MMI 172

AUDIO DESIGN WORKSHOP III - MODERN AUDIO HARDWARE

Lab 0: Diode Circuits and Power Supplies

INTRODUCTION

For a variety of reasons, it is generally advantageous to simulate circuits and electronic systems before assembling in the real world to ensure that it will behave as expected. Furthermore, you may be validating a design before placing a large parts order or sending it off to manufacturing. Taking the time to first simulate designs helps to avoid tiny errors that can manifest into serious blunders.

In this case, we will simulate power supply circuitry, being that it's typically a bad idea to experiment with electricity from your average wall outlet and transformers incur non-negligible cost. Modern audio electronics with solid-state components will usually require a DC voltage between 5 and 15V – however, power is distributed in a 3 phase AC network. The purpose of a power supply is to transform a higher input voltage to the desired range and rectify to DC. Here, we will address what exactly this requires.

OBJECTIVE

This introductory lab is dedicated to the simulation of simple diode circuits and a power supply using the simulation software PSpice. Additional materials and resources can be found on the course wiki.

REQUIRED MATERIALS

- PSpice 9.1, Student Edition [linked on the course website]
- Windows-enabled computer

PART I – GETTING STARTED WITH PSPICE

PSpice is a software suite for simulating circuits, and incorporates several programs to address different needs, such as building different component models or input waveforms. For our purposes, we will only need *Schematics* and *PSpice A/D*. The most straight-forward way of conducting basic transient analysis is detailed for the first circuit of the lab, and will provide you with the necessary skills for the remainder of the course.

A. HALF-WAVE RECTIFIER

The circuit diagram for a half-wave rectifier is given in Figure 1. One of the most important characteristics of the diode as an electrical component is that it allows current to only flow in one direction. This makes half-wave rectification an immediate and simple application. To simulate this circuit, we want to analyze its transient behavior, which can be achieved by the following:

i. Start *Schematics*.

ii. Place a resistor on the grid by pressing the “Get New Part” button and either: Type “R” in the field or scroll down to find the resistor. **Note:** “Get New Part” also found in *Draw* Menu.

iii. Also place the diode (D1N4002), voltage source (VSIN), and analog ground (GND_ANALOG) in the same way.

iv. Connect the components by selecting “Draw Wire” (not “Draw Bus” which is to the right).

v. Set the appropriate parameters by double-clicking on the component you would like to adjust.

[VSIN: VAMP = 15V, FREQ = 60Hz, VOFF = 0V.   R: VALUE = 1k to 20K]

vi. Add a voltage probe to a node of interest by selecting the button; in this case, we would like to observe the voltage across the resistor.

vii. In the menu “Analysis/Setup…”, make sure that *Transient* is the only checked box. Click the corresponding button and set *Final Time* equal to 100ms.

viii. Either click the simulate button or follow the menu “Analysis/Simulate” to run the circuit simulation. A time-voltage plot will automatically be displayed.

Now that you have assembled and simulated a basic HWR circuit, repeat the process for the circuits in Figures 2-3, noting the details in the captions for each figure (including Figure 1).

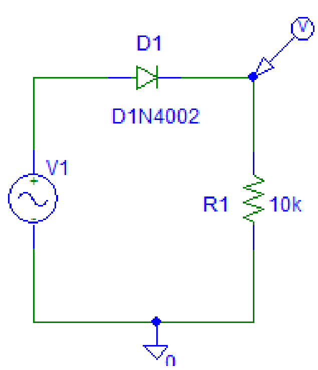

Figure 1 – Half-Wave Rectifier, R={1k:10k}. Reverse the diode (Ctrl+R) and comment on the results.

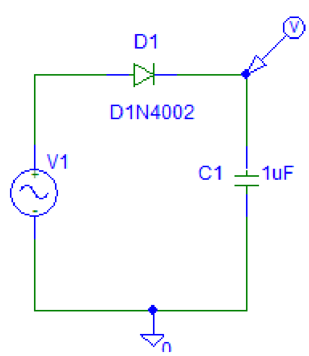

Figure 2 – Peak Rectifier, C={1µF:10µF}.

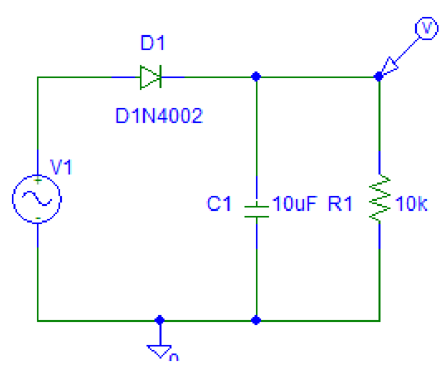

Figure 3 – HWR with Low Pass RC Filter. Try values of τ = RC {0.05:1}. Add a DC Offset (VOFF) and observe the impact on the ripple voltage.

B. FULL-WAVE RECTIFIER

For full-wave rectification, the signal must be fed to four diodes, in a topology known as a “Diode Bridge.” The signal passes through two of the diodes when the input voltage is positive, and the other two when the signal changes polarity. You can think of the circuit as coupling a positive HWR and a negative HWR with a common output. The diode bridge results in a signal through the load resistor that always has the same direction.

For the voltage source (VSIN), set the parameters as [VOFF = 0V, VAMPL = 15V, FREQ = 60Hz | R = {1k:10k}] and run another transient analysis, again with a final time of 100ms. Be sure to place the voltage probe on the correct node.

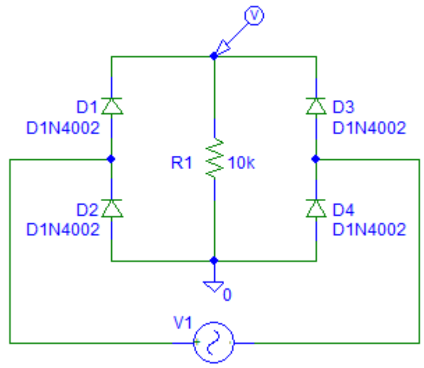

Figure 4 – Full-Wave Rectifier

PART II – A FULL POWER SUPPLY

A DC Power Supply can be easily created by feeding an AC signal to a full-wave rectifier diode circuit. The output of the rectifier is filtered using a simple RC filter to significantly attenuate the ripple voltage.

For the voltage source (VSIN), set the parameters as [VOFF = 0V, VAMPL = 15V, FREQ = 60Hz | τ= {0.02:0.5}] and run another transient analysis, again with a final time of 100ms. Be sure to place the voltage probe on the correct node. Additionally, note that the ripple voltage is given by the equation $V_R=V_S/(2f\tau)$. Compare the actual ripple voltage and the ripple voltage measured in the simulation.

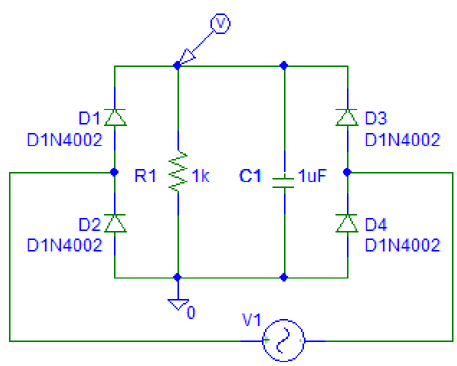

Figure 5 – Full Power Supply