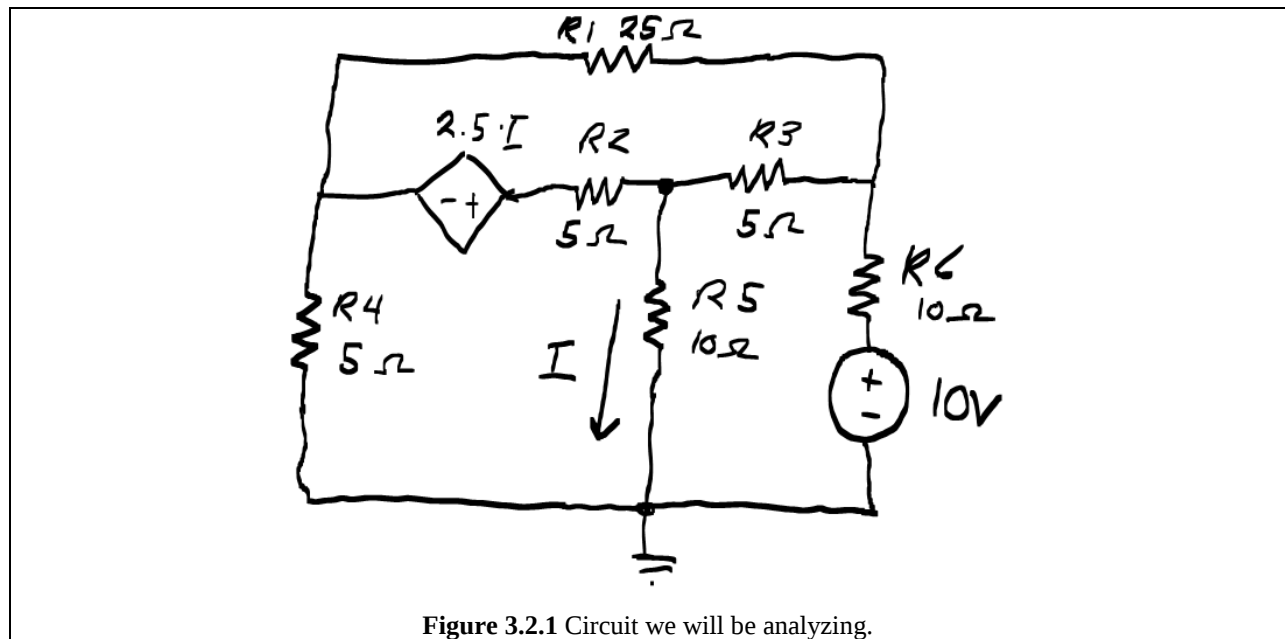


MULTISIM DEMO 3.2: DC CIRCUIT ANALYSIS II

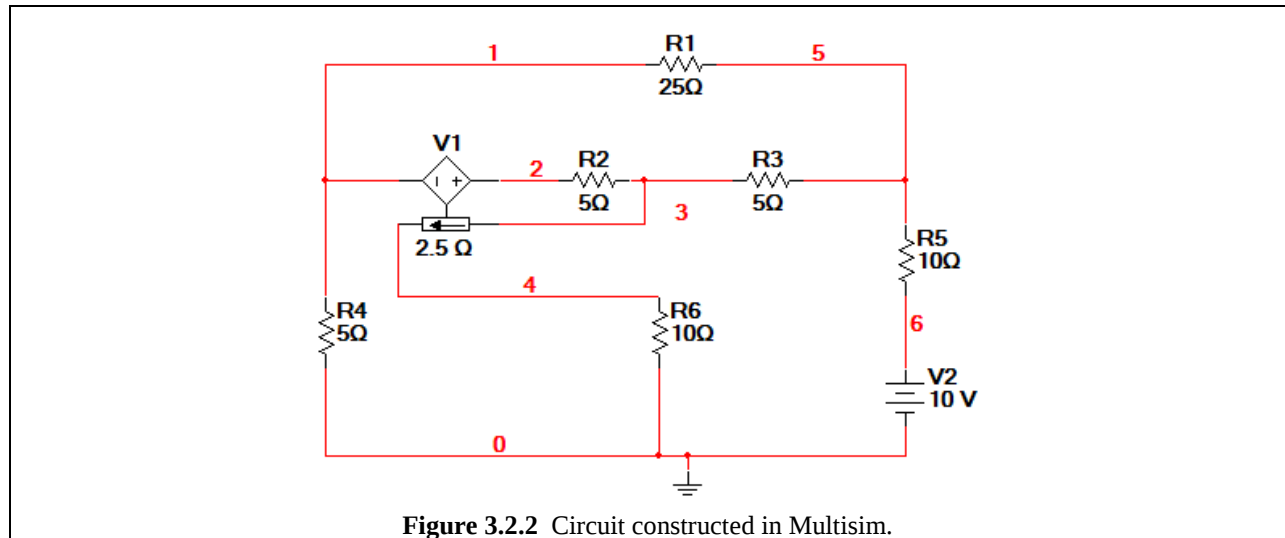
In Multisim Demo 3.1, we analyzed a DC circuit using the Measurement Probes in the Interactive Simulation. There is another way to get DC values called the DC Operating Point Analysis.

All of the simulations which we have run thus far in this tutorial (Demos 2.1 through 2.5 and 3.1) have been Interactive Simulations. These are a sort of real-time simulation of a virtual lab complete with instruments. We “begin time” by pressing F5 or the appropriate button or toggle switch and use instruments or tools to measure circuit parameters as if we were in an actual lab. We “end time” by pressing the stop button.

There is another way to analyze circuits in Multisim. It is through using what are officially called “Analyses.” Analyses are predefined ways to analyze circuits. In many ways they are the more traditional methods used to analyze a SPICE circuit. They include things like parameter sweeps, frequency sweeps, DC operating point sweeps...things which don’t need to be done in a interactive virtual environment. We’ll begin our study of Analyses by working with the DC circuit shown below in Fig. 3.2.1.



When you build the circuit, let’s use a CURRENT_CONTROLLED_VOLTAGE_SOURCE just to get some practice in implementing it. When completed, your circuit should resemble that in Fig. 3.2.2 on the next page.

