

JABATAN KEJURUTERAAN ELEKTRIK

COURSE CODE	EC302	COURSE NAME	ELECTRONIC COMPUTER AIDED DESIGN
COURSE WORK	LAB WORK 3	TOPIC	AC ANALYSIS
DATE			

Learning Outcomes:

1. Apply various types of analyses and tools provided in the simulation package and analyze the simulation results to ascertain its compliance to the electronic circuit theory.
2. Draw the various analogue, digital and mixed-signal circuit schematics and simulate the circuits using a particular simulation package.

Laboratory Equipment:

Computer / MicroSim Eval 8

Theory:

Transient analysis is use to examine the the behavior of a waveform (voltages or current) as time varies. Transient analysis solves some differential equations describing a circuit and obtains voltages and currents versus time. Transient analysis is also used to obtain Fourier Analysis. To perform transient analysis on a circuit using Microsim usually involves these steps (1) drawing the circuit (2) providing specifications and (3) simulating the circuit.

Procedure:

PART A: RC CIRCUIT. [CLO3]

Steps :

1. Draw the circuit as shown in figure 3.1
In order to run a transient analysis on a circuit, the circuit must first be created using **Schematics** and the source must be specified. Sources used in the transient analysis include VSIN.

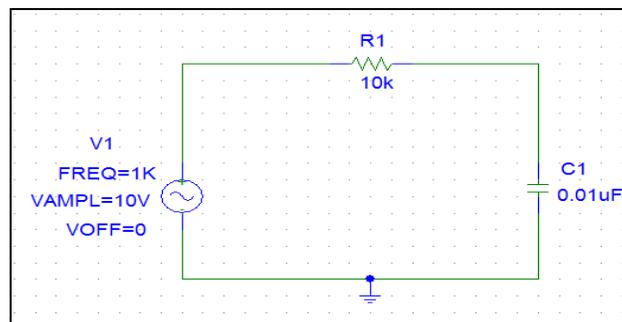


Figure 3.1

- Set the attribute for Vsin as per below:[CLO3]
 Voff = 0
 Vamp = 10V
 Freq = 1KHz
- After the circuit is drawn and the source is specified with its attributes, we need to add some specifications for the transient analysis. Click **Setup Analysis** and click the **Transient** box to open **Transient window** and set the transient as figure 3.2.

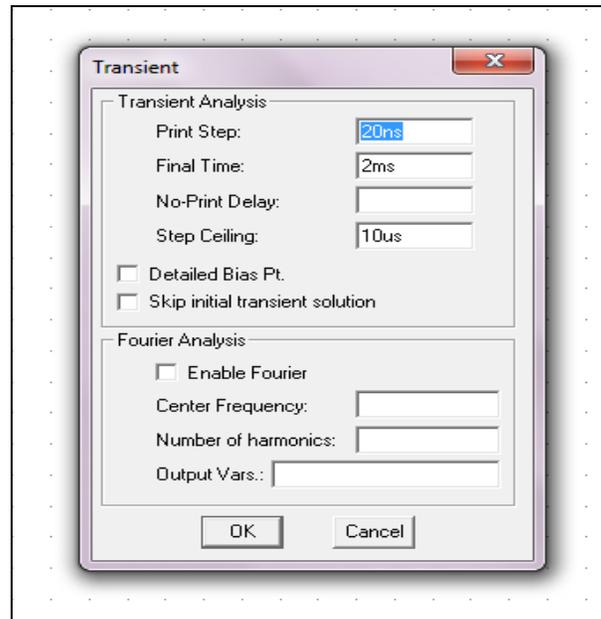


Figure 3.2

- Next, analyze the circuit as below:

The setup process is complete, so we save the file and begin PSpice simulation. (If errors are found, refer to the Message viewer and output file).

When the PSpice window appears (Figure 3.3), note the summary at the bottom, which gives a running account of the *Time step* (time between calculation), the *Time* (present calculation time), and the *End* (finishing time)

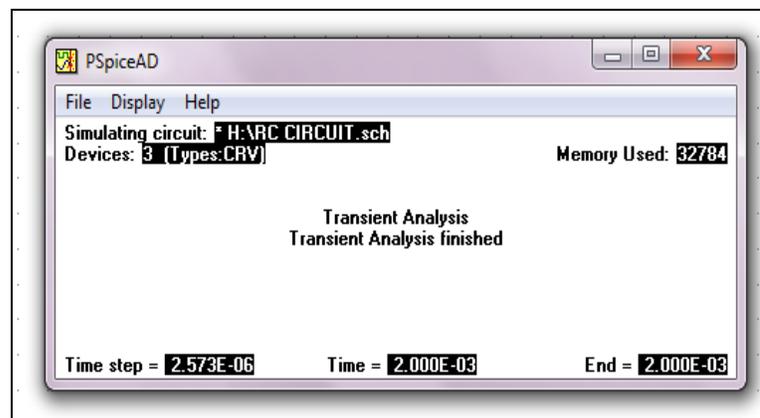


Figure 3.3

5. Then, to display input and output voltages as Y-axis variable, set the two voltage “markers” as shown in Figure 3.4. Drag the voltage marker to the desired node.[CLO3]

