

Experiment 11

DC Analysis of a Diode Circuit Using SPICE

Introduction

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

Pre-Lab

Analyze the circuit of Fig. 1 twice, once assuming the diode to be ideal, and then assuming a constant-voltage-drop (CVD) of 0.7 V in the on state. Find the diode state, the diode voltage, and the diode current. Bring these expected results to the lab.

Equipment Needed

- 1N4148 Small-signal silicon diode
- Current version of PSpice
- Normal lab equipment

Procedure

Turn on the PC and open the current version of PSpice. All simulations will be done using a “DC” analysis. (This is also referred to as a “bias point” analysis.)

1. Simulate the circuit of Fig. 1 using a 1N4148 diode model from the component model library. The results will appear in the output (*.OUT) file. Record the diode voltage and current.
2. Simulate the circuit of Fig. 1 using the following diode model parameters: $I_S = 10^{-15}$ A and $N = 0.01$. These can be entered in PSpice by editing the diode “instance model.” Highlight the diode on the schematic and pick “Edit,” followed by “Model,” followed by “Edit Instance Model.” This gives you access to the text file containing the diode model

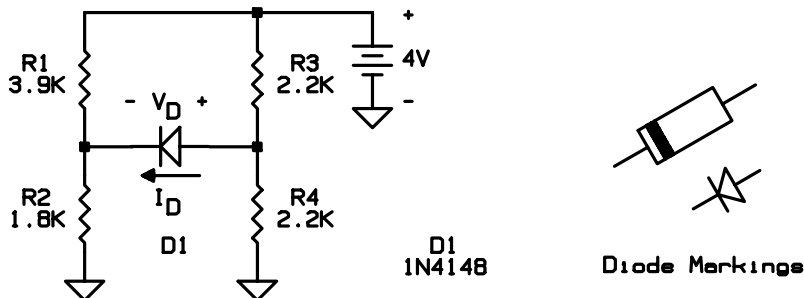


Fig. 1 DC diode circuit.

Experiment 11 DC Analysis of a Diode Circuit Using SPICE

statements. The changes you make to the model will be local to your schematic only (they will not change the library). Delete all existing model statements; enter the two model statements¹ given above. Record the diode voltage and current.

3. Assemble the circuit of Fig. 1 using a 1N4148 and measure the diode voltage and current using a digital multimeter. Record the diode voltage and current.

Report

No formal report is required for this experiment. Simply summarize the results you have obtained in a homework format. The required results are the diode state (on or off), diode voltage, and diode current as given by each of the following techniques:

- (a) Hand calculation based on an ideal diode model.
- (b) Hand calculation based on a CVD diode model.
- (c) SPICE simulation based on a 1N4148 model.
- (d) SPICE simulation based on $N = 0.01$ and $I_S = 10^{-15}$ A.
- (e) Experimental measurement using a 1N4148 silicon diode.

Comment on what happened when N was set to the unrealistically low value of 0.01.

¹ The format of the SPICE model statement should end up similar to the following:
.MODEL 1N4148 D(N=0.01 IS=1E-15)