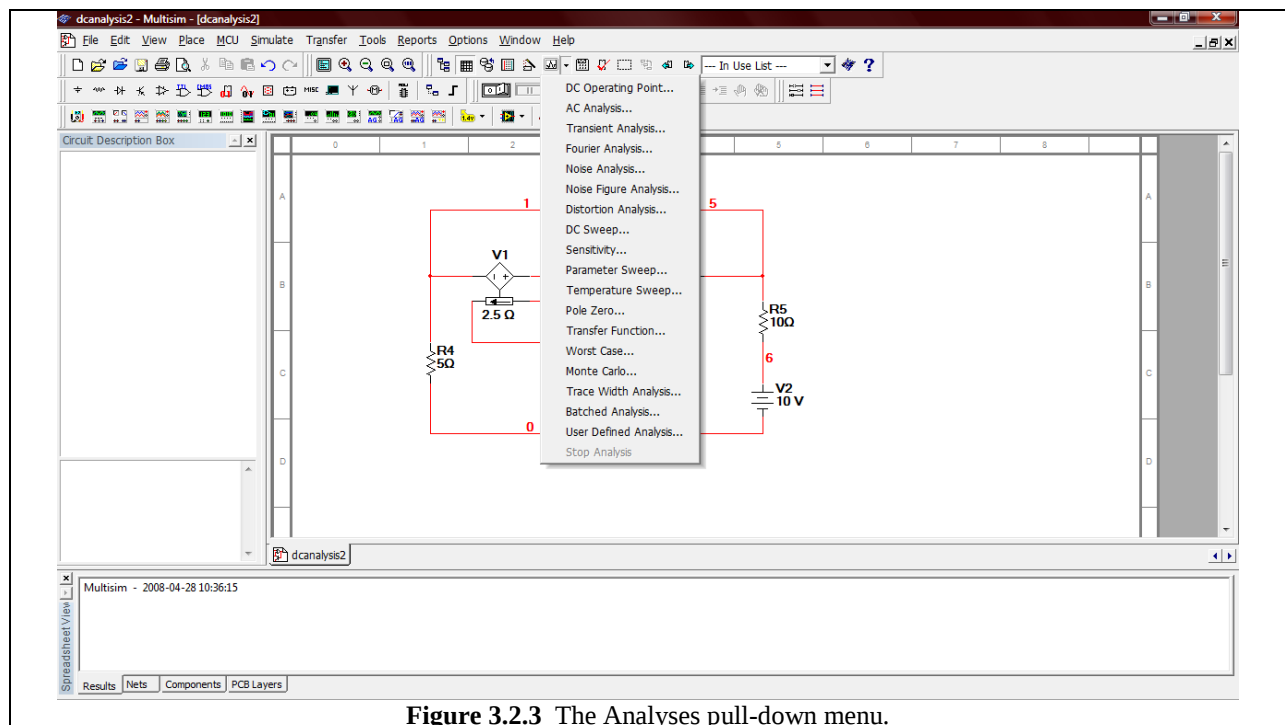


So now how do we conduct one of these so called “Analyses”?

Go to Simulate>Analyses>DC Operating Point...

or

Click on the arrow portion of the  found on your toolbar button to bring out a menu of which should resemble what is shown in Figure 3.2.3 below.



Click on the DC Operating Point option. It will bring up the window shown in Figure 3.2.4 below.

