EXPERIMENT 1
Introduction to MultiSim: DC Analysis

Bei Zhang and Suraj Sindia
Revised by Elizabeth Devore
May 2016

The objectives of this session are to:
- Learn how to write a good lab report
- Help students become familiar with the basic features of MultiSim, a circuit simulation software tool
- Provide an introduction to computer simulation of dc circuits using MultiSim

1 Introduction

This lab experiment is designed to introduce you to the SPICE program to be used throughout the semester and how to make basic electronic measurements. The key to a good lab report is the proper use of figures, tables, lists, calculations, and having just the right amount of explanation, not too little or too much. Record and explain your experiment processes and results in a way that an Electrical Engineer without access to the lab manual could understand what you did.

2 Introduction to MultiSim

National Instruments (NI) MultiSim is used in academia and industry for circuits education, electronic schematic design and SPICE simulation. It is an electronic schematic capture and simulation program, similar to other SPICE programs that can model the behavior of a particular analog or digital circuit. It is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation tool. In MultiSim, the engineer has a tool can model almost any conceivable circuit design, examine the corresponding circuit for values at particular components or probe the behavior of the entire circuit by performing dc, ac, or transient analyses, and much more.

MultiSim obeys the Passive Sign Convention:
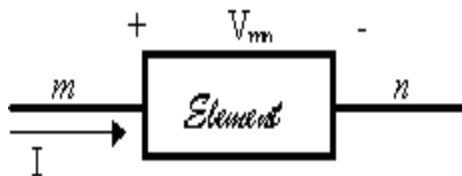
The voltage across the element is defined positive at node *m* with respect to node *n*. Obviously, the ordering of the nodes is quite important. In fact, we'll return to this concept of node *m* versus node *n* in future experiments on nodal analysis. If V_{mn} as calculated by MultiSim is positive, then MultiSim will return a positive number. If the current value returned by MultiSim is positive, then current flows in at node *m* and out of node *n*. For example, if we ask MultiSim its calculations for V_{mn} and *I* and it said, -4.5 and 2.2E-3, we know that node *n* is 4.5 V positive with respect to node *m* and a current of 2.2 mA flows from *m* to *n*. In other words, the element is a source.

When elements such as resistors and voltage sources are given values, it is convenient to use unit prefixes. MultiSim supports the prefixes listed below. Note that the letter must immediately follow the value - no spaces. Also, any text can follow the prefix letter. Finally, MultiSim is case insensitive.   So, there is no difference between 1 Mohm and 1 mohm.

MultiSim Unit Prefixes

```
t - tera - 10¹²     k - kilo  - 10³      n - nano  - 10⁻⁹
g -  giga - 10⁹     m - milli - 10⁻³     p - pico  - 10⁻¹²
meg - mega - 10⁶    u - micro - 10⁻⁶     f - femto - 10⁻¹⁵
```

2   Drawing Circuit Diagrams with MultiSim

2.1   Illustration of MultiSim Workspace

Let's draw the simple dc circuit in Fig. 1 using MultiSim. The initial step is to open MultiSim using the Start → All Programs → National Instruments → Circuit Design Suite 14.0 → MultiSim 14.0 sequence of pop-up menus. (If a window appears asking for the Evaluation License, click the "Evaluate" button.) After MultiSim finishes loading, one should see the screen shown in Fig.2 which is called the Capture and Simulate environment. The term "capture" refers to drawing the schematic in MultiSim; afterwards, one may simulate the circuit behavior.

Fig. 2 shows the location of the toolbars in your MultiSim window. The purpose of each toolbar will become clear as the reader moves through this exercise. To add or remove the toolbars, simply go to the View menu, and go to Toolbars. You can customize your own workspace to decide which toolbars are to be shown. Make sure that you at least have the toolbars shown in Fig. 2 enabled.
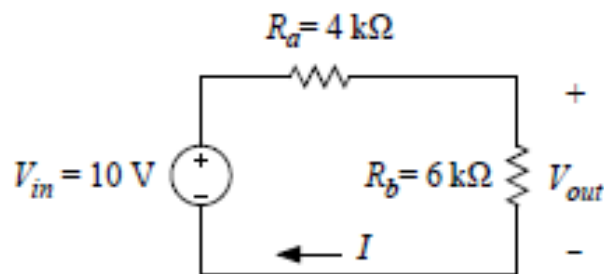

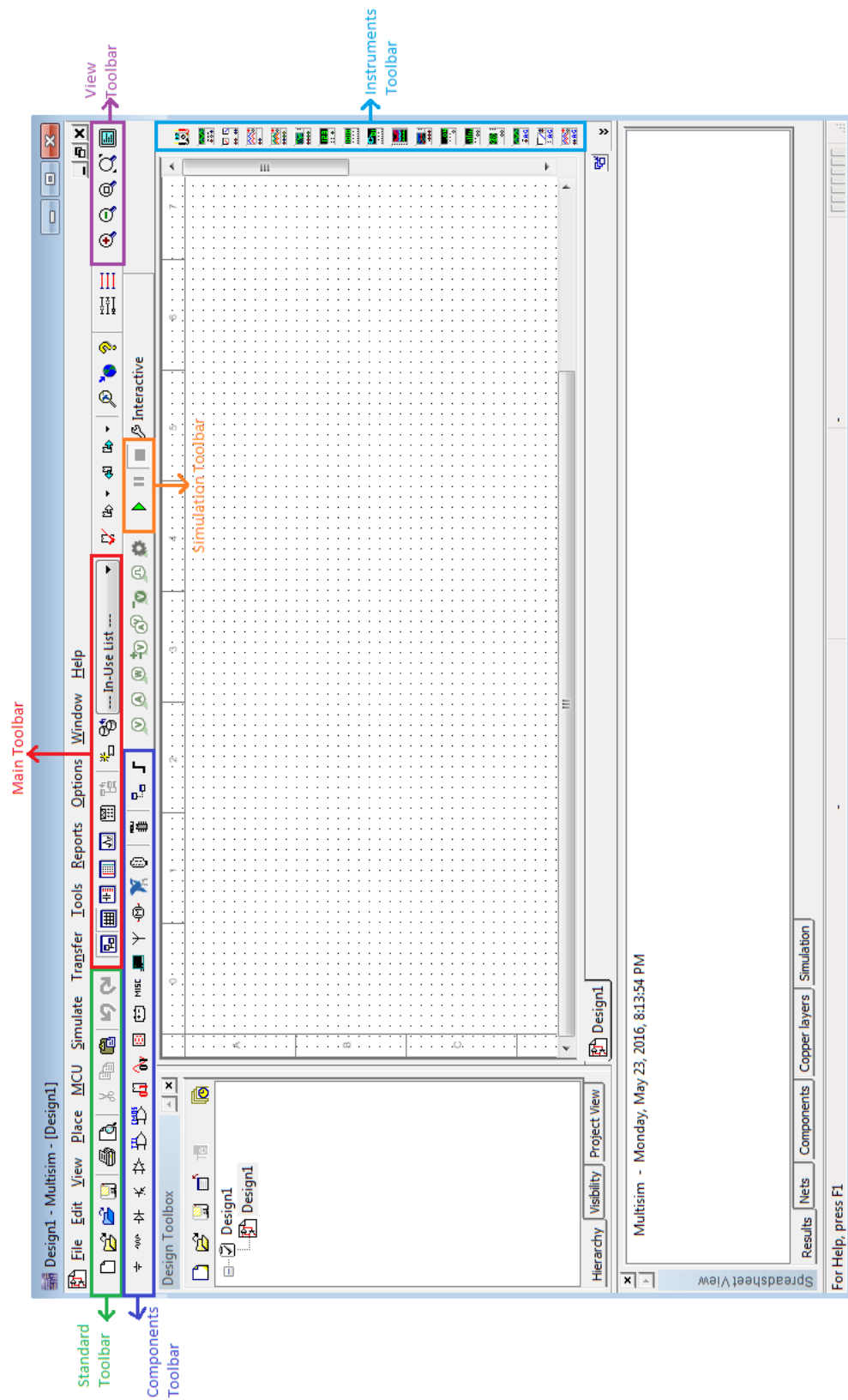
Figure 1: Prototype Circuit

Figure 2: Illustration of Toolbars in MultiSim Workspace

2.2   Getting parts

Generally, we get the components in the circuit by left clicking the Place menu and selecting Component (or by hitting Ctrl+W). You get the window shown in Fig. 3. The shortcut to place component is to use the Component Toolbar shown in Fig. 3.

