East West University

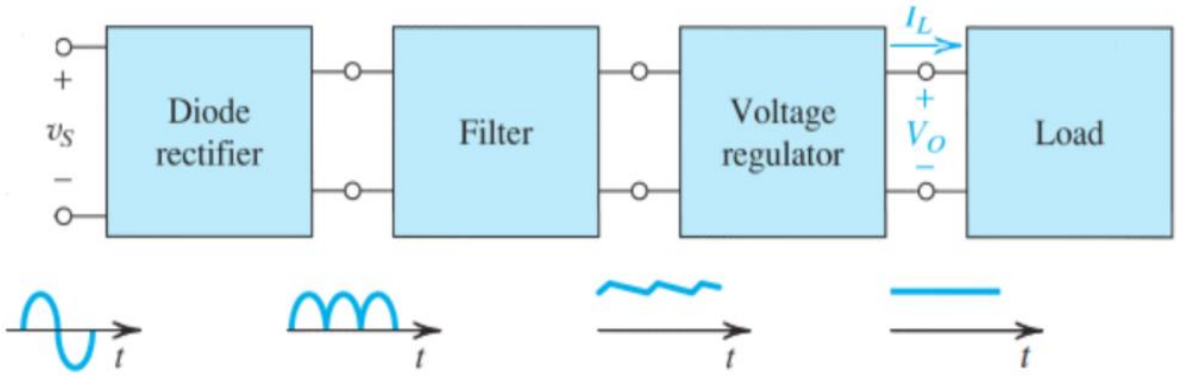
Department of CSE

PROJECT

Course Code and Name: CSE251 Electronic Circuit	
Project no: 02	
Topic: Design a 5V DC Power Supply using Diode for a specified input.	
Semester and Year: Spring 2022	
Name of Student: Iffat Tasnim (2020-2-60-004)	**Course Instructor information:** M Saddam Hossain Khan Senior Lecturer Department of Computer Science and Engineering East West University
Date of Project Submitted: 19-05-2022	

Design a 5V DC Power Supply using Diode for a specified input.

Problem Statement:



The figure shows the block diagram of a dc power supply design process. The design process includes three design segments: a diode rectifier, a filter and a voltage regulator to get the final output V_0.

- The diode rectifier converts the input sinusoid V_S to a unipolar output, which can have the pulsating waveform indicated in figure.
- The variations in the magnitude of the rectifier output are considerably reduced by the filter block.
- The output of the rectifier filter contains a time-dependent component, known as ripple. To reduce the ripple and to stabilize the magnitude of the dc output voltage against variations caused by changes in load current, a Zener shunt voltage regulator can be implemented.

We will design the circuit components, and finally simulate to test the circuit. Then we will use sine wave (24Vp-p) as input signal, and capacitor, resistors and Zener diode of suitable value for the design.

Design Details:

Step 1: Identifying and designing of Voltage source

To get the sine wave as asked in the question we are using Transient sine voltage source or VSIN as input voltage. In this voltage we are giving 24V p-p which means 12V amplitude and 1KHz frequency which give us a smooth sine wave.

