Using Multisim (14.0 or 14.1) for Lab 1: Diodes

First, extract the parameters for I_S and n experimentally. Recall that you want to use data from measurements where the effects of depletion region recombination (low currents,) and high injection and series resistance (high currents) are not significant.

You will use Multisim to simulate the measurements you made and compare the simulations to the measured results. The comparison will give you a good idea of the usefulness of the model and the accuracy of the parameters.

1. Open Multisim (for reference, I am using Multisim 13.0.1)
2. Place components; from the Master Database
    a. Group: Sources; Power Sources-DC Power and Ground
    b. Group: Basic; Resistor-1k
    c. Group: Diodes; Diodes_Virtual; Diode
3. Connect the components as shown below. Note that I have opted to show all net names, which are in this case node numbers. Node 0 is always ground. You can do this with Options->Sheet Properties, Net Names, Show All.
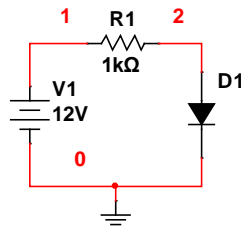

4. Double-click on the diode; then select the Edit Model button. Uncheck the Use default box for the saturation current and the emission coefficient. Below, I have entered 100 pA and 1.72 for the two parameters. Note that you can use engineering prefixes in this dialog box.
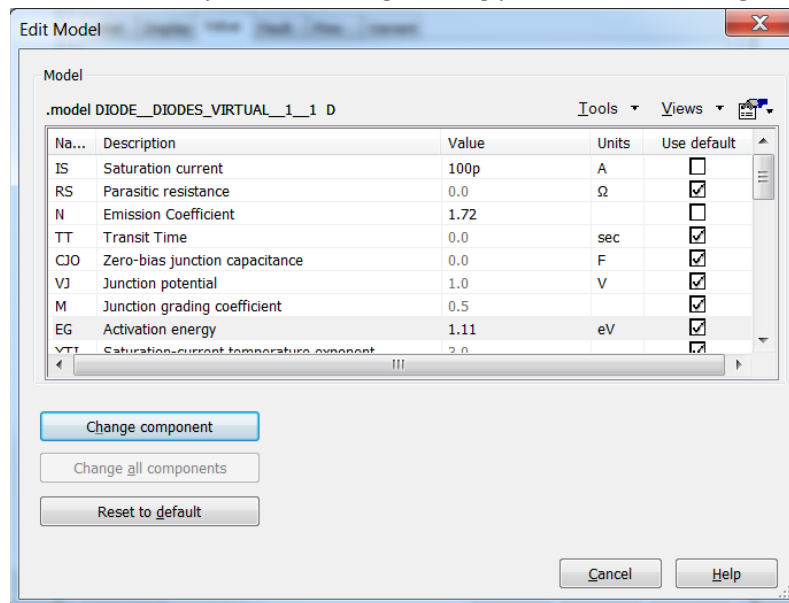