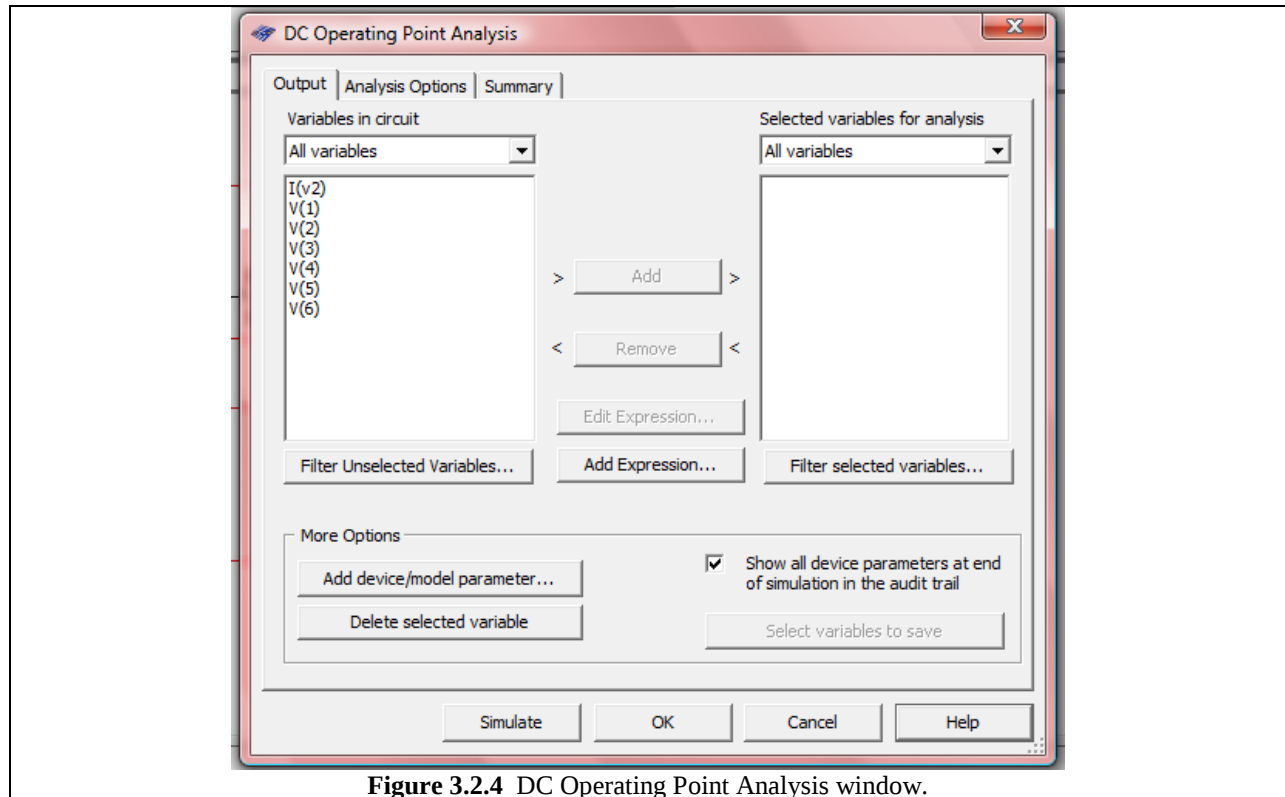


Figure 3.2.4 DC Operating Point Analysis window.

On the right side of the screen under the Variables in circuit title is the list of all variables available in the circuit. Notice that there are six voltage variables, indicated by V(1), V(2),...V(6). These correspond to the node numbers in the schematic found in the Schematic Capture screen.

There is one additional variable visible, I(v2), which is the current through (into the positive terminal of the voltage supply) the voltage source. The direction of this current is demonstrated pictorially in Figure 3.2.5 below.

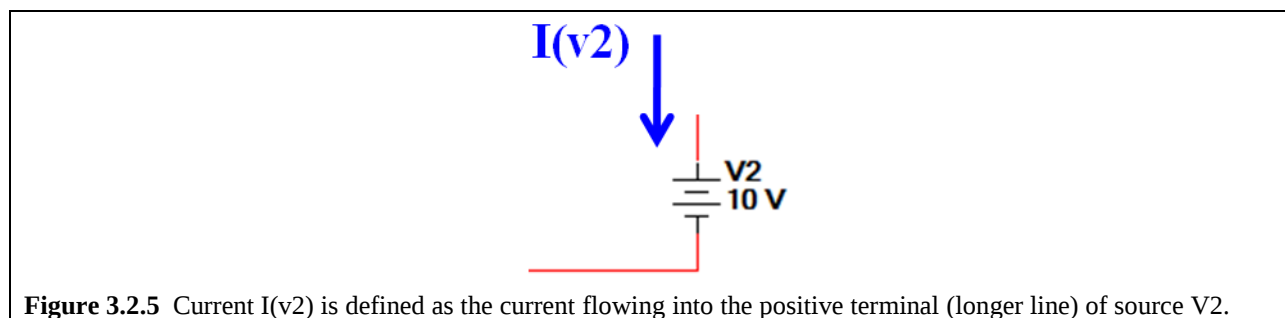


Figure 3.2.5 Current I(v2) is defined as the current flowing into the positive terminal (longer line) of source V2.

Now you may ask, of all the currents in the circuit, why is it that only the current flowing in that single branch, I(v2), is the only current which is available? Well the answer is pretty general...

If we wanted to know the current in a particular branch, what would define which way it is pointing? For example, if we wanted to know the current through Resistor R3 in Figure 3.2.2 above how would it be defined? Would it be left to right...or right to left? Flipping the circuit would change variable names. We could do it from node to node...in which case the current would be called $I(R3, V3, V5)$, but this could become bulky. As a result, in Multisim and SPICE software in general, the way to measure current in simulations and to get a variable which we can manipulate, is to place a 0 V voltage source into the circuit where we want to measure the current. Let's use the example of R3 in Figure 3.2.2. We will place a 0 V source in series with the resistor. The result should look like that shown in Fig. 3.2.6.

