# CMPE 314 Spring 2011
Lab 0 Diode Circuit
Simulation

Using OrCad Capture or similar circuit schematic tool, construct a circuit which consists of a DC voltage supply and a diode in series. A diode model should be used which has simulation parameters. The Dbreak model in the BREAKOUT library can be used. Follow the procedure given below to begin Cadence.

- Start → Cadence→ SPB 16.01→ Design Entry CIS.
- In Cadence Product Choices, choose Allegro PCB Design CIS XL
- From file menu options choose New Project, Select Analog or Mixed A/D
- Provide a name for the project e.g. diode_simulation and save in S drive or in My Documents.
- Choose a blank project in the Create Pspice Project dialogue box.
- In the main project window click on the "place part" menu option .
- Add library from the following location:
    Cadence→SPB_16.01→tools→capture→library→pspice
- Add diode, breakout, source and analog libraries.
- Design the circuit given below and place a current marker.
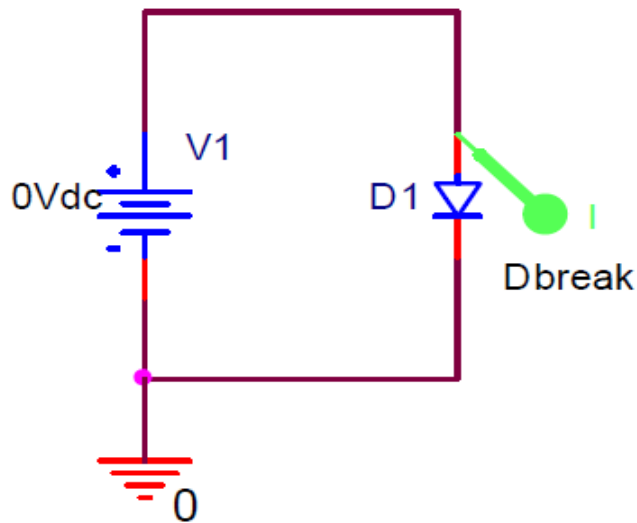
Now set up a DC sweep by first going to PSPICE->New Simulation Profile and then perform a DC sweep varying the voltage of the supply from 0 to 1.0 V. (To do that choose Analysis Type as DC Sweep, the sweep variable name should be same as your voltage source name.) The voltage increments should be small enough to get a good picture (Typically 0.01). The resulting waveform should look something like the following:
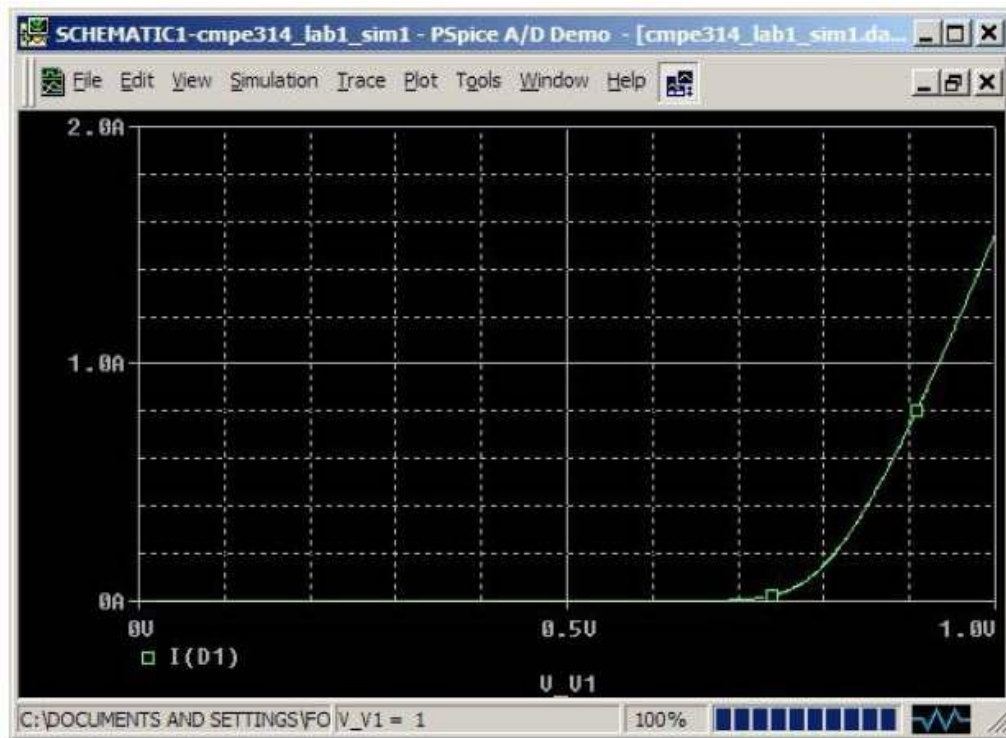