5. Click the Change component button; select OK.

6. Select Simulate: Analyses and Simulation, DC Sweep. On the DC Sweep Analysis parameters tab, set source V1 to start at 0.5 V and stop at 5 V, using increments of 1 mV. Note that you will have to enter either 0.001 V or 1e-3 V. Multisim does not, in general, understand engineering prefixes.
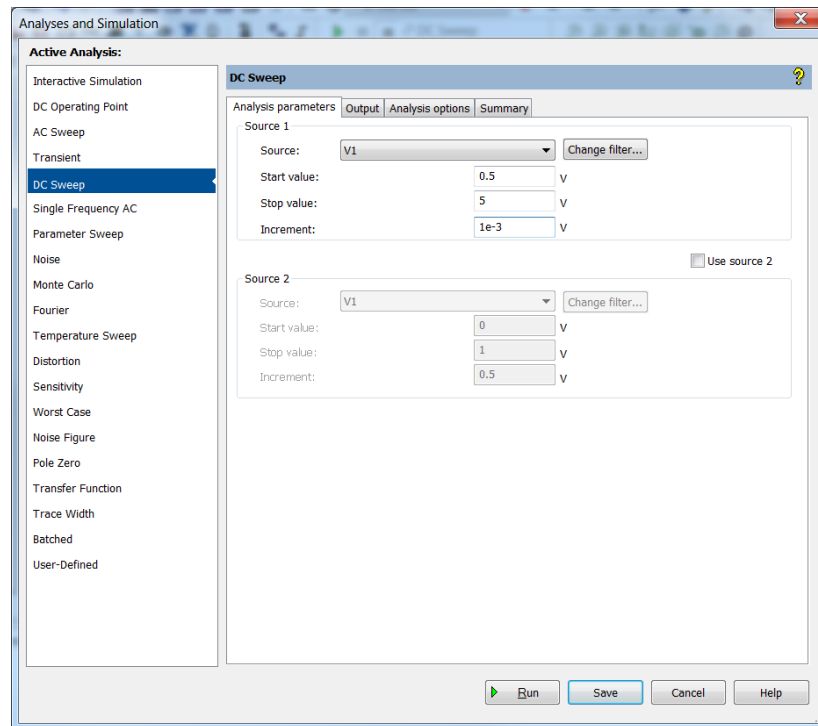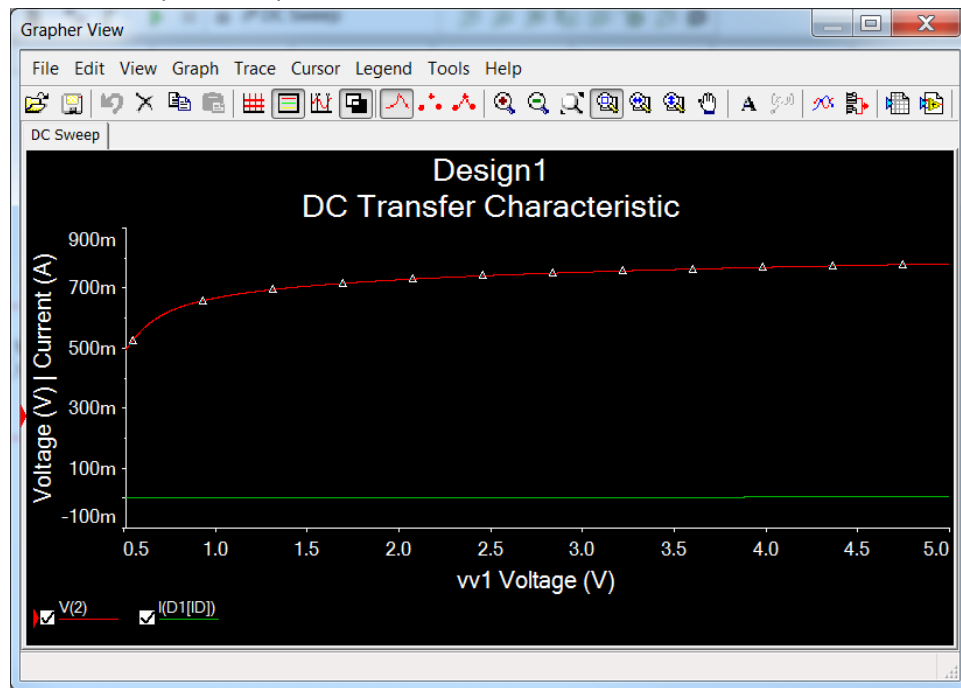

