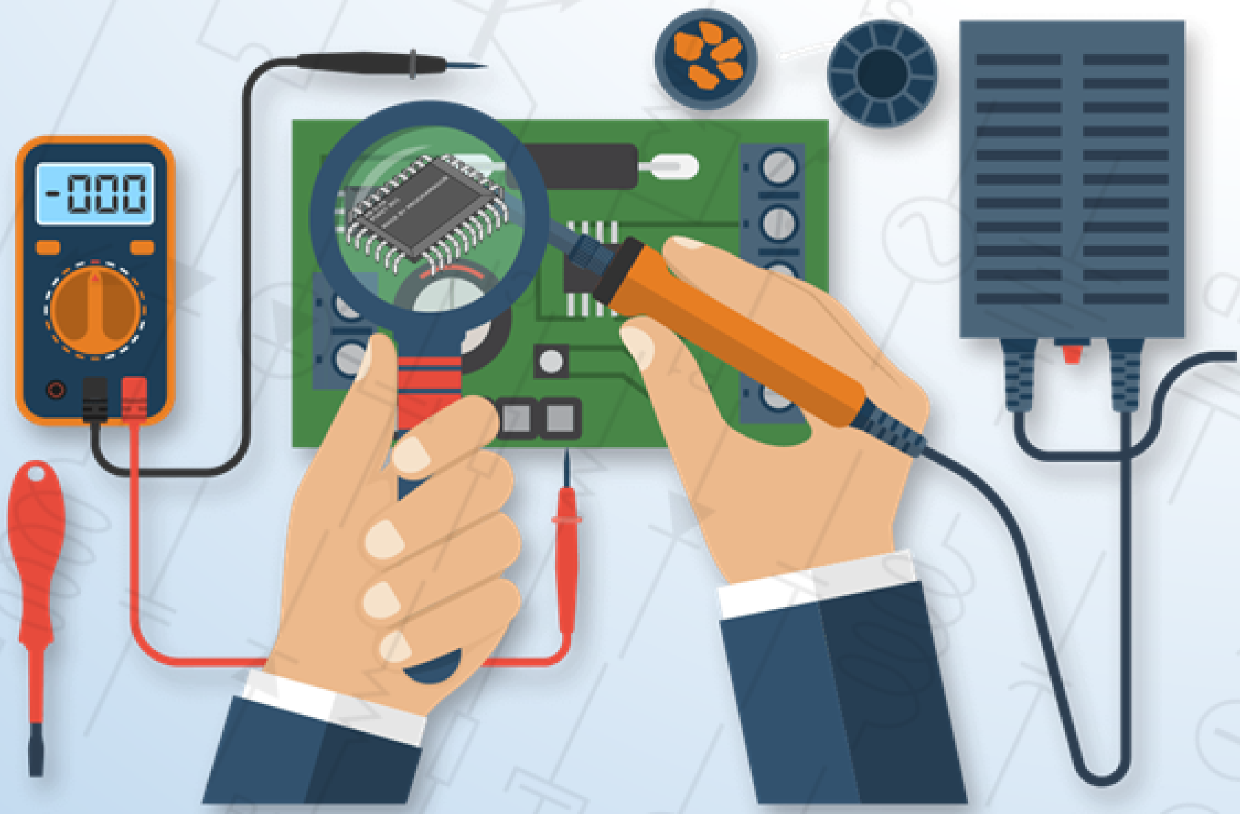
HO CHI MINH CITY UNIVERSITY OF TECHNOLOGY
COMPUTER ENGINEERING

# Electronic Device Component

## Project for Midterm Exam's Report



Date:7/11/2021

Cao Minh Quang - 2052221

# Contents


**Chapter 1. 24 Hour Project for Midterm Exam** 7

1 Equivalent Resistance 1 . . . . . . . . 8

1.1 PSPICE Simulation . . . . . . . . 8

1.2 Theory calculation . . . . . . . . 9

2 Equivalent Resistance 2 . . . . . . . . 10

2.1 PSPICE Simulation . . . . . . . . 10

2.2 Theory calculation . . . . . . . . 11

3 Current calculation . . . . . . . . 12

3.1 PSPICE Simulation . . . . . . . . 12

3.2 Theory calculation . . . . . . . . 14

4 Diode exercise 1 . . . . . . . . 16

4.1 PSPICE Simulation . . . . . . . . 16

4.2 Theory calculation . . . . . . . . 17

5 Diode exercise 2 . . . . . . . . 19

5.1 PSPICE Simulation . . . . . . . . 19

5.2 Theory calculation . . . . . . . . 20

6 Zener Diode . . . . . . . . 21

6.1 PSPICE Simulation . . . . . . . . 22

6.2 Theory calculation . . . . . . . . 23

7 BJT simulation circuit . . . . . . . . 23

7.1 PSPICE Simulation . . . . . . . . 23

7.2 Theory calculation . . . . . . . . 24

7.3 DC Sweep simulation . . . . . . . . 25

7.4 Theory calculation . . . . . . . . 25

8 BJT and Zener . . . . . . . . 26

8.1 PSPICE Simulation . . . . . . . . 26

8.2 Theory calculation . . . . . . . . 27

# CHAPTER 1

## 24 Hour Project for Midterm Exam

# 1 Equivalent Resistance 1

Simulate this circuit in PSPICE, with the **voltage supply between A and B is 24V**.

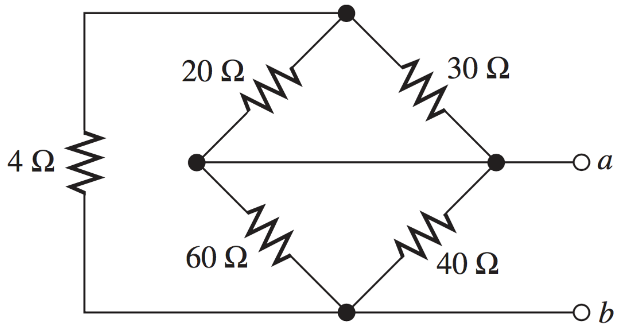



*Figure 1.1*: *Equivalent Resistance*

## 1.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

***Your image goes here***

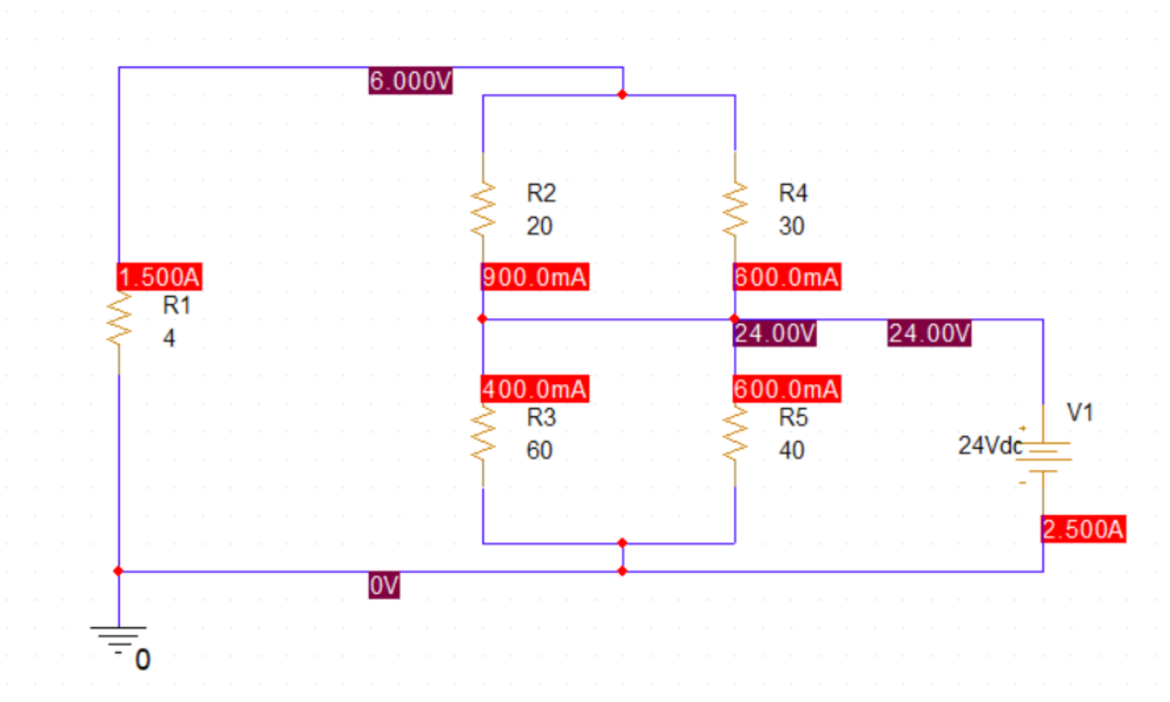



*Figure 1.2*: The bias point simulation of the circuit.

From the simulation results, what is the equivalent resistance of the circuit?

***Your answer goes here***

We have: $R_{eq} = \frac{V_{AB}}{I} = \frac{24}{2.5} = 9.6(\Omega)$

## 1.2 Theory calculation

In order to confirm with the simulations above, your calculations are required to present in this part.

