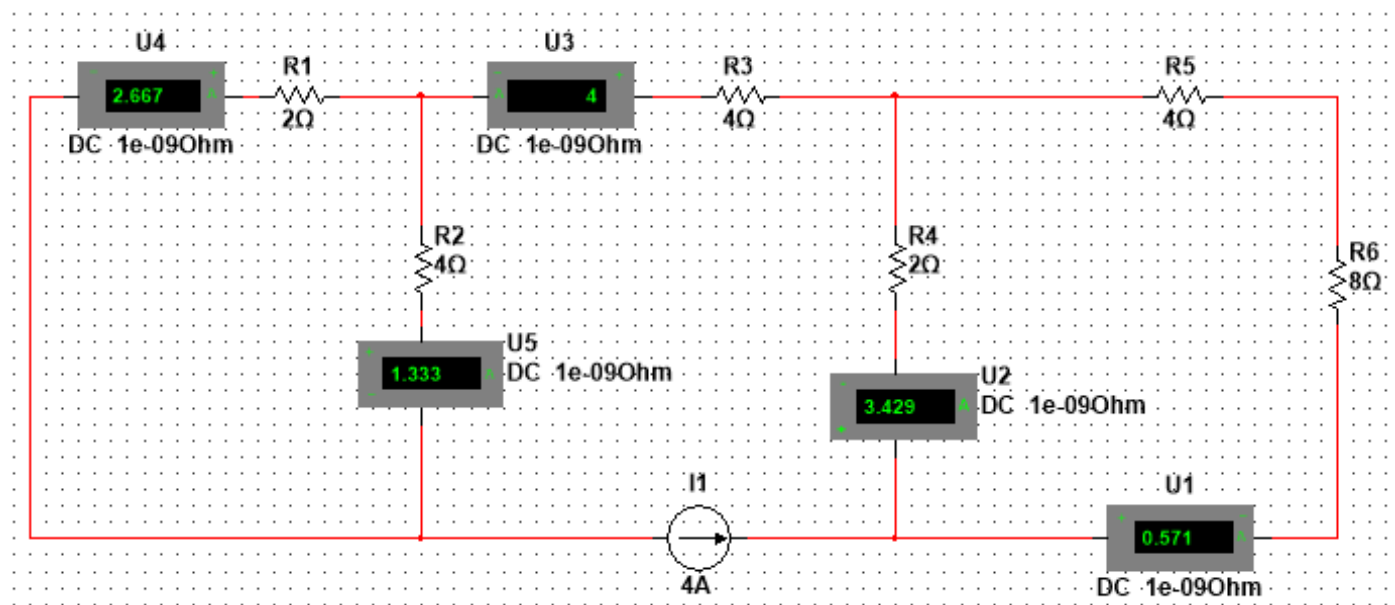


Figure 12: MULTISIM simulation showing all currents from the current source

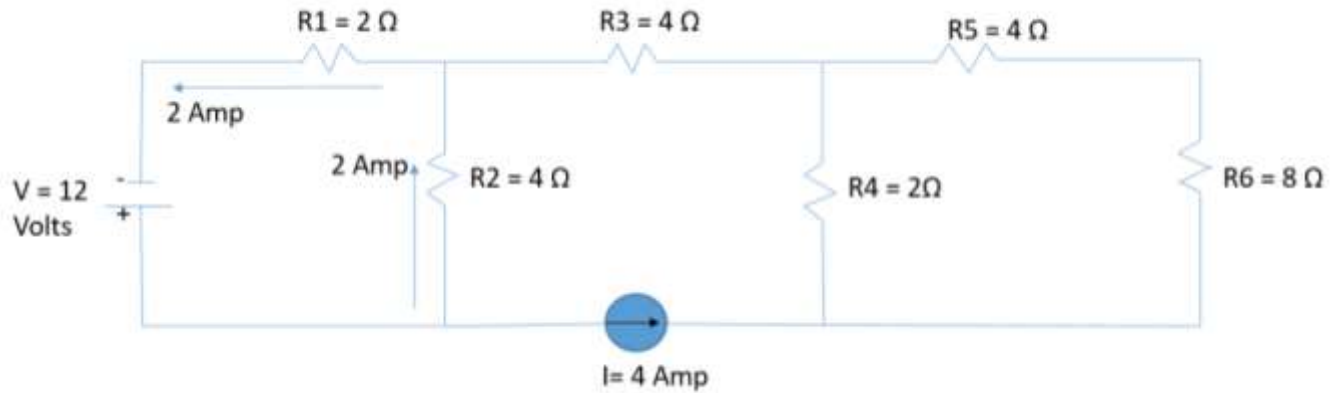Using figure 5, the currents shown in figure 13 are resulting from the voltage source only.