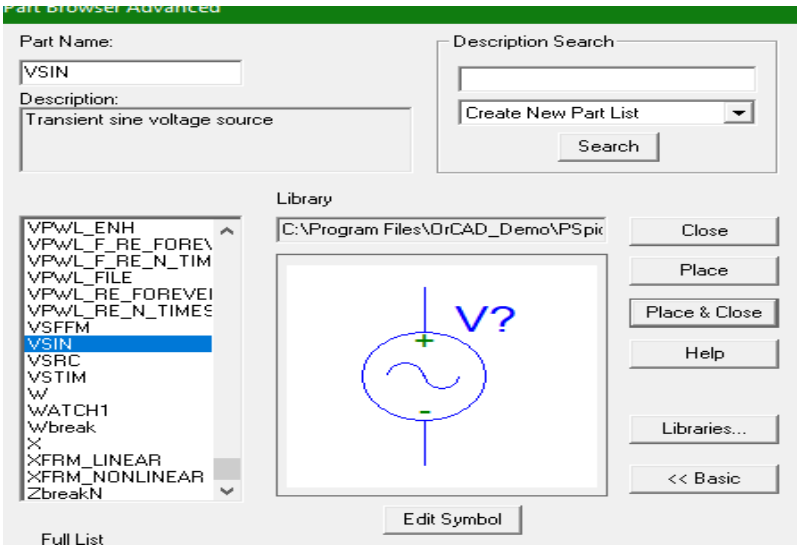


Figure: Voltage Source -VSIN

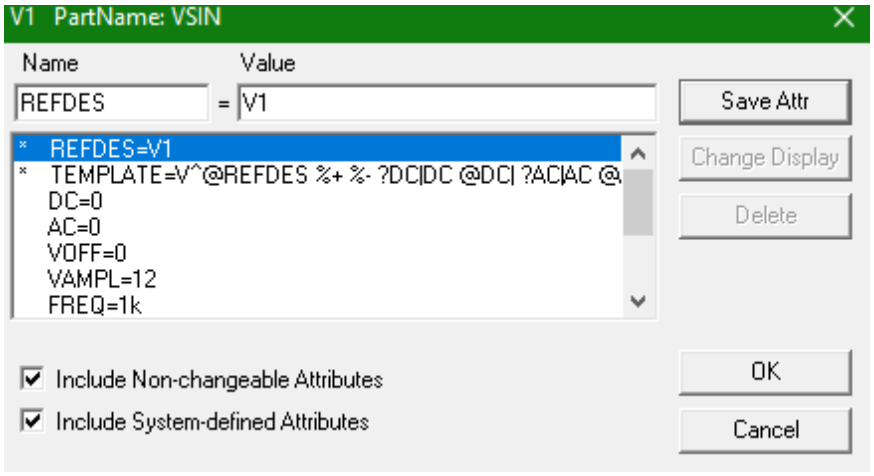


Figure: Values of VSIN

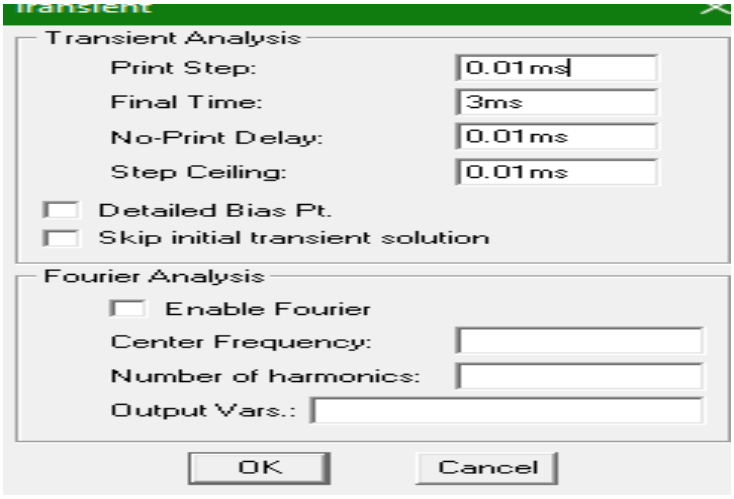


Figure: Transent for wave shapes

Here,

- Amplitude: 12 V
- Frequency: 1KHz
- AC: 0
- DC: 0
- Phase: 0
- Load resistor :100Ω

Step 2: Identifying and designing of diode rectifier

To rectify the input sine wave, we have used bridge rectifiers as full wave rectifiers. To design the rectifier, we have used four D1N4002 as diodes in the simulation.

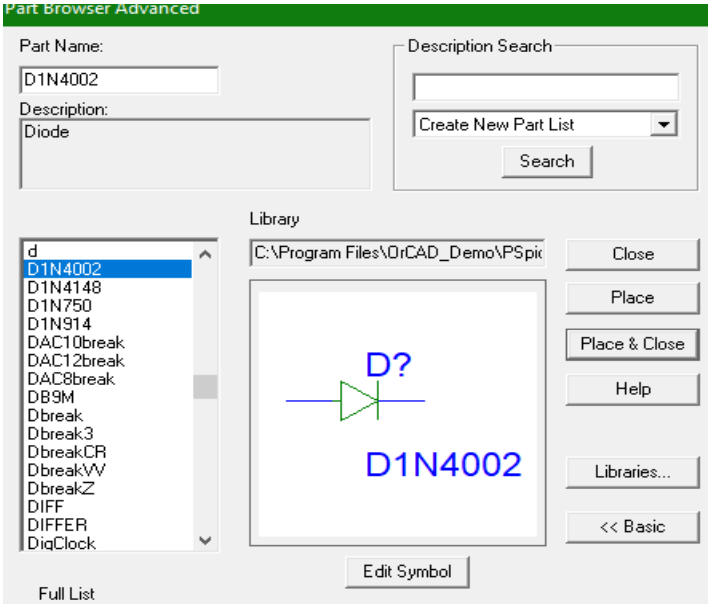
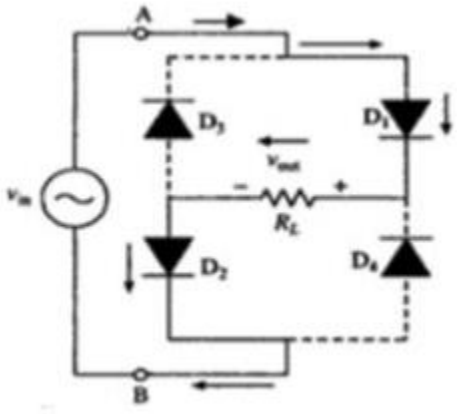
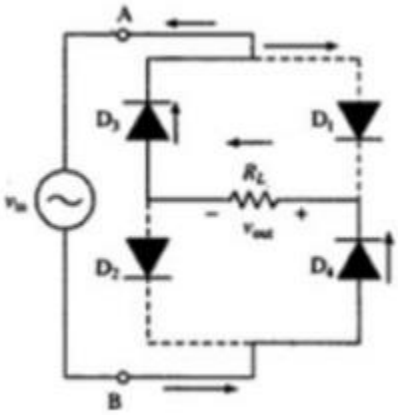