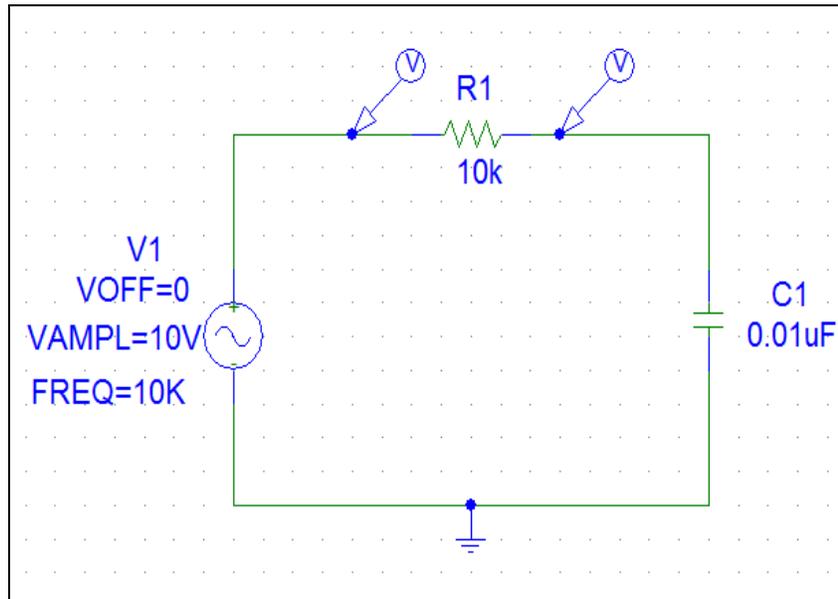


Figure 3.4

(Rubric : Circuit drawing – 2 Marks)

6. Probe generated transient analysis of RC circuit as below:

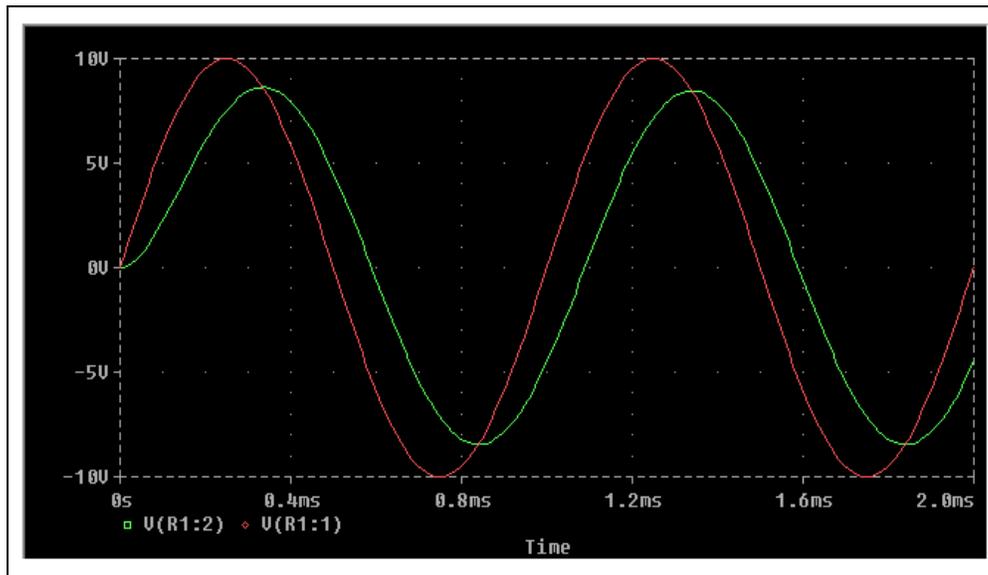


Figure 3.5

(Rubric : Analysis/Result – 4 Marks)

PART B: MULTIPLIER CIRCUIT. [CLO3]

