EGRE 224 Introduction to Microelectronics

Laboratory No. 2

DC Diode Characterization and Simulation
I. Introduction

In this laboratory you will measure the I/V characteristics of a real 1N270 diode using the Hewlett-Packard HP4156B Precision Semiconductor Parameter Analyzer. Then you will simulate the 1N270 diode in Accusim using both the model for the device provided in the AccuParts library and a Spice model for which you devise the characteristics yourself.

II. Procedure

Use the HP4156B Parameter Analyzer in room 241 as instructed to generate a file containing data for \( I_D \) vs. \( V_D \) for the 1N270 diode for a range of \( V_D \) from -1.0V to 1.0V. Save this data to a floppy disk for analysis later.

In the EDA lab (room 213), use Netscape to bring up the class web page:

http://www.people.vcu.edu/~rhklenke/egre224

Click on the Basic Schematic Capture and Analog Simulation with Design Architect and Accusim link under the Mentor Tools item. Go through Tutorial 3. Save the data for later analysis using Excel. After you analyze the data, modify the IS, RS, and N parameters, as explained below to attempt to develop a model of the diode that more closely resembles the measured data.

III. SPICE Diode Models

The SPICE model of a diode is shown in the figure below.
The DC characteristic (I_D vs. V_D) is modeled by the non-linear current source in series with the resistor r_S. The current source follows the ideal diode equation:

\[ I_D = I_S \left( e^{V_D/nV_T} - 1 \right) \]

In the Spice model, three parameters affect the DC characteristics of the diode model, I_S, n, and r_S. I_S is the saturation current, n is the emission coefficient, and r_S is the resistance of the ohmic regions of the diode on either side of the depletion region. The value of I_S and r_S can be found from a log plot of I_D vs. V_D as shown in the figure below [1].

For the 1N270 diode, the log plot of I_D vs. V_D has a very different. From the graph of the actual diode data, it is easier to see the intercept point of I_D if the range of values that are plotted is reduced to V_D < 0.1V. When plotted this way, I_S can be read directly from the graph as the point where the I_D curve appears to cross the V_D=0 axis (obviously, if V_D is equal to 0 then I_D will be as well). The value of r_S can be estimated by taking the inverse slope of the I_D vs. V_D curve for voltages greater than 0.3 or so.

The typical value of the emission coefficient, n, is 1.0. In some cases, n can be as high as 2.0 and can also be slightly less than one.

A plot of the VD vs. ID curves for the actual diode and the two simulated diodes, the AccuParts model and the model constructed using the default values of the three SPICE parameters defined above, is shown in the graph on the next page. Notice that the default model is not a good representation of the actual diode’s characteristics. By varying the values of I_S, r_S, and n, a better model can be constructed.

Varying the value of I_S will shift the “turn-on” point of the diode curve to the left or right on the X-axis. Varying the value of r_S will vary the slope of the curve past the
turn-on point. Finally, varying the emission coefficient, \( n \), will vary the shape of the curve around the turn-on point.

![Diode Characterization Graph]

### III. Write-up

Your task as part of the lab exercise is to vary the parameters in the Spice diode model to attempt to generate a model that is closer to the actual diode’s characteristics than the default model. You may in fact be able to generate a better model than the AccuParts model if you experiment a bit, but that is not required. What is required is to come significantly closer than the default model.

The write-up should include the following sections:

1. Abstract
2. Introduction
3. Results and conclusions - describe your results in constructing a better diode model. You must include a graph including all three model results like the graph above and a log plot of \( I_D \) vs. \( V_D \) which you used to estimate \( I_S \). You must also include a table showing the values of \( I_S \), \( r_S \), and \( n \) that you used in your "improved" Spice model.

### References