Then, Select a Component dialog box in Fig. 4 appears listing all kinds of parts in the evaluate version.
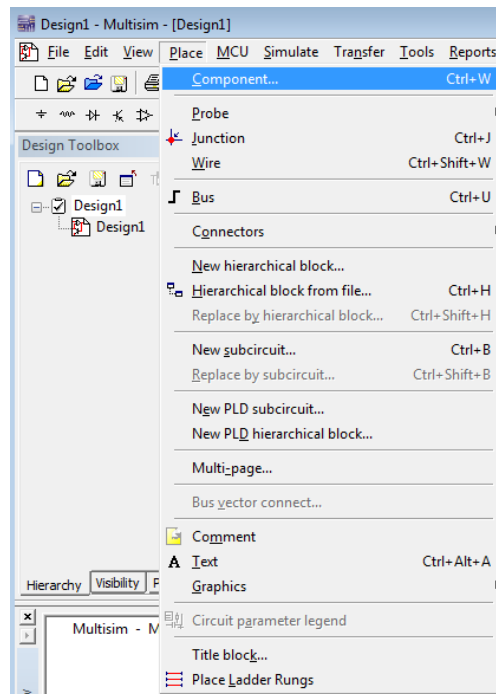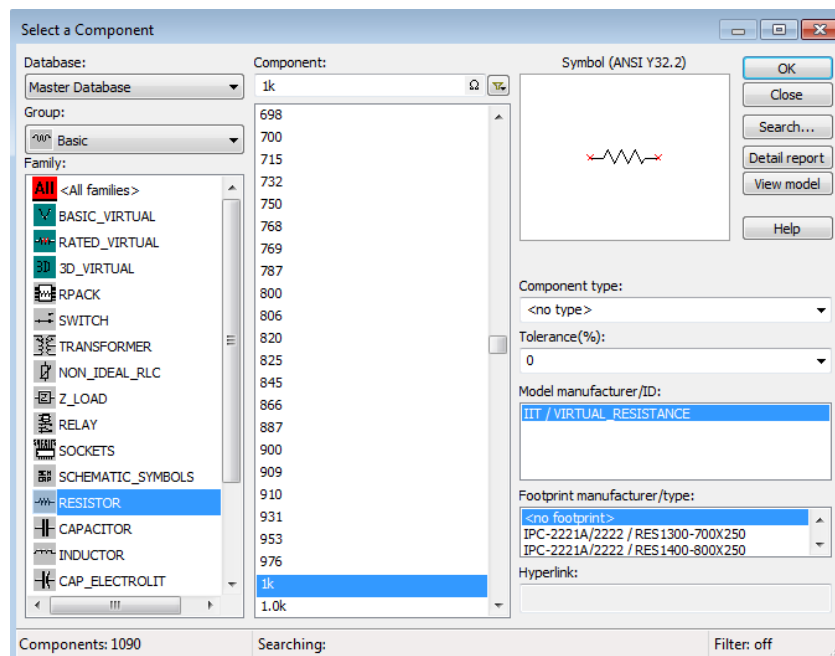
Figure 3: Open Components Library

Figure 4: Select the Component

First, let's get the voltage source. In the Database: menu, select the Master Database. From the Group: item, select the Sources, many different kinds of sources will be shown. See Fig. 5. Here, we choose the DC_POWER component from the POWER_SOURCES family, and click on OK (or double click the DC_POWER item).
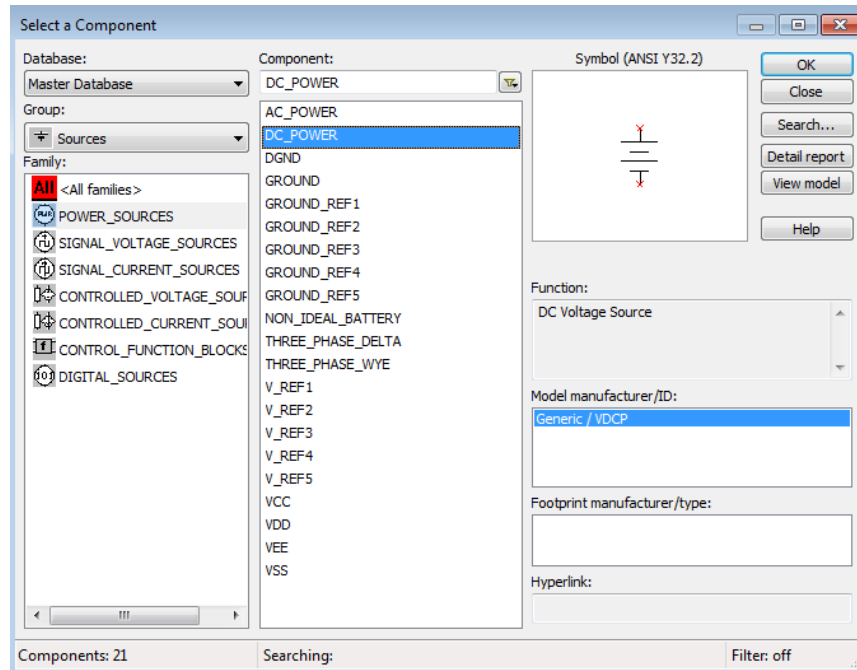
Figure 5: Select DC Source

Then, you are back to the schematic page of Fig. 2. Now, the mouse pointer is a dc voltage source. Place it any place inside the drawing area and left click once. MultiSim calls the source V1, and will call the subsequent voltage Source V2, V3, etc. Note that MultiSim also aligns whatever you draw to the grid dots.

