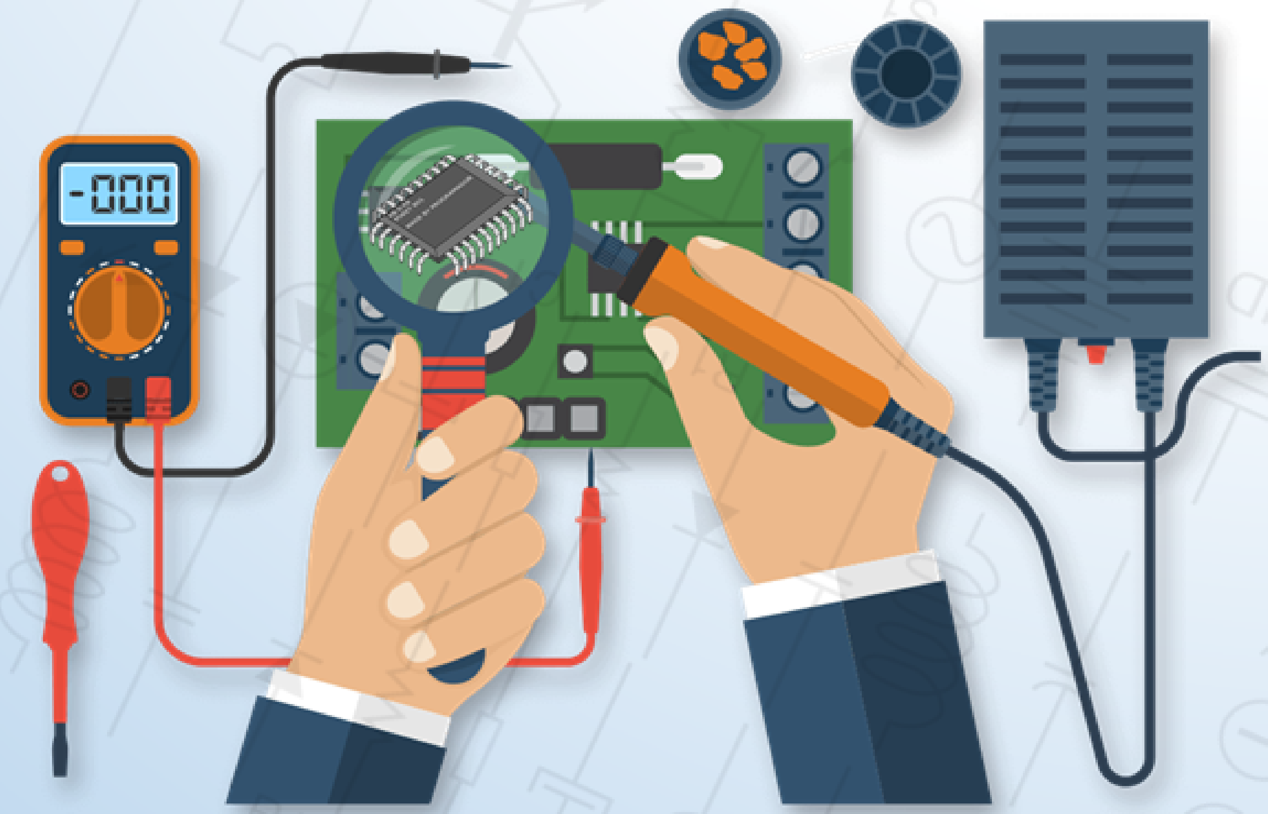
HO CHI MINH CITY UNIVERSITY OF TECHNOLOGY
COMPUTER ENGINEERING

Electronic Device Component

Lab's Report



Date:13/09/2021

Cao Minh Quang - 2052221

Contents

CHAPTER 1

First Project on PSpice

1 Introduction

PSpice for TI is a design and simulation environment that helps evaluate functionality of analog circuits. This full-featured, design and simulation suite uses an analog analysis engine from Cadence®. Available at no cost, PSpice for TI includes one of the largest model libraries in the industry, spanning our analog and power portfolio, as well as select analog behavioral models.

The PSpice for TI design and simulation environment allows you to simulate complex mixed-signal designs with its built-in library. Create complete end equipment designs and prototype your solutions before you commit to layout and fabrication, reducing time to market and development cost.

Within the PSpice for TI design and simulation tool, you can search for TI devices, explore the portfolio, open test benches and simulate your design to further analyze the selected device. You can also run co-simulation of multiple TI devices to better represent your system.

In addition to a full library of preloaded models, you can easily access the latest technical collateral for TI devices within the PSPICE-FOR-TI tool. After you have verified that you have the correct device for your application, you can access the TI store to purchase the product. Using PSpice for TI, you have access to tools to address your simulation needs as you progress through the design cycle, from circuit exploration to design development and verification. Available at no cost, it is easy to get started.

A primary purpose of this lab is for you to become familiar with the use of PSpice and to learn to use it to assist you in the analysis of circuits (e.g. double check the results with your exercise). The software is required to install in your computer. Moreover, it is your responsibility to learn its use in a more detailed way since you will be using it along the course of this semester and in the future. The targets in the first manual are summarized as follows:

- PSpice installation on Windows OS
- Create a project on PSpice
- Create a bias analysis profile
- Run the simulation and obtain results

2 PSpice installation

The homepage to install this tool is located at https://www.ti.com/tool/PSPICE-FOR-TI.

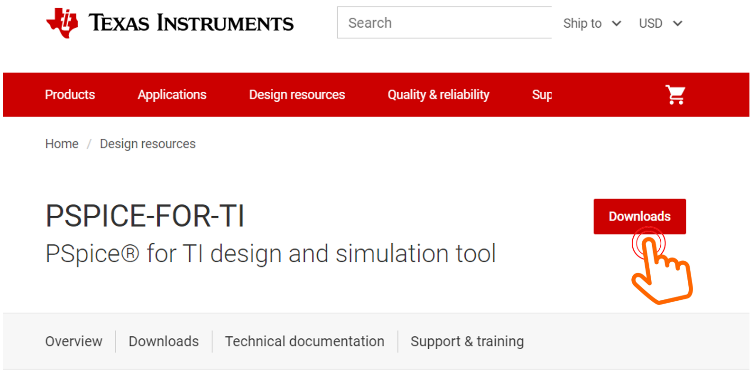


Figure 1.1: Homepage to download PSpice

By clicking on the **Download** button, the website is navigated as follows:

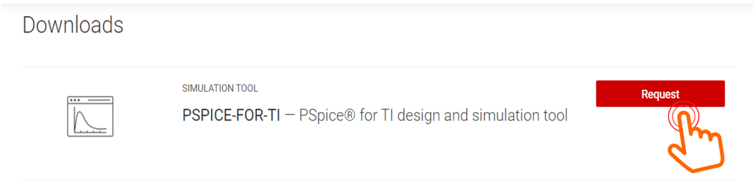


Figure 1.2: Request information for downloading

Basically, you need to login before requesting a setup file. Please follow the manuals from the website to accomplish this process. An email with an access key will be sent to your account to activate the PSpice software.

3 Create a project on PSpice

Step 1: Launch PSpice for TI from windows start menu.

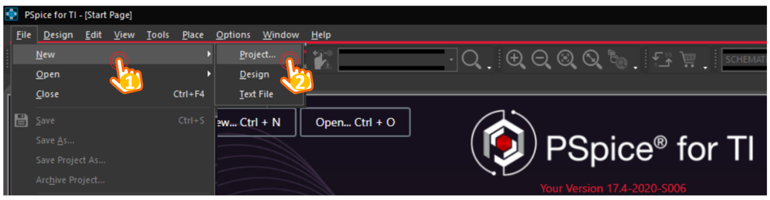


Figure 1.3: Create a new project on PSpice

From menu **File**, select **New**, then select **Project**.
Step 2: Create an empty project as follows.

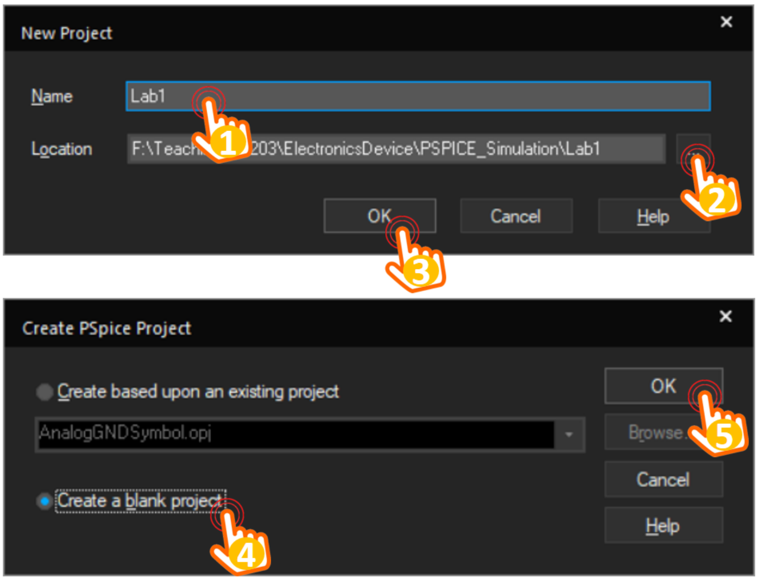


Figure 1.4: Provide the name, location and select a blank project

In the second dialog, please select **Create a blank project**. An empty project is created and the next UI is displayed as follow:

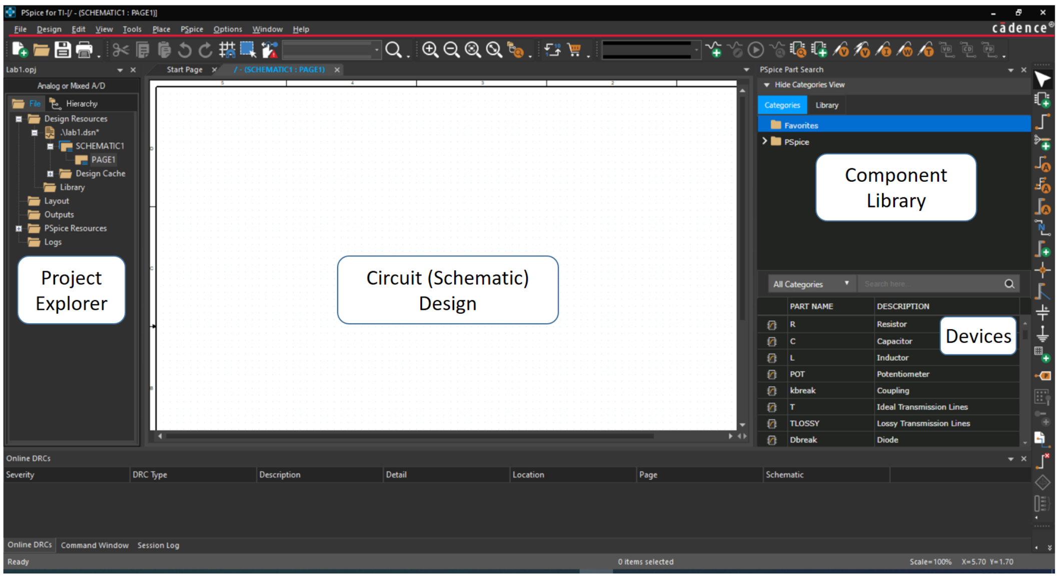


Figure 1.5: An empty project is created on PSpice

4 Design a circuit

Step 1: Double click on the Resistor in the device list and then move the mouse to the schematic design page. This device is in the **Favourite** library in default. While moving, **press R** to rotate the device before placing it (by left-mouse click), as follow:

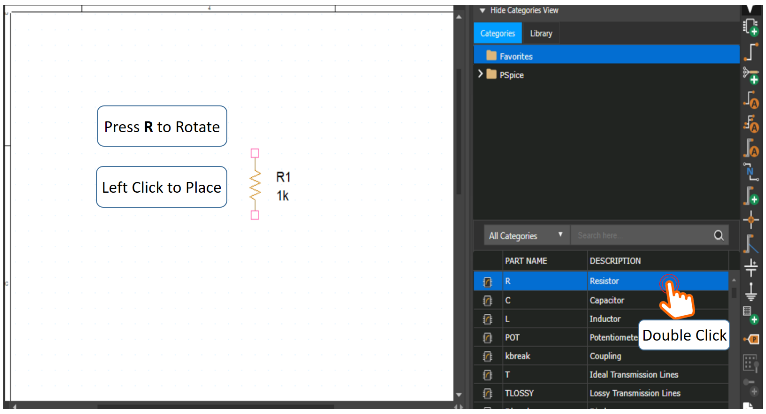


Figure 1.6: Place a resistor in PSpice

Step 2: Double click on the value of the resistor, which is 1k in default, in order to change its resistance.