7. Switch to the Output tab. Add the diode current (also resistor current), and diode voltage, V(2) in this case, to the outputs. You can see why I chose to show the net names now.
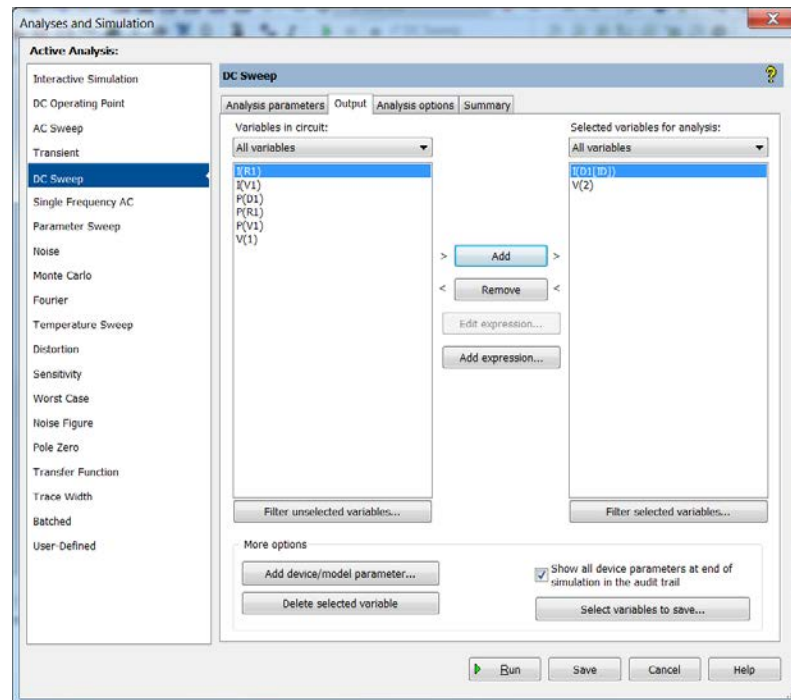
You can then select the Analysis options tab. In this case, there is nothing needed here. Finally, click the Run button at the bottom of the window.

8. The simulation will open a Grapher View.

There are several problems here. Change the background to white by clicking the control with both black and white boxes. Turn on the grid. You will notice that the two traces, diode voltage and diode current use the same axis on the left. The scale is not appropriate for either one. Double-click in the area of the graph and select the Traces tab. Move the trace for the current from the left axis (the default) to the right axis (y-vertical axis). Then select the Right axis tab and enable the right axis and select Auto-range. Then Auto-range the left axis. It would also make sense to relabel the axes at this point.
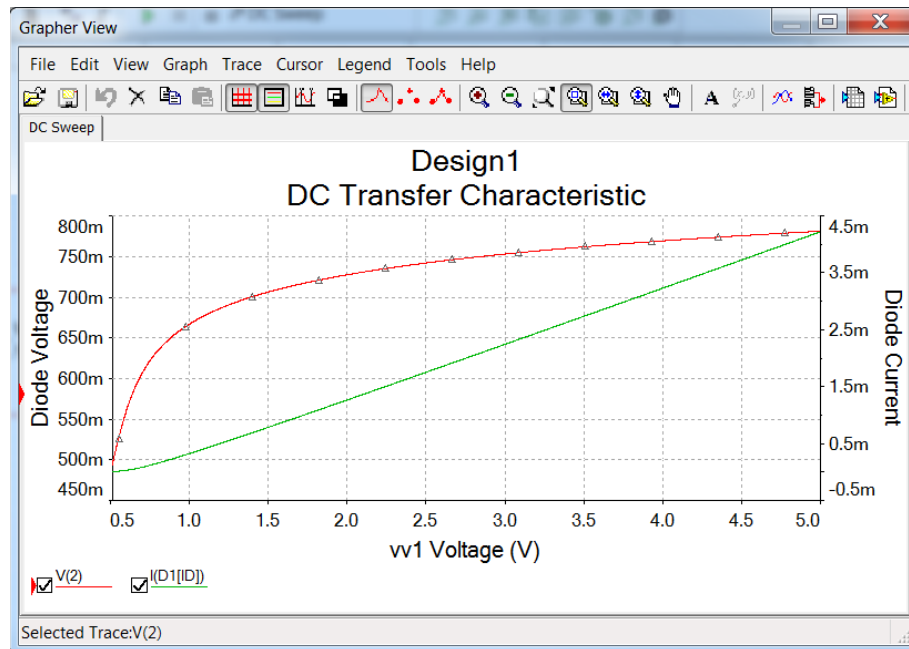
9. The plot shows the diode voltage and current as a function of the DC supply voltage. Unfortunately, that is not quite what we need, which is the diode current as a function of the diode voltage. There are two options at this point: 1) We could export the data to another program like Excel and create the desired plot, or 2) We could adjust the circuit to give make the voltage sweep variable be the diode voltage itself.

