Introduction

SPICE is capable of performing DC, AC and transient analysis of nonlinear circuits containing semiconductor devices such as diodes and transistors. Much of what you will learn about here also applies to transistors. Hand analysis of these types of circuits can be instructive, but perhaps a bit tedious and time consuming. Typically, many simplifying assumptions are made in the hand analysis of a transistor circuit to verify its basic operation. Subsequently, SPICE is used to improve the estimate of the performance of the circuit. SPICE is a circuit design tool and is not a substitute for a good understanding of circuits and devices.

In this lab you will measure device parameters that will be used in SPICE simulations following the session. If you study the format for a SPICE model for diodes and transistors, you will find that there are many (more than 10) parameters that SPICE can use to better describe real parts. You are going to obtain a few of those parameters in the lab and try to make your data agree with your SPICE simulation. During this exercise you will gain insight into the function of nonlinear circuits and the strengths and weaknesses of computer modeling.
Laboratory Tasks

Task 1 - Diode Characteristics: The I-V Curve.

Circuit A: $R_1 \approx 500 \Omega$ and $D_1 = 1N914$

(a) Select a simple silicon diode (e.g., a 1N914) and label it with a piece of masking tape for future reference.

(a) Using a DC power supply and two digital multimeters, measure and make a plot of the current vs. voltage in the range -1 volt to +1 volt. For forward biases choose currents that increase in a 1, 2, 5, 10 mA sequence. Use a series resistor of approximately $500 \Omega$ to protect the device, and do not exceed a forward current of $I = 100 \text{mA}$ even if this occurs before a forward voltage drop of 1 volt.

(c) Describe your results and fit them to the “Shockley” diode equation. Extract the value $I_S$, from your plot assuming the Shockley equation holds.

(d) Real diodes have series resistances of around $100 \Omega$. If you do not account for this when you solve for current, your results may be incorrect. Consider the following model:

\[ I_d \rightarrow D_{\text{Shockley}} \]

\[ V_d \]

\[ R_s \]
Using two data points, you should be able to determine both $R_s$ and $I_s$ for the actual diode -- 100$\mu$A and 10mA are two good current values to use for this type of diode. The following equation describes the model above:

$$V_d = V_T \ln \left( \frac{I_d}{I_s} + 1 \right) + R_s I_d$$

You can either obtain two data points using the suggested current values and solve the simultaneous equations, or you may be able to neglect one of the terms for one of the data points (realizing that $R_s$ should only be around 100$\Omega$ or so). Use both $R_s$ and $I_s$ in your SPICE models.

**Task 2 - Diode Characteristics: Large Signal.**

![Diode Circuit](image)

Circuit B: $R_1 \approx 1k\Omega$ and $D_1 = 1N914$

(a) Wire the circuit above and apply a ±10 volt sine wave with a 0 volt offset at $V_{in}$. With an oscilloscope, view the output across the resistor $R_1$.

(b) Draw, on a piece of linear-linear graph paper, the input ($V_{in}$) and output (voltage across $R_1$) waveforms you observe. Indicate and label the points at the maximum and minimum voltage. Explain the difference between the input and the output.
(c) Now modify your circuit to resemble the circuit above. View the output waveform as you did in part (a) using a ±10 volt peak-peak input signal.

(d) Draw, on a piece of linear-linear graph paper, the input ($V_{in}$) and output (voltage across $R_L$) waveforms you observe. Indicate and label the points at the maximum and minimum voltage. Explain the difference between the input and the output.

**Task 3: Diode Characteristics: Small Signal.**

(a) Apply the output of the function generator to your diode with a 100Ω series resistor as shown in circuit D. Put the ground probes and the ground of the function generator on the diode's cathode (the negative side in this circuit).

(b) Use the offset and amplitude controls to achieve a signal across the diode at 1kHz that has 0.5 (500mV) volts of offset and 20mV peak-peak signal on top of the offset. A function generator capable of doing this is the FG 501A Tektronics function generator with the -20dB button pushed in. You may
have better results if you use the 1x probes on the oscilloscope. Do not worry if you can not get the amplitude up 20mV, just get the correct offset voltage, and then get as close as you can to 20mV. The problem is that the -20dB attenuation in the function generator is attenuating both the AC signal and the DC offset.

(c) Measure the small signal resistance $r_o = \frac{dV}{dI}$ about a DC operating point $I_0$ by measuring the voltage across the resistor using the add/invert function of the scope to determine the current through the diode.

(d) Make two more measurements of $r_o$ at 0.4V (400mV) and 0.6V (600mV) of offset.

Task 4: Improved Half-wave Rectifier.

(a) Wire the circuit above and apply a ±10 volt sine wave with a 0 volt offset at $V_{\text{in}}$. With an oscilloscope, view the output across the resistor $R_L$. What determines the value of the resistor $R_L$?
(b) Draw, on a piece of linear-linear graph paper, the input \((V_{\text{in}})\) and output (voltage across \(R_L\)) waveforms you observe. Indicate and label the points at the maximum and minimum voltage. Explain the difference between the input and the output.

(c) Compare the performance of this half-wave rectifier with the one studied in Task 2 parts (a) and (b).

After Laboratory Tasks

**SPICE Simulation of Diode and Transistor Characteristics -- To be completed after lab session.**

This portion of the lab should be completed on your own after the scheduled lab time. With the parameters obtained from the circuits built and studied in lab you should be able to obtain results from SPICE that are very close to those you have measured.

**Extra SPICE Details**

A diode can be included in the circuit by simply using the format:

```
dxxxx <+ node> <- node> <model name>
```

A transistor can be included in the circuit by simply using the format:

```
qxxxx <collector nodej <base node> amitter node> <model name>
```

In addition, SPICE needs to have the proper parameters describing the electrical characteristic semiconductor diodes and transistors. This information is given by the “.model card.” For this assignment, use:

```
.model dmod diode is=<from lab>
```

The diode model is an ideal semiconductor diode with a saturation current value that you obtained in the lab procedure. See *A Guide to Circuit Simulation and Analysis Using PSpICE* for more detailed information and explanation.

For this assignment, you will need to perform a DC bias analysis that will step DC voltage sources over a range of voltages:

```
.dc <sourceb <start> <stop> ~step> <sourceb <start, <stop>atep>
```
The DC operating point will be computed for the range of values of source 1 and source 2. Source 2 is kept constant while source 1 is stepped through its range of values.

**Diode Characteristics**

(a) Using the above SPICE model for a diode, find $I_d$ vs. $V_d$ for $V_d = -3\text{v}$ to $+3\text{v}$. (use DC bias analysis and remember the SPICE model sets $I_s$ equal to the value you obtained in lab). What is $V_d$ for $I_d = 1\mu\text{A}$?

(b) Calculate the voltage at node 2 when $V_S=2\text{V}$ and $V_S=-2\text{V}$.

(c) Let $V_S$ be a sinusoidal input at 1000Hz and with an amplitude of $0.6\text{V}$. Use SPICE to find the voltage waveform at node 2. Compare the shape of the voltage waveform at node 2 and the current in the diode. Comment on the relationship.

(d) Let $V_S$ be a sinusoidal input at 1000Hz, with an offset of $0.5\text{v}$, and with an amplitude of $1\text{v}$, $100\text{mV}$, and $10\text{mV}$. Use SPICE to find the voltage waveform at node 2. Comment on the amount of distortion and the amplitude of the input.

(e) Use PSPICE to model the full wave rectifier circuit shown in Circuit C. Apply a $\pm10\text{v}$ peak-peak signal to the input and compare with results obtained from the actual circuit in lab.