

***College Of Engineering and Computer Science***

*Electrical Engineering Department*

**EE3310 Device and Circuits**

**Diode Properties and Application**

**Lab’s number 07**

Ravi Patel

UID: 00762945

Bilal Addulhamed

12-4-20

*“I have neither given nor received aid on this assignment, nor have I observed any violation of the Honor code”*

Fall 2020

Name : Ravi Patel Lab Instructor: \_\_\_Bilal Abdulhamed

Date Performed: 9-20-20 Date Due: 09-25-20

# **Purpose**

The purpose of this laboratory is to learn the operation of diodes in various applications.

## **Full wave rectification circuit:**

**Theoretical Prediction:** Consider the following full wave rectification circuit, the AC voltage source is 10cos(1600πt), the load resistor R1=200Ω, and D1 through D4 are silicon diodes (1N4001). You may use Multisim or Pspice to design and simulate the circuit.

Diagram, schematic

Description automatically generated

1. Find the VPeak of the output DC voltage VO and sketch the output wave form with respect to the input AC wave form.
2. Find the average DC voltage of VO.
3. Find the maximum forward biased (peak surge) current following through the diode D1, D2, D3, and D4.
4. Find the peak reverse voltage across the diode D1, D2, D3, and D4.
5. Find the average DC load current following through R1. What is the power dissipated by R1?
6. Ans to the First five Diagram

   Description automatically generated

**Simulation using Multisim:** Construct the full wave rectification circuit in Multisim as follows:

Diagram

Description automatically generated

* Place an AC voltage source as input using “source place” tab.
* Tab “simulation” and select “instrument” from the expanded list. You can find various instruments. Find the “oscilloscope” and place it on to the work area.

A picture containing application

Description automatically generated

* You can also introduce a differential voltage probe (V+ at R1’s positive terminal and V− at the R1’z negative terminal) and a current probe over the load resistor R1 to measure the rectified DC voltage and current.
* Graphical user interface

  Description automatically generatedYou can now set the AC signal generator to generate the AC source. After “Run” and “Stop” the simulation. You can examine the wave form recorded by sliding the time axis. There are two sliding cursors can be used to provide measurements at any given time location.
* Complete the following data table with proper units to compare the simulation results with your theoretical predictions.

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
|  | Vpeak | Ipeak | VDC | IDC |
| Prediction | 8.6 | 43 mA | 6.37 V | 31.85 mA |
| Simulation | 7.06 V | 5.66 mA | 3.06 V | 2.88 mA |
|  |  |  |  |  |

Section 1.1 circuit built on multisim Live online simulator

Diagram, schematic

Description automatically generated

Section 1.2: Simulator results

Chart, line chart

Description automatically generated

**Conclusion :**

It was bit easy compare to last lab. I don’t have a Windows computer at home so I had to use live multisim software to perform this lab. Photos are shown in the *section 1.1* and the results are shown in the *section 1.2*