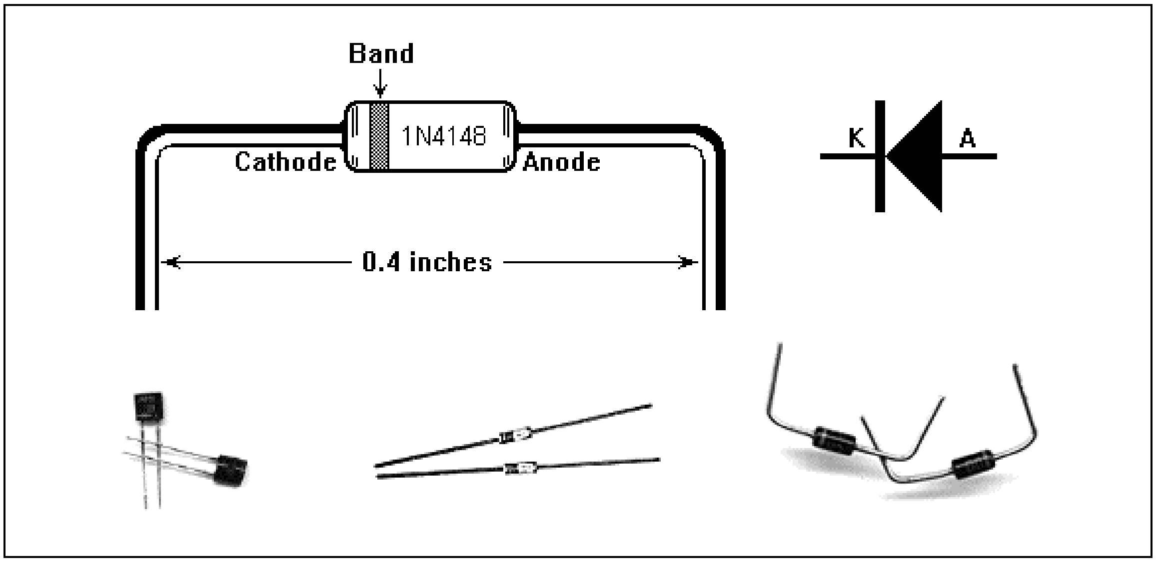


Engineering Sciences 154
Laboratory Assignment 2

Diode Characteristics

Laboratory Measurements and Semiconductor Circuit Simulations with SPICE

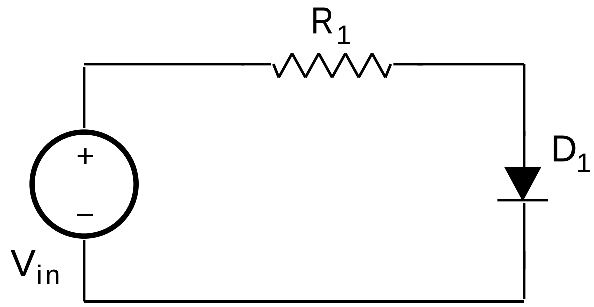
Introduction

SPICE is capable of performing DC, AC and transient analysis of non-linear 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.



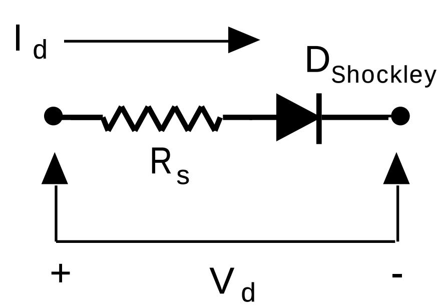
Circuit A: $R_1 \approx 500\Omega$ and D_1 = 1N914

(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 500Ω to protect the device, and do not exceed a forward current of I = l00mA 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 100Ω. If you do not account for this when you solve for current, your results may be incorrect. Consider the following model:

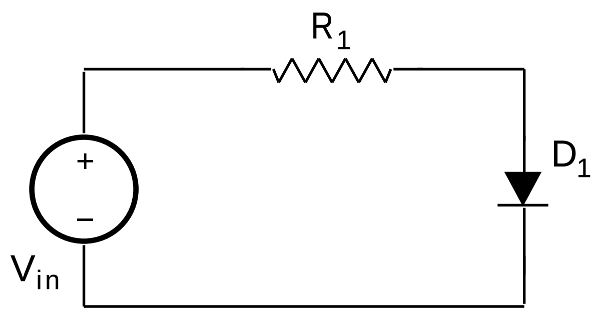


Using two data points, you should be able to determine both R_s and I_S for the actual diode -- 100μA and l0mA are two good current values to use for this type of diode. The following equation describes the model above:

$$V_d = V_T \ln\left[\frac{I_d}{I_s} + 1\right] + R_s\, I_d$$

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 l00Ω or so). Use both R_s and I_s in your SPICE models.

