

Simulation Lab. Rectifier Circuit

Name: Benedict Chin Matric No.: A0284578J Group: 003

OBJECTIVES

1. Be able to construct rectifier circuit in LTSpice with the given diode model.
2. Run transient and average voltage analysis using LTSpice.

INTRODUCTION

Full wave rectifier is a simple diode circuit that helps converting AC voltage into DC voltage. It is constructed using four diodes as shown in Fig. 1.

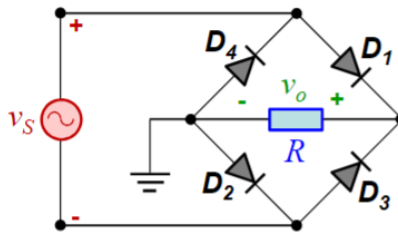


Fig. 1 Full wave rectifier

The $v_o(t)$ and $v_S(t)$ will exhibit waveforms as shown in Fig. 2.

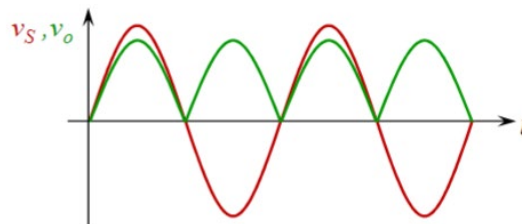


Fig. 2 Full wave rectifier transient waveforms

In fact, both waveforms will exhibit different average DC value and can be predicted as follows:

$$v_{S_avg} = \frac{1}{T} \int_0^T v_{S_pk} \sin(2\pi ft) dt = \frac{v_{S_pk}}{T} \left[-\frac{1}{2\pi f} \cos(2\pi ft) \right]_0^T = 0$$

$$v_{o_avg} = \frac{2}{T} \int_0^{T/2} v_{o_pk} \sin(2\pi ft) dt = \frac{2v_{o_pk}}{T} \left[-\frac{1}{2\pi f} \cos(2\pi ft) \right]_0^{T/2} = \frac{2v_{o_pk}}{\pi}$$

Where T is the period the sine wave, v_{S_pk} and v_{o_pk} are the peak amplitude for $v_S(t)$ and $v_o(t)$ respectively.

LAB EXERCISE

In this lab, you are required to perform transient analysis using LTSpice. You will also extract the average voltage based on the transient analysis.

1. Full Wave Rectifier Circuit

a. LTSpice Circuit Construction

Construct the rectifier circuit using LTSpice as shown in Fig. 3. The V1 source should be set to a sine wave with amplitude of **12 V** and frequency of **80 Hz**, and the resistor R1 should be set to **1 k Ω** . You should label the important nets, such as “Vout”, “V+” and “V-”.

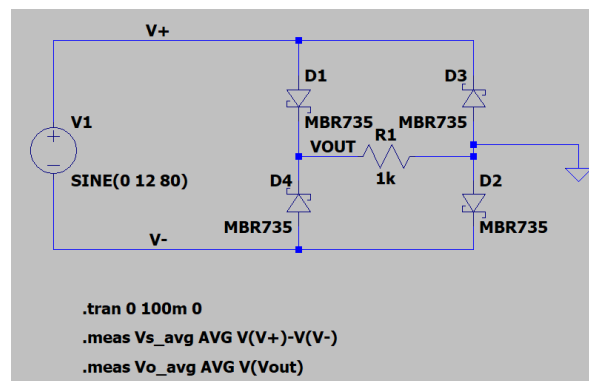


Fig. 3 LTSpice rectifier circuit

To define the diode, right click on the diode symbol. A pop-up window as shown in Fig. 4 will appear. Click “Pick New Diode” button, and select **MBR735**.

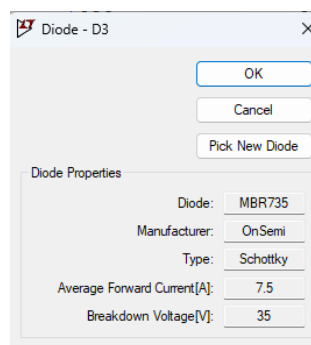


Fig. 4 Pop-up window to define the diode

To define the input sinusoidal source, i.e. V1, similarly right click on the V1 component. A pop-up window as shown in Fig. 5 will appear.