Capture the data as a text file. This can be accomplished from the simulation window (the one above) by going to File->Export->Text. After saving the text file, import the data into MATLAB. This is done by opening MATLAB and navigating to File->Import Data. After selecting the text file which you just saved, you should be in a wizard which will guide you through the rest of the process. Now fit your data to an exponential function. This can be done using the polyfit function built into MATLAB. The following MATLAB code snippet should lead you in the right direction: (use Help)

fit = polyfit (V, log(I), 1). Now plot the exponential function derived from the imported data and the analytical ideal diode equation on the same plot An example of what the plot should look like is presented below. In order to do this you'll first need to find Is (the saturation current for that particular diode). This can be done by right-clicking on the highlighted diode in the schematic and going to Edit PSPICE Model. You don't actually need to do any editing, but the new window should have a value for Is in it. Now you can model this in MATLAB as follows: Id = Is * (exp(V ./ 0.026) - 1), V is a vector of voltage values and the `./` operator says for MATLAB to divide each member of V by 0.026. I got V after importing the data by using the following MATLAB code: V=data(:,1) Next perform another DC sweep varying the voltage of the supply from 0.65 to 0.8 V. The voltage increments should be small enough to get a good picture. The resulting waveform should look something like the following:
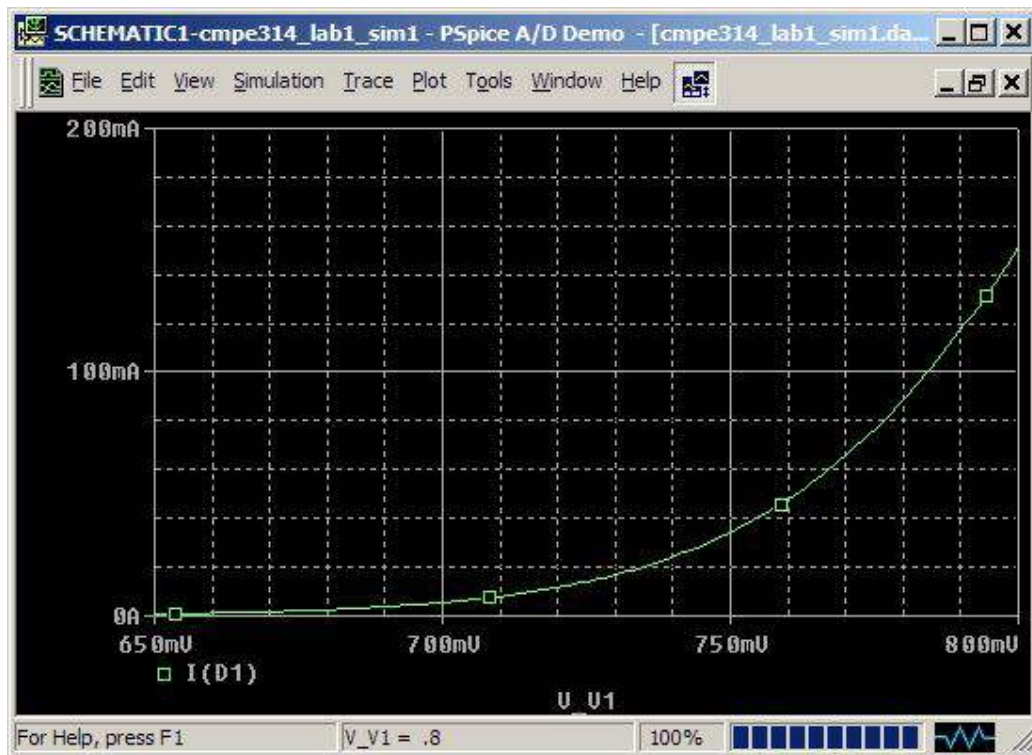