To help you select the next component, the dialog box shown in Fig. 5 will appear again. This time, from the Group: menu, we choose the Basic item and then select the RESISTOR from the Family:. On the Component: side, a long list of resistors with different resistance will appear. We can either select the "4.0k" and "6.0k" resistors or randomly select two resistors, since the attributes of these components can be modified later. Here, two identical resistors with resistance 1kΩ are selected and placed. To stop adding components, click on the Close icon in the Select a Component dialog box. We can see the present condition in Fig. 6 where the blue dashed box surrounding R2 means this component is now selected and may be edited.
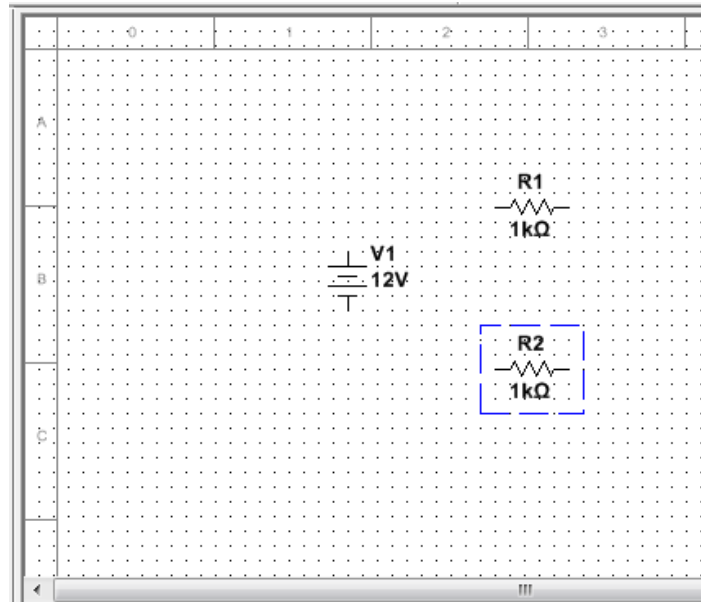
Figure 6: The Circuit Components in Place

2.3   Changing a Part's Attribute

To change the voltage source's attributes (name and value), simply double click on the source and the DC_POWER window in Fig. 7 will appear.
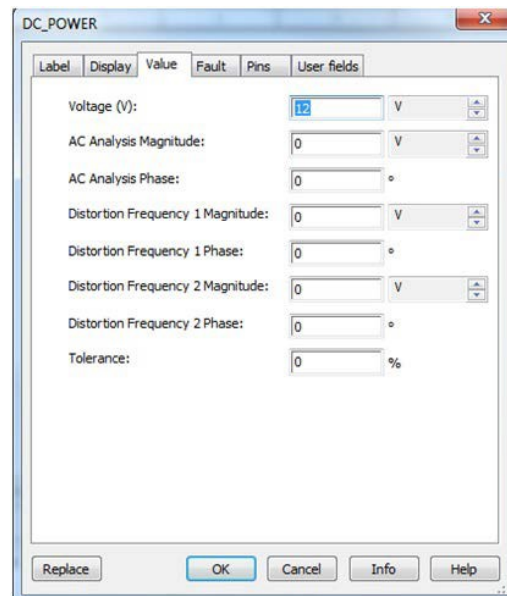

Figure 7: DC power source dialog box

To change the value of the source, enter 10 V to replace 12 V in the Voltage (V): field box. Click on the Label button, type in the RefDes field box the new name Vin. Select OK. Do likewise for the resistors, renaming them Ra and Rb. Also set resistance values at 4kΩ and 6 kΩ, respectively. Then the schematic will look like Fig. 8. Next, we'll arrange the parts for wiring.
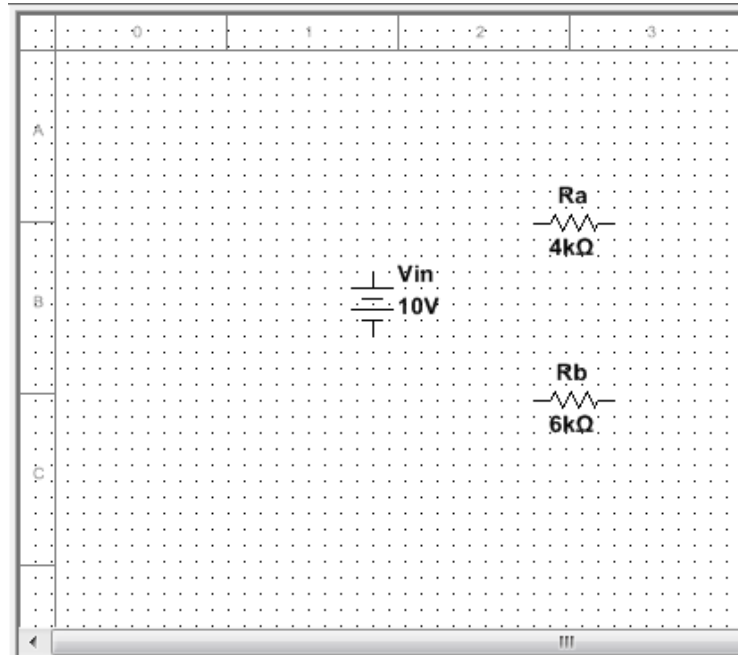
Figure 8: Attributes of Components Modified

2.4   Arranging Parts and Pin Numbers

Every pin on every part in MultiSim is numbered. When a part is placed, it is automatically oriented either vertically or horizontally. Oriented parts have pin 1 on the left and pin 2 on the right. For vertically placed parts, pin 1 can be at the top or at the bottom. As shown in Fig. 9, the voltage source has pin 1 at the top while the current source has pin 1 at the bottom.
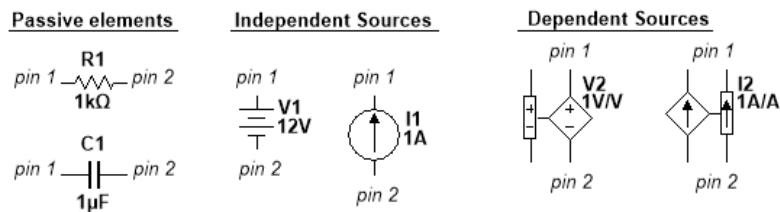
*NOTE: Dependent/controlled sources are connnected with the:
    -Diamond portion connected to the circuit where source is located.
    -Rectangle portion connected where the voltage or current is defined by.
    (current = series connection, voltage = parallel connection).

Figure 9: Currents measured by MultiSim Flow into Pin 1 and Out of Pin 2

Now consider Rb in our circuit in Fig. 8. We need to rotate it for appearance sake. Also, the current of interest, I, flows downward through Rb in Fig. 1. Click on the Edit button, and select Orientation, you can find 4 ways of rotating the components: Flip Vertical, Flip Horizontal, 90 Clockwise, and 90 CounterCW. By using Ctrl+R, we can spin Rb by 90° clockwise, putting pin 1 at the top so as to get the positive current. Finally, we can get the diagram shown in Fig. 10.
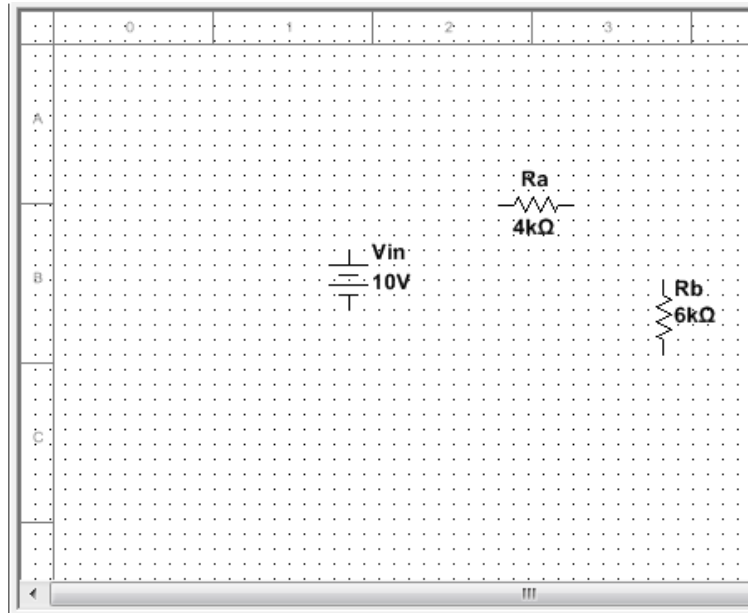
Figure 10: Circuit Ready for Wiring

