

Experiment 12

Analysis of a Diode Clipper Using SPICE

Introduction

The purpose of this experiment is to gain practical experience with the mechanics of a SPICE simulation using a dc sweep, and to compare the results of the simulation with a hand-worked approximate analysis.

Pre-Lab

Analyze the diode clipper circuit of Fig. 1, assuming the diode to be ideal. Sketch a plot of the output voltage v_{OUT} vs. the input voltage v_{IN} for the range $-10 < v_{IN} < +10$. (This is known as the “transfer function.”) Bring these expected results to the lab.

Equipment Needed

- Current version of PSpice

Procedure

Turn on the PC and open the current version of PSpice. All simulations will be done using a “DC Sweep” analysis. Set up the dc sweep analysis to vary the independent input voltage v_{IN} from -10 V to +10 V in steps of 0.1 V. This will cause probe to use v_{IN} as the x axis variable; the y axis can then be used to plot the results of the dc analyses done for the 200 incremented values of v_{IN} .

1. Simulate the circuit of Fig. 1 using a 1N4148 diode model from the component model library. Use probe to plot v_{OUT} vs. v_{IN} , using an x axis of -10 V to +10 V. Record these results.

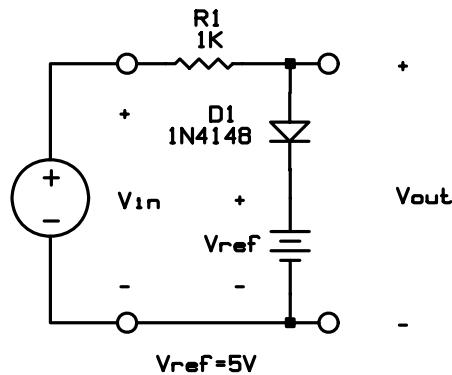


Fig. 1 Diode clipping circuit.

Experiment 12 Analysis of a Diode Clipper Using SPICE

2. Use probe to plot the diode current as a function of the diode voltage. You may use the results of the simulation already run by using the probe dialog sequence “Plot,” “X Axis Settings,” “Change Axis Variable” to make the diode voltage the x axis of the plot. Set the range of the x axis to 0 to +1 V. Record the plot of i_D vs. v_D .
3. Make the y axis (diode current) logarithmic. You will get an error statement if you do not first restrict the y axis to a positive range of values (why?). Set the y axis range to 1 μ A to 10 mA. Then set the y axis for log scaling. Record the resulting plot of i_D vs. v_D .

Report

No formal report is required for this experiment. Simply summarize the results you have obtained in a homework format. Briefly answer the following questions:

1. Compare the transfer function (plot of the response v_{OUT} vs. the input v_{IN}) sketched by hand in the pre lab with that obtained using a SPICE dc sweep.
2. Assuming the input voltage to this “clipping” circuit to be a sine wave having an amplitude of 5 V, what would the output voltage look like? Sketch the output voltage vs. time.
3. What evidence can you give that SPICE is actually modeling the current in a pn diode to be an exponential function of the diode voltage in the forward region? Explain.
4. If you were to simplify this diode to a constant-forward-voltage-drop model in its forward region, what voltage would you use?