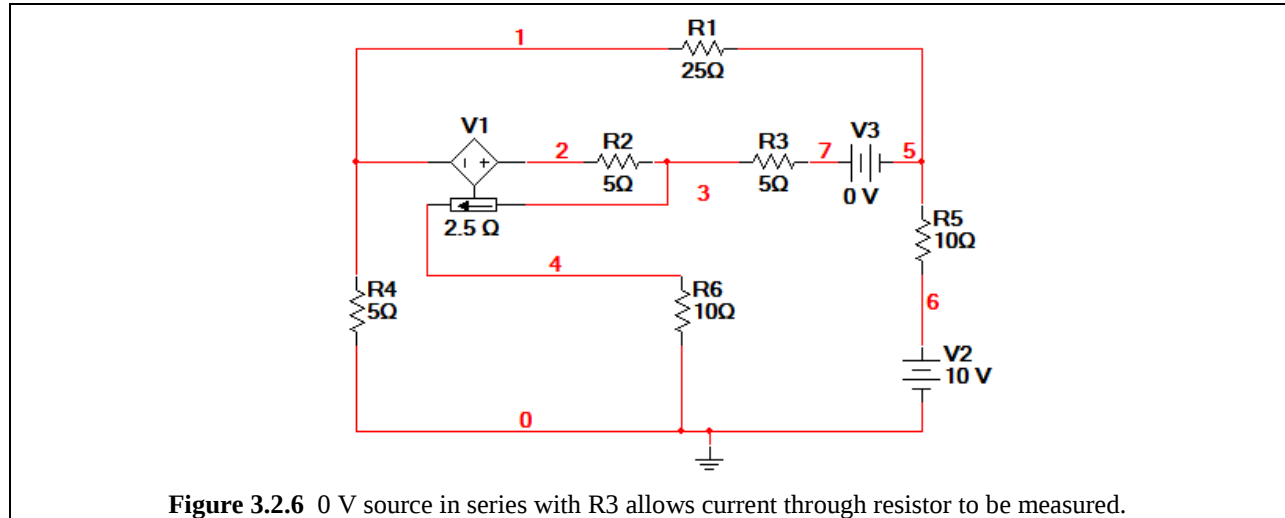


Figure 3.2.6 0 V source in series with R3 allows current through resistor to be measured.

So now when we go to do a DC operating Point Analysis, the window will look like Fig. 3.2.7.