First of all, we need to rearrange the circuit to have a better insight of the resistors connection.

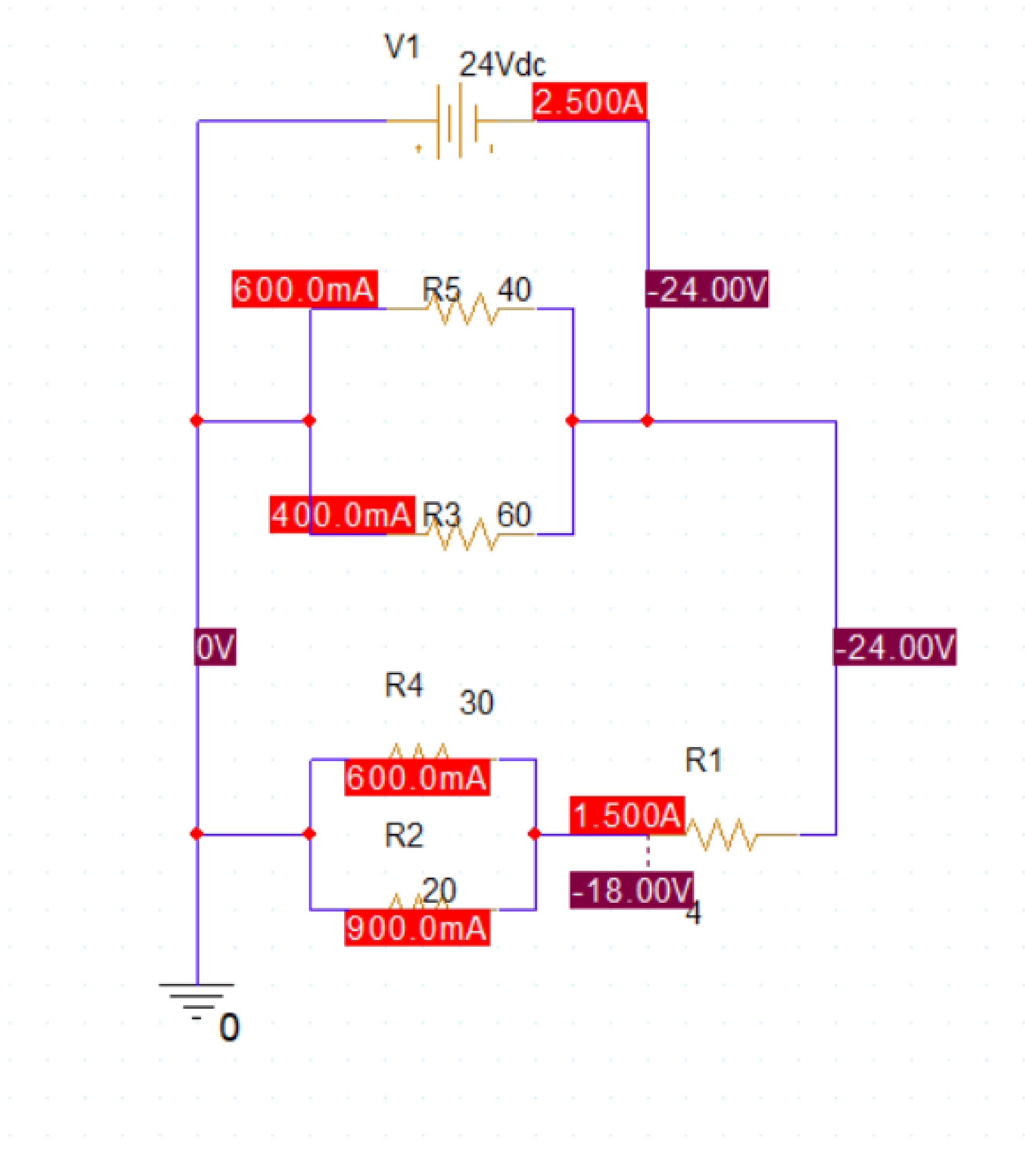



*Figure 1.3*: The rearranging circuit.

Now, it is obvious the 30Ω and 20Ω resistors are in parallel and together they are in series with the 4Ω resistor. The connection of the 40Ω and 60Ω resistors are parallel and this block is also in parallel with the previous one formed by the 30Ω, 20Ω and 4Ω resistors.

Consequently, we can obtain:

The equivalent resistance of the lower block is: $R_{eq1} = \frac{30 \times 20}{30 + 20} + 4 = 16(\Omega)$

The equivalent resistance of the upper block is: $R_{eq2} = \frac{40 \times 60}{40 + 60} = 24(\Omega)$

The equivalent resistance of the whole circuit is: $R_{eq} = \frac{16 \times 24}{16 + 24} = 9.6(\Omega)$

This value matches with the above one which obtained by the simulation.

# 2 Equivalent Resistance 2

Simulate this circuit in PSPICE, with the **voltage supply between A and B is 24V and all resistances in the circuit are 1k (R=1K)**.

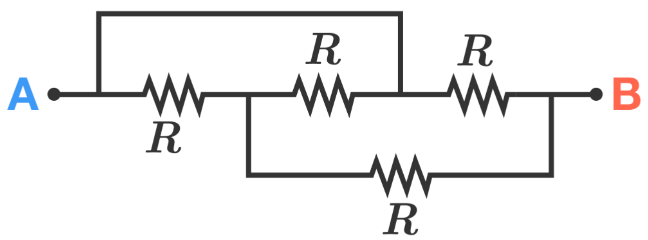



*Figure 1.4: Equivalent Resistance*

## 2.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

***Your image goes here***

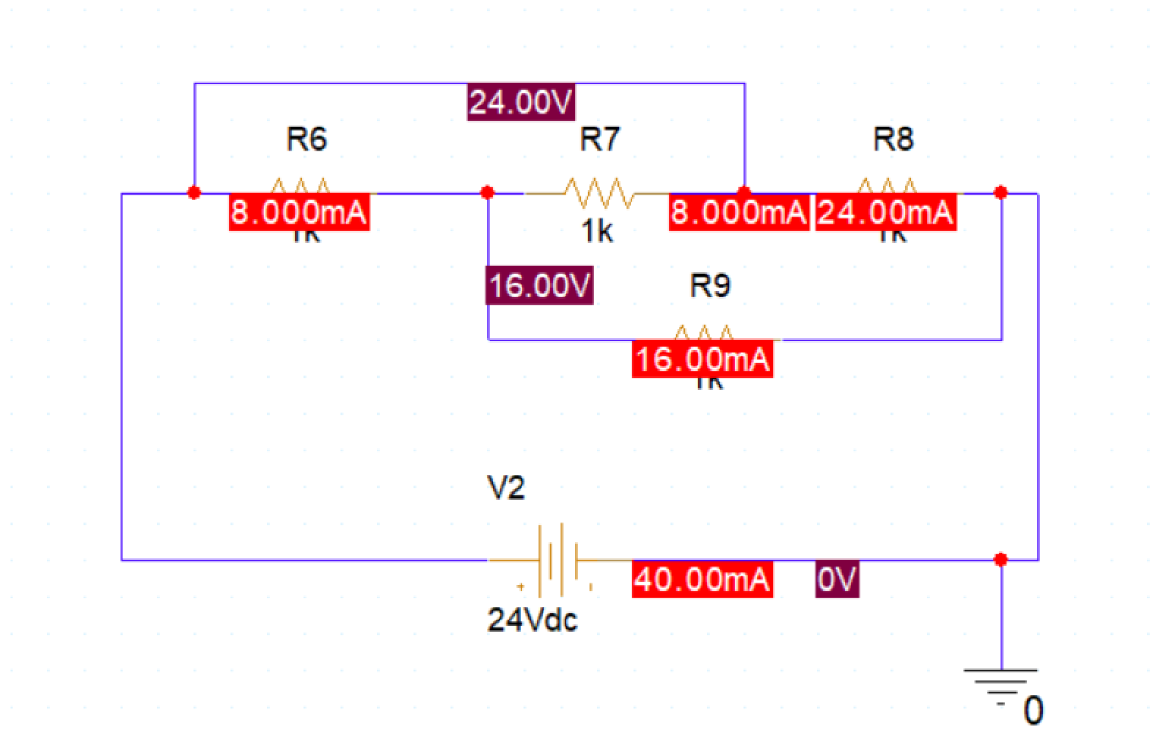



*Figure 1.5*: The bias point simulation of the circuit.

From the simulation results, what is the equivalent resistance of the circuit?

***Your answer goes here***

$$R_{eq} = \frac{V_{AB}}{I} = \frac{24}{40 \times 10^{-3}} = 600(\Omega)$$

## 2.2 Theory calculation

In order to confirm with the simulations above, your calculations are required to present in this part.

Rearranging the circuit gives us a better sight to recognize the connection of the resistors.

