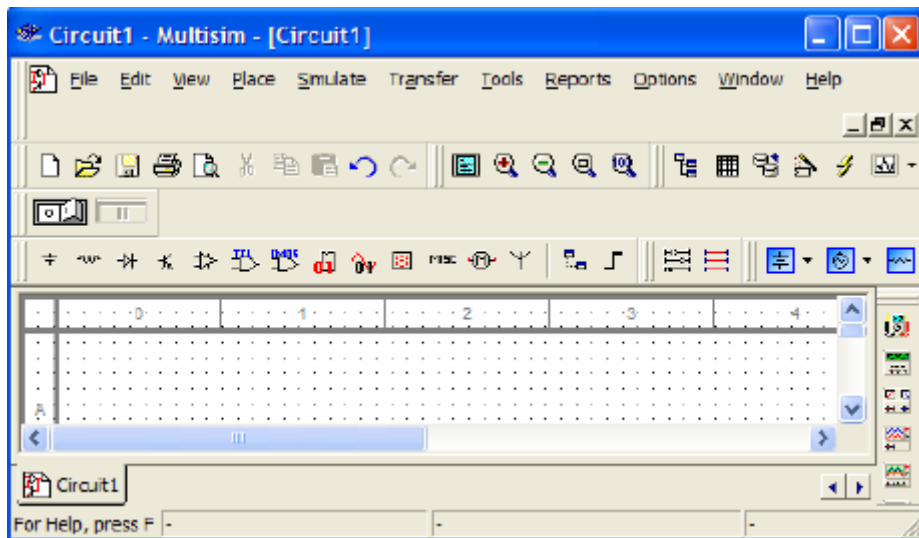


# Introduction to MultiSim – Part 1

Prepared by:  
**Mohamad Eid**  
**Summer 2007**

The purpose of this document is to introduce the many features of MultiSim. Begin by first opening up MultiSim. For Windows users the default location can be found by clicking: Start ->All Programs -> Electronics Workbench -> DesignSuite Freeware Edition 9 -> MultiSim 9. You should see a screen similar to Figure 1 below. This is called as a “Capture and Simulate” environment because you “Capture” your schematic by drawing it in MultiSim and then you “Simulate” it.



**Figure 1.** MultiSim startup screen

Figure 2 shows the different parts of the MultiSim workspace. Note that the location of the toolbars on your MultiSim window may be different.