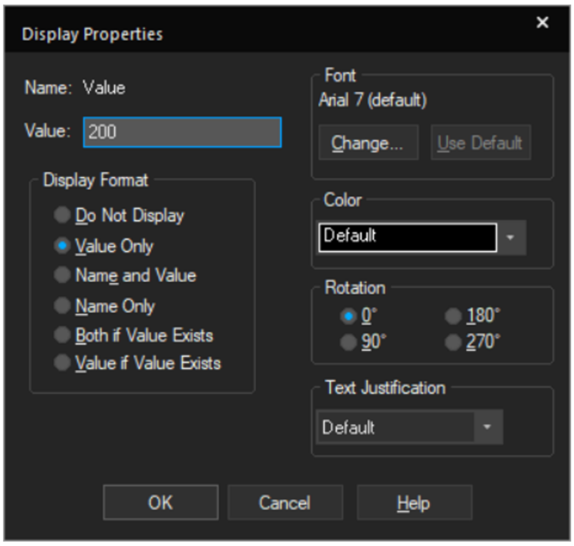


Figure 1.7: Assign the resistance

If the resistance is Ohm, no unit is required in the **Value** field. Repeat the first 2 steps to finalize all resistors in the circuit.

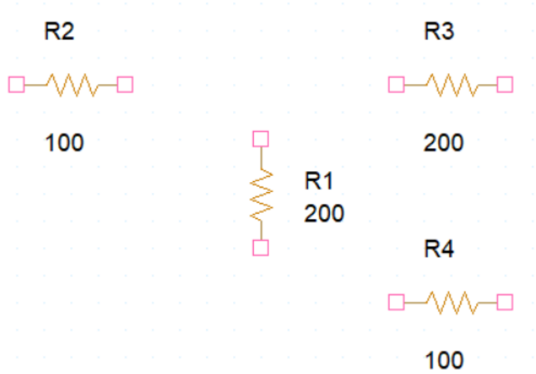


Figure 1.8: Place other resistors in PSpice

Some hot keys including **Ctrl + C** and **Ctrl + V** can be used to copy an old device.

Step 3: In order to place a voltage supple, find the component named **VDC** (DC Voltage Source) in the favourite list, or filter it on the search area. Double click on the voltage supply and change to a desired value. The result after this step is expected as figure bellow.

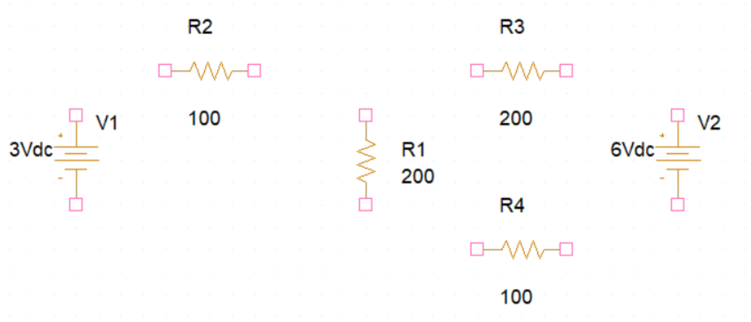


Figure 1.9: Place the power supply using VDC component

Step 4: Wire all components by selecting the Wire command on the right panel (hot key is W).

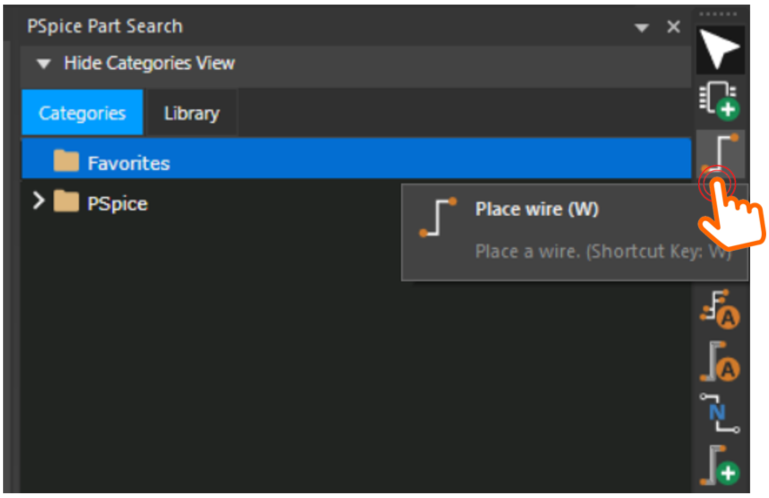


Figure 1.10: Wire the whole circuit

By clicking a start point and an end point, a wire is placed. In order to delete, select the wire and press **Delete**. The picture of the circuit after this step is depicted as follow.

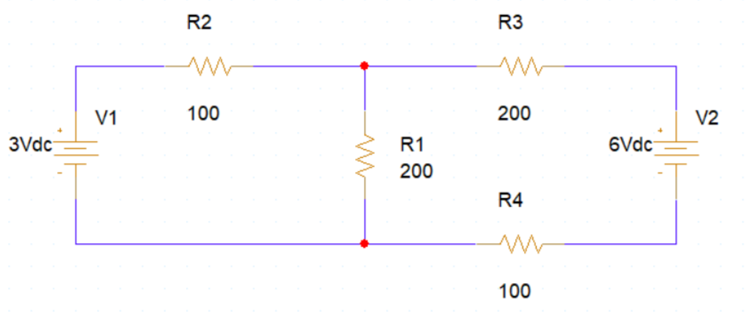


Figure 1.11: Finish all wire in the circuit

Step 5: Place a Ground symbol. This step is very important to analyze the voltage in a circuit as a reference voltage (0V) is required. The ground symbol is available on the right panel of the software.

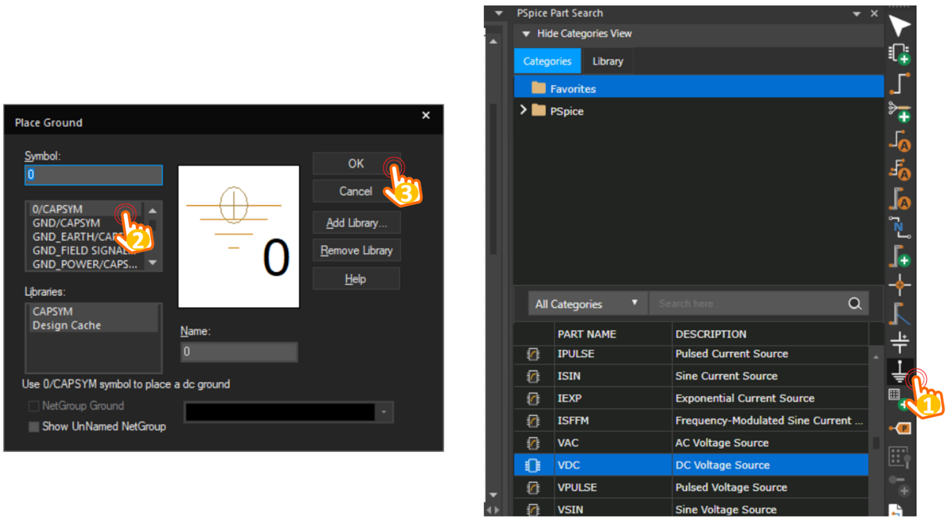


Figure 1.12: Add a ground symbol to the circuit

Wiring the ground to a point in the circuit, the final result should be like the figure bellow.

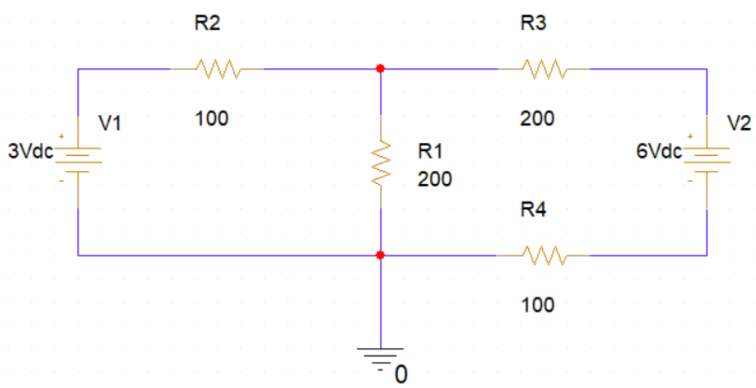


Figure 1.13: A ground point is added to the circuit

5 Create a simulation profile

Before the simulation is started, a simulation profile (or the simulation configuration) is required. From menu **PSpice**, select **New Simulation Profile** as follow:

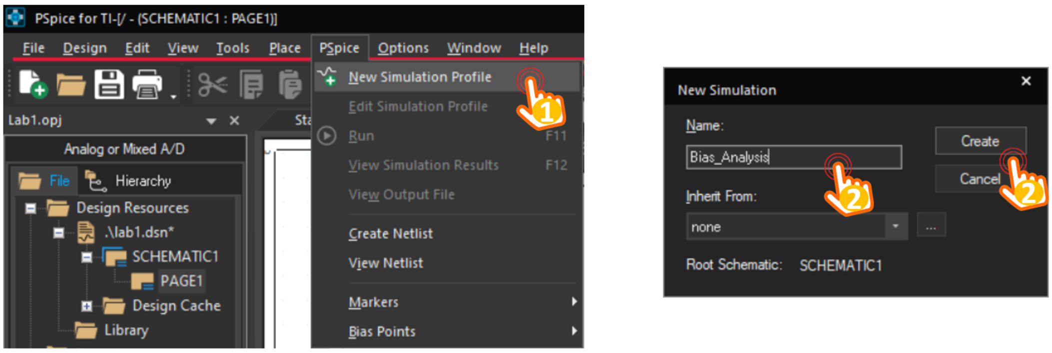


Figure 1.14: Create a simulation profile

When the simulation setting dialog is appeared, please select the **Bias Point** for analysis type.