Capture and plot the data as a text file and repeat what was done for the first simulation. Your resulting plot should look like the following:
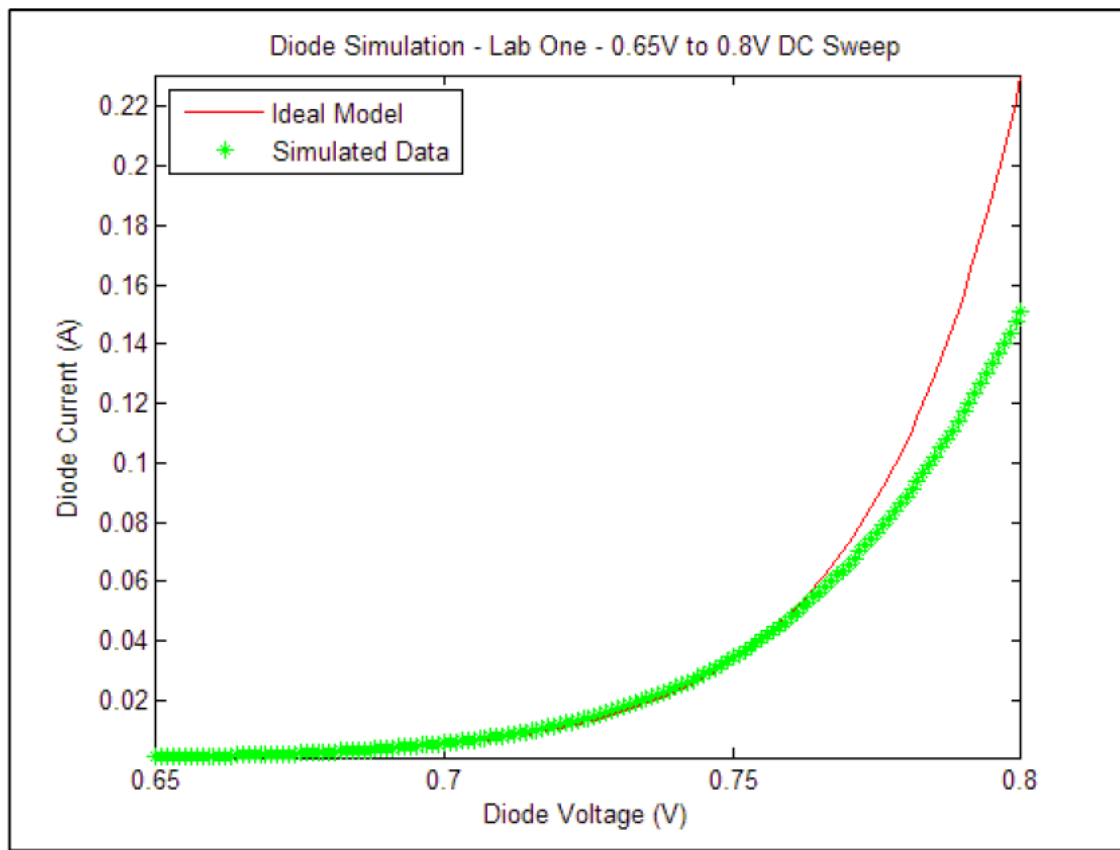
In both plots, you'll notice a divergence of the simulated data from the analytical. Include in your conclusions what you believe may be the cause of this deviation. Don't just guess at it though; analyze the situation. There is a parameter that was left out of the ideal diode equation above because it is usually assumed to be one, but can, in fact, vary. Look it up; it's called the emission coefficient. Your analysis might include analyzing the deviation and deriving the error in fitting the polynomial. For extra credit, you might explore ways to achieve better agreement between the experimental (simulation, in this case) and the analytical solutions. You might also decide to insert a resistor or a range of resistances (maybe do a parametric sweep) in series in the simulation circuit and analyze what happens.

Your lab report should include waveforms and plots. It should be neat and easy to read. All plots should be well labeled. Do not include unlabeled plots or reams of raw data. This is only confusing and uninteresting. Also, your plots should be in-line with your text, not appended to the report. Big plots that are the size of a sheet of paper are not necessary most of the time and look amateurish and lazy. This is the sort of thing you'll be turning in to management and presenting to colleagues if you become a professional engineer. In fact, a lot of engineering is writing reports and analyzing data. Treat this assignment accordingly.