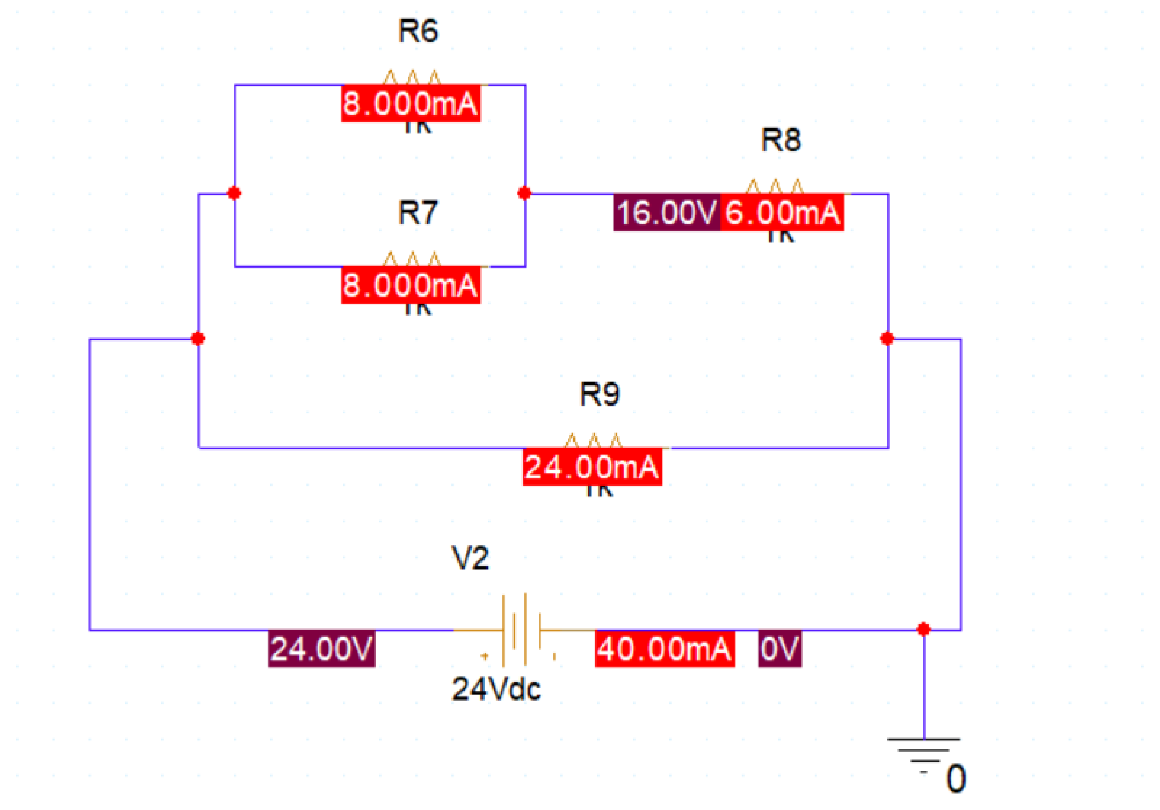



*Figure 1.6*: The rearranging circuit.

We have, the equivalent resistance of the upper branch is: $R_{eq1} = \frac{1 \times 1}{1+1} + 1 = 1.5(k\Omega)$

The equivalent resistance of the whole circuit is: $R_{eq} = \frac{1.5 \times 1}{1.5+1} = 0.6(k\Omega) = 600(\Omega)$

The result calculated by theory is exactly the same with those obtained by simulation.

# 3 Current calculation

Simulate this circuit in PSPICE, and find all the currents in the circuit. Hint: Delta-wye transform can be used to simplify the circuit.

## 3.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

***Your image goes here***

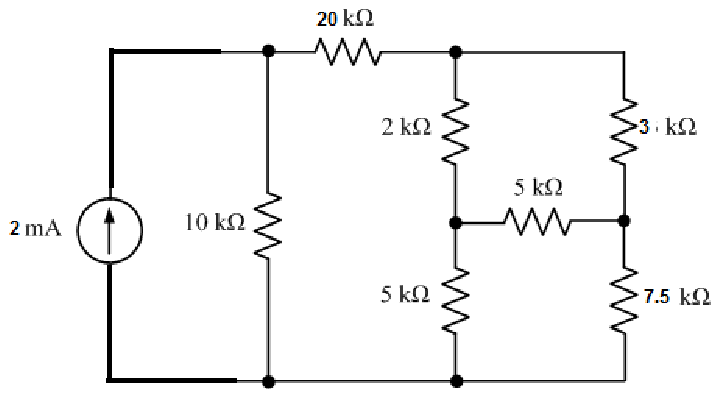



*Figure 1.7: Current calculations*

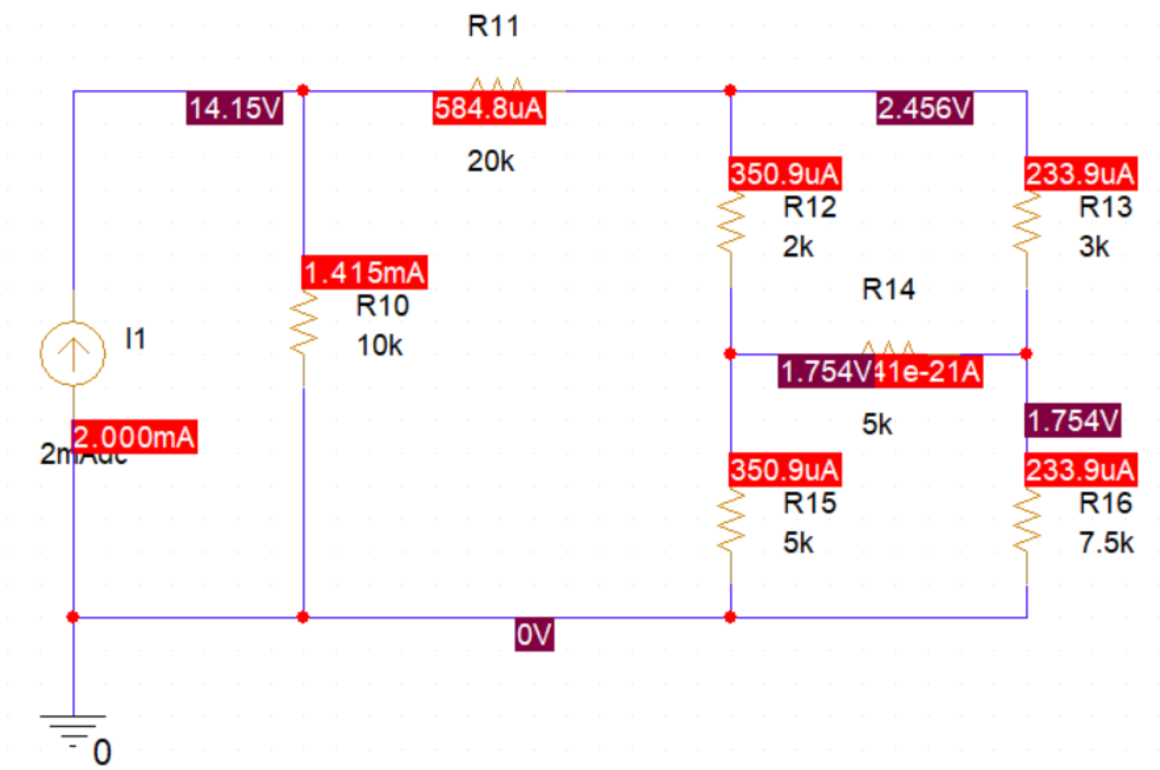



*Figure 1.8*: The bias point simulation of the circuit.

From the simulation results, what is the equivalent resistance of the whole circuit?

***Your answer goes here***

$$R_{eq} = \frac{V}{I} = \frac{14.15}{2 \times 10^{-3}} = 7075(\Omega)$$

### 3.2 Theory calculation

In order to confirm with the simulations above, your calculations are required to determine **all the current passing through resistors**.

We will divide the current at each node to find the value through each branch.

