**pn Junction Diodes HW (30 points)**

(jas, pn Junction Diodes HW.docx, 12/28/2024)

Clearly indicate answers and include units for numeric answers, using 3 or 4 significant figures. Show at least the major steps of your work so that if necessary partial credit can be awarded.

Utilize the Shockley Diode Equation, , along with the equation for Thermal Voltage for the following two problems, given n = 1:

1. Calculate the reverse saturation current Is for a diode operating at 10C, given ID = 2 mA at VD = 0.5 V. Use the calculated value for , where k is Boltzmann’s constant, TK is the temperature in Kelvin, and q is the magnitude of the charge of an electron or proton, i.e. . (Hint: Answer should be somewhere between 2 and 3 pA.) (3 points.)
2. Given the Shockley diode equation along with the relation that VT = kTK/q, derive the expression for the diode forward voltage VD in terms of n, k, TK, q, ID and IS. This result is to be used for question 6 below, so check your work to make sure your answer contains a constant term multiplied by a natural log term. (3 points.)
3. If you haven’t already done so, download and install LTspice from the Analog Devices web site using the following link: [LTspice](http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html). While LTspice offers a version for the Mac OS, it is not recommended because it is less documented and well known. Also the Mac version does not have some useful LTspice features such as Efficiency Reports for Switching Regulators. There is now an older and newer version of LTspice, with some nomenclature differences between the two versions. The older version included a **.op** for Operating Point icon in the top toolbar, whereas the newer version uses **.t** for the same function, which opens a **Spice Directive** window. A **Spice Directive** window can also be opened by choosing **Spice Directive** in the **Edit** pull-down menu in the upper left-hand corner of the LTspice window. A **Spice Directive** window is where simulation commands are entered for inclusion on an LTspice schematic. Consult the available LTspice Help as needed to become more proficient with this simulation tool used frequently in ECEN 350.

Using an LTspice schematic window connect the circuit shown below using the diode

1. symbol to place silicon pn junction diodes. After placement of the diodes, they are to be changed from the generic LTspice silicon pn junction diode to 1N4007 diodes. This can be accomplished by hovering the hand symbol over the diode and the right clicking to open a diode parameter pane, which includes a **Pick New Diode** button. The **Pick New Diode** button opens a **Select Diode** pane with many available diode simulation models, including the 1N4007. Sorting the list by clicking on **Part No.** will position the 1N4007 diode model either near the top or bottom of the list. Click on the 1N4007 Part No. and then hit OK to configure the diode symbol on the schematic to use the 1N4007 diode simulation model. The diode type can also be changed simply by editing the value from D to 1N4007 on the schematic, although becoming familiar with the LTspice parts libraries is helpful.

The independent current source is named **current** and can be found in the parts library that contained the independent voltage source. As a reminder, to access the parts library, click on the AND gate symbol  on the toolbar, which then opens the **Select Component Symbol** pane. Then choose **current** to place an independent current source on the schematic. Next add a **.step temp -55 125 10** parameter sweep statement by means of **Edit** 🡪 **Spice Directive**. The **step temp -55 125 10** parameter sweep statement causes the operating point to be determined at 10 different temperatures ranging from -55 ˚C to +125 ˚C. Add the net name **Vd** to your schematic as shown below, by means of the Net Name icon  on the top toolbar.

A diagram of a step temp

Description automatically generated

Simulate the resulting constant current biased diode circuit and then determine the temperature coefficient (**TC**) of the diode forward voltage by displaying the diode forward voltage **V(vd)** in the plot pane.

Note: LTspice uses a dark background for the default plot pane, which makes it hard to visibly see some trace colors such as dark blue. The **Tools** 🡪 **Control Panel** 🡪 **Waveforms** 🡪 **Color Scheme** 🡪 **Background** allows you to modify the background color with Red = Green = Blue = 255 resulting in a white background. This is an optional change that you may find improves waveform viewing. If you make this change, it is also recommended to navigate to **Tools** 🡪 **Control Panel** 🡪 **Waveforms** and change the **Data Trace** and **Cursor width** from 2 to 3, along with using a Bold Font for the waveform labels so that the traces, cursors are waveform labels are more easily visualized.

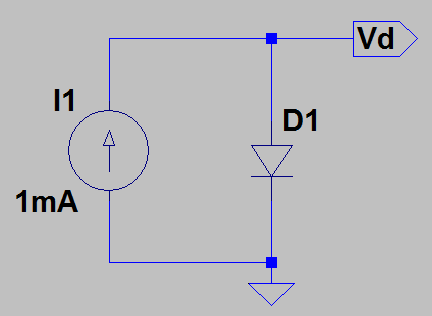
The resulting TC of the constant current biased diode can be easily obtained in the plot pane using **Cursors**. To activate cursors, hover the mouse pointer near the **V(vd)** trace label located near the top of the plot pane. When the hand icon appears, right click on it and an **Expression Editor** box opens, which includes an **Attached Cursor** option. Choose the **1st& 2nd** cursor option, which will attach two movable cursors to the **V(vd)** versus temperature trace in the plot pane. With the mouse, drag the cursors apart so that one cursor is at or near the coldest temperature and the other at or near the hottest temperature in the plot pane. A Cursor box appears when cursors are added, indicating the Horizontal and Vertical values at each cursor location, along with the difference values between cursors and the slope between the cursors. The TC of the diode forward voltage VD is calculated as ∆VD/∆Temperature and can be read directly from the cursor box as the Slope, displayed in the lower right corner of the box in units of V/˚C between the two different located cursors. Note the slope is the same independent of which cursor is placed at the coldest temperature and which is placed at the hottest temperature. Drag the cursor box onto the plot pane to document your simulated values. Also annotate your plot pane with your name by means of the **Plot Settings** → **Notes and Annotations** → **Place Text options** available by accessing the Plot Settings pulldown menu above the toolbar. Replace the resulting plot pane below with your version. Note: You can adjust the aspect ratio of the Plot Pane in LTspice so that it is taller and narrower, as was done for the plot pane below. (14 Points.)

A graph on a screen

Description automatically generated

1. Based on your LTspice Plot Pane results, record the simulated Diode Forward Voltage Temperature Coefficient (TC) below using 2 significant digits, including the sign and units.

**Simulated Diode Forward Voltage TC equals \_\_\_\_\_\_\_\_\_\_.** (3 Points.)

1. The adjacent current sourced biased diode has a forward bias diode voltage Vd = 0.6 V at 0 ˚C, and a diode voltage TC of –2 mV/˚C. Determine the operating temperature of the current sourced biased diode when the diode has a forward voltage of 0.46 V, using at least 2-significant digits. Include at least two major steps of your calculations below. (4 points total. 2 points for including at least 2 steps.)

**Diode Operating Temperature for Vd = 0.46 V equals \_\_\_\_\_\_\_\_\_\_.**

1. Based on the expression you previously derived for the diode forward voltage, what parameter is responsible for the resulting negative TC of the diode forward voltage VD for a constant ID by means of a current source (1 point) and why (2 points.)?

(3 points total.)