Figure: Diode



Positive half cycle



Negative half cycle

The bridge is composed of four diodes. During the positive half-cycle of input voltage vin, the terminal ‘A’ is at positive potential with respect to the terminal ‘B’ and because of this diodes D1 and D2 are forward biased whereas diodes D3 and D4 are reverse biased (Fig: 4). The current therefore flows through diodes D1, D2 and load resistor RL. During the negative half-cycle of input voltage waveform, the diodes D3 and D4 are forward biased whereas the diodes D1 and D2 are reverse biased (Fig: 5). As a consequence, current flows through diodes D3 and D4.

Step 3: Identifying and designing of filter

To filter the rectified wave, we are using a capacitor which is placed parallelly to the load resistor. Here, Capacitor: 100µF, Load resistor :100Ω. As a result, we get a ripple voltage wave.

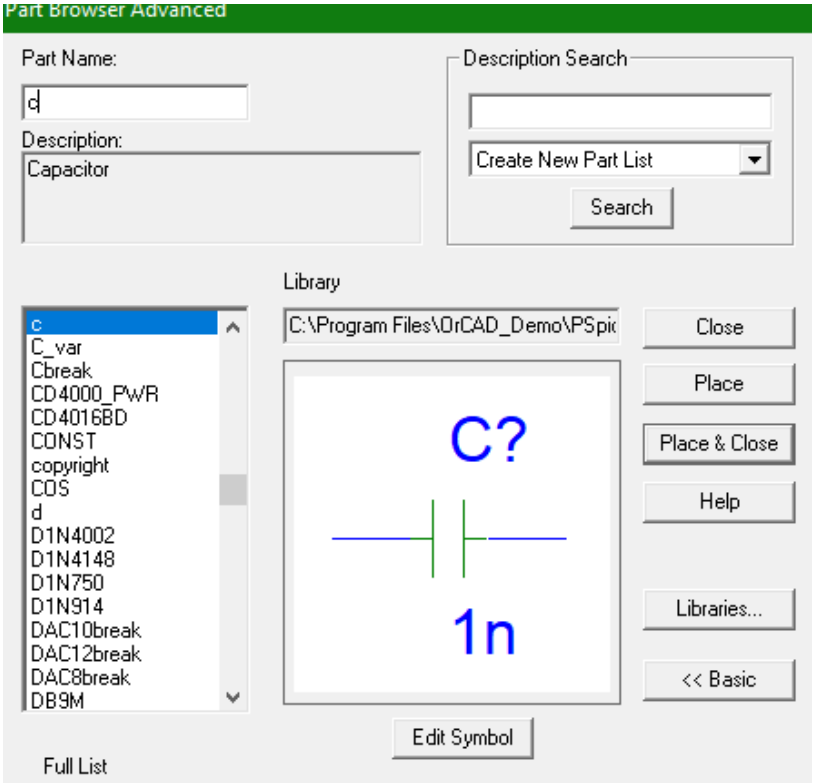


Figure: Capacitor

Step 4: Identifying and designing of Voltage Regulator

To stabilize the ripple voltage and fixed the output voltage in 5V we have used a Zener Diode as a voltage regulator.

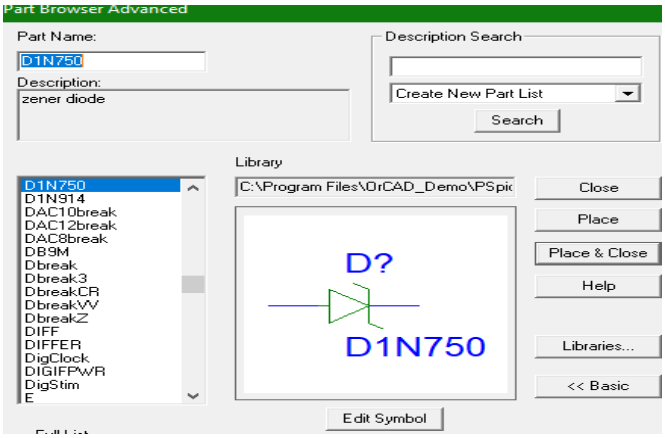


Figure: Zener Diode.