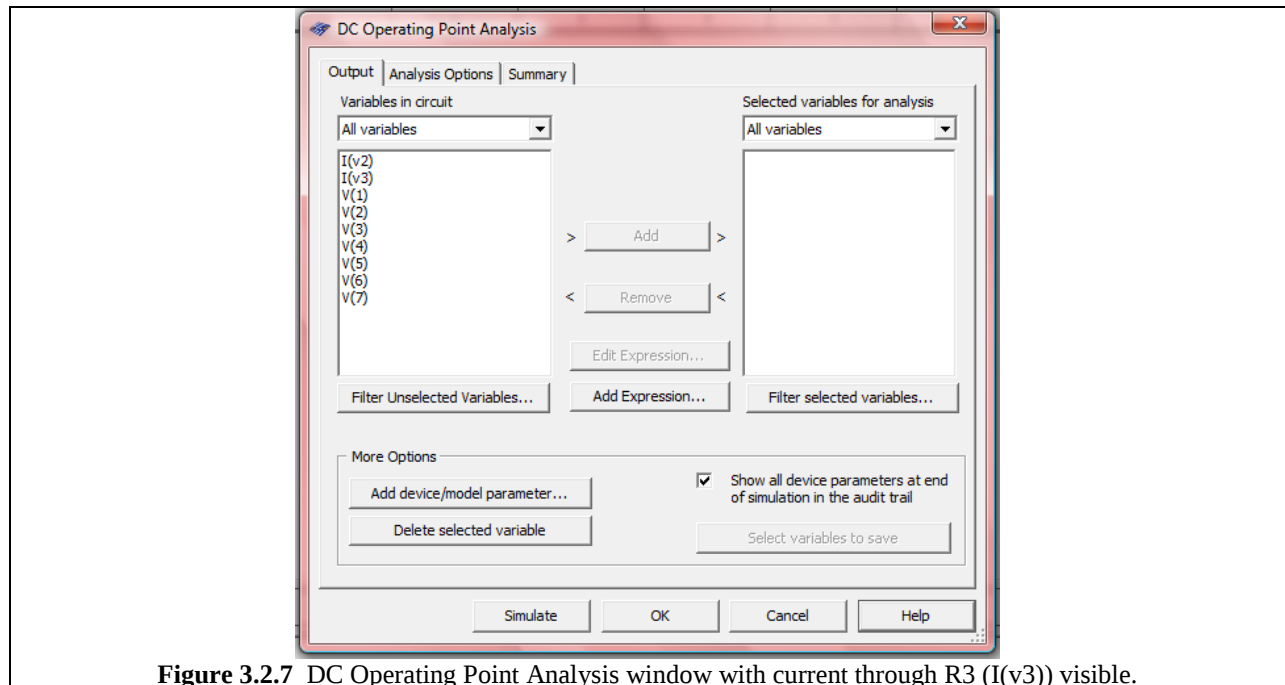


Figure 3.2.7 DC Operating Point Analysis window with current through R3 ($I(v3)$) visible.

Note that in Fig. 3.2.7 we now have two currents: $I(v2)$ and $I(v3)$. $I(v3)$ is the current defined as that moving from Node 7 to Node 5, which is the same thing as the current moving from Node 3 to Node 7 (which is the current moving through the resistor R3.) We now have that current as a variable which we can plot.

We will do more with this in a little bit.

Now we still haven't done any simulations, so to do that, let's go and select some variables for analysis. To do this, in the DC Operating Point Analysis window, click on one of the variables on the left in the "Variables in circuit" field and press the "> Add >" button. This will move the variable over to the field under the title "Selected variables for analysis". If you want to get rid of a variable from the analysis field, simply click the "< Remove <" button. This will remove it from the analysis.

Let's analyze the current through R3, the voltage $V(2)$, and the voltage across R1. You should be able to do the first two of these easily. Click $I(v3)$ and then click Add, and do the same for $V(2)$. The voltage across R1 is not a simple variable presented up there, however...it is actually a difference of two variables. To get this difference, we need to go to the Add Expression button found in the middle of the window. Clicking on that button will bring up the window shown in Figure 3.2.8.