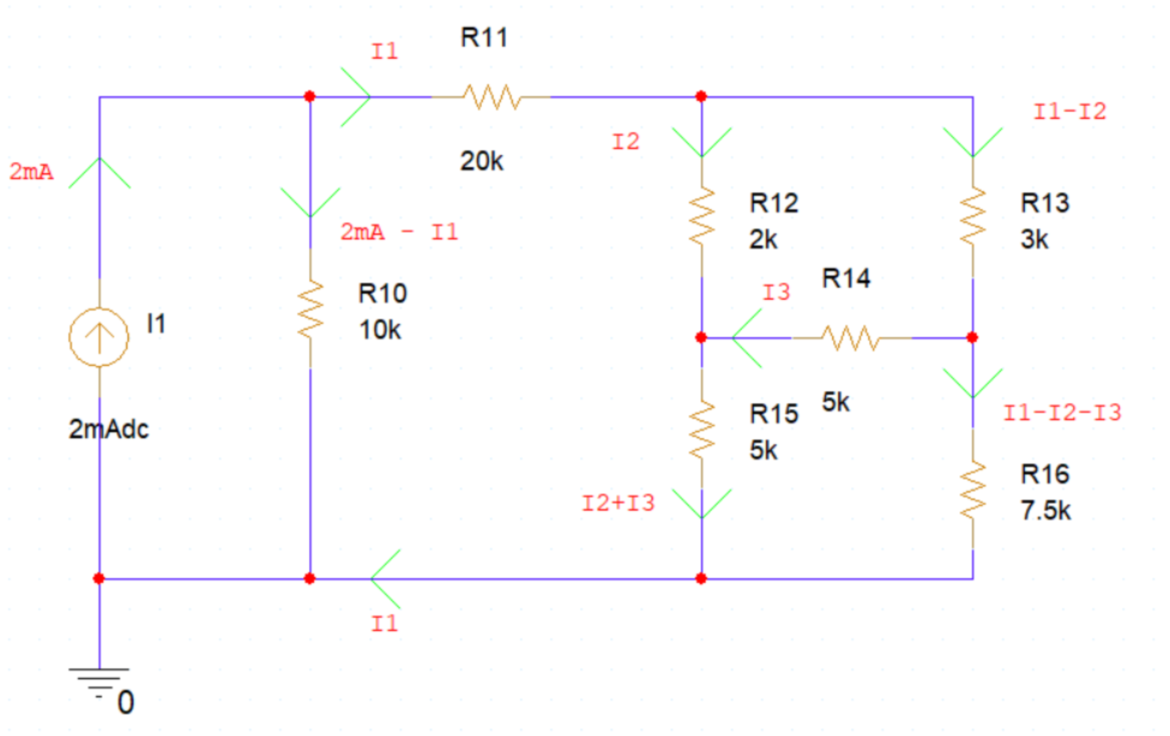



*Figure 1.9*: The current division.

According to the Kirchoff's Voltage Law, we can establish a system of 3 equations with 3 unknowns. Solving this system gives all the current passing through each resistor.

$$\begin{cases} 20000I_1 + 2000I_2 + 5000(I_2 + I_3) = 10000(2 \times 10^{-3} - I_1) \\ 3000(I_1 - I_2) + 5000I_3 = 2000I_2 \\ 7500(I_1 - I_2 - I_3) = 5000(I_2 + I_3) + 5000I_3 \end{cases}$$

=>

$$\begin{cases} 30000I_1 + 7000I_2 + 5000I_3 = 20 \\ 3000I_1 - 5000I_2 + 5000I_3 = 0 \\ 7500I_1 - 12500I_2 - 17500I_3 = 0 \end{cases}$$

=>

$$\begin{cases} I_1 = 584.79(\mu A) \\ I_2 = 350.87(\mu A) \\ I_3 = 0(A) \end{cases}$$

In order to confirm the value of $R_{eq}$, we can transform the lower right corner Delta connection loop into the Wye connection

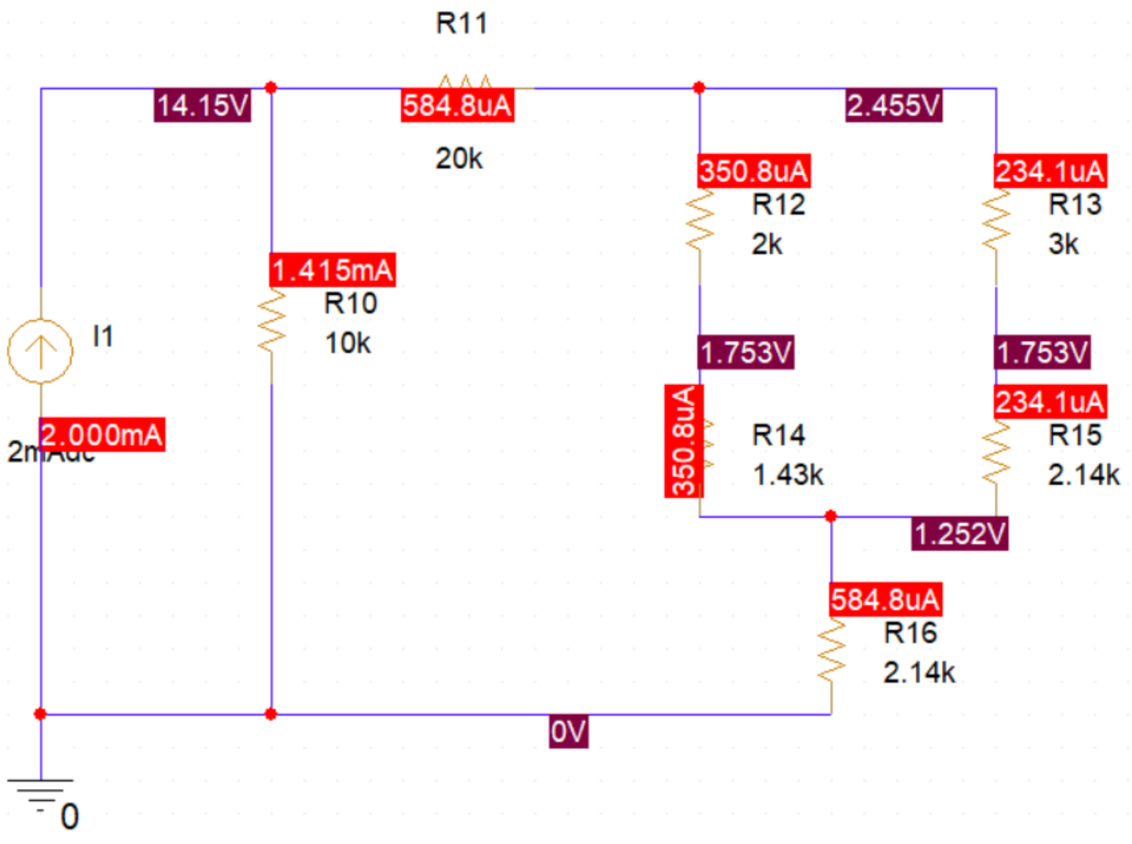



*Figure 1.10*: The Delta-Wye transformation.

The value of the three resistor in Wye connection are:

$$R_1 = \frac{5 \times 5}{5+5+7.5} = 1.43(k\Omega)$$

$$R_2 = \frac{5 \times 7.5}{5+5+7.5} = 2.14(k\Omega)$$

$$R_2 = \frac{5 \times 7.5}{5+5+7.5} = 2.14(k\Omega)$$

We have, $R_{eq1} = \dfrac{(2+1.43) \times (3+2.14)}{2+3+1.43+2.14} = 2.06(k\Omega)$

$$R_{eq2} = 20 + R_{eq1} + 2.14 = 24.2(k\Omega)$$

$$R_{eq} = \frac{10 \times R_{eq2}}{10 + R_{eq2}} = 7.076(k\Omega)$$

The values obtained by theory match with those obtained by simulation.

# 4 Diode exercise 1

Simulate this circuit in PSPICE to determine the current passing through the diode.