2.5 Wiring Parts

In MultiSim, placing wire is very simple. Move the mouse pointer toward the pins of the components until it turns into a cross cursor with a middle black dot, left click once and release, and drag the wire to the destination node, left click again to finish the connection. To delete the unwanted line, just select it and press Delete on keyboard. Fig. 11 shows the wired schematic.
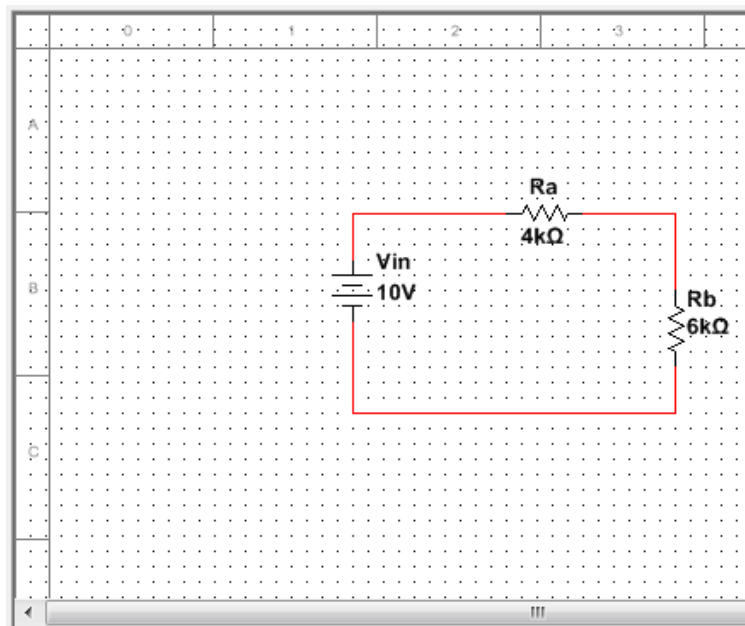

Figure 11: Wired Schematic

2.6   The Ground

MultiSim also requires that all schematics have a ground, or reference terminal. The reason for this is SPICE (the underlying simulation engine) uses nodal analysis to solve circuits. The first step in nodal analysis is to pick a ground node. The part you need is either the analog ground (GROUND) or the digital ground (DGND). You can find them from the POWER_SOURCES family in the Sources Group. In this example, we will get the GROUND part. It doesn't matter where we ground the circuit, for consistency, let's place it at the bottom of the schematic as shown in Fig. 12.
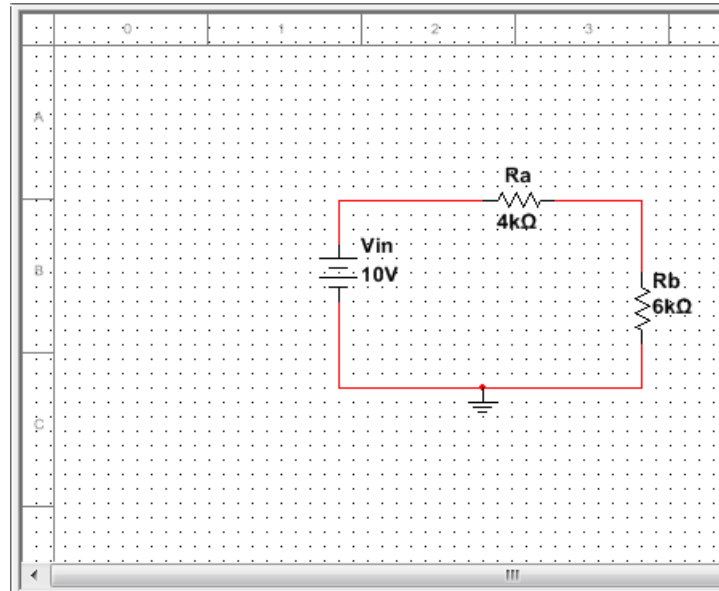
Figure 12: Wired Circuit with Ground Added

2.7   Naming Nodes and Changing Wire Colors

You will receive the option to name a node by double clicking on a wire. Type in the wire names Vin and Vout. Remember to click the show box. You can also select the net color to make debugging easier. This is especially useful in larger circuits. However, you should try to stick to electronics wire color conventions. For example: RED for power and BLACK for ground. After adding nodes and changing colors, the schematic should look like Fig. 13.
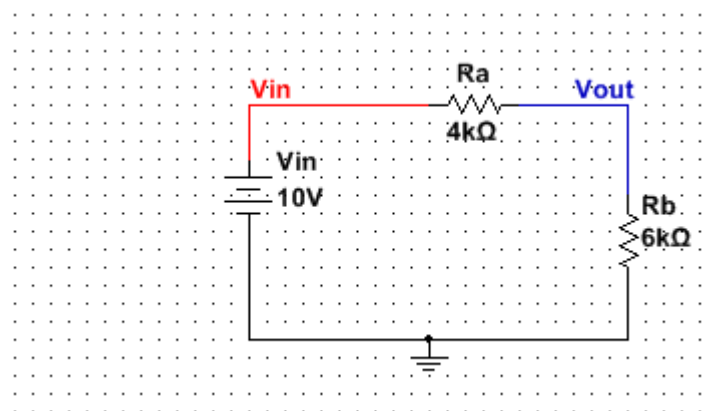
Figure 13: Circuit Ready for Simulation

2.8 Saving Schematics

Simply press Ctrl+S to save your design which is given the extension .ms14. MultiSim will try to save your design in the My Documents directory. It's much better to save your work in a different directory. Here the file name is saved as "Simple DC Circuit".

3 Using MultiSim for Bias Point Detail Analysis

From the Simulate menu, choose Analyses and simulation, the DC Operating Point Analysis window should pop up, → DC Operating Point (on the left). Next you select the variables in the circuit for analysis. V(vin), V(vout) are the voltages and wire names you have given; I(VIN) is the current through the voltage supply. Add all these three variables for analysis.

Press the Run button to start the simulation. The Grapher View window shown in Fig. 14 would thus appear. We can see the results are fully in accordance with what we expect to see. Note the current flowing top-down through our source is -1 mA, which verifies that MultiSim obeys the passive sign convention. Also note that the background is black, and should be changed to white by clicking the button circled in Fig. 14.
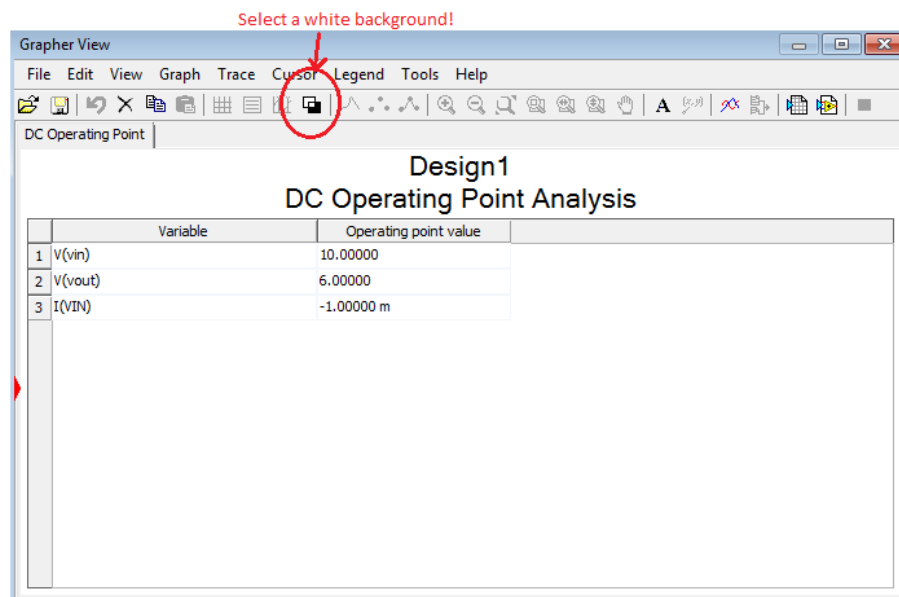
Figure 14: Simulation Result for the DC Operating Point Analysis

