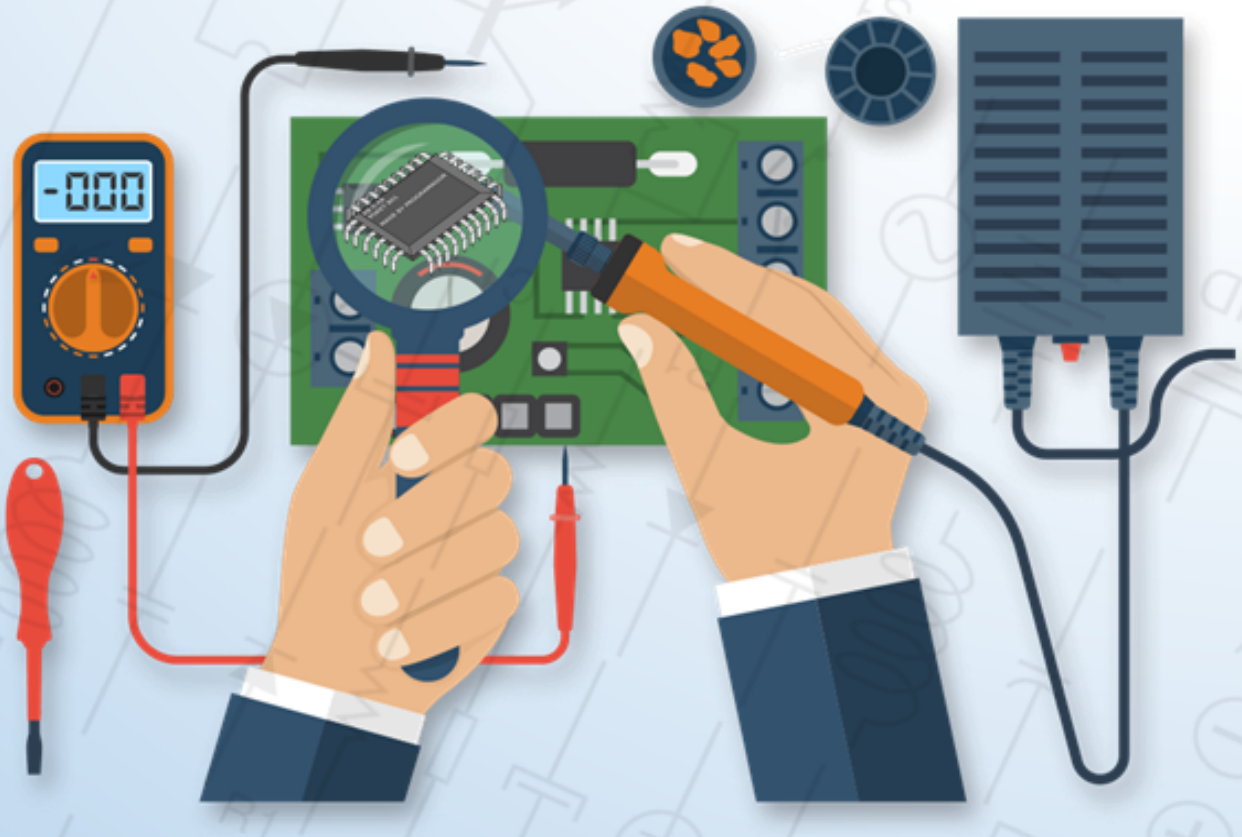
HO CHI MINH CITY UNIVERSITY OF TECHNOLOGY
COMPUTER ENGINEERING

Electronic Device Component



Dr. Le Trong Nhan

Contents

Chapter 1. 24 Hour Project for Midterm Exam **5**
1 Equivalent Resistance 1 . . . 6
1.1 PSPICE Simulation . . . 6
1.2 Theory calculation . . . 6
2 Equivalent Resistance 2 . . . 6
2.1 PSPICE Simulation . . . 7
2.2 Theory calculation . . . 7
3 Current calculation . . . 7
3.1 PSPICE Simulation . . . 7
3.2 Theory calculation . . . 7
4 Diode exercise 1 . . . 8
4.1 PSPICE Simulation . . . 8
4.2 Theory calculation . . . 8
5 Diode exercise 2 . . . 8
5.1 PSPICE Simulation . . . 8
5.2 Theory calculation . . . 8
6 Zener Diode . . . 9
6.1 PSPICE Simulation . . . 9
6.2 Theory calculation . . . 9
7 BJT simulation circuit . . . 9
7.1 PSPICE Simulation . . . 10
7.2 Theory calculation . . . 10
7.3 DC Sweep simulation . . . 10
7.4 Theory calculation . . . 10
8 BJT and Zener . . . 10
8.1 PSPICE Simulation . . . 11
8.2 Theory calculation . . . 11

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**.