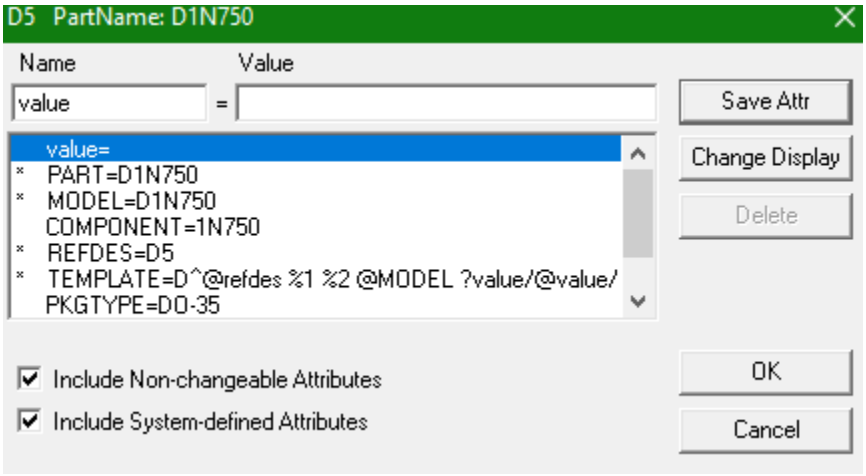


Figure: Values of Zener Diode.

To fixed the output voltage at 5V,

Here,

- Capacitor: 1mF or 1000µF
- Load Resistor: 100Ω
- Resistor in series with Zener diode: 5.5Ω

Circuit Diagram:

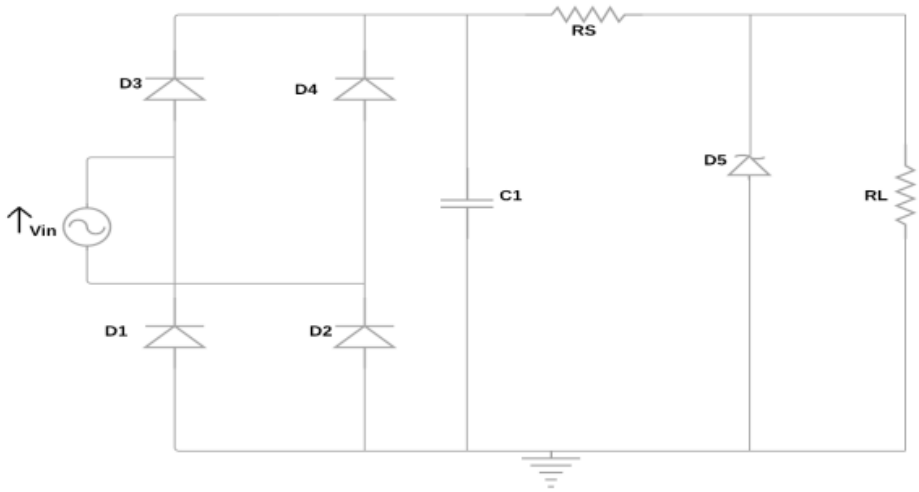


Figure: Rectifier Circuit with filter and voltage regulator

Simulation Results:

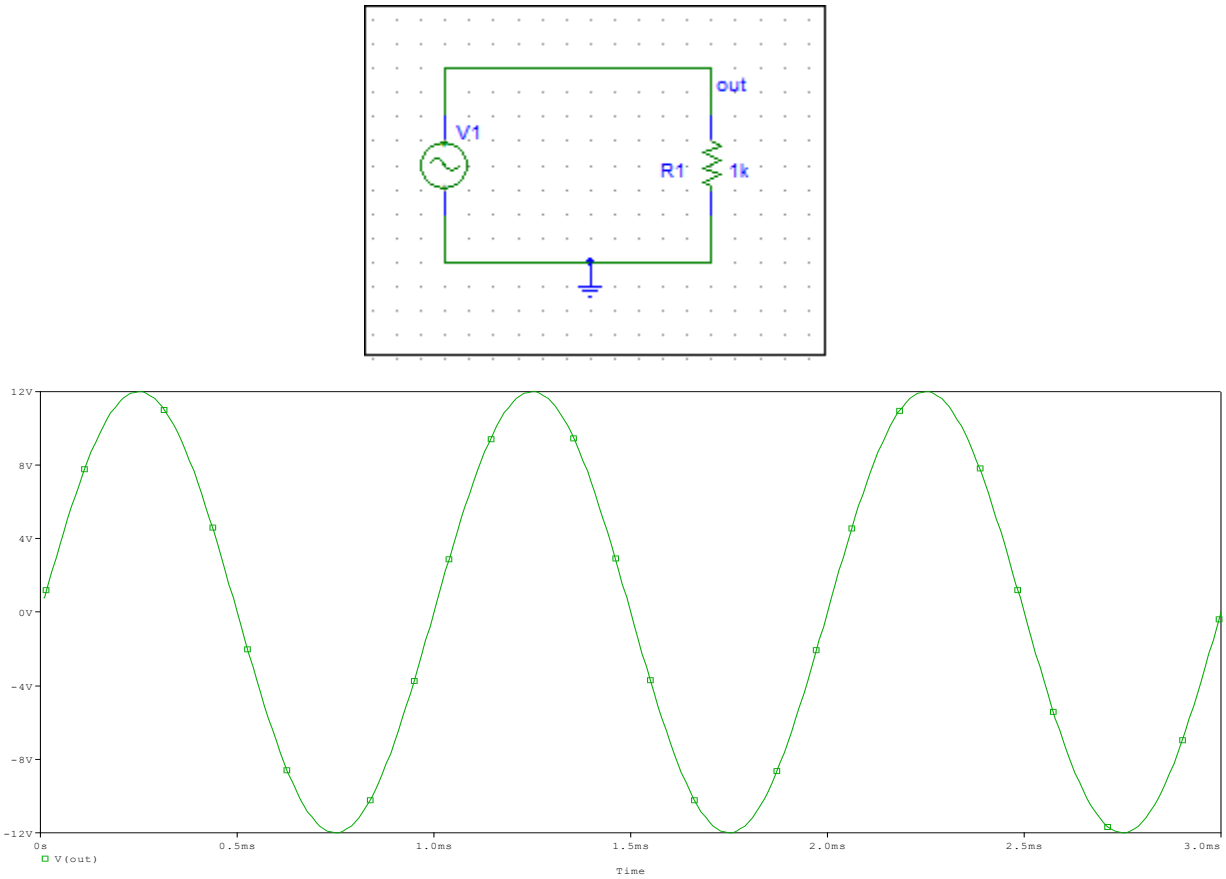


Figure: Input Wave

Diode Rectifier:

