

American University of Beirut  
DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING

EECE 310 – Electronics

Fall 2011 – 2012

*Due Wednesday October 12, 2011 at 9:00 am*

### **CAD Assignment 1**

Use PSpice to analyze the circuit of Problem 3 in Homework 2.

Start by placing 4 resistors with the values shown, and a 9V DC voltage source.

*Make sure that your ground node(s) are named 0.*

The diode part to use is Dbreak from the BREAKOUT library. After placing the diode, right-click on its symbol, then click on Edit Pspice Model. Replace the .model line in the Model Editor with the following line:

```
.model D_ideal D Is=1e-14 n=1e-6
```

The unusually small value of  $n$  ( $10^{-6}$ ) in this model makes the diode behave like an ideal one.

Create a new simulation profile, (under the PSpice menu), and call it DC Analysis. What we are interested in for this circuit is an Analysis Type of “Bias Point”.

Simulate your circuit, and observe the node voltages and branch currents, by clicking on the “V” and “I” buttons on the Capture toolbar.

To simulate the circuit now with a real diode, right-click on the diode symbol, then click on Edit Pspice Model. Replace the .model line in the Model Editor with the following line:

```
.model D_real D Is=1e-14 n=1.2
```

This value of  $n = 1.2$  makes the diode behave like a real one, with a value of  $I_S = 10^{-14}$  A.

Simulate your circuit, and observe the node voltages and branch currents, by clicking on the “V” and “I” buttons on the Capture toolbar.

- Submit a printout of your Capture windows showing the values of currents and voltages for both cases (ideal diode and real diode).
- List the values of the diode voltage and current for the real diode case, as obtained from SPICE, and compare them with the values obtained in Problem 3, part (b-i).