10. Choosing the second option, we note that the diode voltage varies from around 480 mV to about 780 mV. Go back to the diagram and remove the resistor. Reconnect the circuit. Select Simulate: Analyses: DC Sweep. Set the V1 source to vary over the range given above. Make sure the diode current is still in the output list. Then Simulate.

11. Fix the graph up again (white background and grid). Then change the left axis (now for current) to logarithmic.
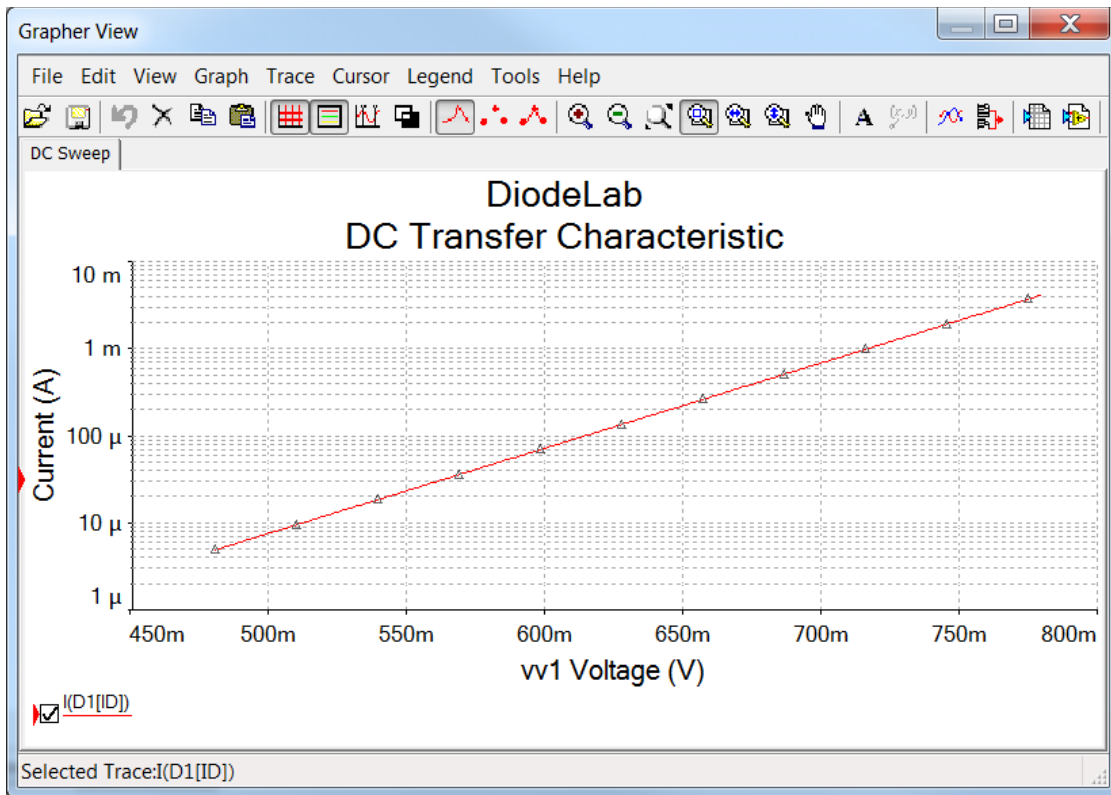
12. Compare the graph shown to your measurements.