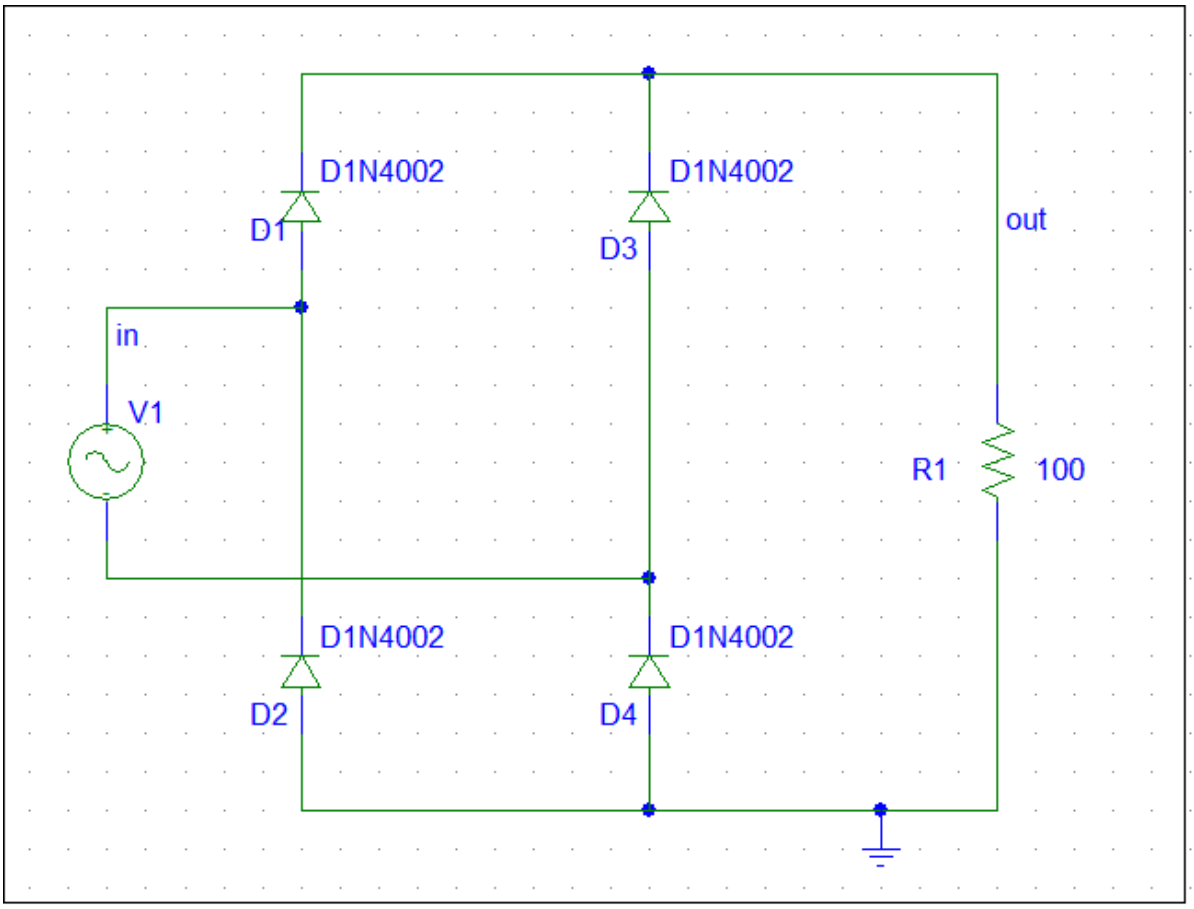


Figure: Rectifier Circuit

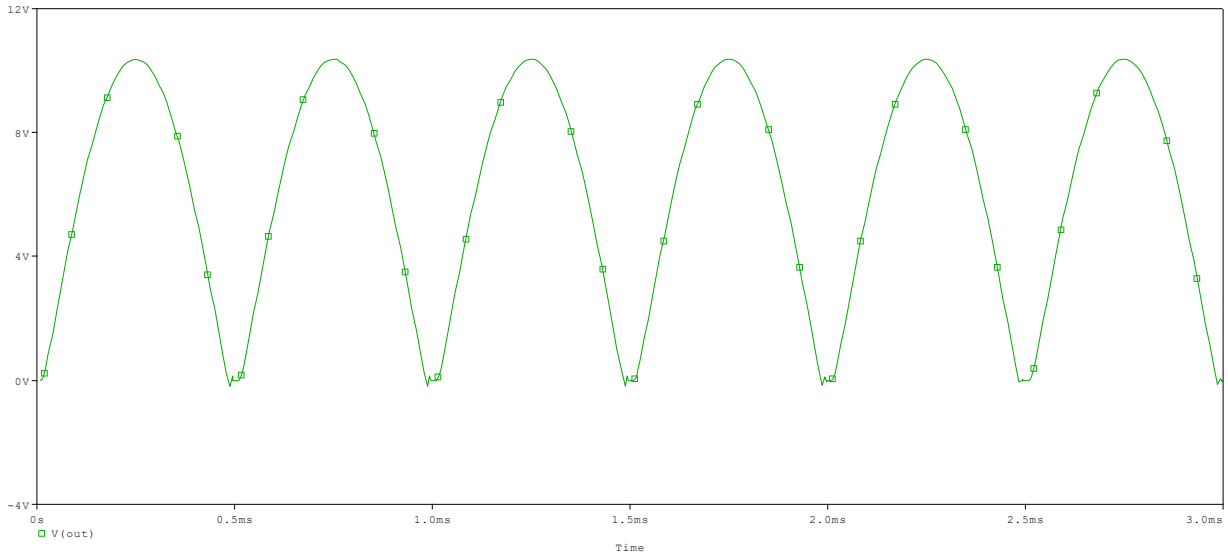


Figure: Output Wave after rectification

Filler:

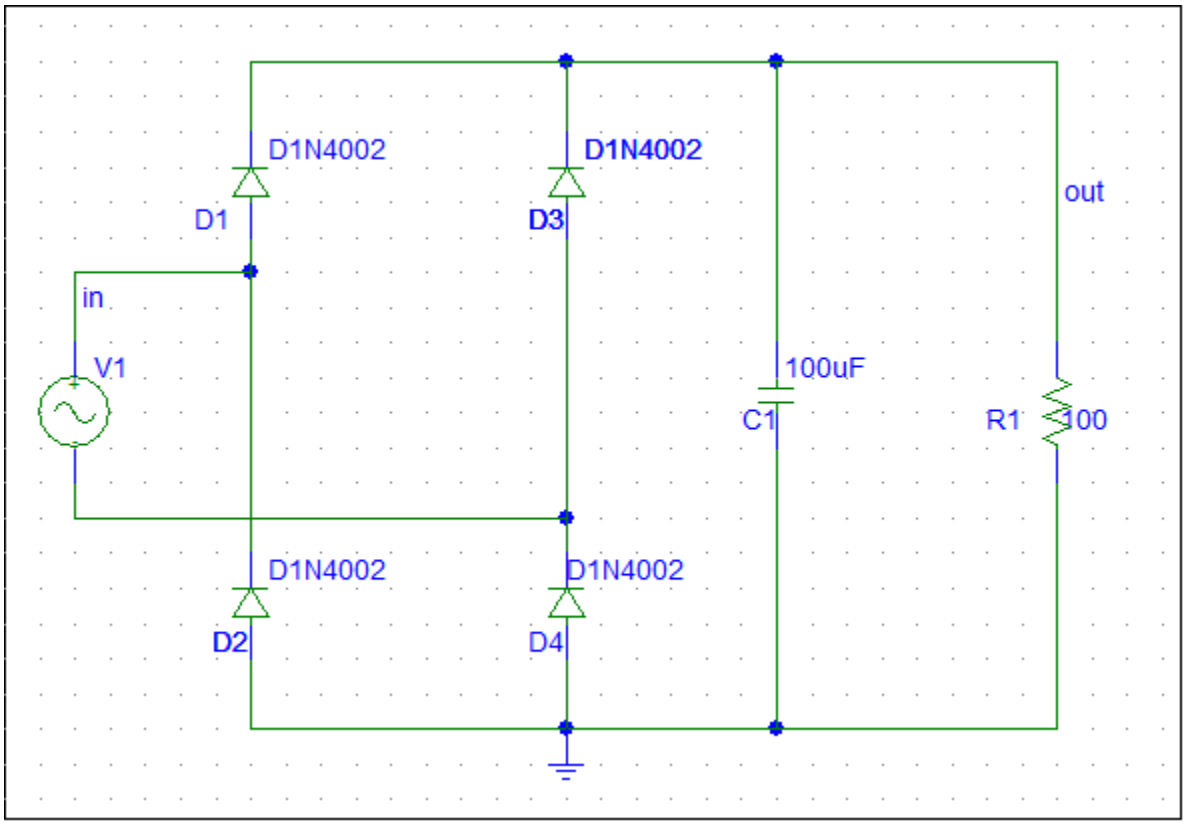


Figure: Rectifier Circuit with filter capacitor

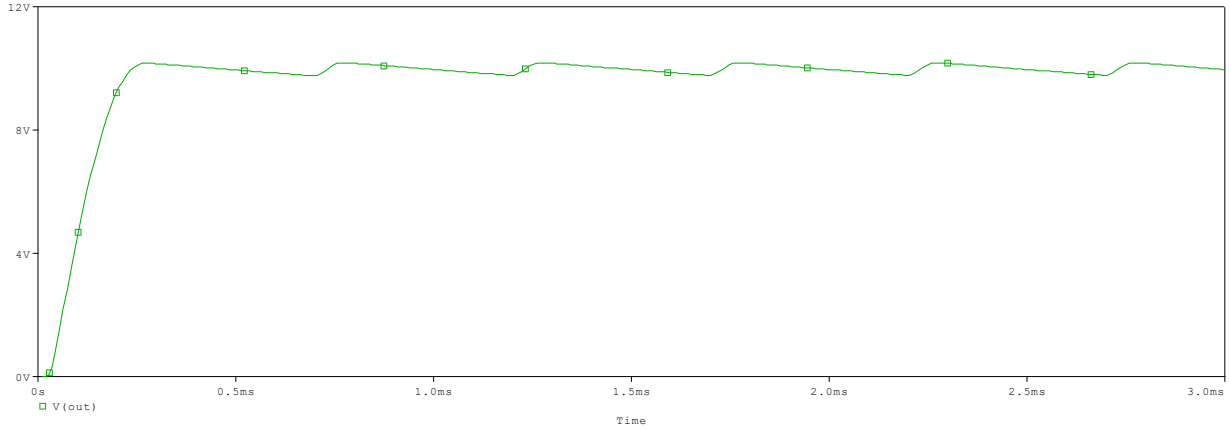


Figure: Output wave after filtration.