4 DC Sweep in MultiSim

From the Simulate menu, choose Analyses and simulation → DC Sweep. Change the Analysis parameters as shown in Fig. 15. Then click the Output tab and add V(vin) and V(vout) to the Selected Variables for Analysis box as shown in Fig. 16. Click the Run button to obtain the results shown in Fig. 17.
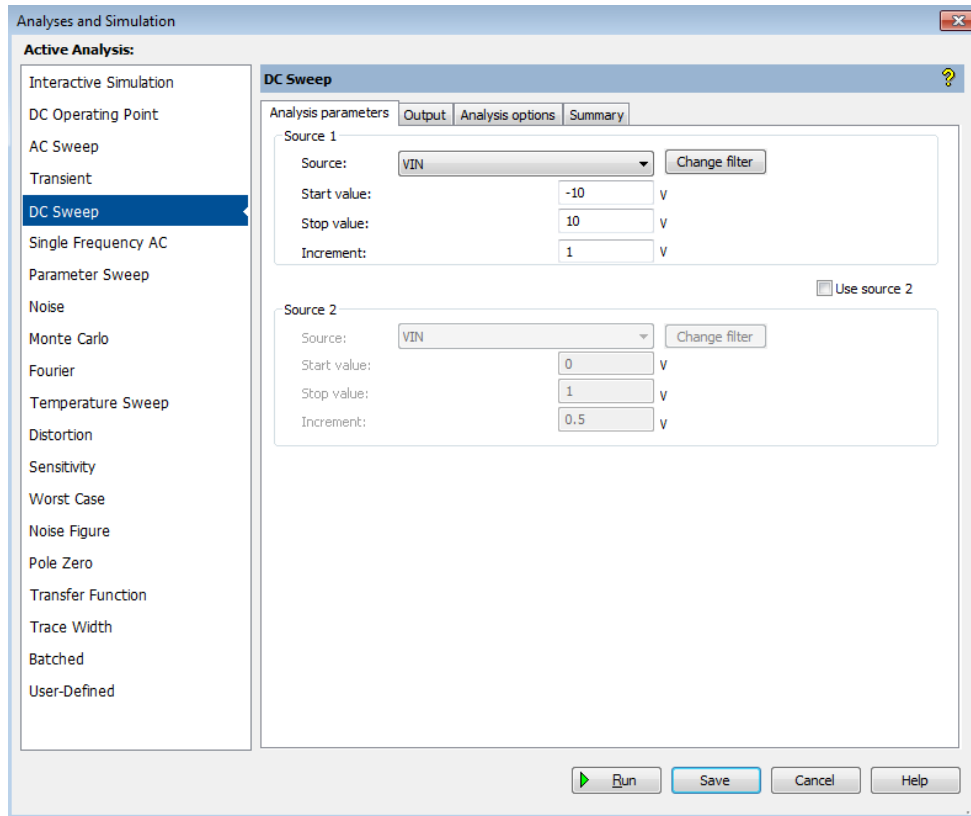
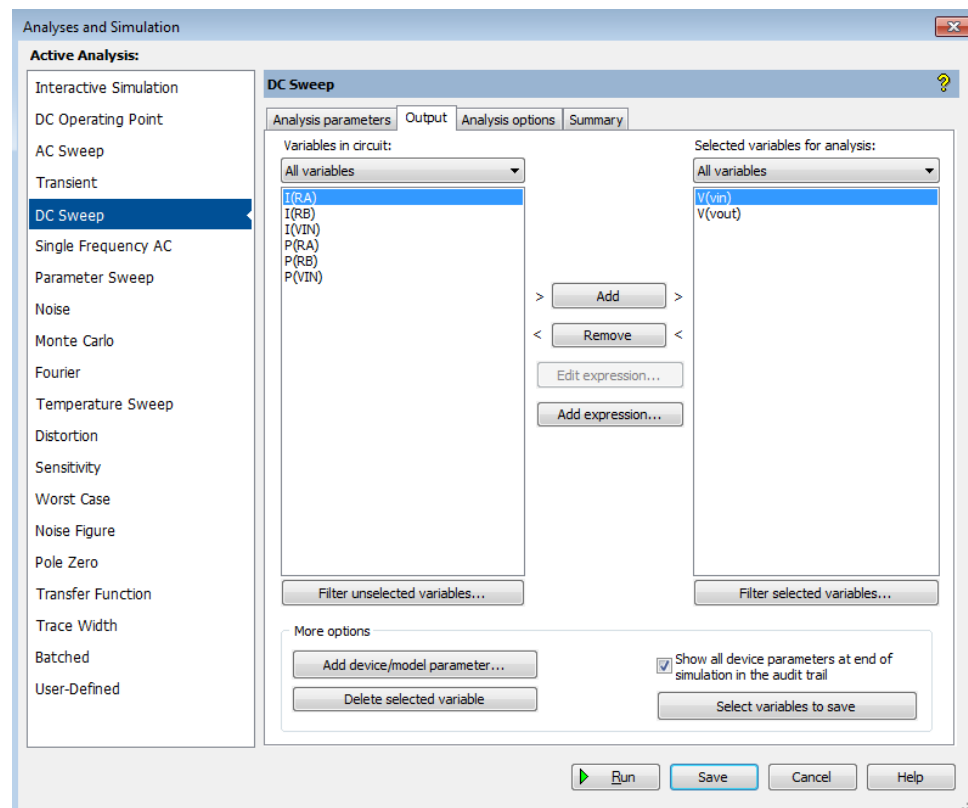Figure 15: Analysis Parameters for DC Sweep Analysis

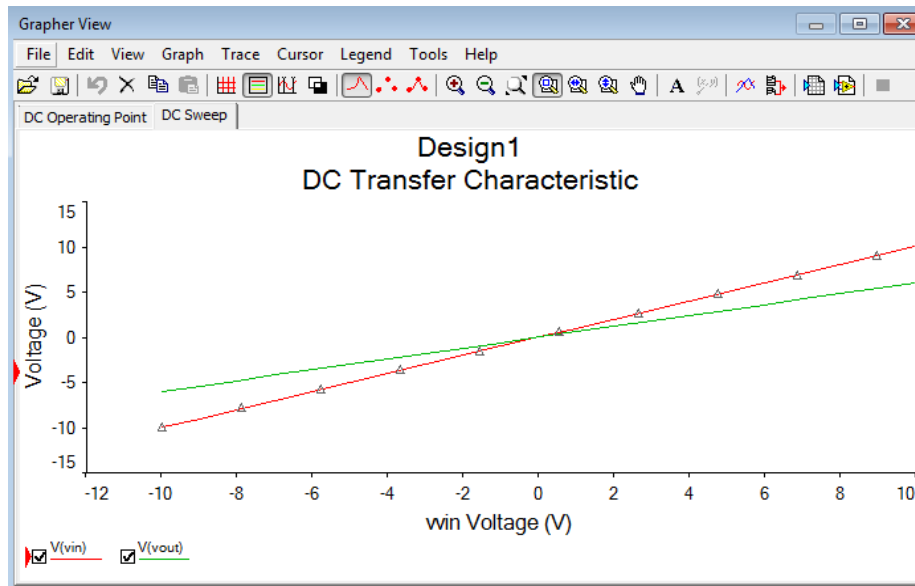Figure 16: Output Variables for DC Sweep Analysis