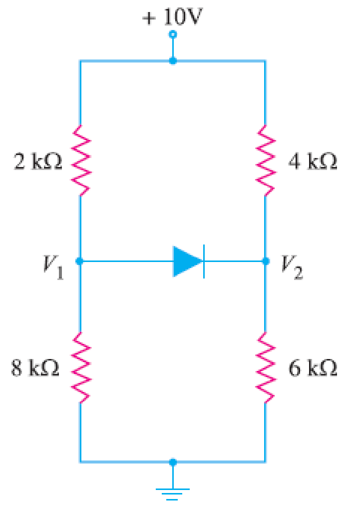



*Figure 1.11: Diode current*

## 4.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

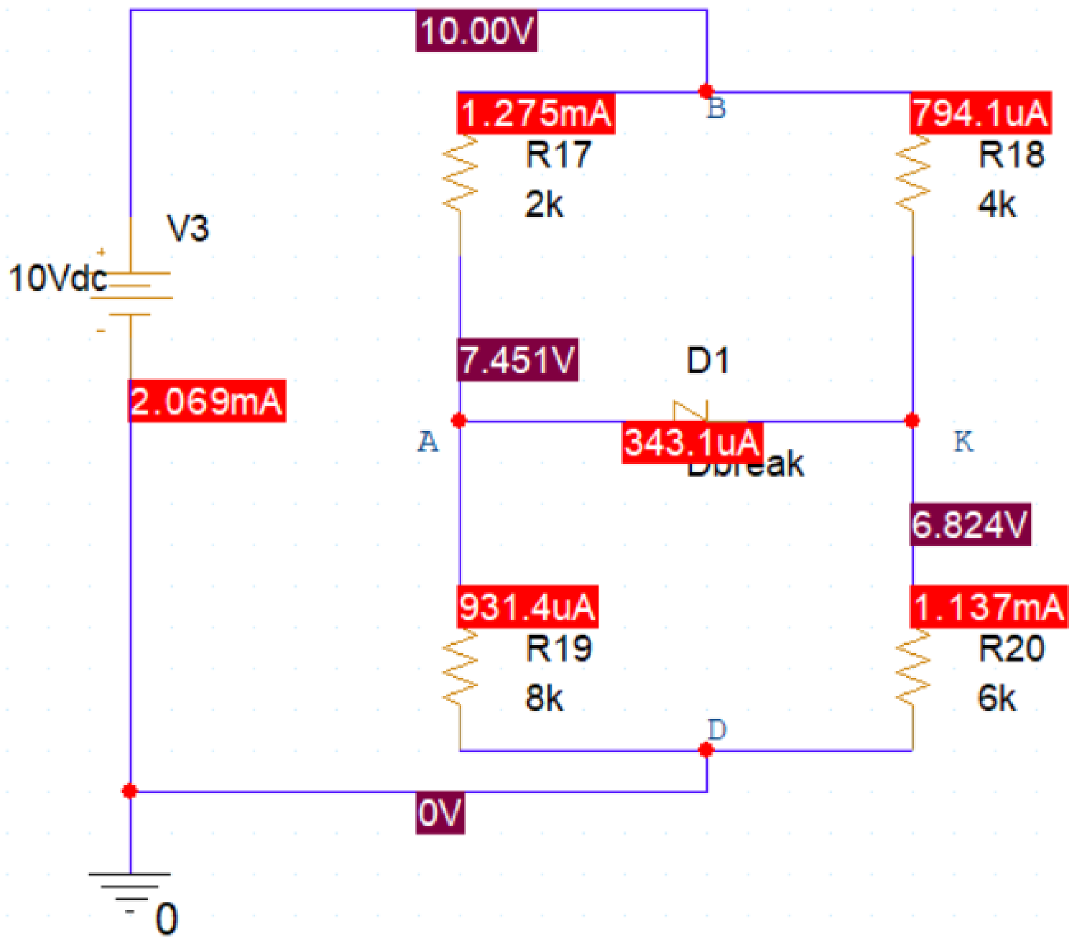



*Figure 1.12*: The bias point simulation.

## 4.2 Theory calculation

In order to confirm with the simulations above, your calculations are required to determine the current passing through the diode. Please use the practical diode ($V_F = 0.7V$) model to evaluate.

We will divide the current at each node to find it values through each branch.

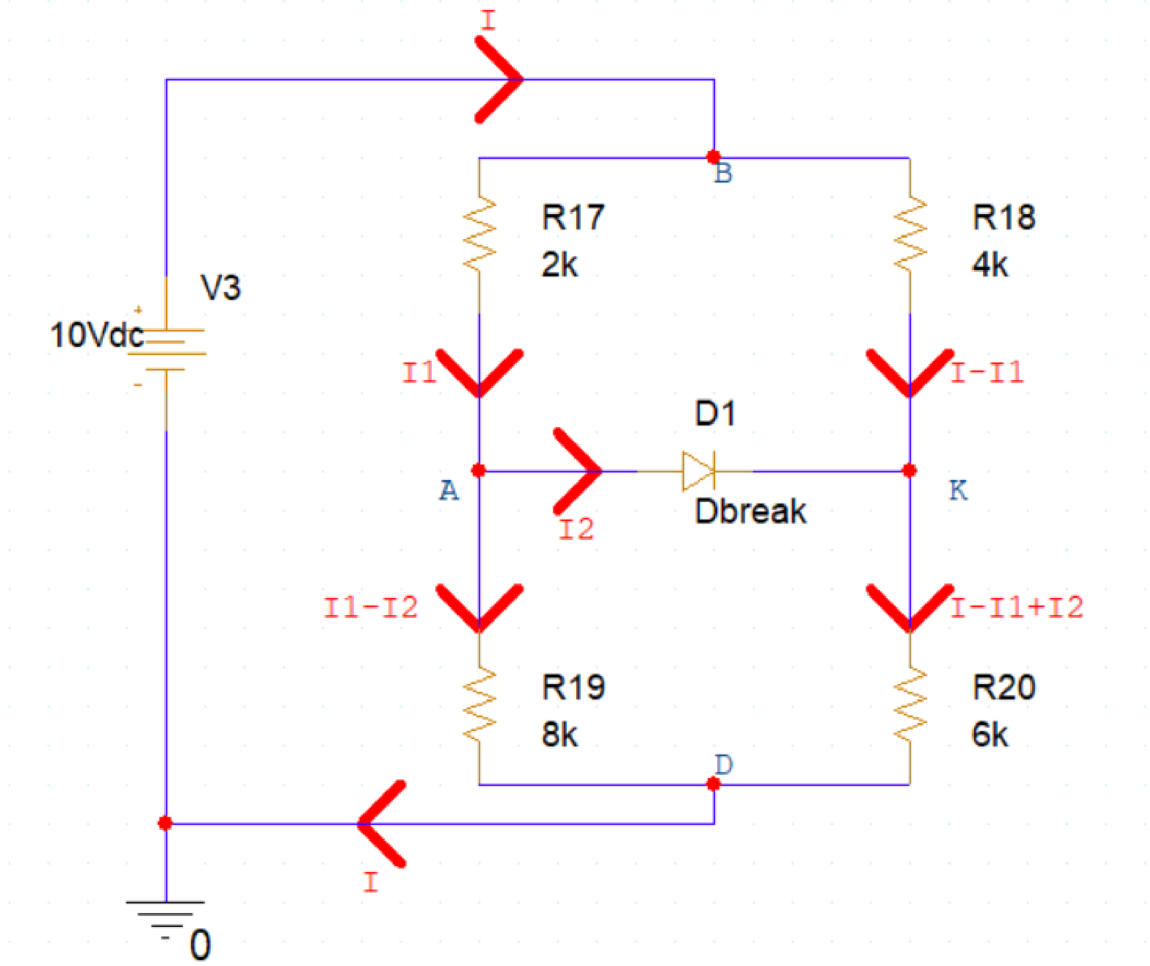



*Figure 1.13*: The current division.

We have the following system of 3 equations with 3 unknowns. Solving this system will give us the value of $I_2$ which is also the current through the diode.

$$\begin{cases} V_{AK} = V_{AB} + V_{BK} \\ V_{AK} = V_{AD} + V_{DK} \\ V_{BD} = V_{BK} + V_{KD} \end{cases}$$

=>

$$\begin{cases} 0.7 = -2000I_1 + 4000(I - I_1) \\ 0.7 = 8000(I_1 - I_2) - 6000(I - I_1 + I_2) \\ 10 = 4000(I - I_1) + 6000(I - I_1 + I_2) \end{cases}$$

=>

$$\begin{cases} I = 2.065(mA) \\ I_1 = 1.26(mA) \\ I_2 = 325(\mu A) \end{cases}$$

The result is quite close to simulation.

# 5 Diode exercise 2

Simulate this circuit in PSPICE to determine the current passing through the diode.