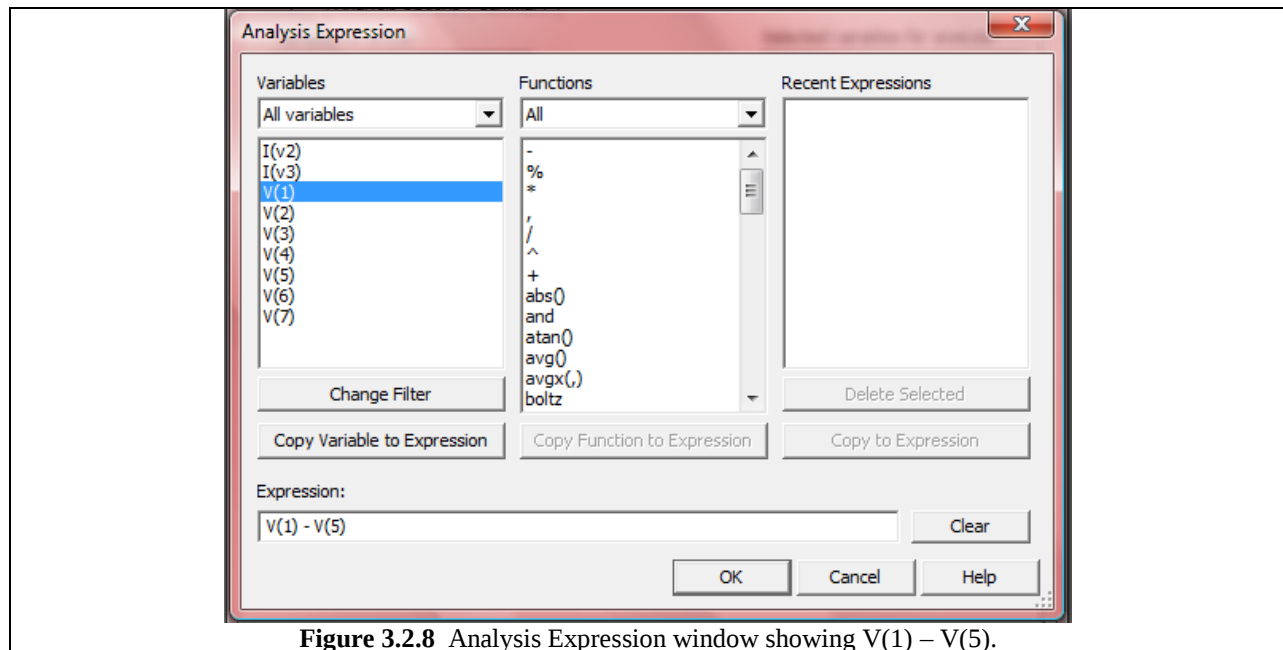


Figure 3.2.8 Analysis Expression window showing $V(1) - V(5)$.

The window shown in Fig. 3.2.8 allows you to create expressions using the variables from the circuit. Since the voltage across R1 is not determined by a single variable, but instead the difference of the two of the know variables, we can type in an expression to get that value. The voltage across R1 is simple $V(1) - V(5)$.

REMEMBER: The variable names used in this Demo are for the author's circuit. When you build the circuit, the appropriate nodes may be numbered differently depending on the order in which you placed your wires. You can rename them to correspond to the examples presented here, but it is not necessary. Just make sure that the variables correspond to your specific schematic and not the one presented here!

To enter this value:

1. In the Expression: field type: $V(1) - V(5)$.
2. Press OK. Your DC Operating Point Analysis window should now look like that shown in Figure 3.2.9.