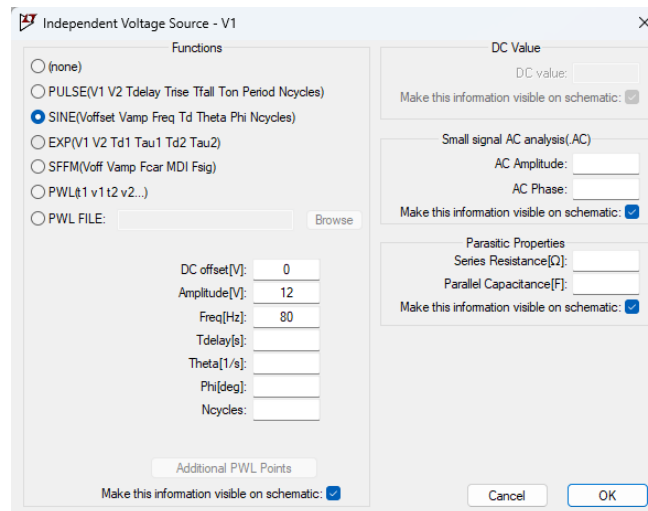


Fig. 5 Pop-up window to define the V1 parameters

You should choose “SINE”, and define the “DC offset(V)” to be 0 V, “Amplitude(V)” to be 12 V, and “Freq(Hz)” to be 80 Hz.

b. LTSpice Transient Simulation

Perform transient simulation, by selecting “Simulate->Edit Simulation Cmd”. Choose “Transient” tab. A pop-up window as shown in Fig. 6 will appear. Set the “Stop time” to 100 ms.

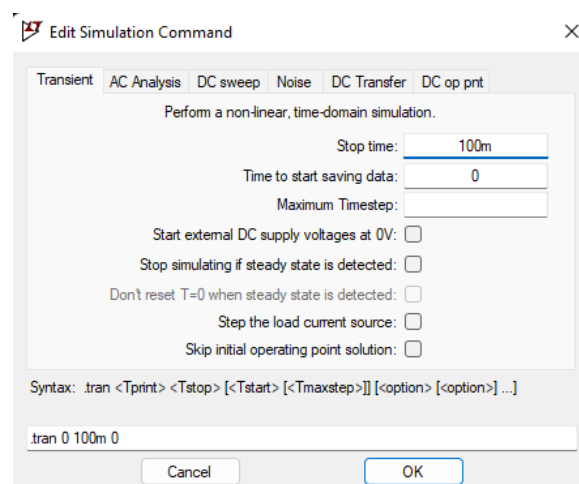


Fig. 6 Pop-up window for transient simulation

Print out the transient waveforms of “V+ – V-” and “Vout”. Label the voltage at the peak of the “V+ – V-” and “Vout”, and note down their difference below. The printout should be attached together with this lab sheet for submission.

Peak Voltage of “V+ – V-”= 11.997 (V)

Peak Voltage of “Vout”= 11.596 (V)

Peak Voltage Difference = $\underline{11.997 - 11.596 = 0.401 \text{ (V)}}$

Note: This peak voltage difference accounts for the voltage drop across the two diodes. This allows us to deduce the turn-on voltage for the single diode.

c. LTSpice Average Voltage Estimation

You can use “.meas” SPICE directive to measure the average voltage. Here, “V(V+)-V(V-)” is the node voltage of the $v_s(t)$, “V(Vout)” is the node voltage of $v_o(t)$. Include the following SPICE directives into the schematic as shown in Fig. 7, i.e.

“.meas Vs_avg AVG V(V+)-V(V-)”

“.meas Vo_avg AVG V(Vout)”

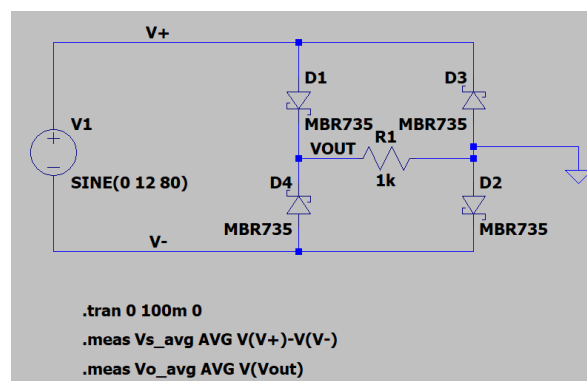


Fig. 7 Setup for .meas SPICE directive

Select “View->SPICE Error Log”. A pop-up window will appear, **note down the Vs_avg and Vo_avg value. You should attach a copy of the SPICE Error Log printout showing Vs_avg and Vo_avg in your lab submission as well.**