Figure 13: Currents in the circuit assuming the only power source is V= 12 volts

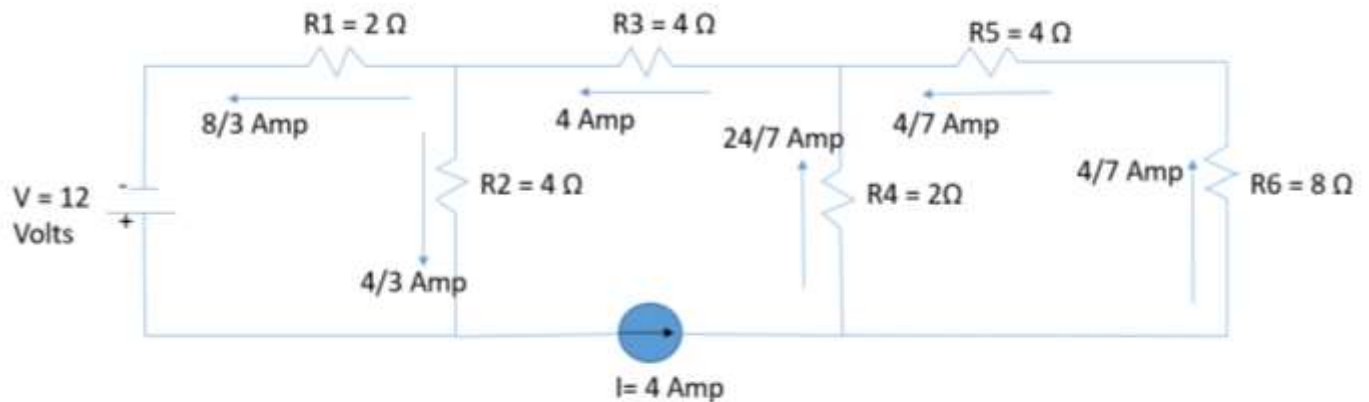Using figures 9 and 10, the currents shown in figure 14 are resulting from the current source only.



Figure 14: Currents in the circuit assuming the only power source is I= 4 Amp

Summing the currents of figures 13 and 14 algebraically (accounting for their direction) results in the following calculations and results.

IR1 = (8/3 Amp) + (2 Amp) = 14/3 Amp
IR2 = (2 Amp) – (4/3 Amp) = 2/3 Amp
IR3 = (4 Amp) + (0 Amp)= 4 Amp
IR4= (24/7 Amp) + (0 Amp)= 24/7 Amp
IR5 = (4/7 Amp) + (0 Amp) = 4/7 Amp
IR6 = ((4/7 Amp ) + (0 Amp) = 4/7 Amp