Figure 17: Results of DC Sweep Analysis

5 Parametric Sweep in MultiSim

We will now see how to do parametric sweep of 4 kΩ resistance Ra in Fig. 13 using MultiSim. We will plot the power consumed as a function of resistance. Here we go:

1. Go to Analyses and simulation → DC Sweep → Parameter Sweep.
2. Go to Analysis parameters tab and refer to Fig. 18; you should change Device type to Resistor, Name to RA, and Parameter to resistance. Give Start value as 40 and Stop value to be 4kΩ. Analysis to sweep should be changed to DC Operating Point.
3. Then go to Output tab and add P(RA) as a variable for analysis as shown in Fig. 19. Then click Run. You should be able to see the plot of power versus resistance RA as shown in Fig. 20.

Figure 18: Setting up Analysis Parameters for Parameter Sweep in MultiSim
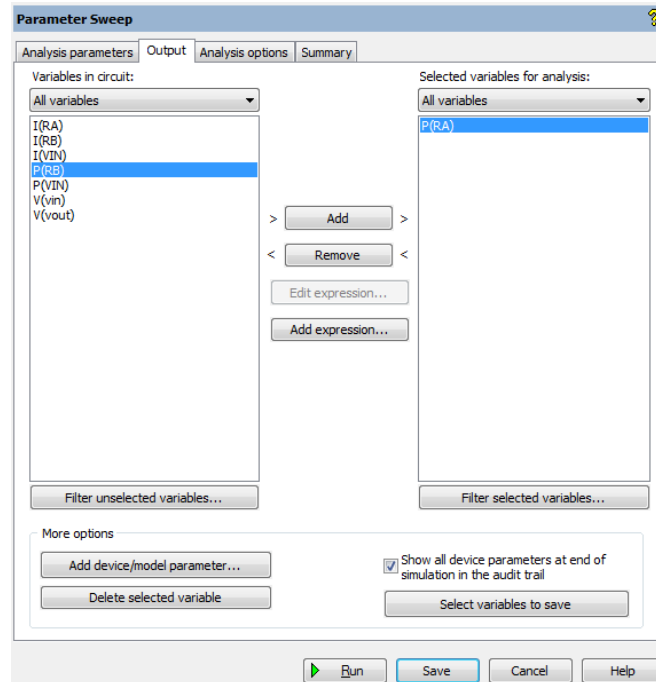
Figure 19: Setting up Output Parameters for Parameter Sweep in MultiSim
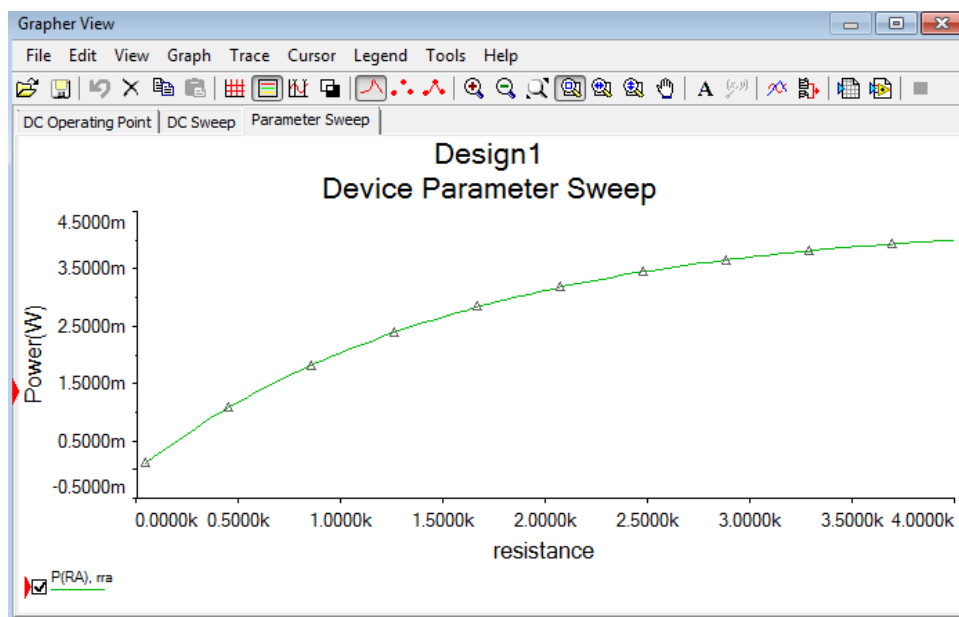
Figure 20: Power versus Resistance Ra Plot

6   Alternative Ways of Circuit Simulation in MultiSim

This section describes an alternative way to perform circuit simulation in MultiSim. It involves using virtual instruments which are just like those in the laboratory. Let's measure the current through the dc source and the voltage drop across Rb for example.

Click the Multimeter and drag the Multimeter to your workspace. Do the same thing for the Agilent Multimeter . The Schematic will look like Fig. 21.
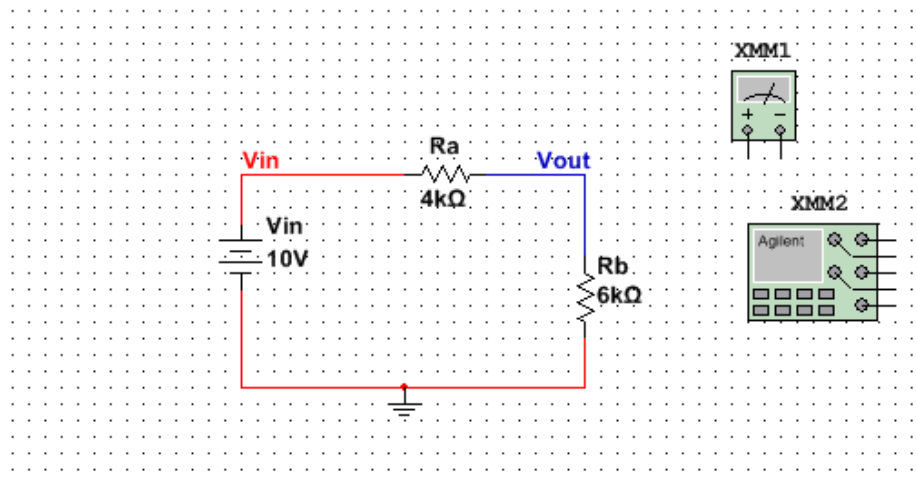
Figure 21: Multimeters Placed on Workspace