Multisim can perform a wide variety of analyses. "**DC operating point**" analyzes a circuit containing non-linear devices to find the DC voltages and currents. These are important in determining how the non-linear devices will function when (small) changes are made to the DC values. "**AC analysis**" performs analysis over a range of sinusoidal input frequencies. "**Transient analysis**" is useful for determining how the circuit will respond to non-sinusoidal signals. Note that AC analysis is performed using a linearization based on the DC operating point. If linearity is not maintained, by using small inputs, the results will not predict how the circuit will perform in practice.

The next section of the lab requires you to compare the regulation offered by two different circuits, one consisting of five 1N4002 diodes in series and the other with a 1N4279A Zener diode. Recall that the regulation is the change in output voltage divided by the change in the input voltage, usually expressed as a percentage. The two circuits are both drawn in the same schematic below.
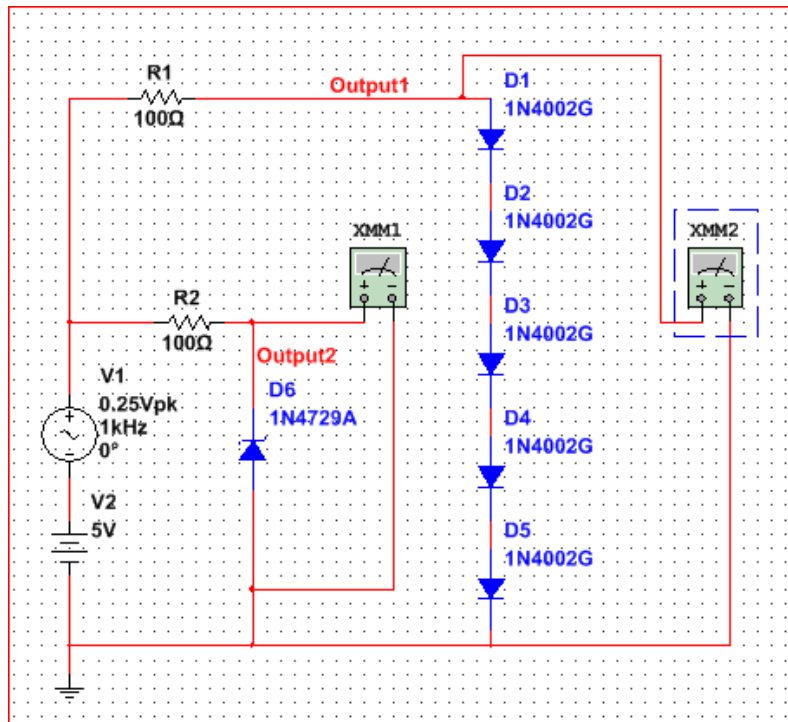
I have put two multimeters into the circuit to determine the DC voltages. Remember that DC voltages and currents will be important in determining how the diodes perform. By selecting Simulate: Run or clicking the green arrow a simulation will begin. Double-clicking on the two meters displays their measurements, as shown below.
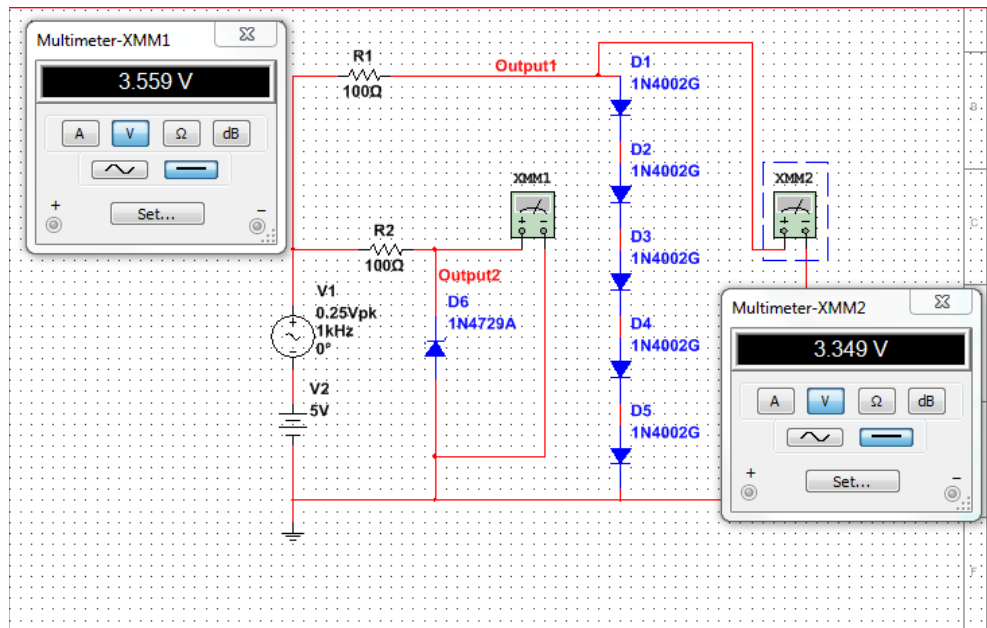
But this is not enough information to determine the regulation, which depends on changes. One way to determine the changes is to switch the meters from DC measurements to AC measurements.
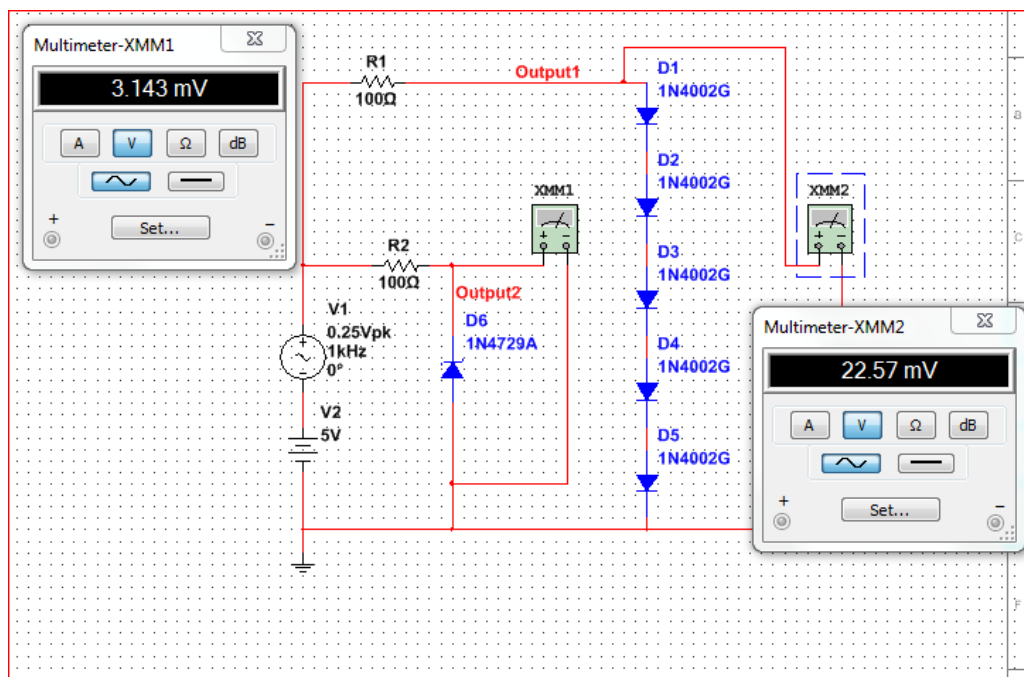
I am still not sure of what the meters are showing. I don't know if these are rms values, peak values or peak-to-peak values.