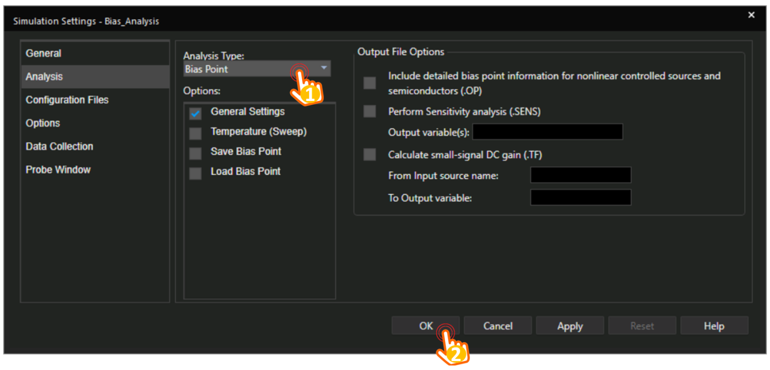


Figure 1.15: Select Bias Point simulation

In PSpice, the bias point analysis calculates the node voltages and currents through the devices in the circuit. Bias point analysis also takes into account any voltage sources applied to the circuit and any initial conditions set on devices or nodes in the circuit.

Finally, click on menu **PSpice** to select **Run**, or press **F11** to start the simulation. The simulation results are displayed directly on the circuit as follow:

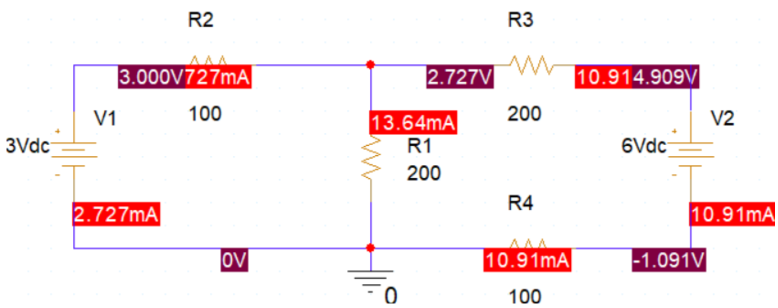


Figure 1.16: Volate and Current in the whole circuit

In order to display simulation results, go to menu **PSpice**, select **Bias Point** and enable the information you need. Students are also proposed to double check their solutions with the simulation results.

6 Exercise and Report

In this section, students are proposed to work in some circuit analysis, mostly based on resistors. Some explanations are required and will be considered as a part of the report. Note that the calculation subsection expects to see formulas and equations rather than only the results.

6.1 Exercise 1

Given the following circuit. Calculate the value of the voltage v_0 and the current i. Then, simulate the circuit to check it out.

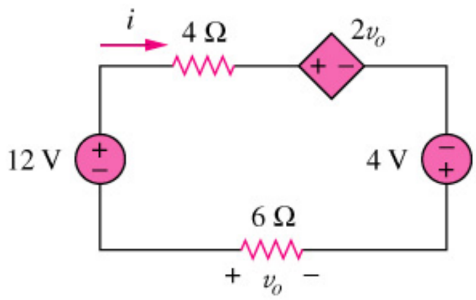


Figure 1.17: Find the voltage and the current in the given circuit using KVL

6.1.1 Calculation

Notes:
Explanations, formulas, and equations are expected rather than only results.

According to the Kirchhof Voltage Law and the Ohm Law, we have the system of two equation:

$$\begin{cases} 4i - 2v_0 + 4 + 6i + 12 = 0 \\ 6i = v_0 \end{cases}$$

Then, we can obtain:

$$\begin{cases} i = 8(A) \\ v_0 = 48(V) \end{cases}$$

6.1.2 Simulation

Tips:
To get the Voltage Controlled Voltage Source (VCVS) from the PSpice, under the ***Place*** menu, find ***PSpice Component*** > ***Source*** > ***Controlled Sources*** > ***VCVS***.

A circuit used for the simulation in this exercise maybe like this:

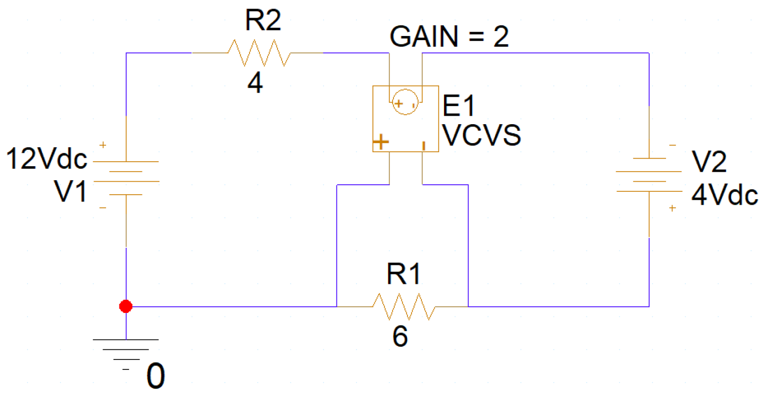


Figure 1.18: A circuit containing a Voltage Controlled Voltage Source in PSpice

Simulation result (image):

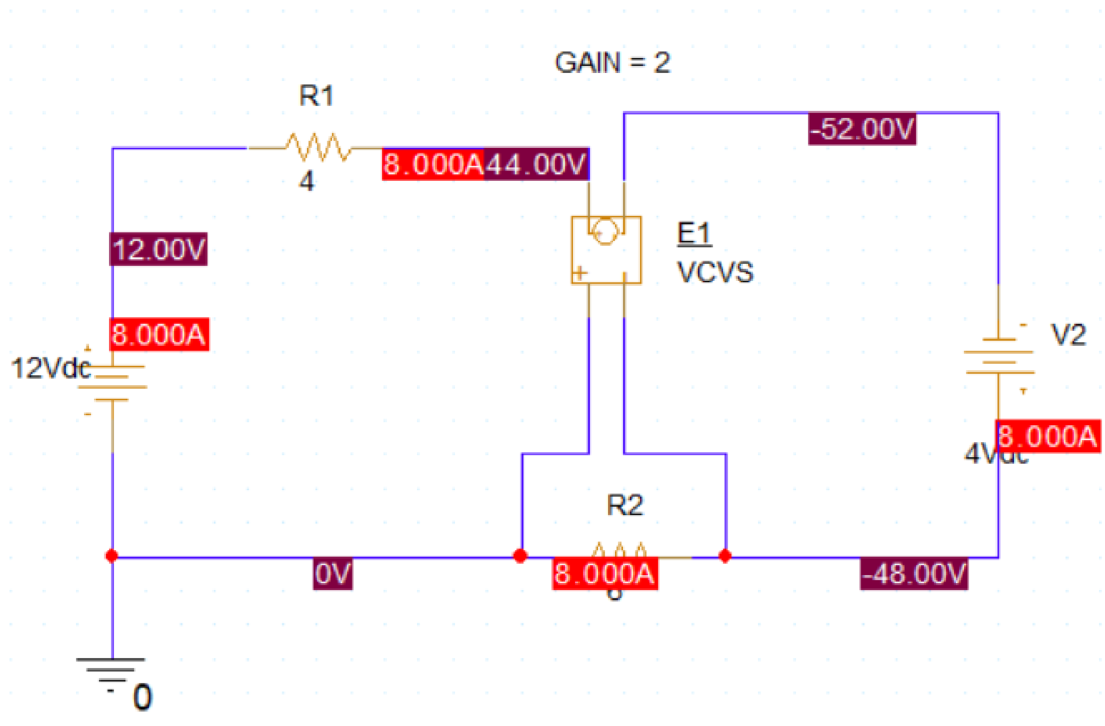


Figure 1.19: The result of Exercise 1

The voltage at the VCVS is:

V = 44 - (-52) = 96 (V)

This result equals to $2v_0$ so $v_0 = 48(V)$

6.2 Exercise 2

Given the following circuit, students rearrange the circuit to clarify its serial and/or parallel topology. Then, apply the knowledge you've learned to find the equivalent resistance value between two circuit terminals A and F. Finally, perform the simulation to check if the current through the whole circuit is correctly calculated.

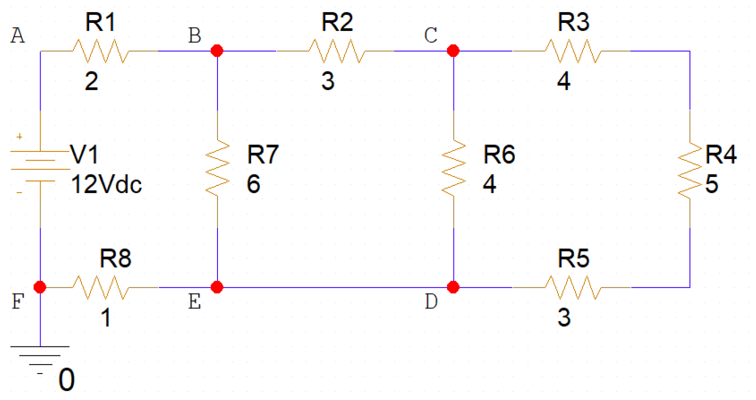


Figure 1.20: Find the equivalent resistance value between terminals A and F

6.2.1 Rearrange the circuit

Insert the rearranged circuit here. Don't forget the resistance values and the nodes' names.

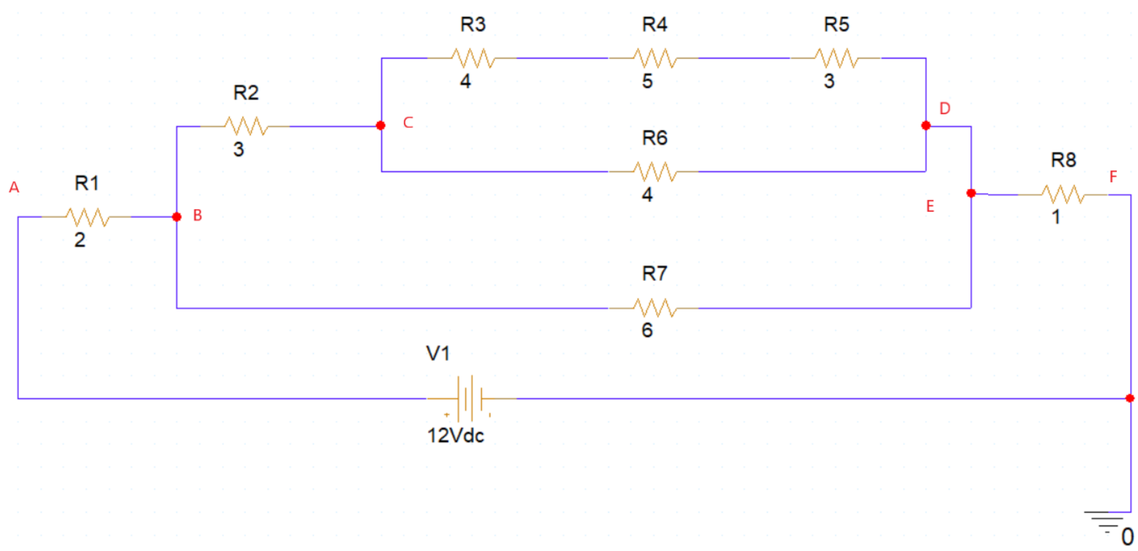


Figure 1.21: The rearranged circuit.

6.2.2 Calculation

Convention:
The equivalent resistance between the two terminals A and B of a circuit segment containing only R1, R2, R3, and R4 may be named R_{AB_1234}.