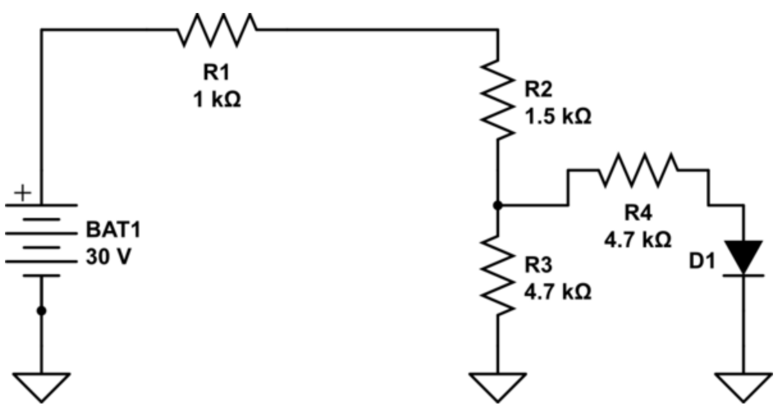



*Figure 1.14: Diode current*

## 5.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

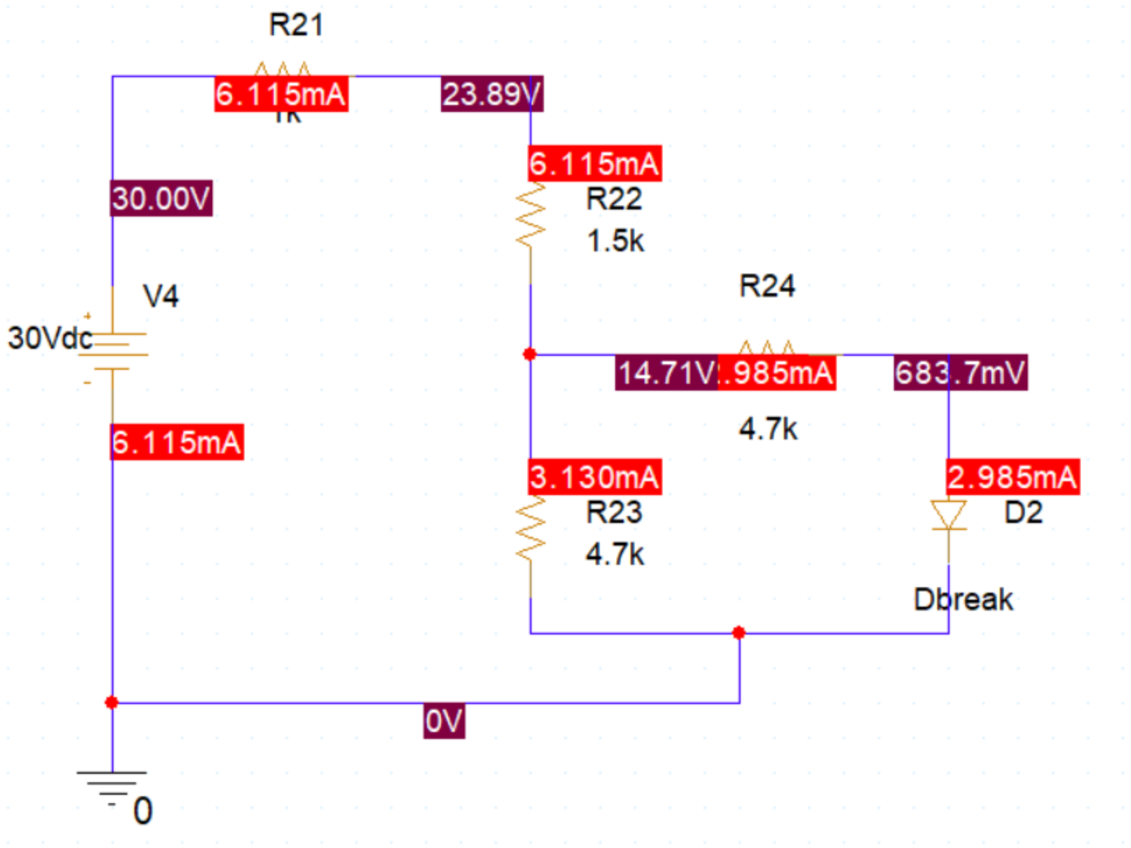



*Figure 1.15*: The bias point simulation.

## 5.2 Theory calculation

In order to confirm with the simulations above, your calculations are required to determine the current passing through the diode. Please use the practical diode ($V_F = 0.7V$) model to evaluate.

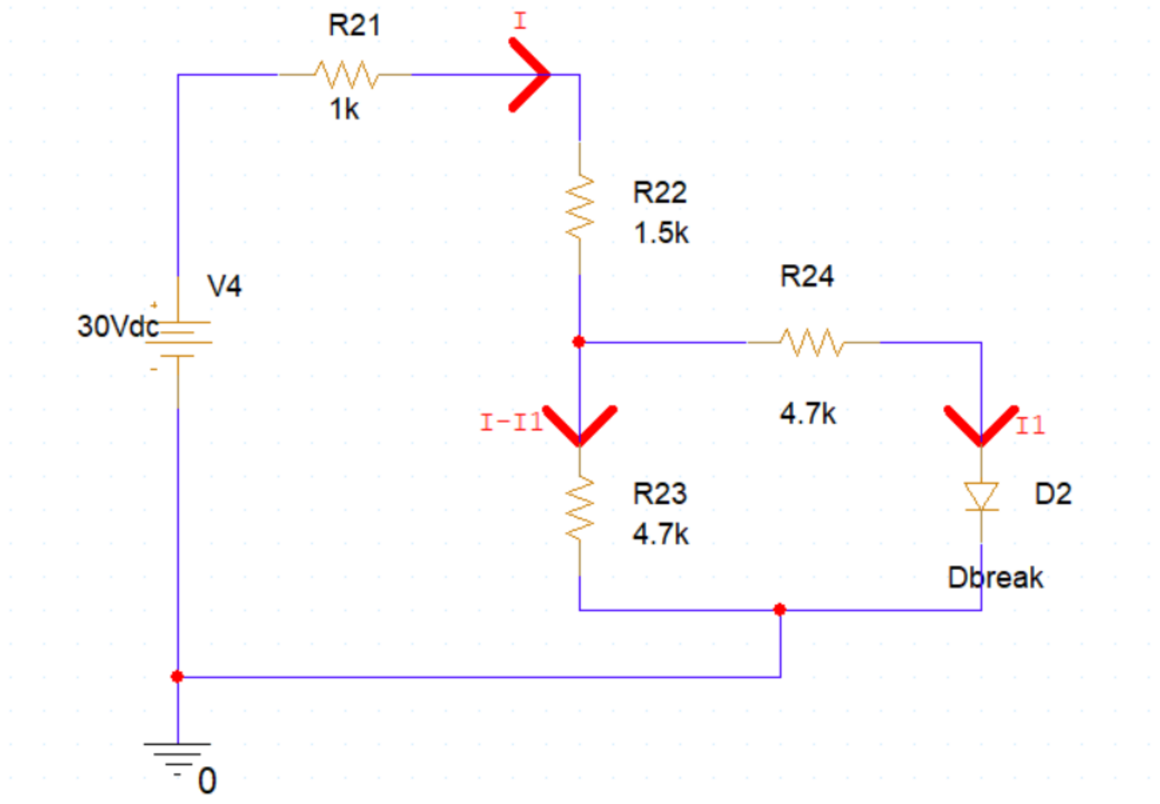



*Figure 1.16*: The current division.

According to the Kirchoff's Voltage Law, we have the following system of 2 equations:

$$\begin{cases} 30 = 1000I + 1500I + 4700(I - I_1) \\ 4700I_1 + 0.7 = 4700(I - I_1) \end{cases}$$

=>

$$\begin{cases} 7200I - 4700I_1 = 30 \\ 4700I - 9400I_1 = 0.7 \end{cases}$$

=>

$$\begin{cases} I = 6.113(mA) \\ I_1 = 2.982(mA) \end{cases}$$

The values are exactly the same with the simulation.

# 6 Zener Diode

In the circuit shown bellow,the voltage across the load is to be maintained at 12 V. Simulate this circuit with two different values of $R_L$, including 1k and 10k

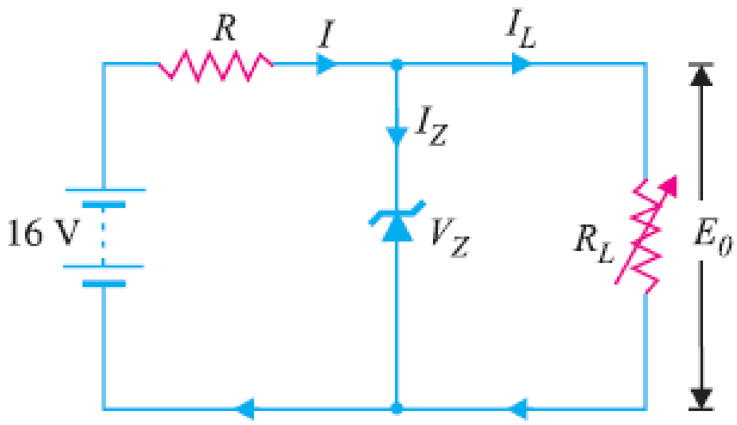



*Figure 1.17*: *Diode current*

## 6.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

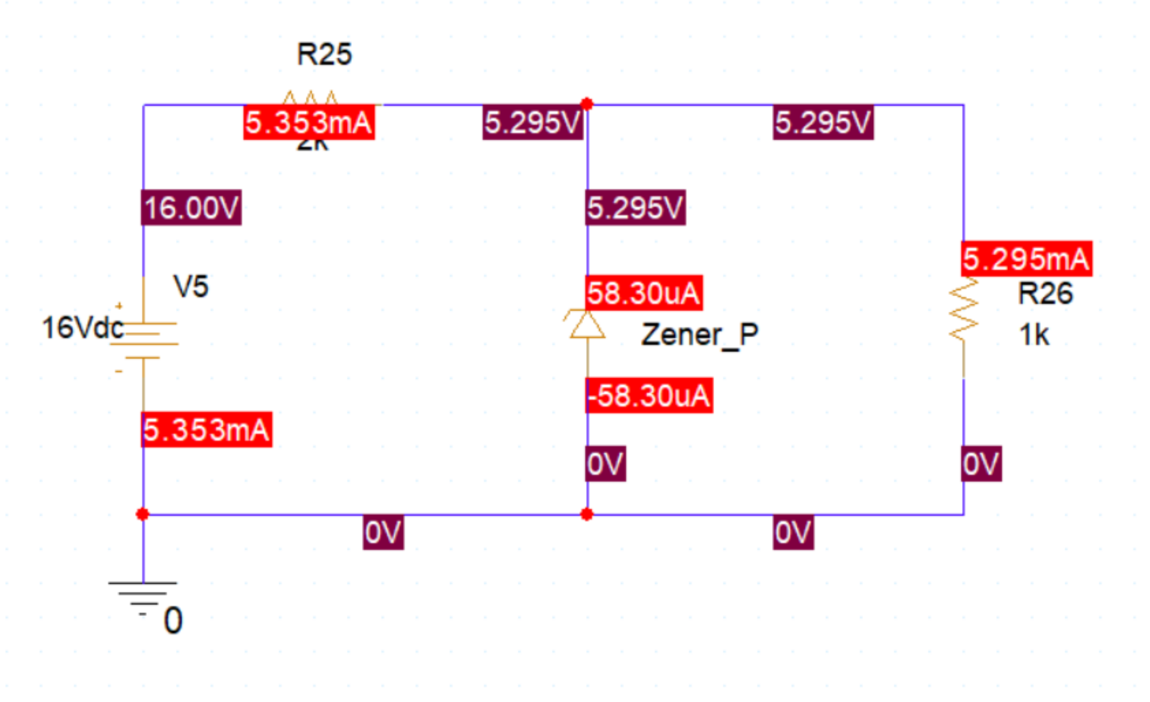



*Figure 1.18*: The bias point simulation when $R_L = 1k\Omega$.