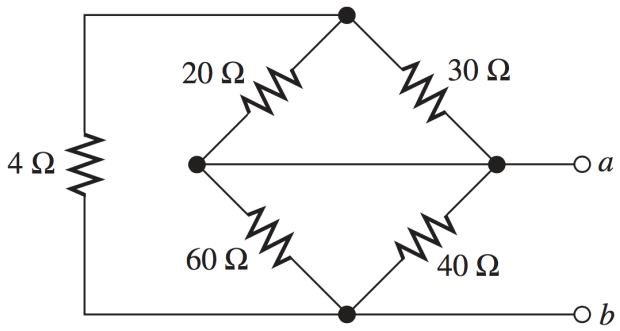


Figure 1.1: Equivalent Resistance

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

Your answer goes here

1.2 Theory calculation

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

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)**.

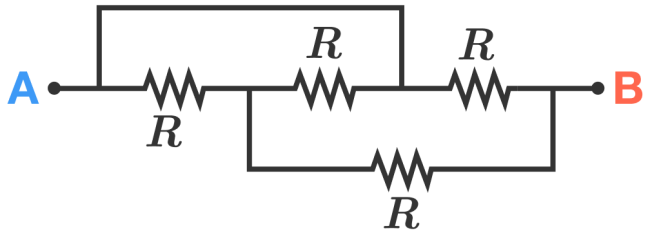


Figure 1.2: Equivalent Resistance

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

Your answer goes here

2.2 Theory calculation

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

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.

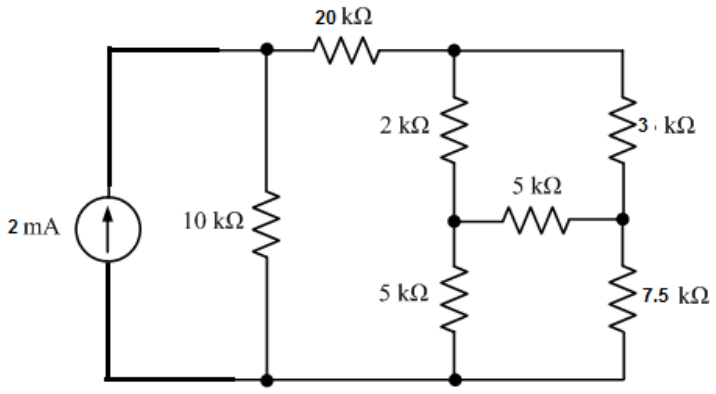


Figure 1.3: Current calculations

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

Your answer goes here

3.2 Theory calculation

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

4 Diode exercise 1

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

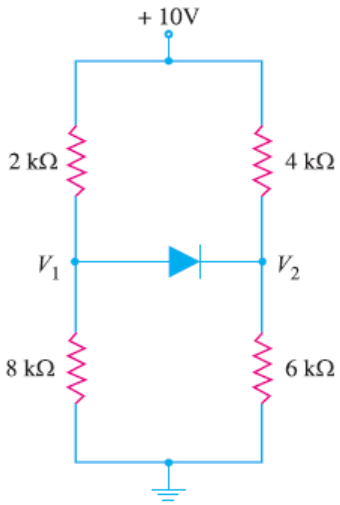


Figure 1.4: Diode current

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.

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.

5 Diode exercise 2

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

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.

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.