Double-click these two instruments respectively, we can see their front panels in Fig. 22 and 23. The Multimeter will be connected in series in the circuit to measure the current. So, click the [A] button on the Multimeter to use it as ammeter. The Agilent Multimeter will be connected in parallel with Rb to measure the voltage drop. So, click the [Power] button to first turn on the Agilent Multimeter, and then click the [DC V] button to measure the dc voltage. Finally, draw wires from the Multimeter terminals to the circuit as shown in Fig. 24. To simulate the circuit, press the ▶ button in the simulation toolbar. Fig. 25 and 26 show what we get from the instruments. Also, as you can see from Fig. 26, MultiSim will mark the terminals on the front panel as you make the connections.
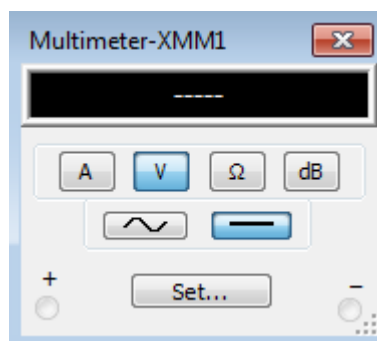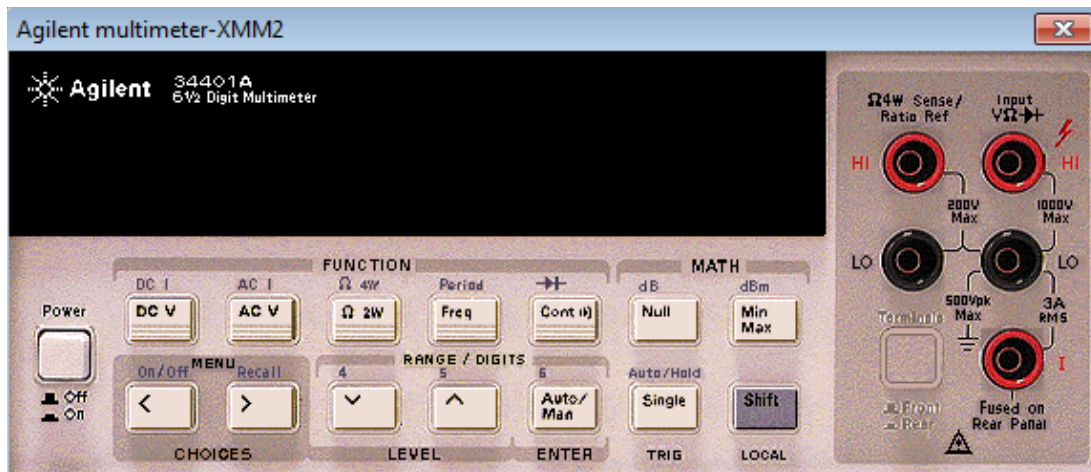

Figure 22: Front Panel of the Multimeter

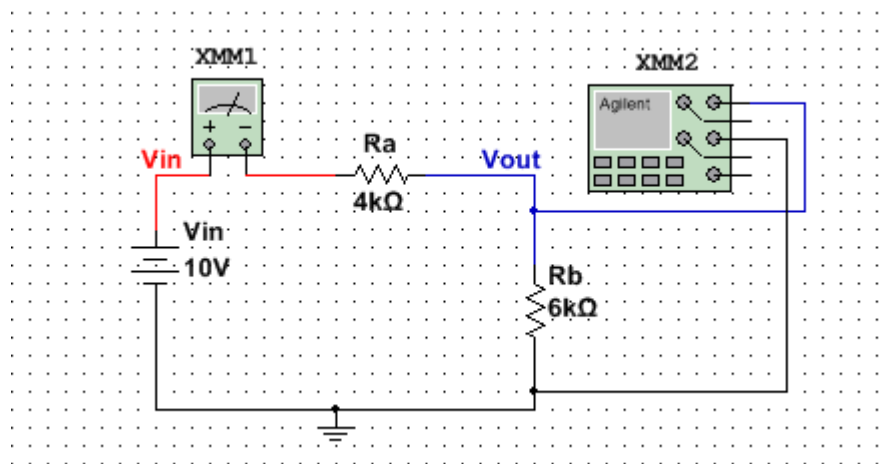Figure 23: Front Panel of the Agilent Multimeter

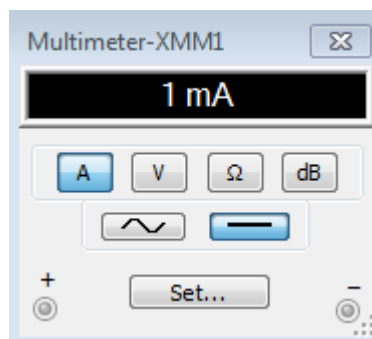Figure 24: Circuit Ready for Simulation (virtual instruments involved)

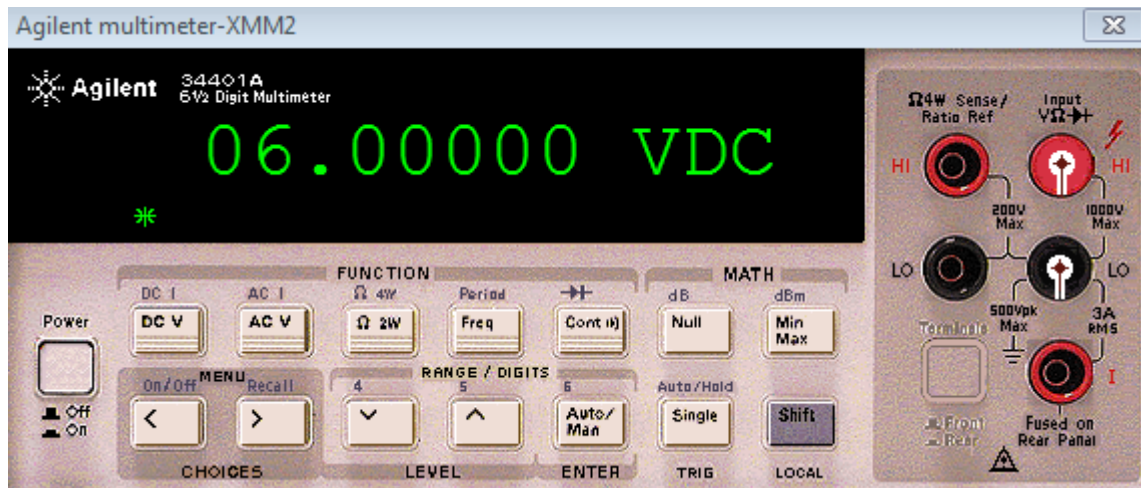Figure 25: Result Shown on Multimeter

Figure 26: Result Shown on Agilent Multimeter

7 Conclusions

This document gives a basic view of MultiSim. There are many other features that are not mentioned. The best way to learn is to experiment; don't be afraid to try out complicated circuits and MultiSim's new features.

8 Exercises

You will need to include screenshots for the various exercises (MultiSim schematic and results) in your lab report. Be sure whatever values are asked for are clearly demarcated and recorded in your lab report.