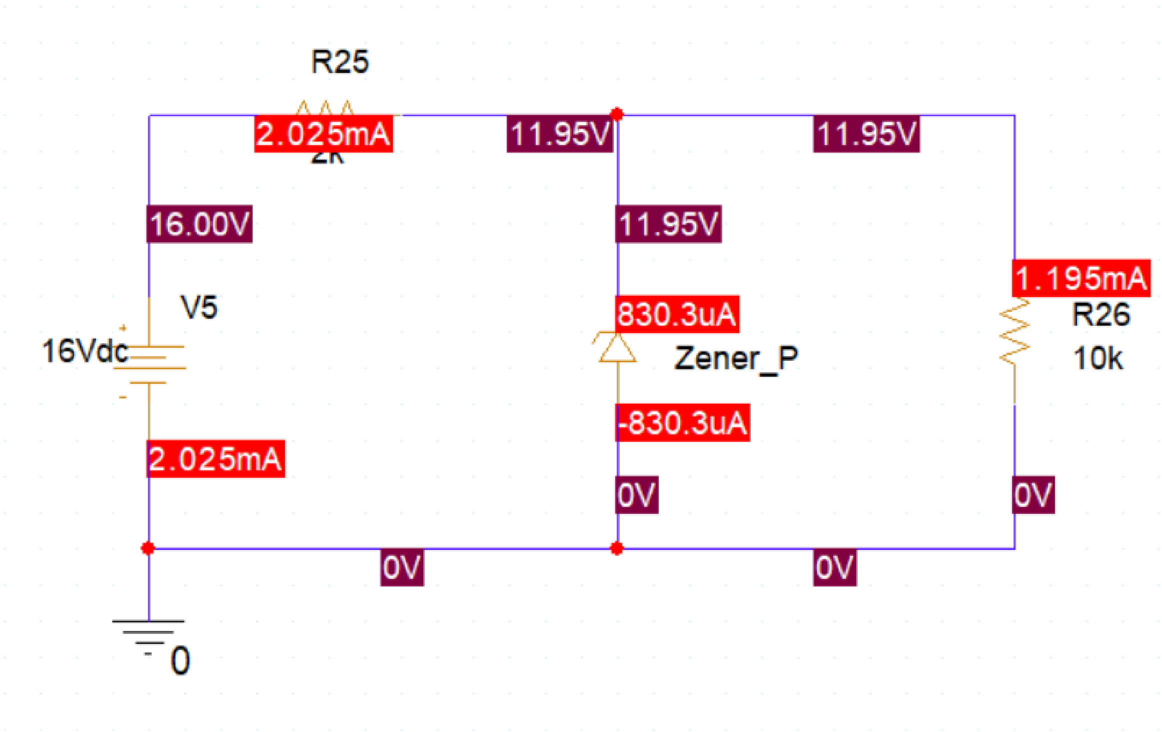



*Figure 1.19*: The bias point simulation when $R_L = 10k\Omega$.

## 6.2 Theory calculation

Only $R_L = 10k$ is required to calculate in this part, to confirm the current passing through the diode with your simulation results.

$$I_L = \frac{12}{R_L} = \frac{12}{10000} = 1.2(mA)$$
$$I = \frac{16-12}{R} = \frac{16-12}{2000} = 2(mA)$$
$$I_Z = I - I_L = 2 - 1.2 = 0.8(mA)$$

The result is approximate with those obtained by simulation.

# 7 BJT simulation circuit

Implement the following circuit in PSPICE. The component used in this circuit is **QBreakN NPN**, which can be found in the Favorites list. The default transistor gain is $\beta = 100$, and the saturated voltage $V_{CE(Sat)} = 0.65V$

## 7.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

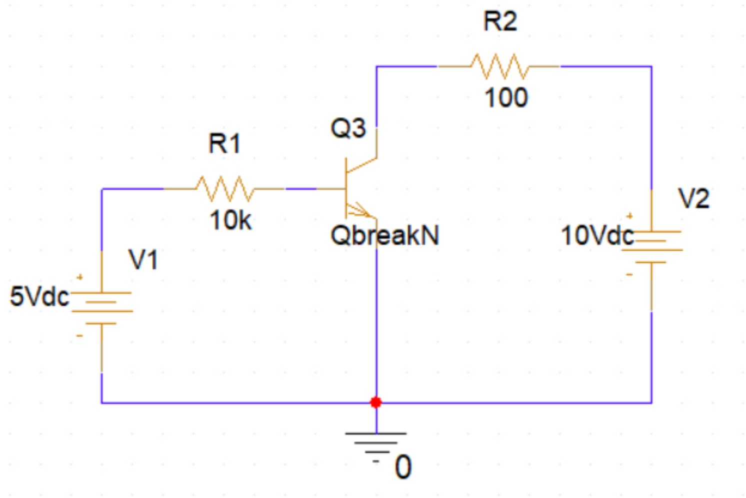



*Figure 1.20*: *BJT simulation*

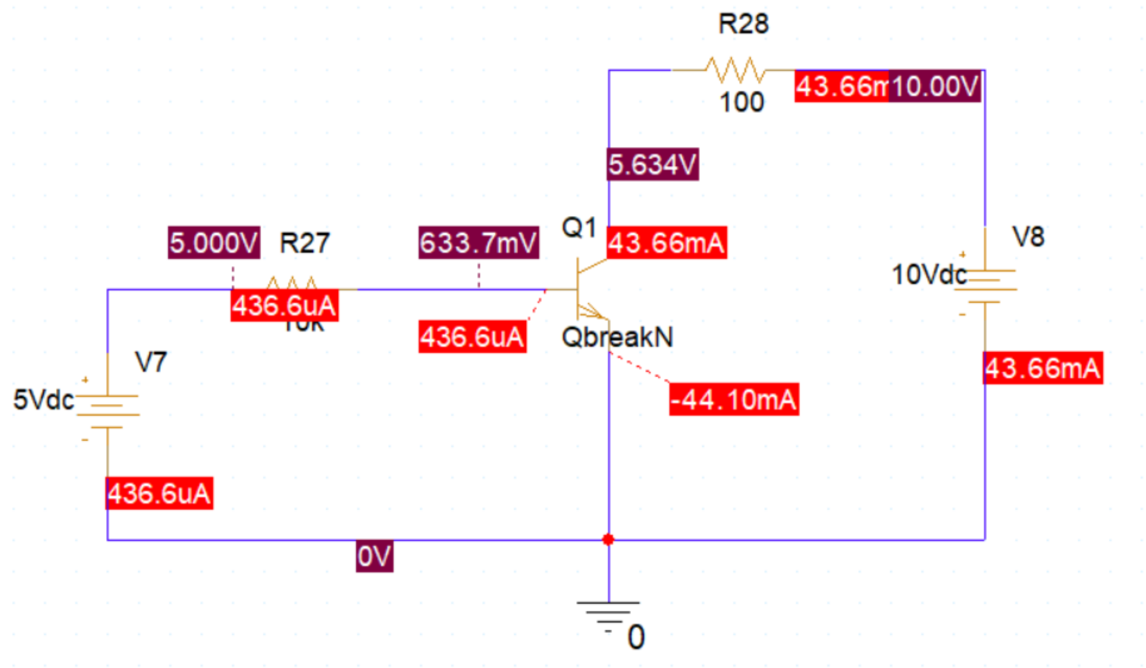



*Figure 1.21*: The bias point simulation.

## 7.2 Theory calculation

Determine $I_B$, $I_C$ and $I_E$ in the circuit by your calculation.

Assume that the transistor is working in linear mode.

We have, $I_B = \dfrac{V_1 - 0.7}{R_1} = \dfrac{5 - 0.7}{10000} = 0.43(mA)$

$I_C = \beta \times I_B = 100 \times 0.43 \times 10^{-3} = 43(mA)$

$I_E = I_B + I_C = 43.43(mA)$

$V_{CE} = V_C - 0 = V_2 - I_C \times R_2 = 10 - 43 \times 10^{-3} \times 100 = 5.7(V)$

Since $V_{CE} = 5.7(V) > 0$, our assumption is correct.