Notes:
Explanations, formulas, and equations are expected rather than only results.

Sine R_3, R_4, R_5 are in series and they are parallel with R_6,so:

$$R_{CD\_3456} = \frac{(R_3 + R_4 + R_5).R_6}{(R_3 + R_4 + R_5) + R_6} = \frac{(3+4+5).4}{(3+4+5)+4} = 3(\Omega)$$

Because R_{CD_3456}, R_2 are in series and they are in parallel with R_7, then:

$$R_{BE} = \frac{(R_{CD\_3456} + R_2).R_7}{(R_{CD\_3456} + R_2) + R_7} = \frac{(3+4).6}{(3+3)+6} = 3(\Omega)$$

As R_1, R_{BE} and R_8 are in series so the equivalent resistance of the circuit is:

$$R_{AF} = R_1 + R_{BE} + R_8 = 2 + 3 + 1 = 6\Omega)$$

$$I_{AB} = \frac{V_1}{R_{AF}} = \frac{12}{6} = 2(A)$$

6.2.3 Simulation

Simulation result (image):

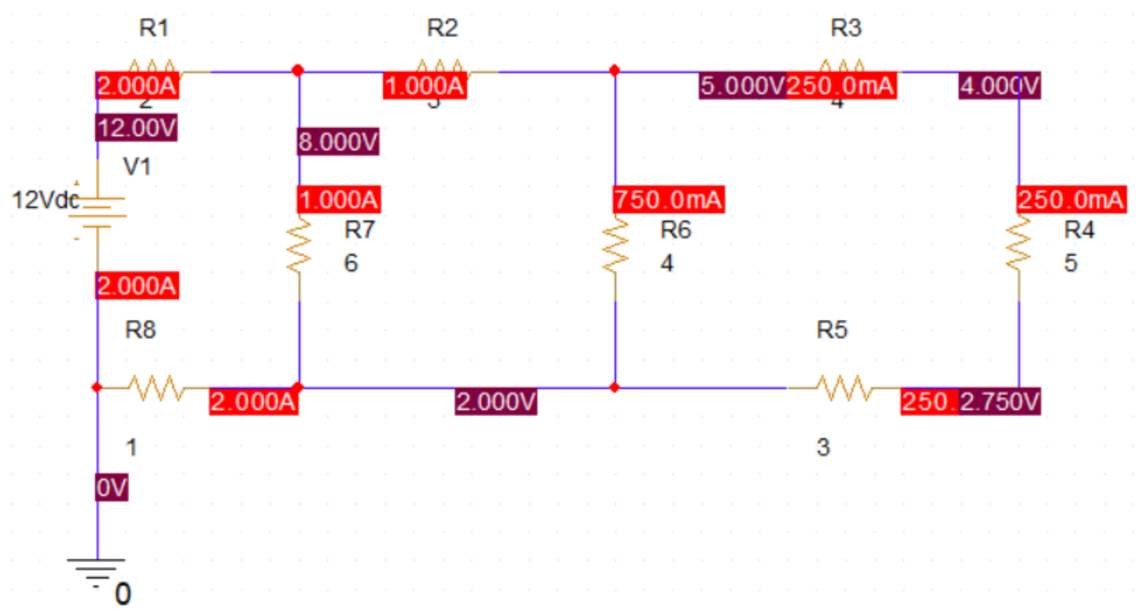


Figure 1.22: Results of both the rearranged circuit and the original circuit are the same.

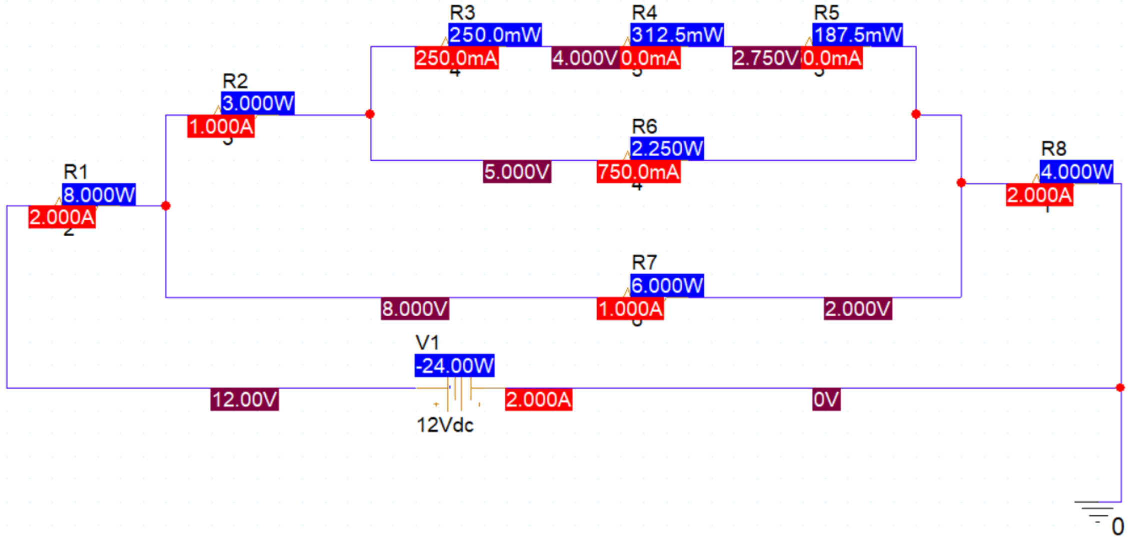


Figure 1.23: Results of both the rearranged circuit and the original circuit are the same.

6.3 Exercise 3

Given the following circuit, students rearrange the circuit to clarify its serial and/or parallel topology. Next, apply the knowledge you've learned to find the equivalent resistance value between two circuit terminals A and F, the voltage values at A, B, C, D, and E. Finally, perform the simulation to check your calculation.

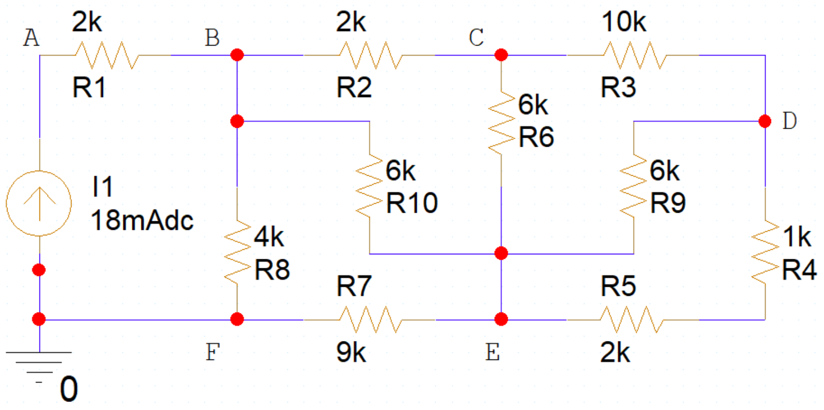


Figure 1.24: Find the whole-circuit equivalent resistance and the voltages at A, B, C, D, and E

6.3.1 Rearrange the circuit

Insert the rearranged circuit here. Don't forget the resistance values and the nodes' names.

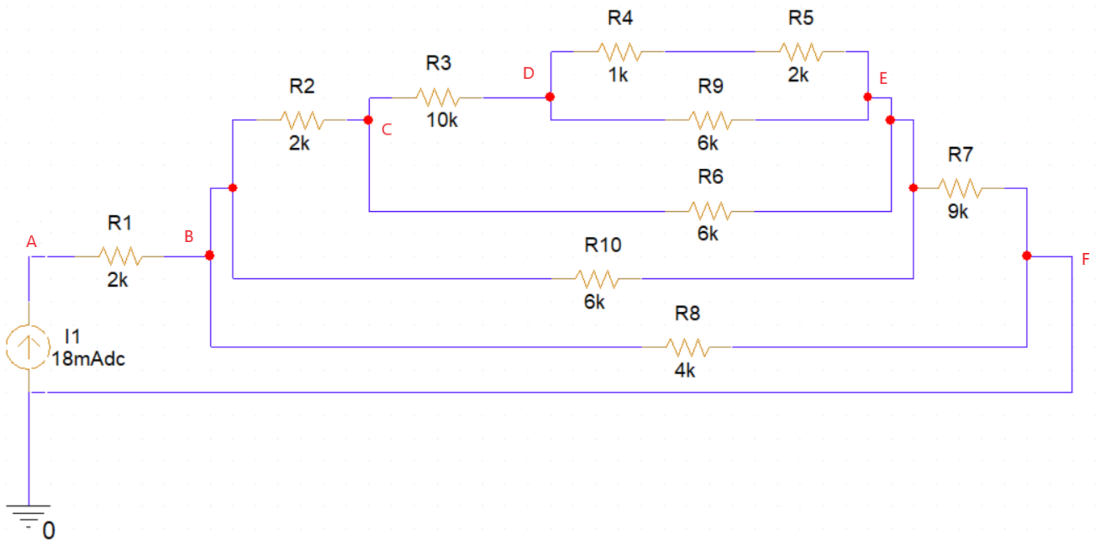


Figure 1.25: The rearranged circuit.

6.3.2 Calculation

Notes:
Explanations, formulas, and equations are expected rather than only results.

After rearranging the circuit we can obtain:

$$\begin{cases} R_{DE} = \dfrac{(R_4+R_5).R_9}{(R_4+R_5)+R_9} = 2(k\Omega) \\ R_{3DE6} = \dfrac{(R_3+R_{DE}).R_6}{(R_3+R_{DE})+R_6} = 4(k\Omega) \\ R_{2C10} = \dfrac{(R_2+R_{3DE6}).R_{10}}{(R_2+R_{3DE6})+R_{10}} = 3(k\Omega) \\ R_{B78} = \dfrac{(R_{2C10}+R_7).R_8}{(R_{2C10}+R_7)+R_8} = 3(k\Omega) \end{cases}$$

=> $R_{AF} = R_1 + R_{B78} = 5(k\Omega)$

$V_A = I.R_{AF} = 18.10^{-3}.5.10^3 = 90(V)$

$U_{AB} = I.R_1 = 18.10^{-3}.2.10^3 = 36(V)$

=> $V_B = V_A - U_{AB} = 90 - 36 = 54(V)$

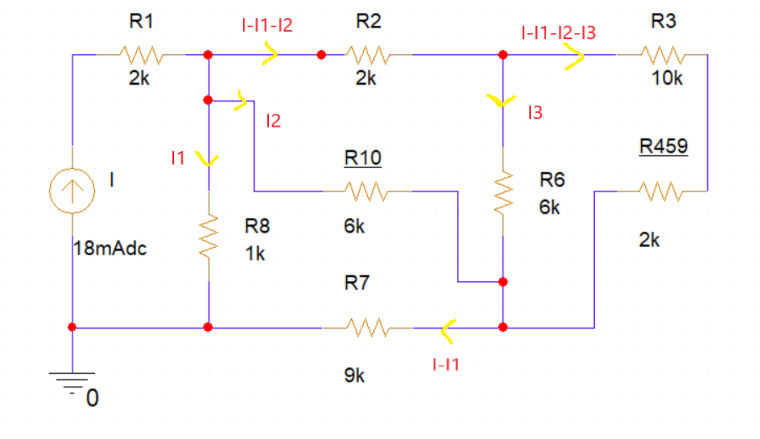


Figure 1.26: Dividing the current to find the value through each branches.