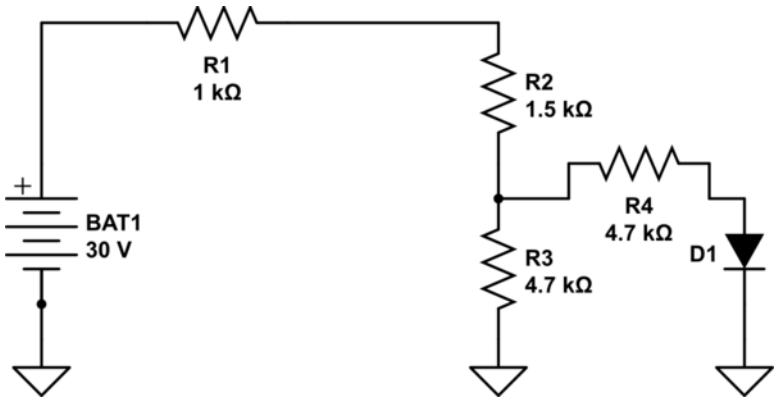


Figure 1.5: Diode current

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

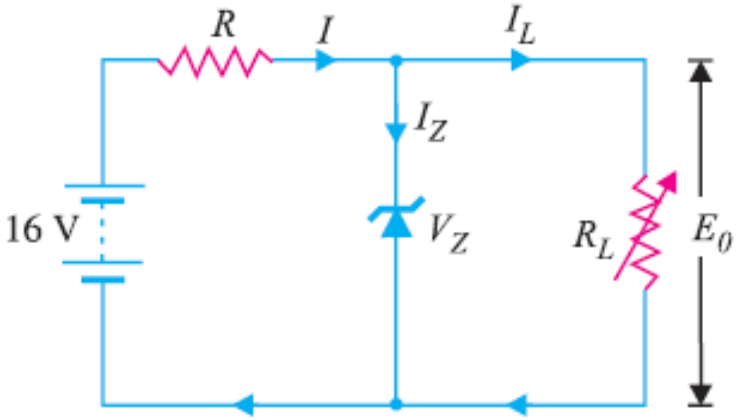


Figure 1.6: Diode current

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.

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.

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 $V_{CE(Sat)} = 0.65V$

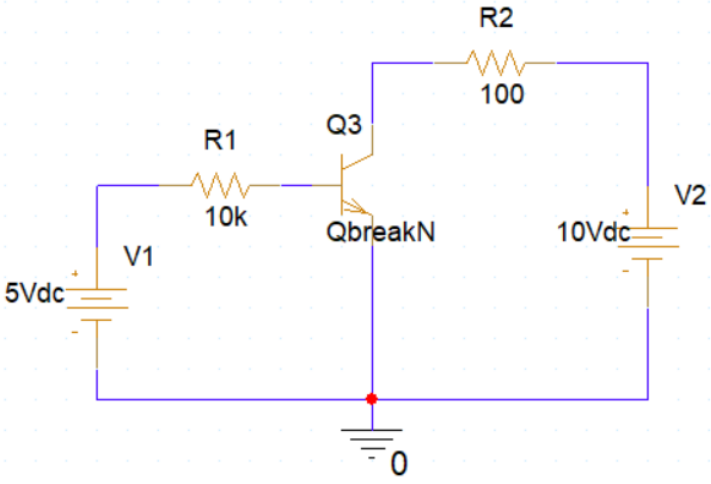


Figure 1.7: BJT simulation

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.

7.2 Theory calculation

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

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

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

7.4 Theory calculation

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

8 BJT and Zener

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

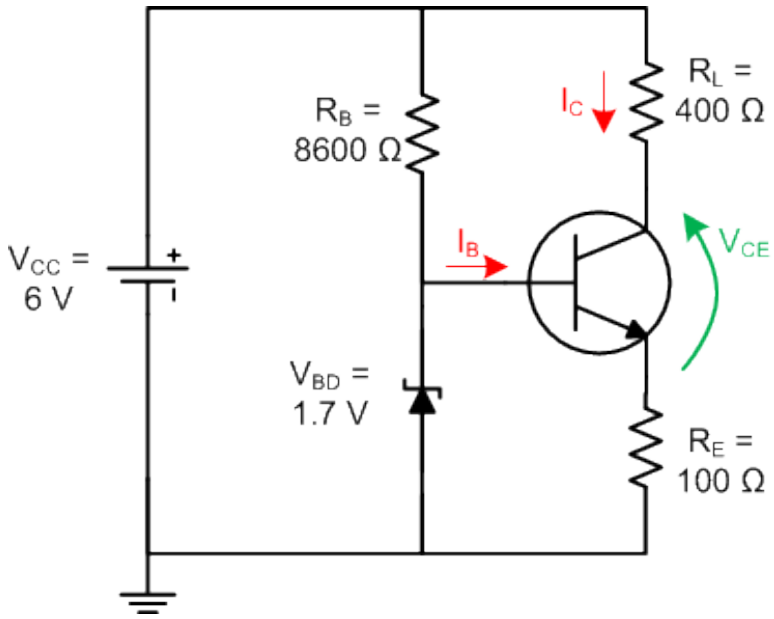


Figure 1.8: BJT and Zener

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.

8.2 Theory calculation

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