Vs_avg = $\underline{3.934 \times 10^{-5} \text{ (V)}}$

Vo_avg = $\underline{7.2917 \text{ (V)}}$

Vo_avg (Calculated) = $\underline{2(11.596) / \pi = 7.382 \text{ (V)}}$ (based on the peak voltage of Vout from the transient simulation)

$V_{avg} = 2V_p / \pi$

SPICE Output Log: C:\Users\Ben\Documents\LTspice\SimLab.log

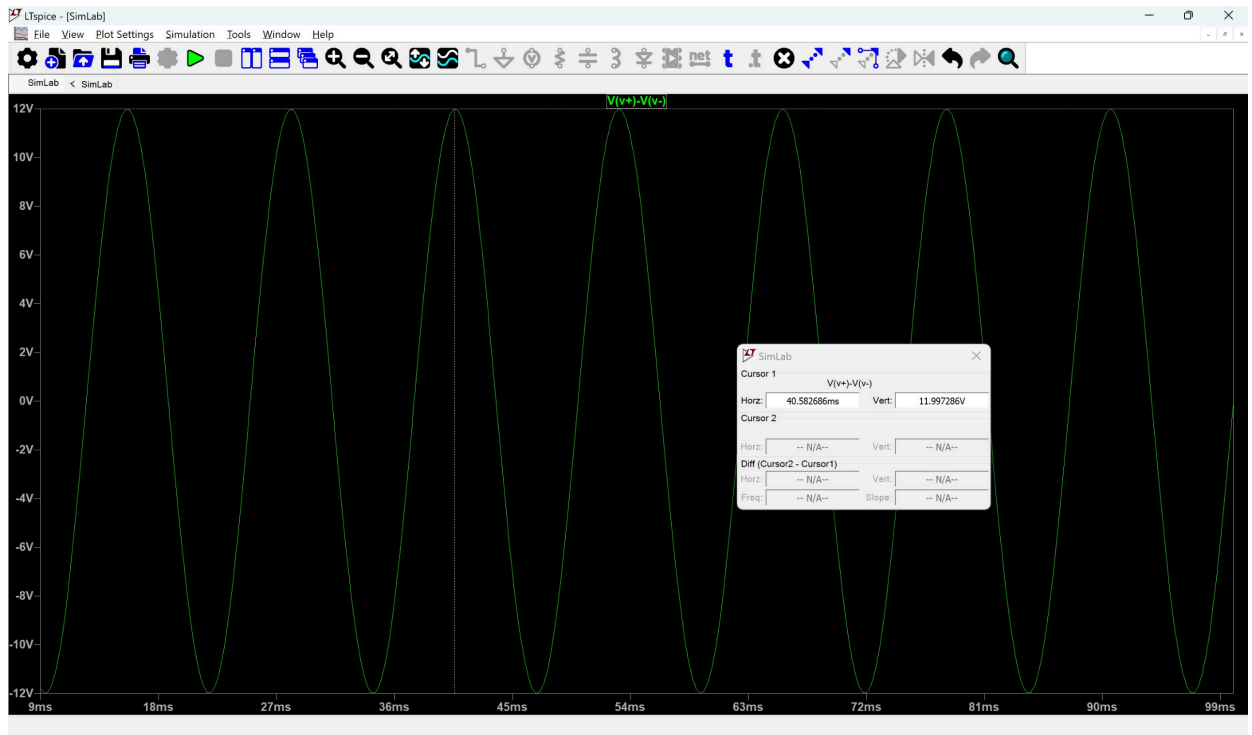
```

LTspice 24.0.12 for Windows
Circuit: * C:\Users\Ben\Documents\LTspice\SimLab.asc
Start Time: Thu Oct 3 13:22:05 2024
solver = Normal
Maximum thread count: 20
tnom = 27
temp = 27
method = modified trap
.OP point found by inspection.

vs_avg: AVG(v(v+)-v(v-))=3.93404e-05 FROM 0 TO 0.1
vo_avg: AVG(v(vout))=7.2917 FROM 0 TO 0.1

Total elapsed time: 0.296 seconds.
  
```

$V_+ - V_-$ graph



V_{out}