Applying the Kirchhoff Voltage Law for three inner loops giving us the system of three equation:

$$\begin{cases} 6000.I_2 + 9000(18.10^{-3} - I_1) = 4000.I_1 \\ 2000(18.10^{-3} - I_1 - I_2) + 6000.I_3 = 6000.I_2 \\ 12000.(18.10^{-3} - I_1 - I_2 - I_3) = 6000.I_3 \end{cases}$$

$$=> \begin{cases} I_1 = \dfrac{27}{1000}(A) \\ I_2 = \dfrac{9}{4000}(A) \\ I_3 = \dfrac{3}{2000}(A) \end{cases}$$

$U_{BC} = R_2.(I - I_1 - I_2) = 4.5(V)$

$=> V_C = V_B - U_{BC} = 54 - 4.5 = 49.5(V)$

$U_{CD} = R_3.(I - I_1 - I_2 - I_3) = 7.5(V)$

$=> V_D = V_C - U_{CD} = 49.5 - 7.5 = 42(V)$

$U_{BE} = R_{10}.I_2 = 13.5(V)$

$=> V_E = V_B - U_{BE} = 54 - 13.5 = 40.5(V)$

6.3.3 Simulation

Simulation result (image):

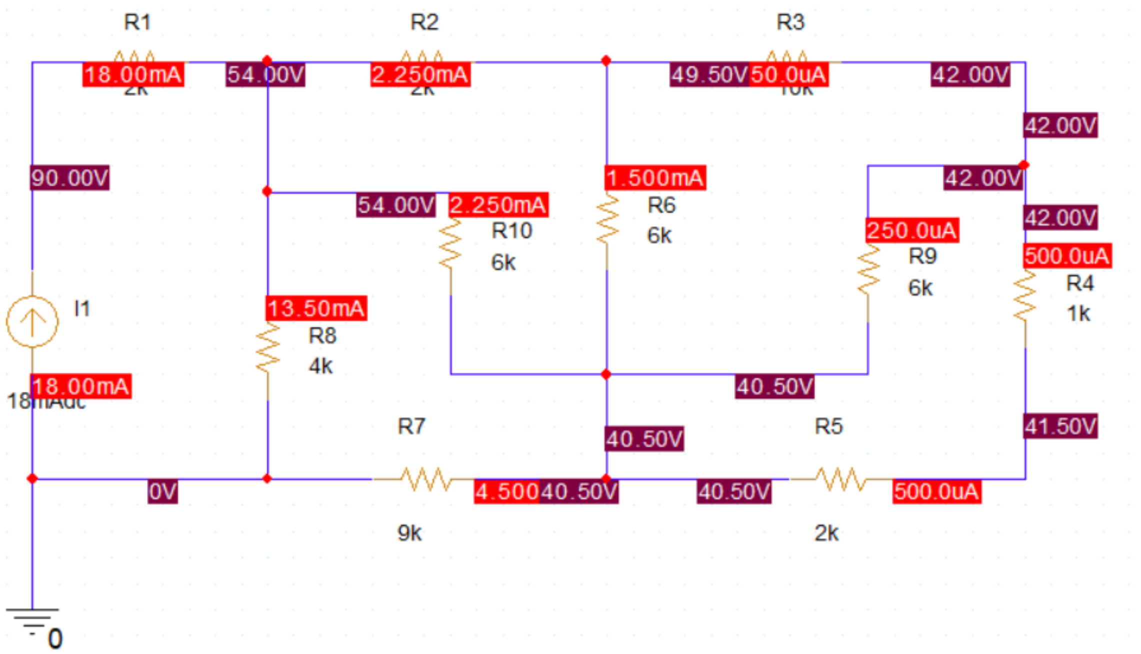


Figure 1.27: Our calculations match the result using simulation tool.

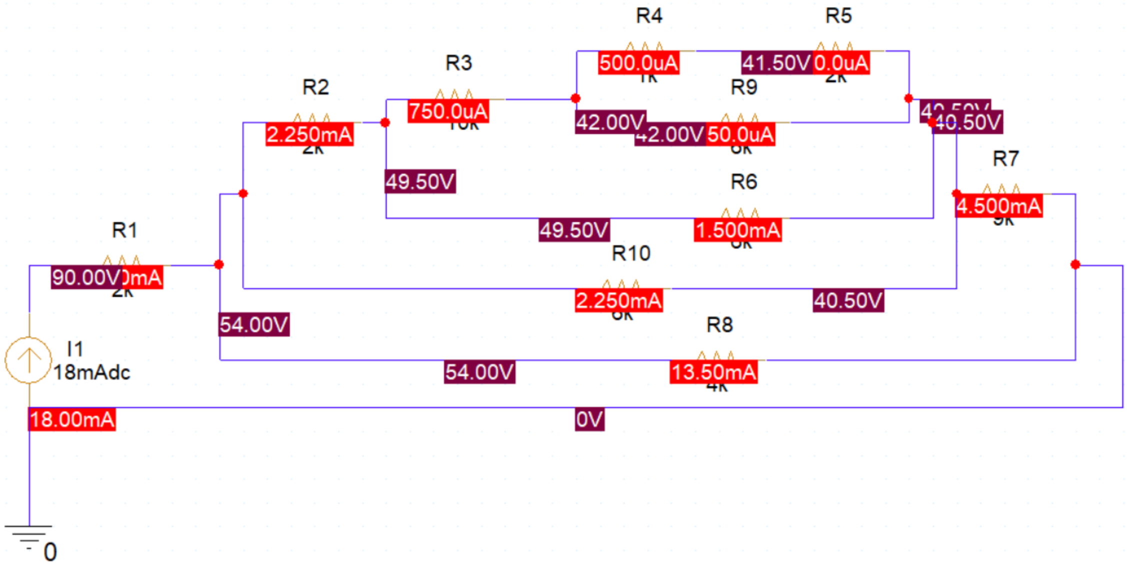


Figure 1.28: Our calculations match the result using simulation tool.

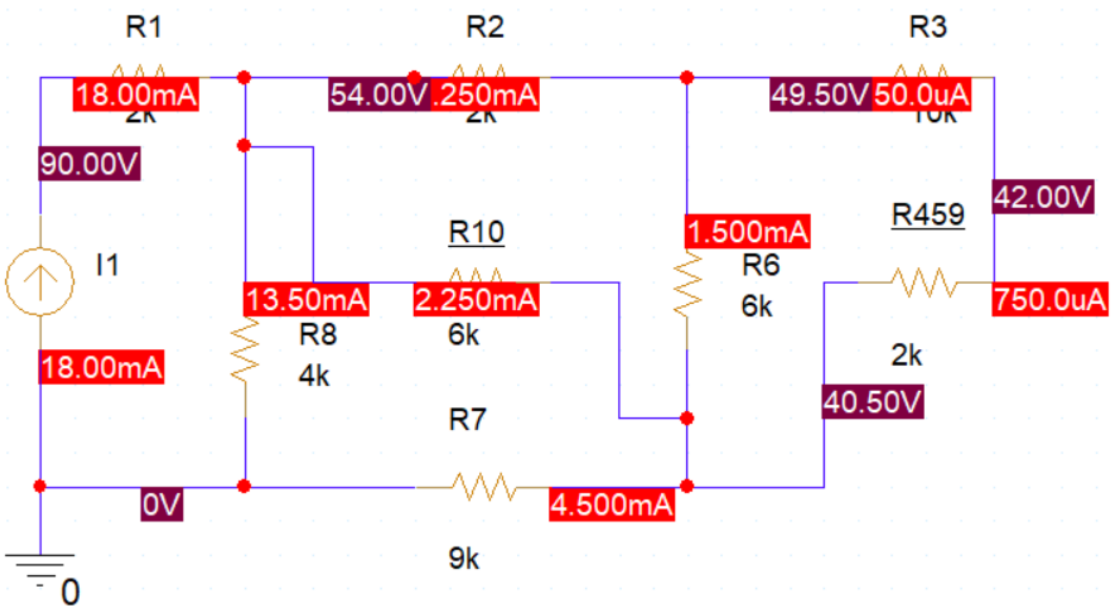


Figure 1.29: Our calculations match the result using simulation tool.

6.4 Exercise 4

Given the following circuit, find I_1, I_2, I_3, V_a, and V_b. Present your calculation steps and check them out by performing the simulation.

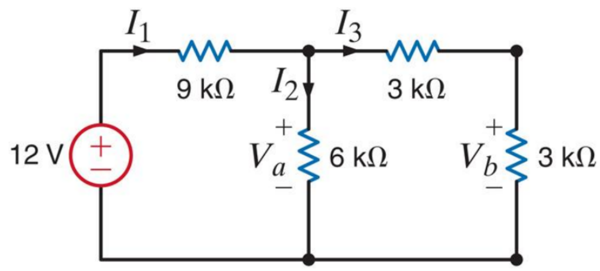


Figure 1.30: Find I_1, I_2, I_3, V_a, and V_b

6.4.1 Calculation

Notes:
Explanations, formulas, and equations are expected rather than only results.

It is obviously that the two 3kΩ resistors on the left are in series and be parallel with the 6kΩ one then the equivalent resistor of the left loop will be in series with the 9kΩ.

So the whole circuit equivalent resistance: $R_{eq} = \frac{(3000+3000).6000}{(3000+3000)+6000} + 9000 = 12(k\Omega)$

$I_1 = \frac{V}{R_{eq}} = \frac{12}{12.10^3} = 1(mA)$

Applying the Kirchhoff Voltage Law, we have:

$$\begin{cases} I_1 = I_2 + I_3 \\ 9000.I_1 + 6000.I_2 = 12 \end{cases} => \begin{cases} I_2 = 0.5(mA) \\ I_3 = 0.5(mA) \end{cases}$$

$V_a = 6000.I_2 = 3(V)$

$V_b = 3000.I_3 = 1.5(V)$

6.4.2 Simulation

Simulation result (image):

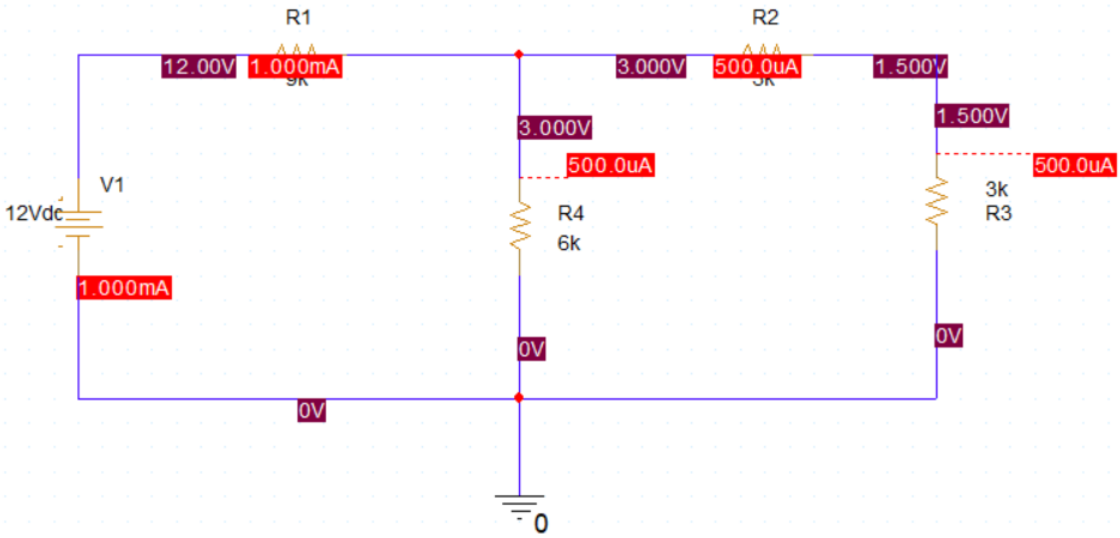


Figure 1.31: The result using simulation tool.