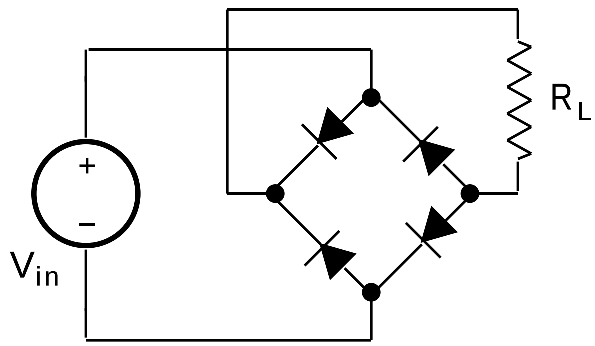
Task 2 - <u>Diode Characteristics: Large Signal</u>.



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.

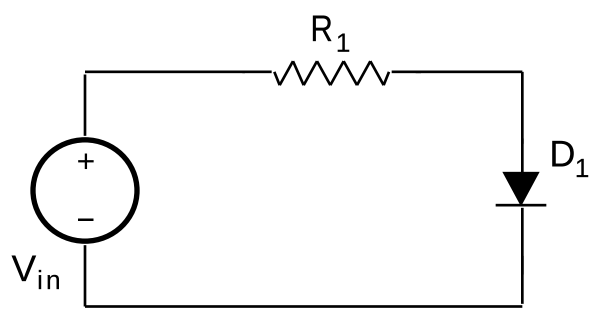


Circuit C: $R_1 \approx 1k\Omega$ and D's are 1N914s

(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.



Circuit D: $R_1 \approx 100\Omega$ and D_1 = 1N914

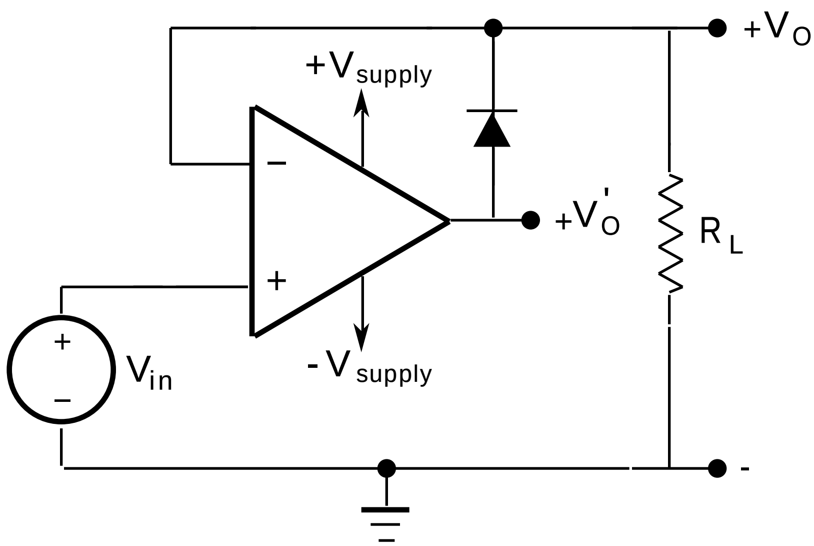
(a) Apply the output of the function generator to your diode with a l00Ω series resistor as shown in circuit D. Put the ground probes and the ground of the function generator on the diode's cathode (the negative side in this circuit).

(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 $r_o = dV/dI$ about a DC operating point I_o 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: <u>Improved Half-wave Rectifier</u>.



Circuit E

(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_L. What determines the value of the resistor R_L?

(b) 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.

(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 -- <u>To be completed after lab session</u>.

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.

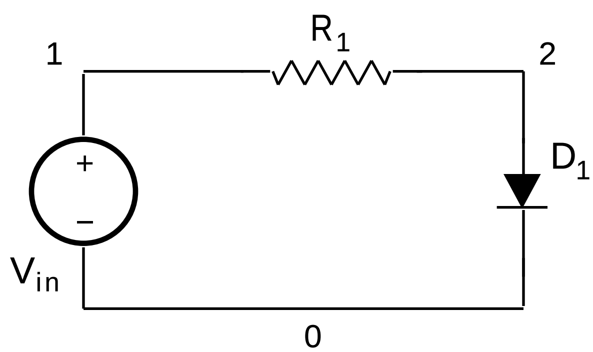
For this assignment, you will need to perform a DC bias analysis that will step DC voltage sources over a range of voltages:

.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 l 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 = -3v to +3v. (use DC bias analysis and remember the SPICE model sets Is equal to the value you obtained in lab). What is V_d



for I_d = luA?

Circuit F: $R_1 \approx 100\Omega$ and D_1 = 1N914

(b) Calculate the voltage at node 2 when VS=2V and VS=-2V.

(c) Let VS be a sinusoidal input at l000Hz and with an amplitude of.6V. 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 VS be a sinusoidal input at l000Hz, with an offset of 0.5v, and with an amplitude of 1v, 100mV, and 10mV. 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 ±10v peak-peak signal to the input and compare with results obtained from the actual circuit in lab.