1. Use MultiSim to find V_0 and I_x in the circuit in Fig. 27. Include a screenshot of the schematic and the simulation results.
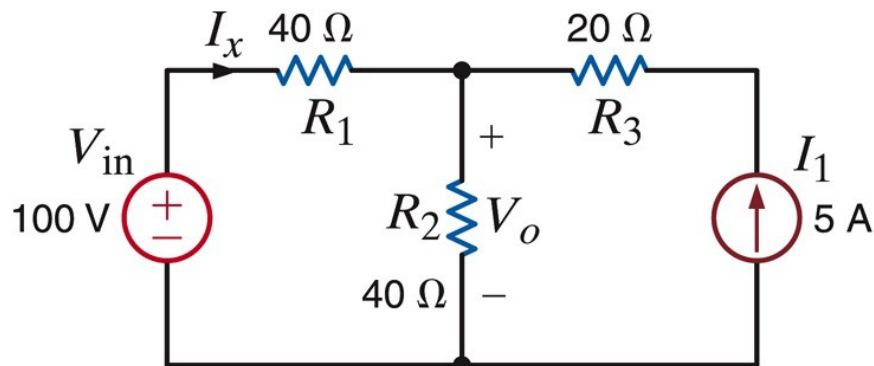

Figure 27: Circuit for Exercise 1

2. Use MultiSim to find V_0 and the power supplied by the 6-V source in Fig. 28. Include a screenshot of the schematic . Also include the power calculations in your report.
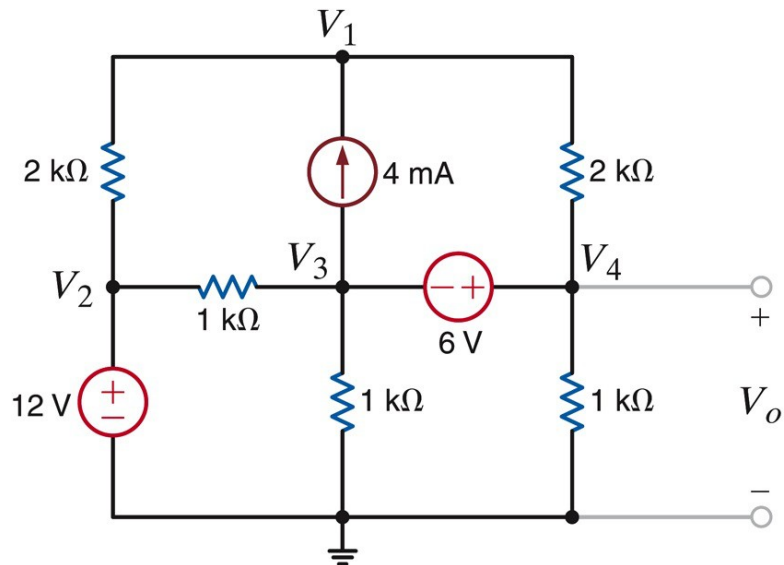

Figure 28: Circuit for Exercise 2

3. Use the dc sweep feature of MultiSim to plot V_0 as the voltage V_{in} is varied between 50 V and 150 V in steps of 10 V in the circuit of Fig. 29. Include the plot in your report. Set I_1 = 5 A. Now set V_{in} = 100 V. Use the dc Sweep feature to plot I_x as the current I_1 is varied from -5 A to 5 A in steps of 1 A. Include the plot in your report. Include a screenshot of your MultiSim schematic.
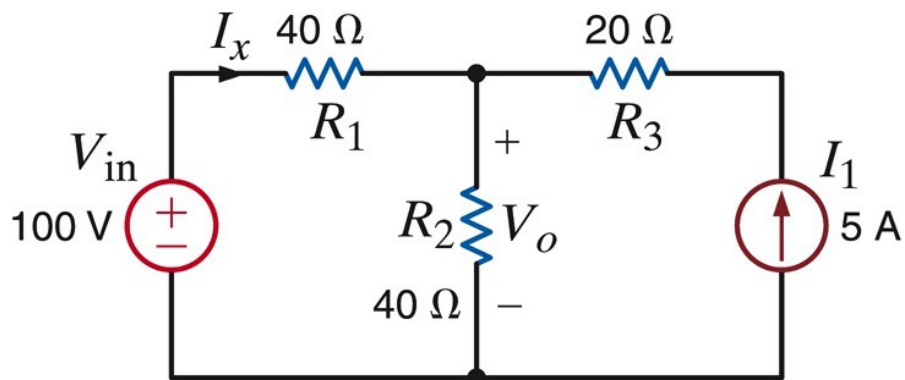

Figure 29: Circuit for Exercise 3

4. For the circuit of Fig. 30, use MultiSim to solve for the voltage V_0 as R_b varies from 250 Ω to 3 kΩ in increments of 25 Ω. Also solve for the power dissipated in R_b (P_0) for each value of resistance. Plot V_0 as a function of R_b and P_0 as a function of R_b. You must make these plots by performing the parameter sweep outlined in Section 5 of this lab manual. Include both plots in your report and a screenshot of your MultiSim schematic.
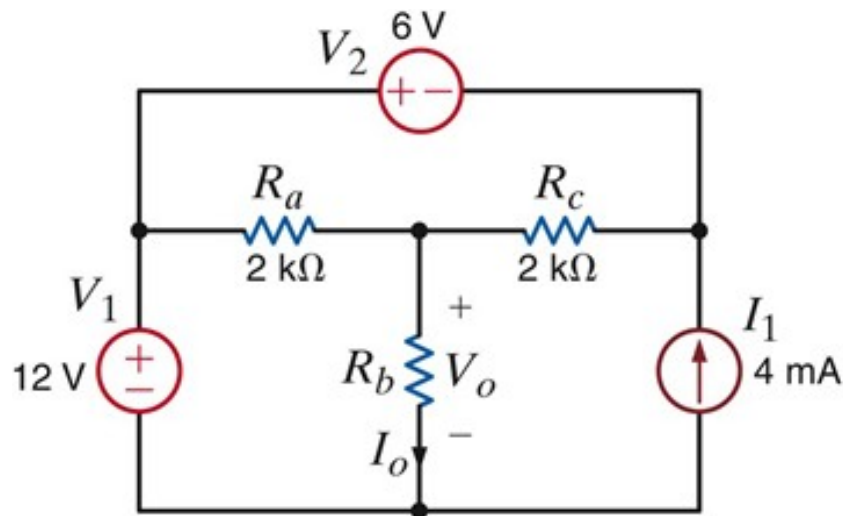

Figure 30: Circuit for Exercise 4

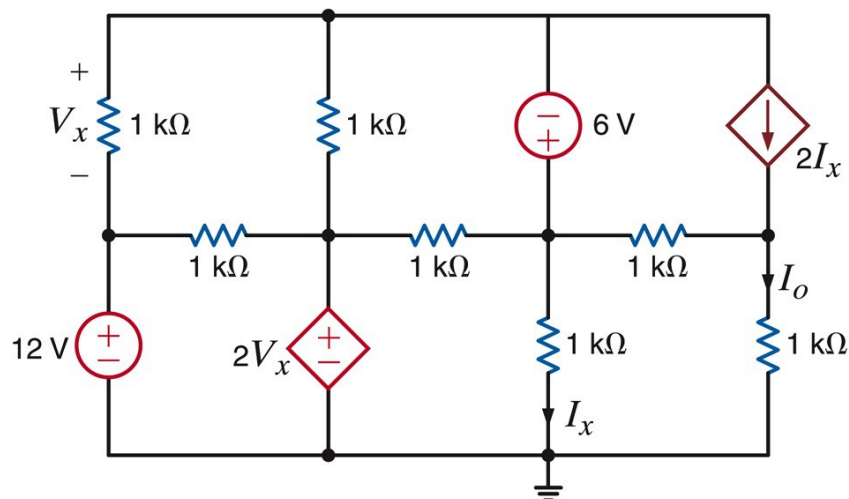5. Determine I_0 in the circuit in Fig. 31 using MultiSim. Include in your report a screenshot of the schematic and also the simulation results.

Figure 31: Circuit for Exercise 5