6.5 Exercise 5

Given the network as shown below:

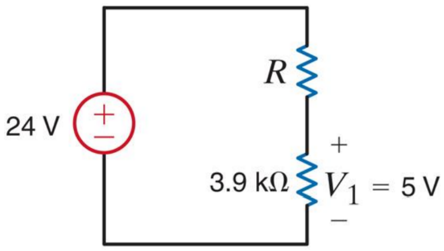


Figure 1.32: Select resistor R from the standard resistors list and do the following requirement

Notes:
Explanations, formulas, and equations are expected rather than only results.

a. Find the required value for the resistor R.

$$I = \frac{V_1}{R_1} = \frac{5}{3.9 \times 10^3} = 1.282(mA)$$

$$V_R = V - V_1 = 24 - 5 = 19(V)$$

$$R = \frac{V_R}{I} = \frac{19}{1.282 \times 10^{-3}} = 14.82(k\Omega)$$

b. Use Table 2.1 in the lecture slide to select a standard 10% tolerance resistor for R. R in the circuit may be a single resistor or a combination of many resistors as long as these resistors are meet the standard values and are available in the market.

According to the Table, we will the standard value that nearest to the one we have calculated above.

The selected resistor: 15kΩ

c. Using the resistor selected in (b), determine the voltage across the 3.9k resistor.

$$I_1 = \frac{V}{R + R_{V1}} = \frac{24}{(15 + 3.9) \times 10^3} = 1.2698(mA)$$

The value of V_1 according to the selected resistor R:$I_1.R_{V1} = 4.9524(V)$

d. Calculate the percent error in the voltage V1, if the standard resistor selected in (b) is used.
The percent error in the voltage V1: $error = \frac{4.9524 - 5}{5} \times 100 = -0.952\%$

e. Determine the power rating for this standard component.

$$P_R = R.I_1^2 = 15 \times 10^3.(1.2698 \times 10^{-3})^2 = 24.18(mW)$$

6.5.1 Simulation

Simulation result (image):

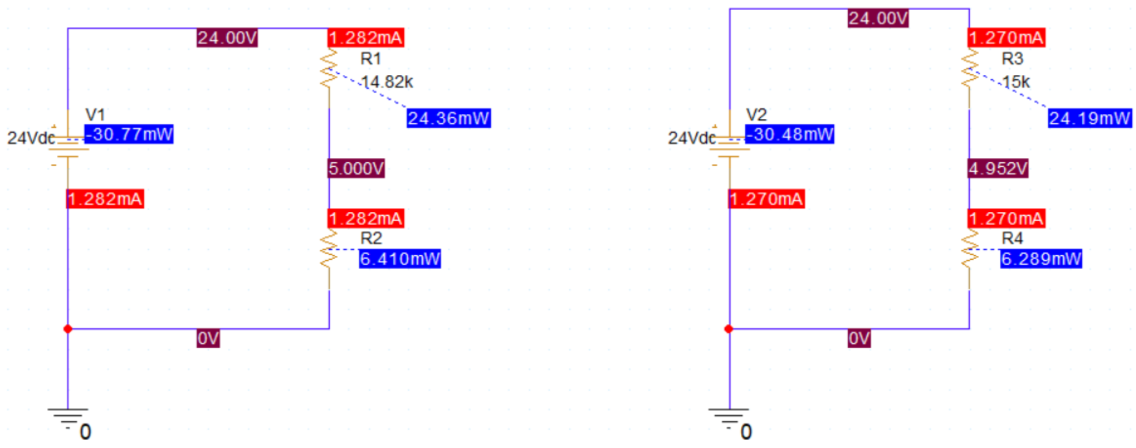


Figure 1.33: Results on simulation are the same with the calculation above.

6.6 Exercise 6

Given the following circuit. Apply the knowledge you've learned to transform it into another form in which you can find total equivalent resistance R_{ab} more easily. Next, find the value of the current i through the circuit and perform a simulation to check it out.

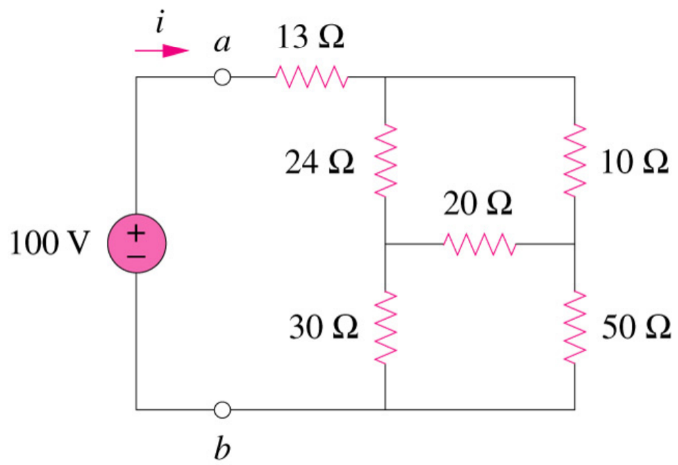


Figure 1.34: Transform the circuit, then find the equivalent resistance R_{ab} and the current i through the circuit.

6.6.1 Circuit transformation

Insert the transformed circuit here.

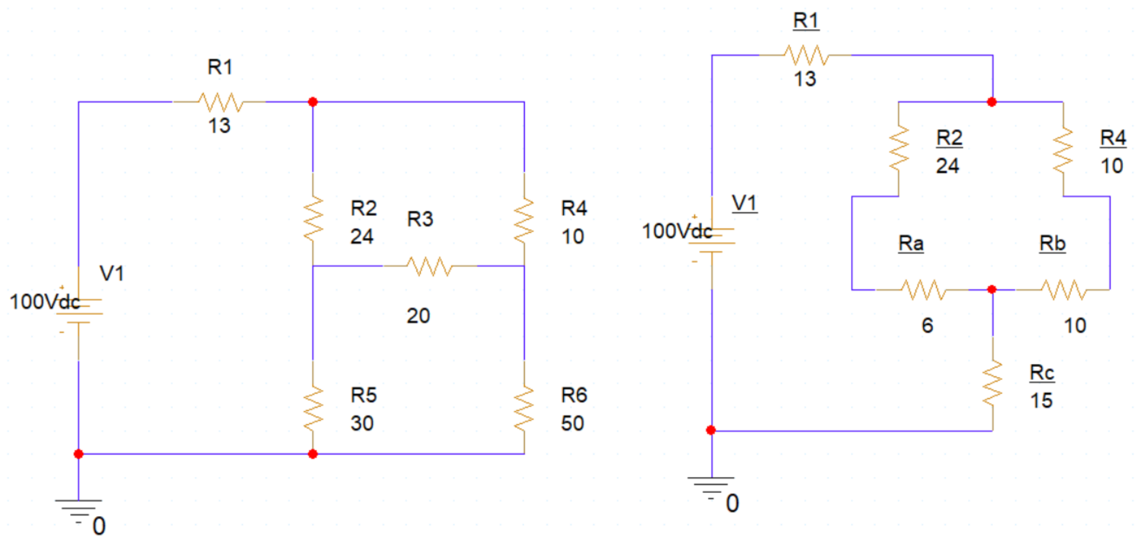


Figure 1.35: The transformed circuit.

Explain any calculations (i.e., write out the formulas you used).

It is certainly that R_3, R_5, R_6 are in the delta connection, so we can convert them into the star connection with the corresponding resistors R_a, R_b, R_c.

$$R_a = \frac{R_3.R_5}{R_3 + R_5 + R_6} = 6(\Omega)$$
$$R_b = \frac{R_3.R_6}{R_3 + R_5 + R_6} = 10(\Omega)$$
$$R_c = \frac{R_5.R_6}{R_3 + R_5 + R_6} = 15(\Omega)$$

6.6.2 Calculation

Notes:
Explanations, formulas, and equations are expected rather than only results.

In the inner loop, we have two pairs of resistors that are in series which are R_a, R_2 and R_b, R_4, respectively. Besides, these two pairs are also in parallel.

So the equivalent resistance of this loop is: $R_{eq1} = \dfrac{(R_a + R_2).(R_b + R_4)}{(R_a + R_2) + (R_b + R_4)} = 12(\Omega)$

$$R_{ab} = R_1 + R_{eq1} + R_c = 40(\Omega)$$

$$i = \frac{V}{R_{ab}} = 2.5(A)$$

6.6.3 Simulation

Simulation result (image):

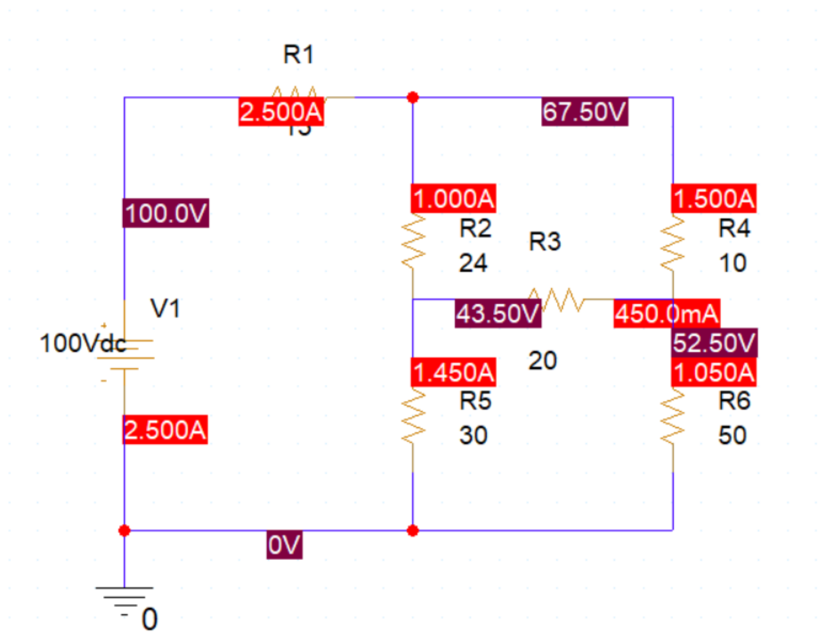


Figure 1.36: The simulation's results.

6.7 Exercise 7

Given the following circuit. Apply the knowledge you've learned to transform it into another form in which you can find total equivalent resistance more easily. Next, find the value of the current I_S through the circuit and perform a simulation to check it out.

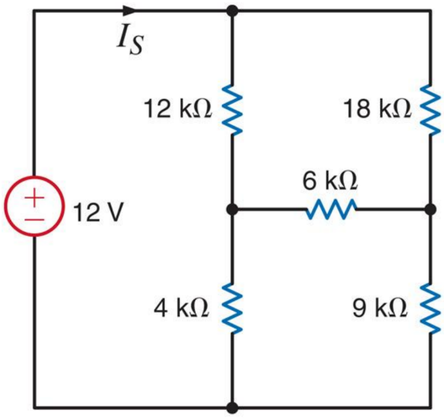


Figure 1.37: Transform the circuit, then find the equivalent resistance and the current I_S through the circuit.