I need a different tool to see that. I will use the transient analysis tool to make a clearer measurement. To prepare for the measurement, I will label the three nodes that I am interested in.

Then, stop the interactive simulation if it is still running by clicking the red stop box. Select Simulate: Analyses and Simulation: Transient. On the Transient Analysis tab set the start and stop times and the maximum time step. My time-varying source is a sinusoid of 1 kHz, so I used 1 µs as the maximum time step. To see several cycles of the sinusoid I chose a start time of 0 and a stop time of 10 ms. Note again that I had to enter 0.01 s. On the output tab, select the three named nodes. Then Run.
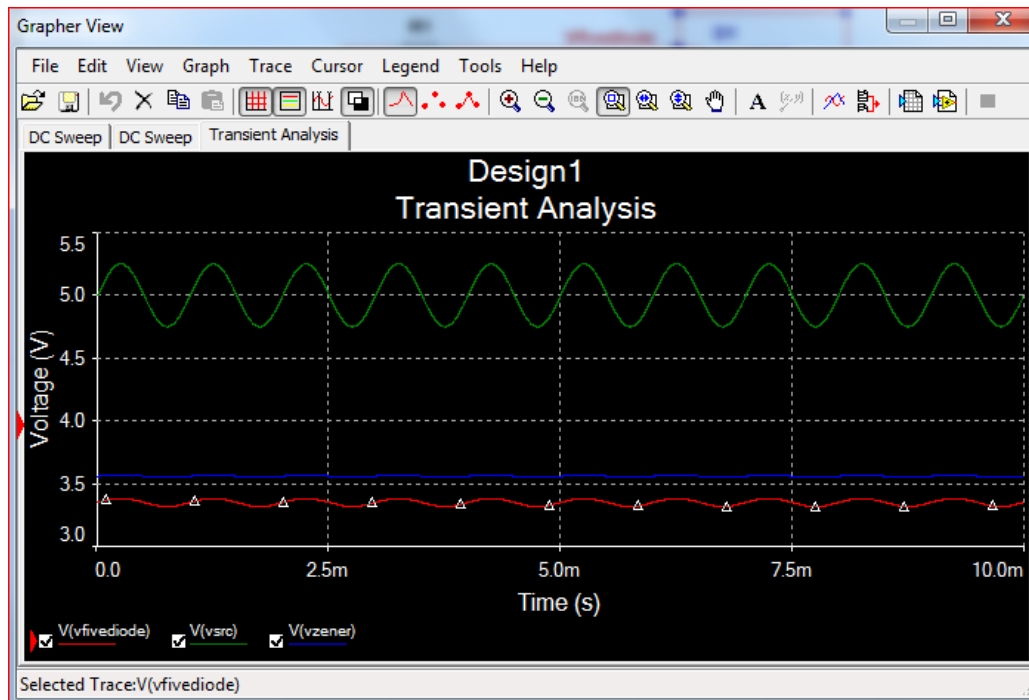