## 7.3 DC Sweep simulation

Run the simulation again with DC Sweep mode. The source V1 is start from 2V to 10V, with step size is 0.1V. Present the simulation results to show $V_C$ in the Y axis and $V_1$ in the X axis.
***Your image goes here***

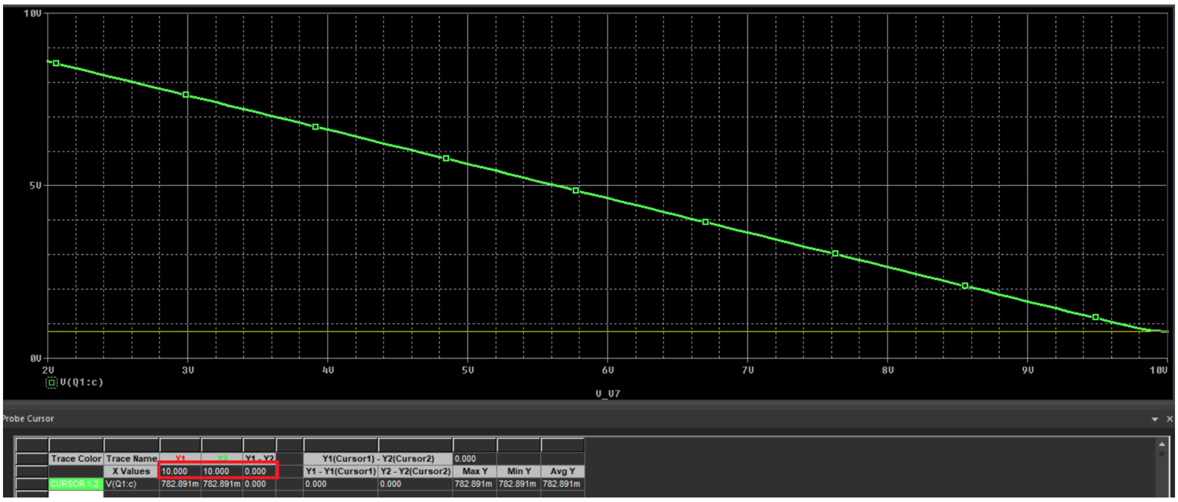



*Figure 1.22*: The DC Sweep mode.

**From the simulation, it can be estimated that when $V_1$ =??, the transistor is in saturation mode**.

Using the function cursor min on the DC Sweep mode, we can estimate that when $V_1$ = $10(V)$, the transistor is in saturation mode.

This is an obvious property since the increase of $V_1$ leads to the rise of $I_B$ and because $I_C$ is a hundred time larger than $I_B$, it will rise significantly. This make $V_{CE}$ decrease dramatically. But when it come to a point where $V_2$ can no longer amplify $I_C$ then the BJT will be in saturation mode and $V_{CE}$ will be kept at a fixed level.

## 7.4 Theory calculation

Your calculations are required here to confirm the value of V1, which is the point that the BJT starts saturation.

$$I_{C(sat)} = \frac{V_2 - V_{CE(sat)}}{R_2} = \frac{10 - 0.65}{100} = 93.5(mA)$$

The value $I_{B(}min)$ for the transistor still working in linear mode is equal to $\frac{I_{C(sat)}}{\beta} = \frac{93.5 \times 10^{-3}}{100} = 0.935(mA)$

So whenever $I_B > I_{B(}min)$ the BJT starts saturation.

This condition is equivalent to: $\frac{V_1 - 0.7}{10000} > 0.935 \times 10^{-3}$

=> $V_1 > 10000 \times 0.935 \times 10^{-3} + 0.7 = 10.05(V)$

The results is quite close to the values in simulaiton.

# 8 BJT and Zener

Implement the circuit bellow in PSPICE. The Zener diode has a regulated voltage at 1.7V.

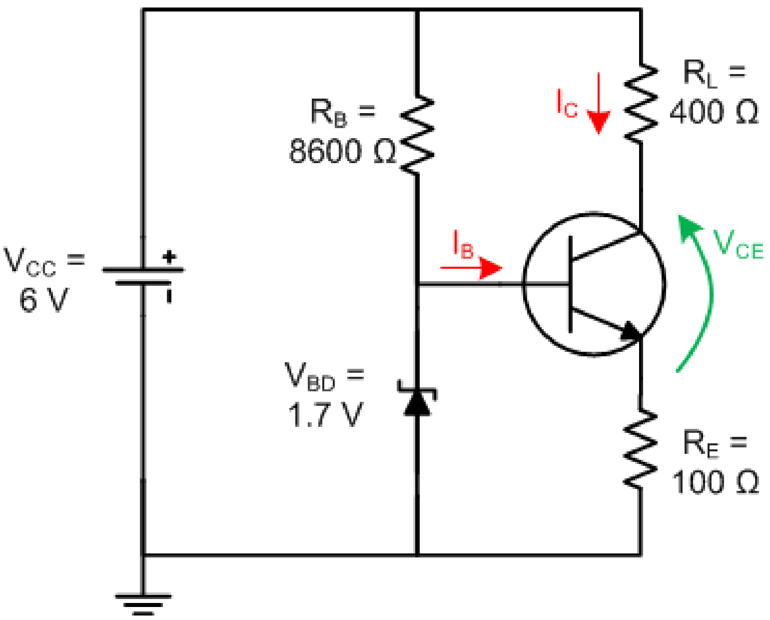



*Figure 1.23: BJT and Zener*

## 8.1 PSPICE Simulation

Run the bias simulation in PSPICE and capture the screen having all the components and the values of voltage and current. Place the picture in this part of the report.

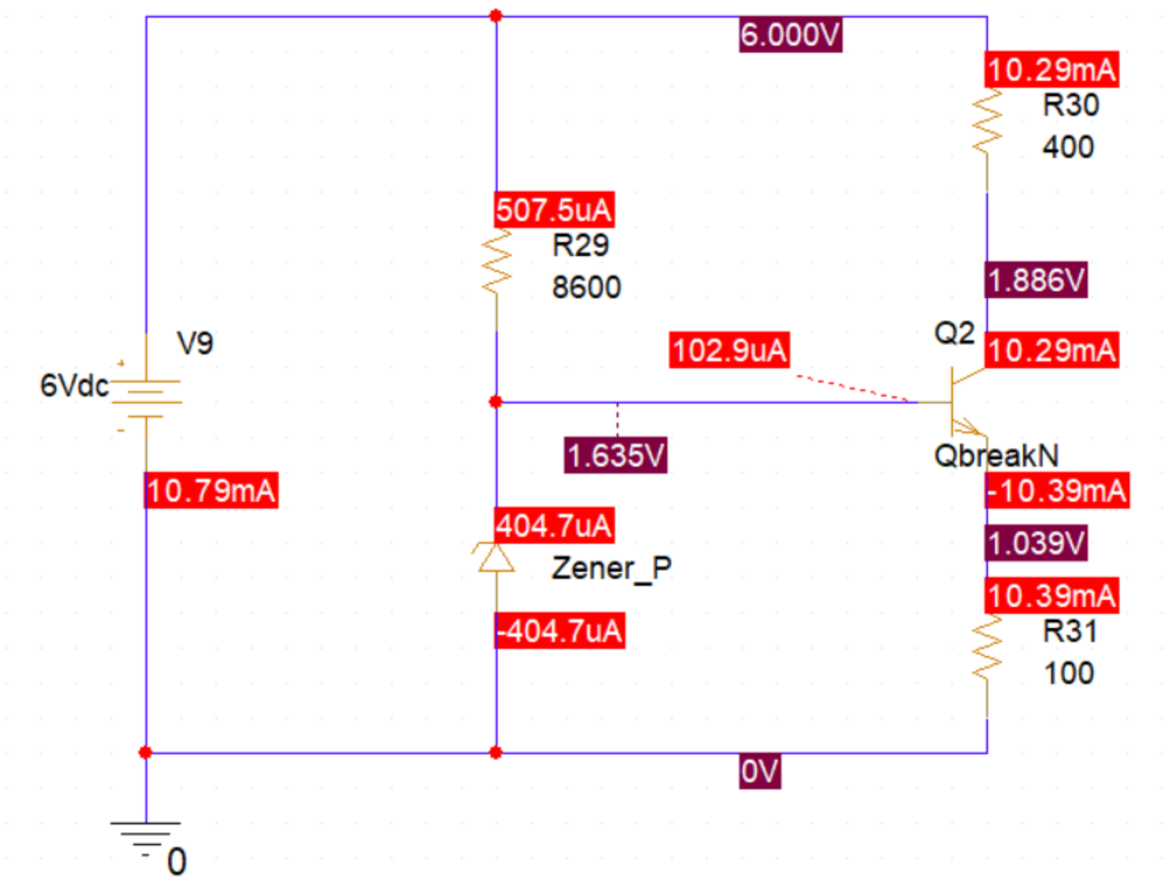



*Figure 1.24*: The bias point simulation.

## 8.2 Theory calculation

Your calculations are required here to confirm the value of $I_B$, $I_C$ and $I_E$.

The Zener diode will maintain a fixed voltage of 1.7V between anode and cathode. So we have, $V_B = 1.7(V)$

$$V_{BE} = V_B - V_E \Rightarrow V_E = V_B - V_{BE} = 1.7 - 0.7 = 1(V)$$

$$I_E = \frac{V_E}{R_E} = \frac{1}{100} = 10(mA)$$

$$I_C = \alpha.I_E = \frac{\beta}{\beta+1}.I_E = \frac{100}{101} \times 10 \times 10^{-3} = 9.9(mA)$$

$$I_B = I_E - I_C = 10 \times 10^{-3} - 9.9 \times 10^{-3} = 100(\mu A)$$

The values calculated by the theory are close enough to that obtained by simulation.

# ***THE END***