1. Draw the circuit as shown in figure 3.6:

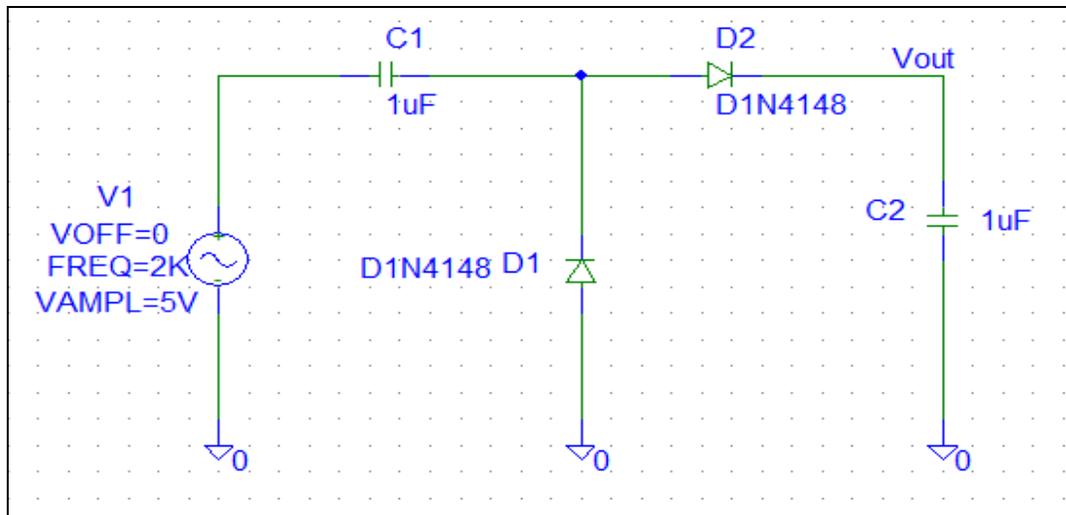


Figure 3.6

(Rubric : Circuit drawing – 3 Marks)

Steps:

1. Follow step 1-3 as above (change the attribute follow the circuit).
2. Using Multi Sim, generate the output waveform from 0 to 4ms

ANALYSIS/RESULT:

3. Find the output voltage waveform
4. Is Vout is still increasing
5. Change the display back to 0 to 4ms , display the new waveform , and compare with the result. Does the circuit approach steady state sooner.

(Rubric : Analysis/Result – 8 Marks)

PART C: RECTIFIER CIRCUIT. [CLO3]

Draw the circuit as below:

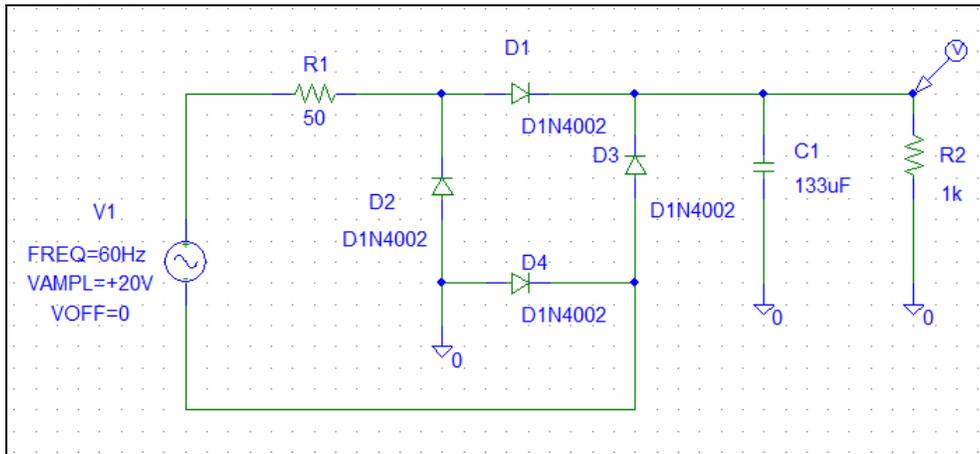


Figure 3.7

(Rubric : Circuit drawing –5 Marks)

Steps:

1. Follow step 1-3 as above ((change the attribute follow the circuit).
2. Generate the output waveform
3. Setting a **step ceiling** to generate more data points

ANALYSIS/RESULT:

1. Discuss the output waveform.
2. What are the solution we add to shorts the AC ripple to ground.
3. What controls do we have over the transient analysis process

(Rubric : Analysis/Result – 8 Marks)

Conclusion :

Write the conclusion based on your findings

(Rubric : Conclusion - 5 Marks)

PREPARED BY	CHECKED BY	APPROVED BY
COURSE LECTURER	COURSE COORDINATOR	HEAD OF PROGRAM
NORISNIYATI ABDUL HAMID	NORHANANI ABD RAHMAN	MOHAMMAD FADZIL BASIR AHMAD
DATE : 20/1/2012	DATE : 20/1/2012	DATE : 20/1/2012