The information in the green trace (the source voltage) is already known, and it is preventing me from seeing what I really need to see, the variations in the blue and red traces. Unchecking the trace for the
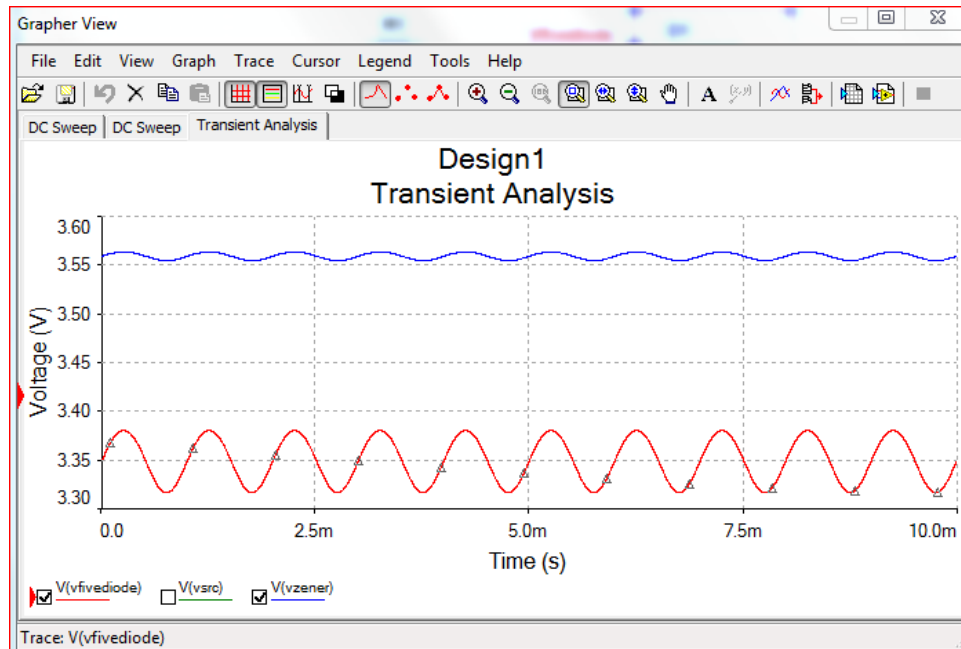
But it will still be hard to measure the amplitude of the trace for the Zener. I turned on the cursors and found that the five diodes gave a peak-to-peak voltage of 63.8047 mV, while the Zener gave a peak-to-peak voltage of 8.8940 mV.
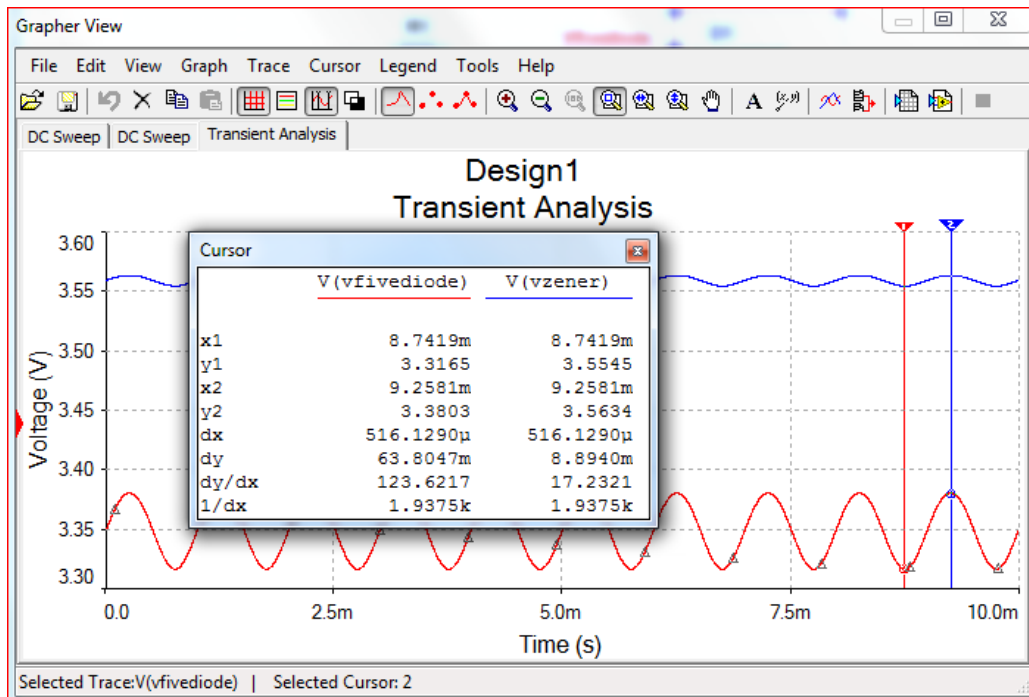
Notice that these do not seem to match with the multimeter measurements above. Recall however that the peak value of a sinusoid is the square root of 2 times the rms value and the peak-to-peak value is

twice the peak value. Assuming that the multimeter measurements are rms, they would give 63.838 mV and 8.8897 mV, pretty close to the measurements from the graph.

The final thing to do is calculate the regulation. I know that the input amplitude is 500 mV peak-to-peak. Therefore the regulation of the five-diode circuit is 12.76% and the regulation of the Zener circuit is 1.779%.

The conclusion from the simulation is that the Zener circuit is much better, providing a regulation that is less than one sixth the regulation provided by the five-diode circuit.