6.7.1 Circuit transformation

Insert the transformed circuit here. Insert the transformed circuit here.

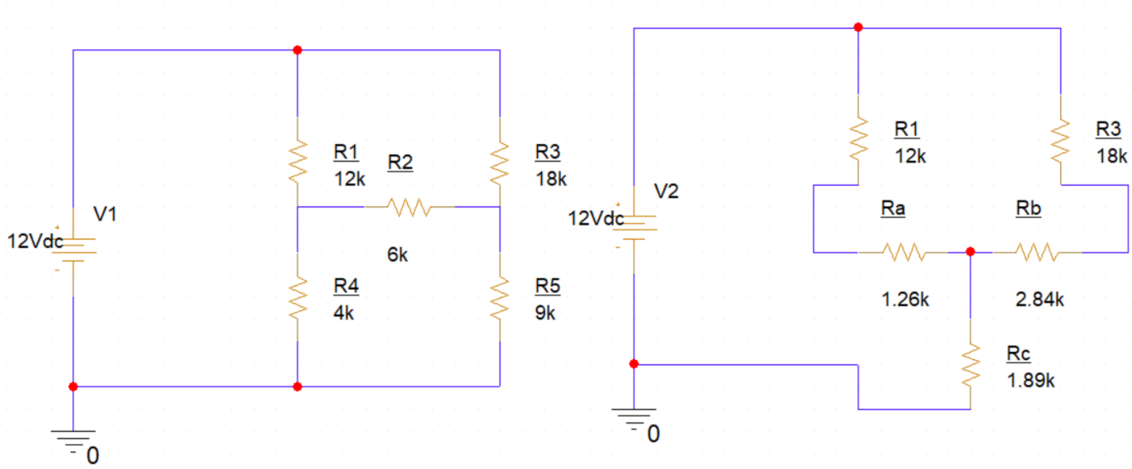


Figure 1.38: The transformed circuit.

Explain any calculations (i.e., write out the formulas you used).

It is certainly that R_2, R_4, R_5 are in the delta connection, so we can convert them into the star connection with the corresponding resistors R_a, R_b, R_c.

$$R_a = \frac{R_2.R_4}{R_2 + R_4 + R_5} = \frac{24}{19}(k\Omega)$$
$$R_b = \frac{R_2.R_5}{R_2 + R_4 + R_5} = \frac{54}{19}(k\Omega)$$
$$R_c = \frac{R_4.R_5}{R_2 + R_4 + R_5} = \frac{36}{19}(k\Omega)$$

6.7.2 Calculation

Notes:
Explanations, formulas, and equations are expected rather than only results.

In the inner loop, we have two pairs of resistors that are in series which are R_a, R_1 and R_b, R_3, respectively. Besides, these two pairs are also in parallel.

So the equivalent resistance of this loop is: $R_{eq1} = \dfrac{(R_a + R_1).(R_b + R_3)}{(R_a + R_1) + (R_b + R_3)} = \dfrac{154}{19}(k\Omega)$

$$R_{ab} = R_{eq1} + R_c = 10(k\Omega)$$

$$i = \frac{V}{R_{ab}} = 1.2(mA)$$

6.7.3 Simulation

Simulation result (image): Simulation result (image):

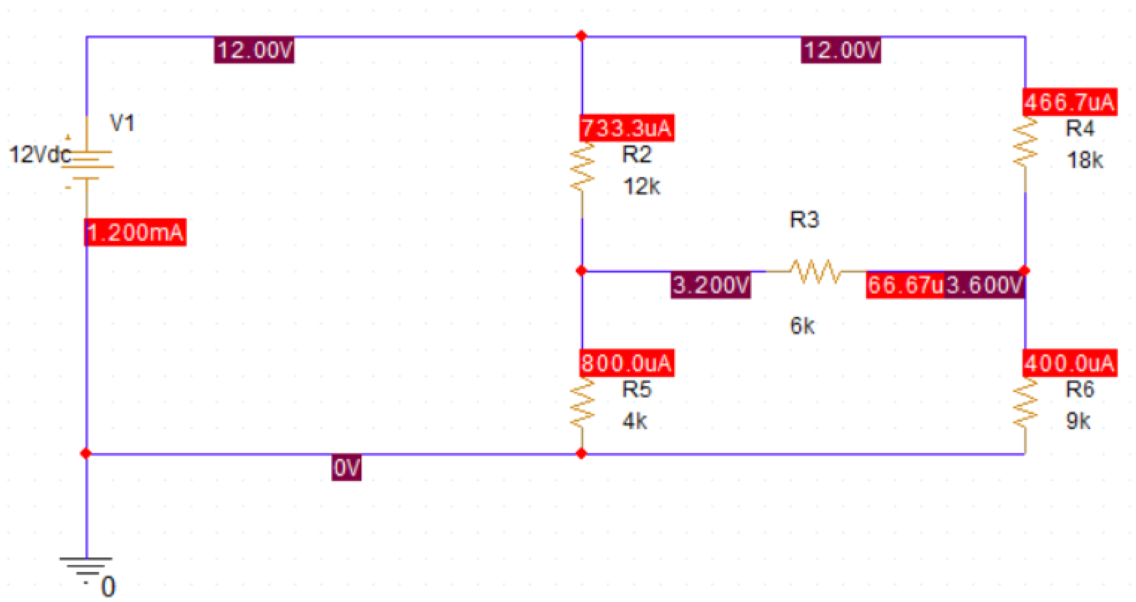


Figure 1.39: The simulation's results.

6.8 Exercise 8

Given the following circuit with p_2, p_3, and p_4 are absorbing powers of unknown electrical elements. First, use the knowledge you've learned to identify whether they are active or passive elements (supplying or absorbing power). To an element absorbing power, use a pure resistor with a proper value as a representative. To a power element, use an ideal DC voltage source with the corresponding value as a representative. Next, redraw the circuit and calculate the power that each element absorbs. Note that here we use the passive sign convention. Then, perform a simulation with the elements determined by the previous step.

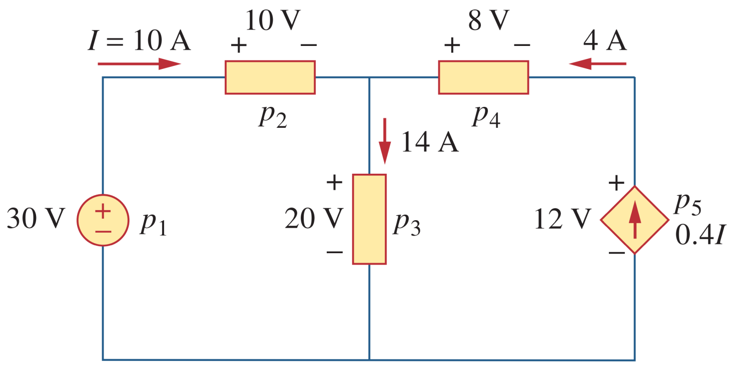


Figure 1.40: Determine the unknown elements and calculate the absorbing power of each

6.8.1 Identify the unknown elements

In Figure 1.40:
The 10 A current enters the positive pole of element p_2 and the power $P_2 = 10.10 = 100(W)$ is positive so p_2 is absorbing power. Similarly, p_3 is also an absorbing element.

On the left-hand side loop, the 4 A current approaches the negative pole of element p_4 and then leaves the positive pole. The power $P_4 = 4.8 = 32(W)$ is positive so p_4 is an supplying element. Eversince p_4 delivers power,the value of its power becomes negative number.

6.8.2 Redraw the circuit for simulation

Simulation result (image):
p_2 and p_3 are replaced by pure resistors with resistance of 1 Ω and 1.4286 Ω, respectively.

p_4 is replaced by an ideal DC voltage source with value of 8 V.

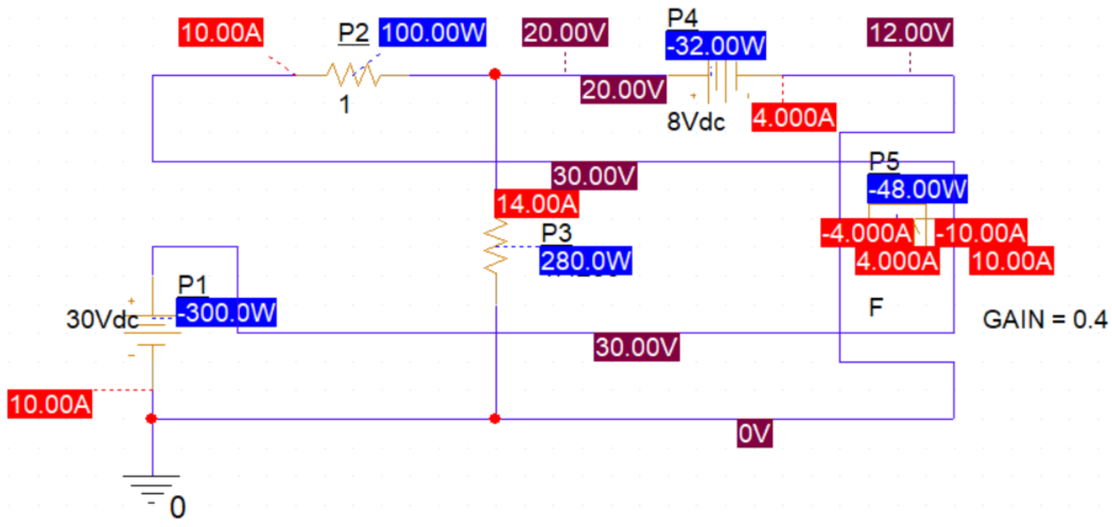


Figure 1.41: The results using simulation.

6.9 Exercise 9

Given the following circuit. Find the voltage v and the current i_x. According to the result, determine the elements whose absorbing power respectively p_1 and p_2 are active or passive (calculations are required). Note that here we use the passive sign convention. If an element consumes power, use a pure resistor with an appropriate value as a representative. If it is a power supply element, use a corresponding ideal DC voltage source to represent it. Perform a simulation to check how the circuit works.

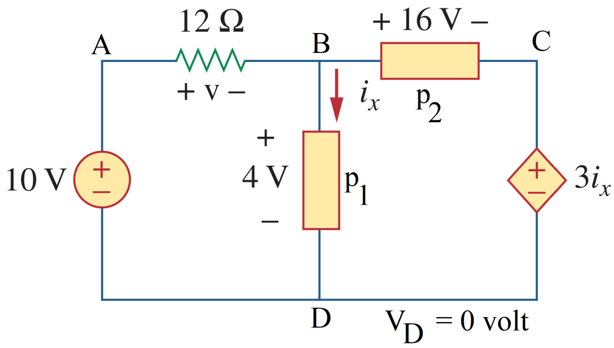


Figure 1.42: Find the unknown elements and variables, then check them out by simulation

6.9.1 Calculation

Notes:
Explanations, formulas, and equations are expected rather than only results.

Considering the left-hand side loop as a Gauss surface, we can have the 12 Ω resistor is in series with the p_1 element and both of them are in parallel with the 10 V voltage source.

Hence, we have:

$$v + V_{p_1} = V_{source} => v = V_{source} - V_{p_1} = 10 - 4 = 6(V)$$

If we let the outermost loop be a Gauss surface, then the 12 Ω resistor, the p_2 element and the CCVS are in series. Besides they are also in parallel with the 10 V voltage source.