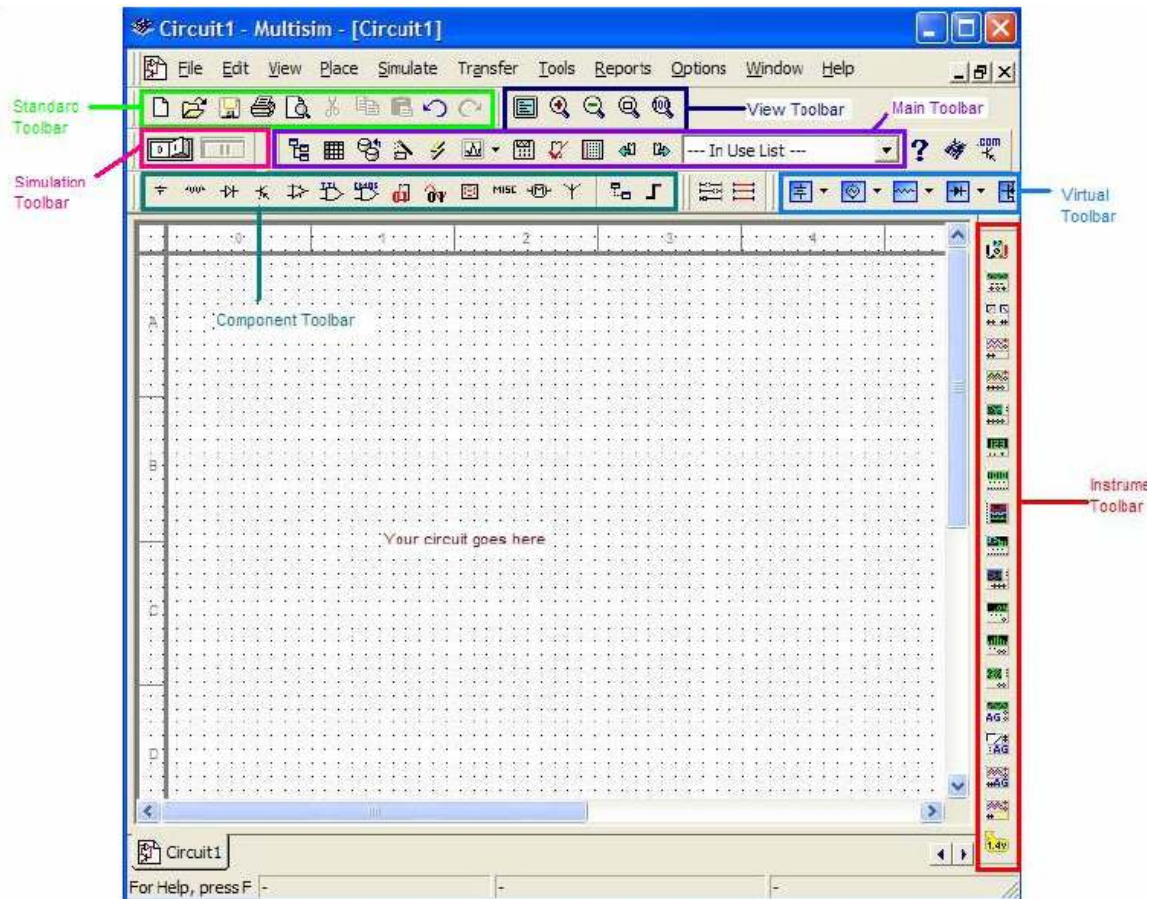


Figure 2. The most important components in the MultiSim workspace

If you don't see the toolbars shown above, **Left-click** on the View Menu and go to Toolbars. Make sure the toolbars shown in figure 2 are checked as shown in figure 3.

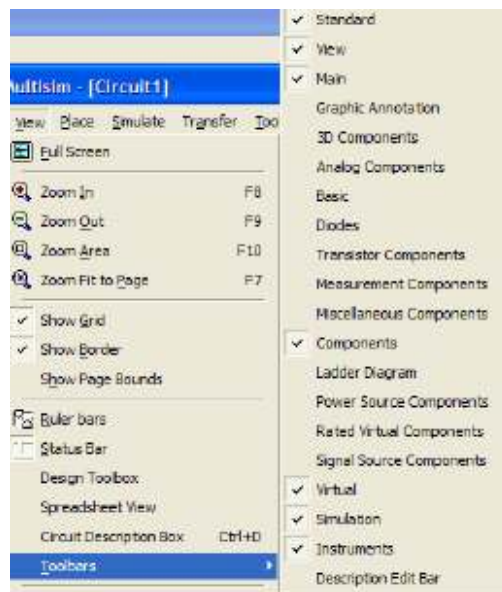


Figure 3. Viewing the toolbars

Please familiarize yourselves with the location of each toolbar as it appears in your MultiSim window. We will be repeatedly referring to the toolbars throughout this document (using the color codes from Figure 2, example: the Virtual Toolbar).

### Example 1: Simple DC Analysis in MultiSim

Let's construct the simple circuit shown in figure 4.