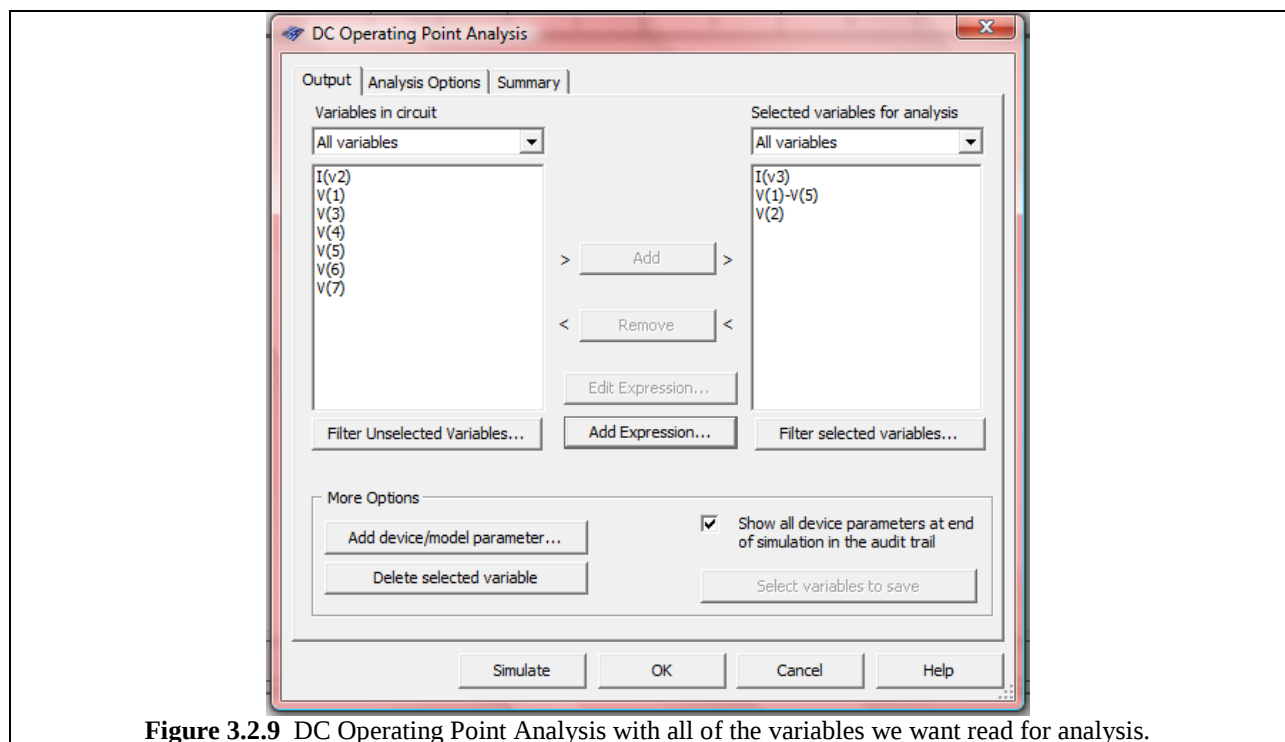


Figure 3.2.9 DC Operating Point Analysis with all of the variables we want read for analysis.

All that is left for us is to press the Simulate button. Doing so will bring up the window shown in Fig. 3.2.10 on the next page. As you can see, the voltage $V(2)$ is 2.000 V, the current $I(v3)$, that is the current going from node 3 to node 7 in $R3$ is -400.00 mA, and the $V(1) - V(5)$, that is the voltage across $R1$, is -3.333 V.

Go ahead and use the DC operating Point Analysis to generate different circuit values. By adding other 0 V sources you can measure the current in any branch of the circuit or any circuit you choose to analyze.

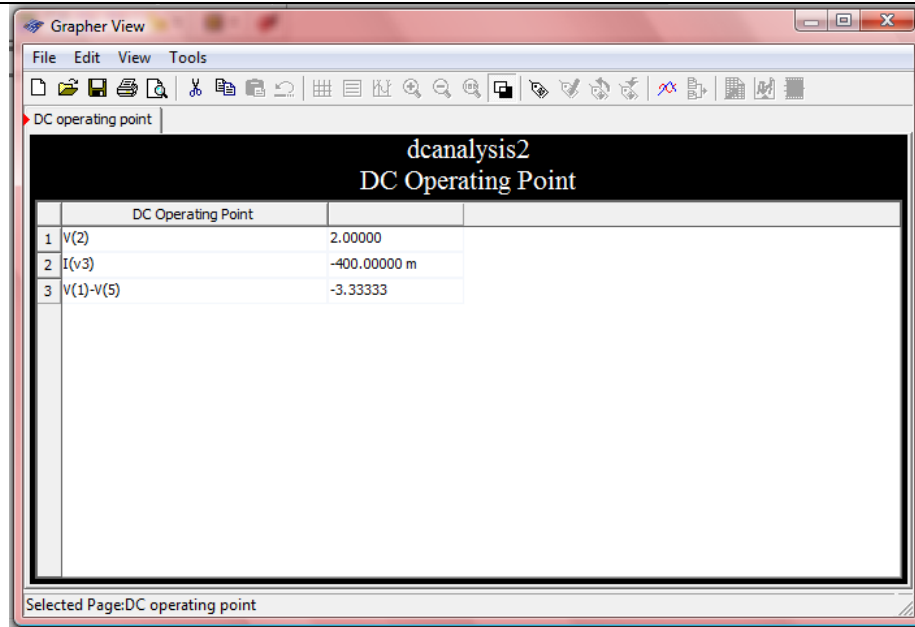


Figure 3.2.10 Output of the DC Operating Point Analysis.