So, we have:

$$v + V_{p_2} + 3i_x = V_{source} => 3i_x = V_{source} - v - V_{p_2} = 10 - 6 - 16 = -12 => i_x = \frac{-12}{3} = -4(A)$$

The i_x current enters the positive pole of p_1 element and its power $P_1 = 4.-4 = -16(W)$ is negative number so p_1 is a supplying element. It can be replaced by an ideal DC voltage source of 4 V.

$$I_{AB} = \frac{v}{R} = \frac{6}{12} = 0.5(A)$$

Since the value of i_x is negative, it should be in the opposite direction.
Therefore, $I_{BC} = I_{AB} + i_x = 0.5 + 4 = 4.5(A)$

I_{BC} enters the positive pole of p_2 element and its power $P_2 = 16 \times 4.5 = 72(W)$ is positive number so p_2 is a consuming element. It can be replaced by a pure resistor with resistance of $\frac{V_{p_2}}{I_{BC}} = \frac{16}{4.5} = 3.56(\Omega)$

$U_{CD} = U_{BD} - U_{BC} = 4 - 16 = -12(V)$

6.9.2 Simulation

Tips:
To get the Current Controlled Voltage Source (CCVS) from the PSpice, under the ***Place*** menu, find ***PSpice Component*** > ***Source*** > ***Controlled Sources*** > ***CCVS***.

A circuit used for the simulation in this exercise maybe like this:

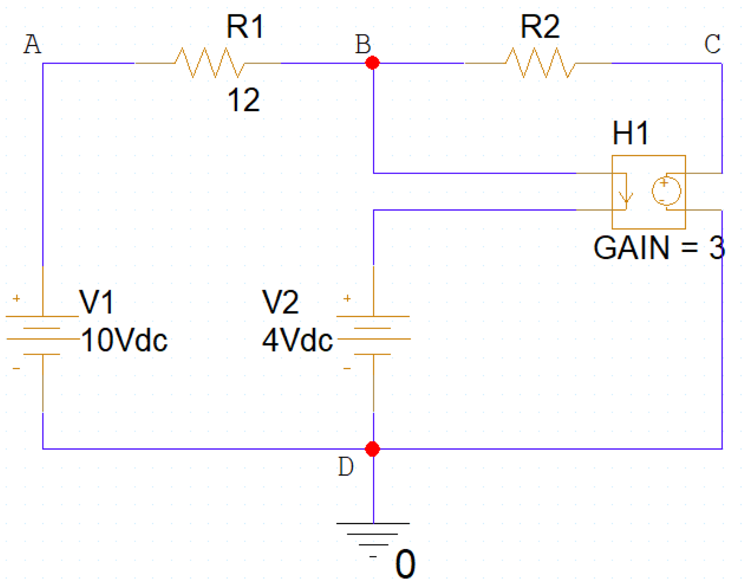


Figure 1.43: A circuit containing a Current Controlled Voltage Source in PSpice

Simulation result (image):

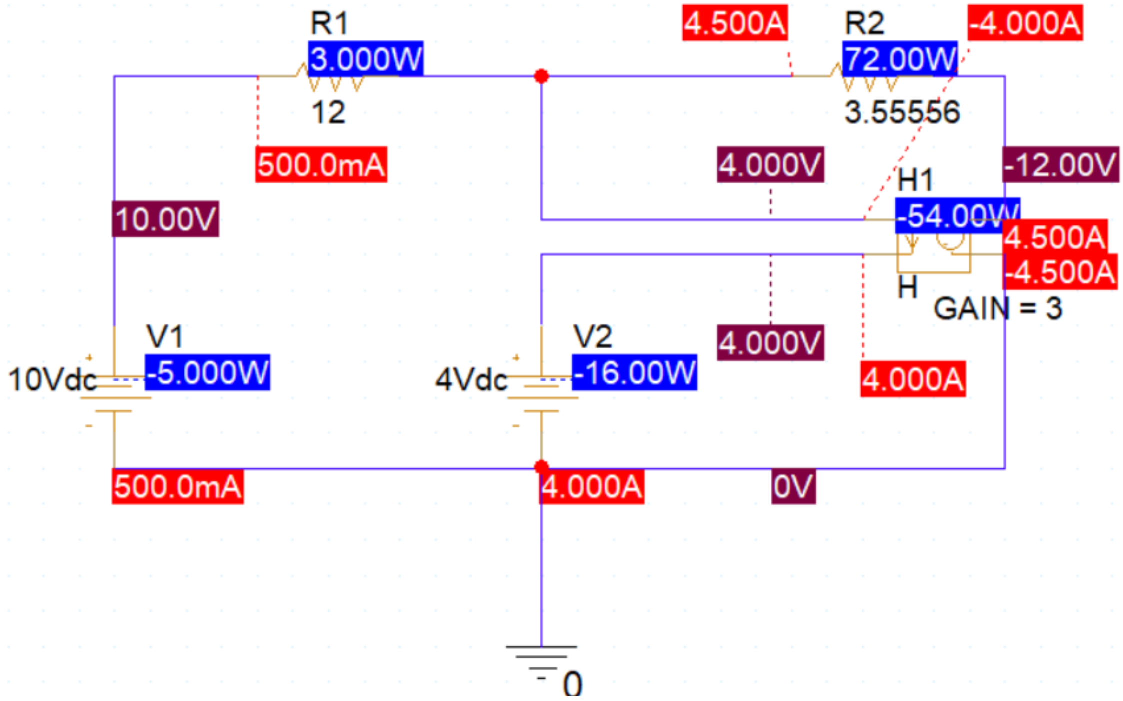


Figure 1.44: The results exactly match with those obtaining by simluation.

6.10 Exercise 10

Given the following circuit. Find the voltage V. You can do this in any way but remember to explain it in detail. Then simulate the circuit to check the result.

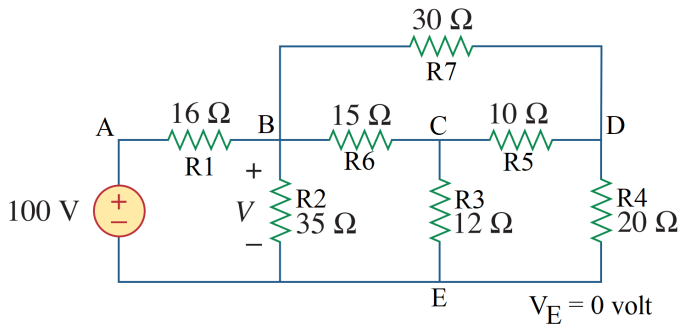


Figure 1.45: Find the voltage V

Answer:

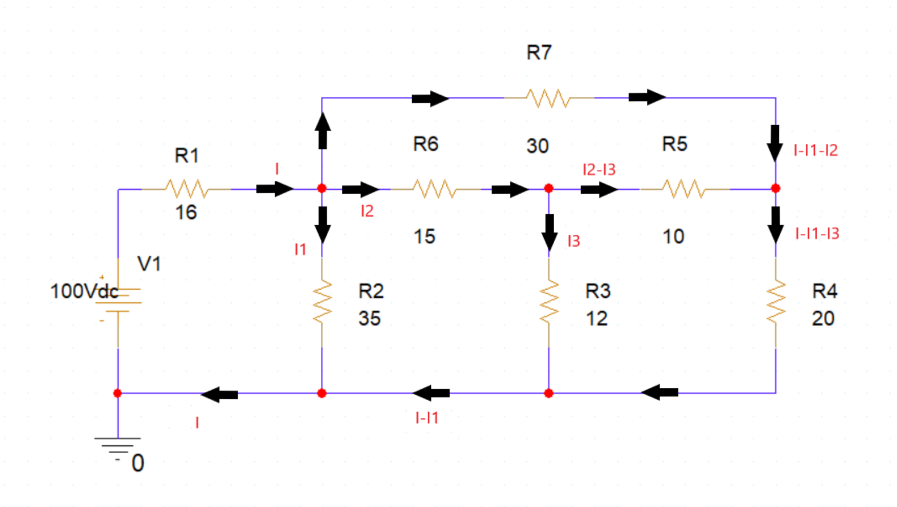


Figure 1.46: Dividing the current to find the value through each branches.

Applying the Kirchhoff Voltage Law for 4 inner loop we have the system of 4 equation:

$$\begin{cases} 16I + 35I_1 = 100 \\ 15I_2 + 12I_3 = 35I_1 \\ 10(I_2 - I_3) + 20(I - I_1 - I_3) = 12I_3 \\ 30(I - I_1 - I_2) = 15I_2 + 10(I_2 - I_3) \end{cases} <=> \begin{cases} 16I + 35I_1 = 100 \\ 35I_1 - 15I_2 - 12I_3 = 0 \\ 20I - 20I_1 + 10I_2 - 42I_3 = 0 \\ 30I - 30I_1 - 55I_2 + 10I_3 = 0 \end{cases} <=> \begin{cases} I = 3.614(A) \\ I_1 = 1.205(A) \\ I_2 = 1.591(A) \\ I_3 = 1.526(A) \end{cases}$$

Therefore, the voltage V is:

$V = R_2.I_1 = 42.175(V)$

Checking the results using simulation:

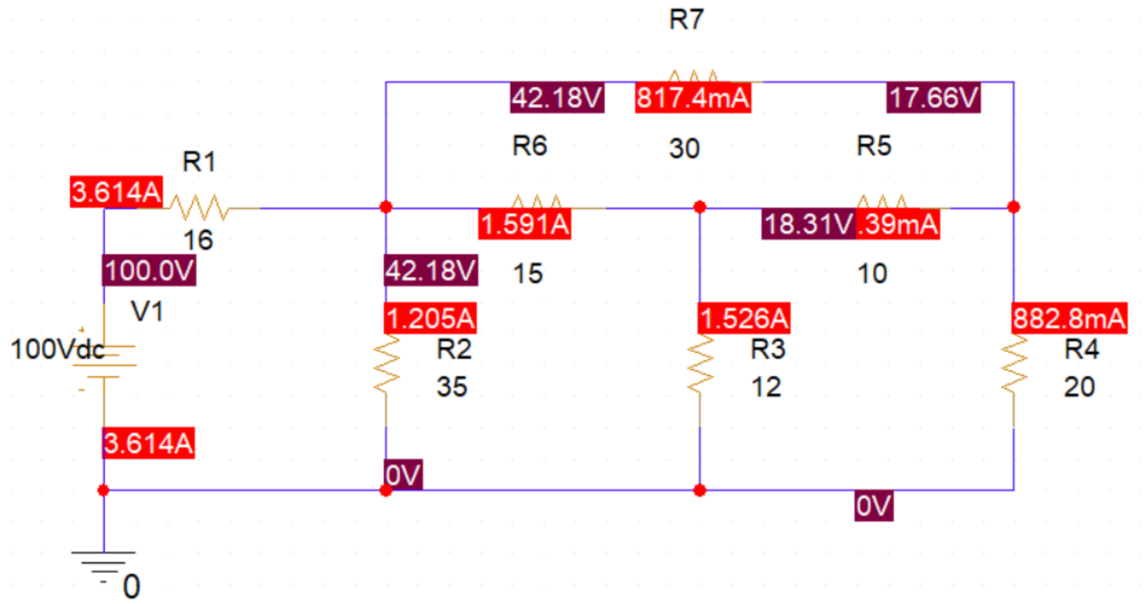


Figure 1.47: The simulation's values match with those obtaining above.

THE END