Voltage Regulator:

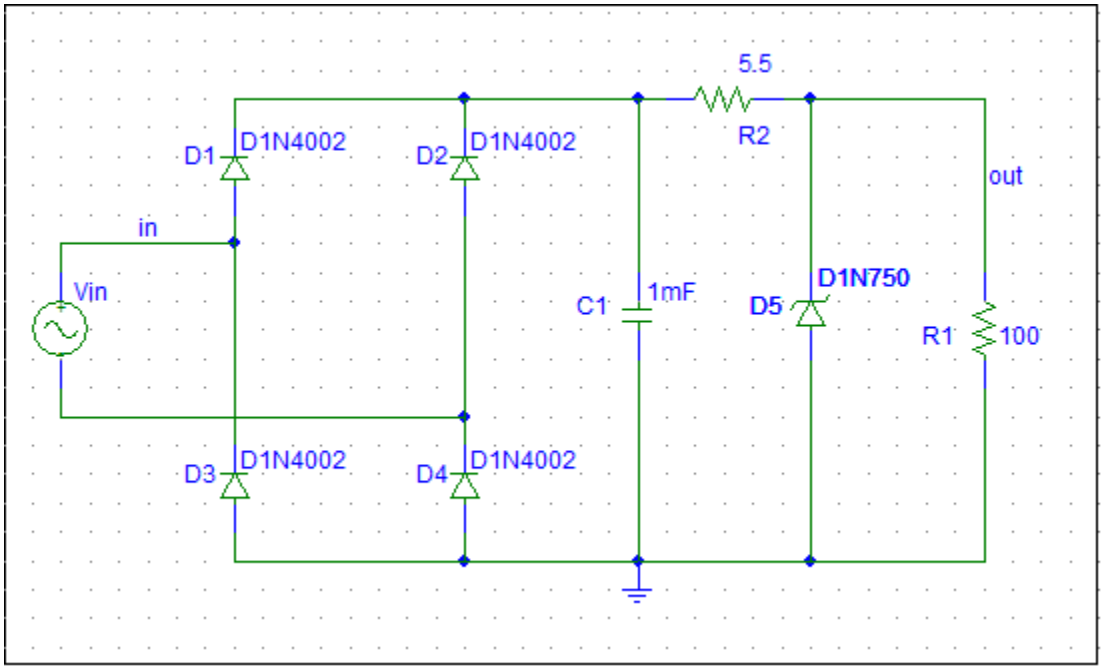


Figure: Rectifier circuit with regular voltage

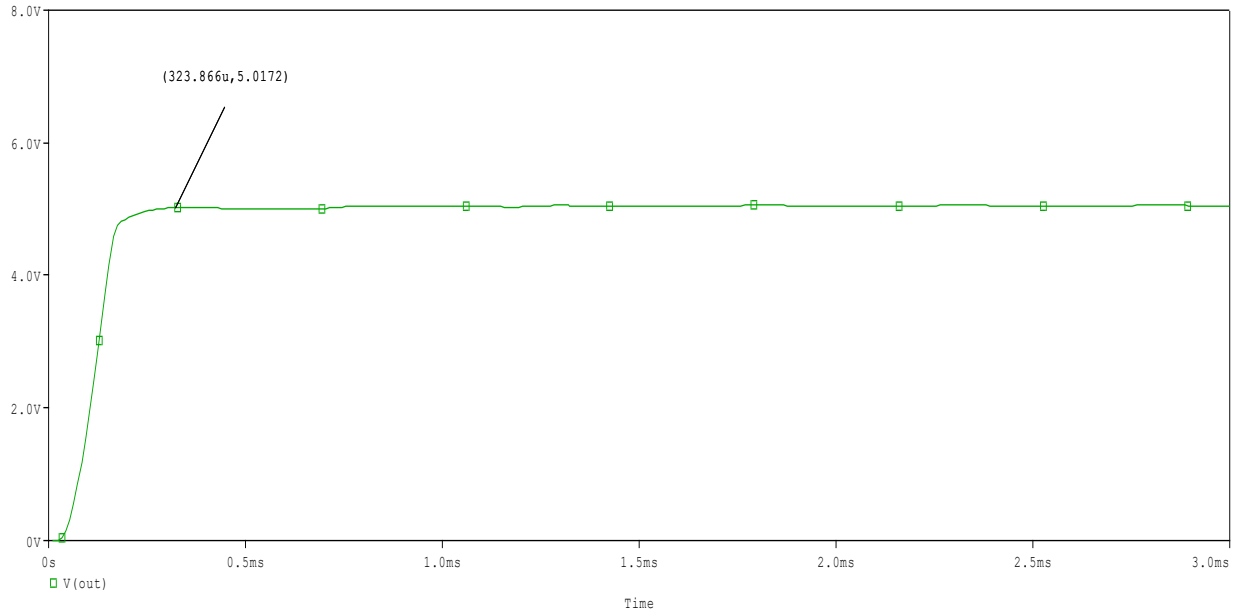


Figure: Output wave after stabilizing at 5V.

Conclusion:

Since all the simulations have been conducted virtually through PSpice, it was quite easier to conduct the experiments because there was no interruption. As a result, there is a very slight discrepancy between the results obtained.