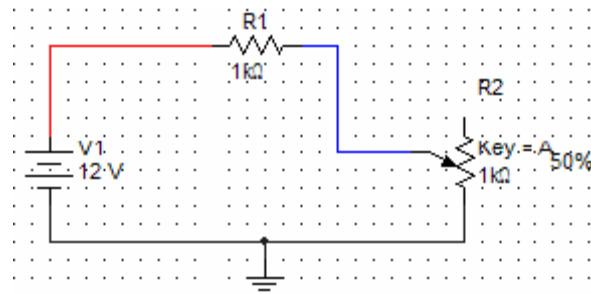


Figure 4. A simple series circuit constructed in MultiSim

Constructing this circuit in MultiSim is easy:

1. **Left-Click** on the Power Source Family in the Virtual Toolbar.
2. The Power Source Components will pop up.
3. **Left-Click** on the DC Power Source icon and drag a battery onto the circuit workspace. Figure 5 shows the result.

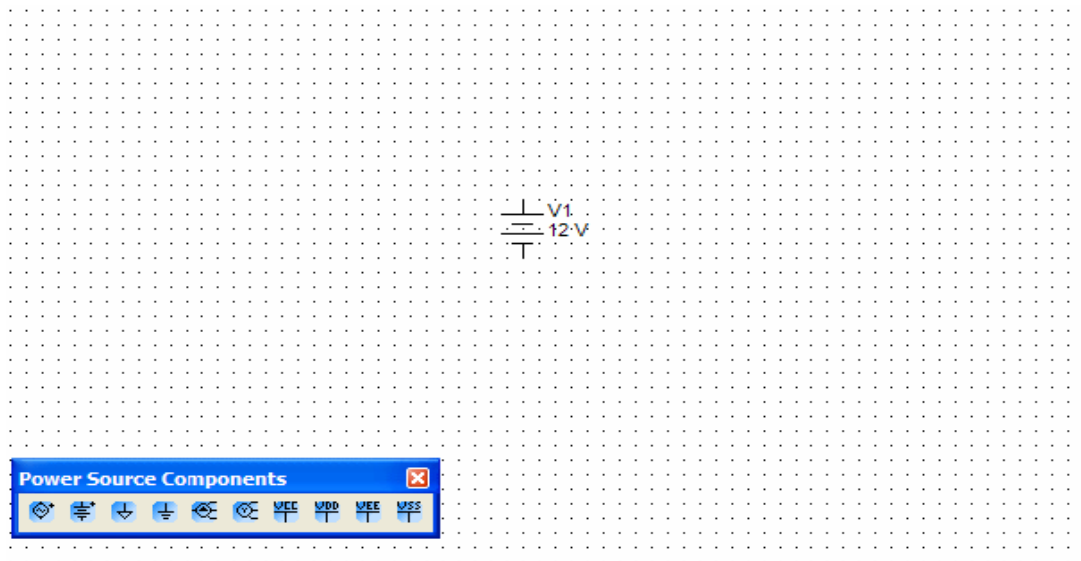
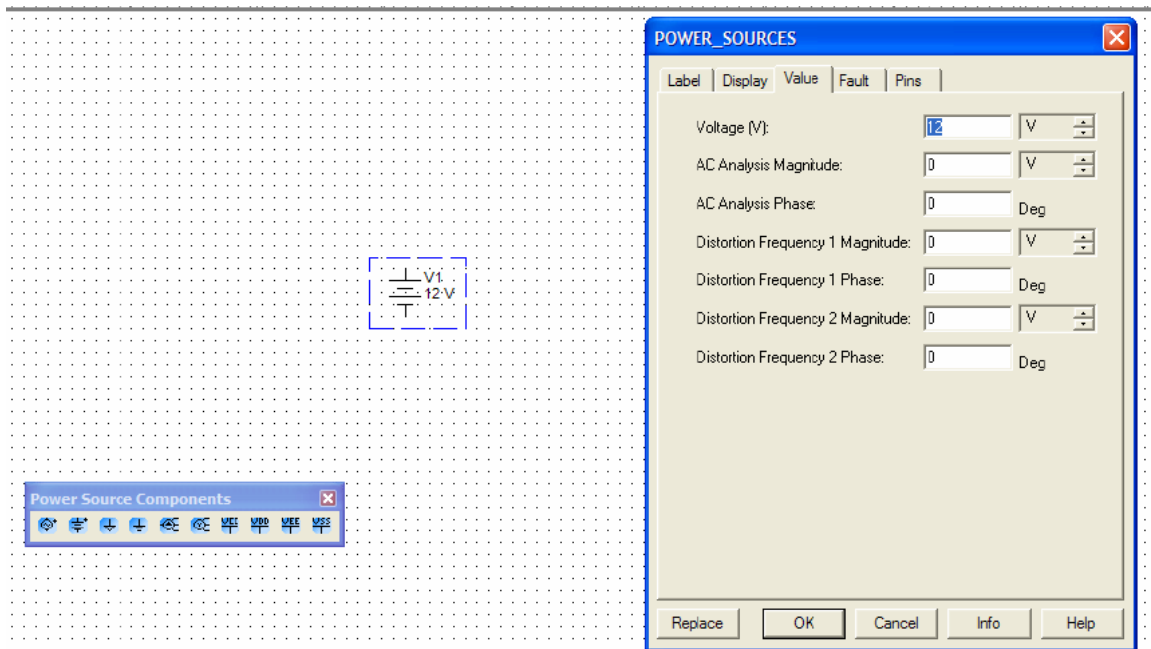