Figure 15 shows the total currents resulting from V & I which are obtained by algebraic summation of the currents of figures 13 & 14.
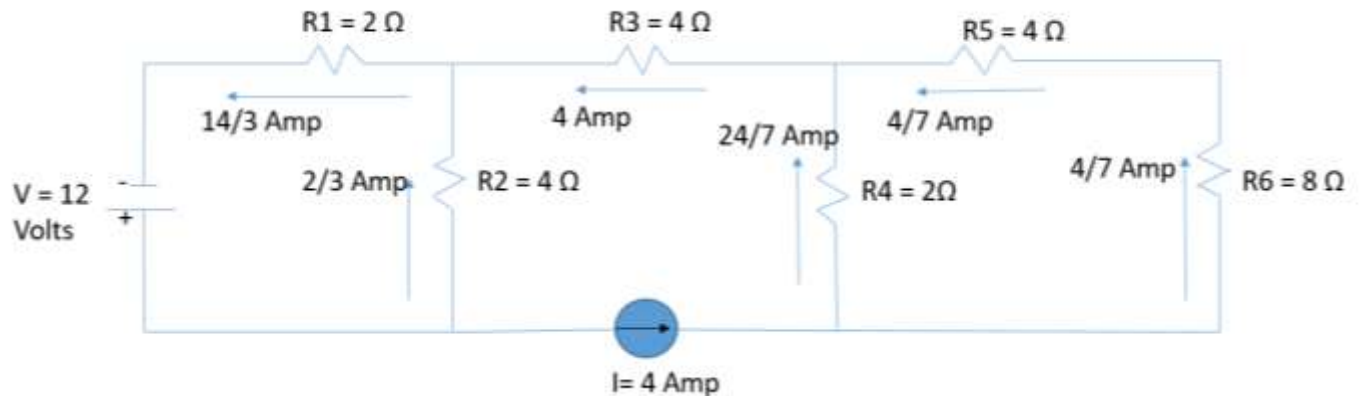


Figure 15: Currents resulting from the entire circuit consisting of
all the resistors, voltage sources & current source

The following are the calculations for the voltages for the complete circuit.

V1 = (R1)(I1)= (2 Ω)(14/3 Amp) = 28/3 Volt
V2 = (R2)(I2)= (4 Ω)(2/3 Amp) = 8/3 Volt
V3 = (R3)(I3)= (4 Ω)(4 Amp) = 16 Volt
V4 = (R4)(I4)= (2 Ω)(24/7 Amp) = 48/7 Volt
V5 = (R5)(I5)= (4 Ω)(4/7 Amp) = 16/7 Volt
V6 = (R6)(I6)= (8 Ω)(4/7 Amp) = 32/7 Volt

In MULTISIM, voltages have the same polarity as current flow which is from the positive pole to the negative pole. The results shown on figure 15 and the voltage calculations match the results shown in figure 2 which is showing the simulation results for the entire circuit.

**Conclusion & Summary**
In this article the use of MULTISIM simulation software as a physical lab substitute and as a supplement for enhancing student understanding of the superposition technique in circuit analysis is introduced.

By using the mixed approach of using a hand calculation technique and MULTISIM, the students have the opportunity to verify the accuracy of every step of the process which will improve the students' learning and will allow the students to find and correct their own errors.

In order to evaluate the suitability of the technique when Covid restrictions are lifted, a thorough evaluation of students' end of semester comments were performed. 80% of the comments indicated that the detailed instructions that were presented, enabled students to learn the software use efficiently, and 40% of the comments indicated that the hybrid use of the software as an addition to the hand calculation techniques were useful. The techniques are not technically

complicated. However, that is precisely the point. The techniques are suitable as teaching aids for students that are just beginning to learn electrical engineering concepts.

Some unique issues regarding exams were encountered as a result of the introduction of the completely automated capabilities of MULTISIM. The exam questions had asked for a complete classical technique solution. Some students used the software and provided the correct answers without doing the hand calculation solutions.  These cases were not treated as academic dishonesty cases, and the students were given a chance to do the hand calculation solutions and turn them in. The fact that the students used the software in this manner indicates that the technique indeed enhanced student understanding.

**References**
[1] Foundations of Electronics Circuits and Devices by Meade
[2] Principles of Electric Circuits by Floyd
[3] Lab Manual; Introduction to Circuits by National Instruments Corporation