Figure 5. A DC power source in MultiSim

If you want to change the value of the power source, **Double-click** the battery. This opens up the Power\_Sources dialog box shown below.



**Figure 6.** Power Sources Dialog box. Use this to change the value of the battery voltage.

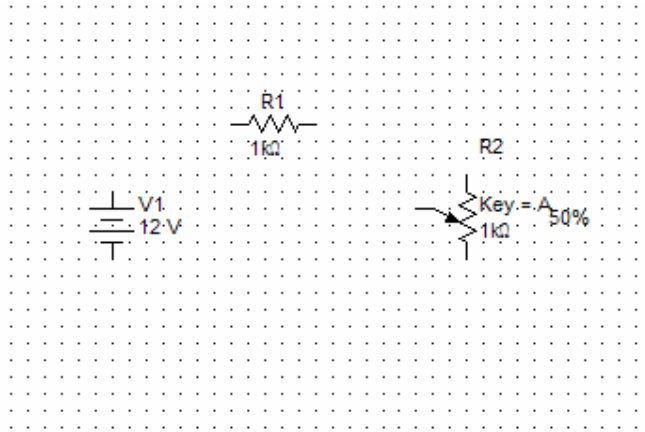
Let's add the resistor and the potentiometer. **Left-Click** the Basic Components Family in the Virtual Toolbar.

5. The Basic Components will pop up.

6. **Left-Click** on the Virtual Resistor tool and drag a resistor onto the workspace. As in the case of the battery, you can **Double-click** the resistor to change component values.

7. Let's complete the circuit by placing the potentiometer. **Left-Click** on the Potentiometer Tool and drag a potentiometer onto the workspace. You can increase (decrease) the resistance on the potentiometer by pressing the "A" (Shift+A) key. Note: The increase and decrease refers to the resistance between the middle leg and the bottom leg of the potentiometer. **Double-click** the potentiometer to change the total resistance of the potentiometer and the increment or decrement in the resistor value.

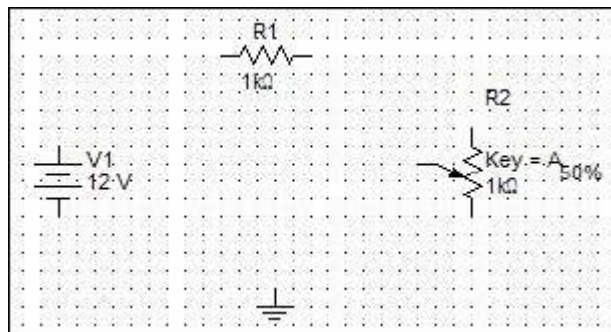
Figure 7 shows the circuit components placed on your workspace. The "50%" next to the potentiometer means that the resistance between the middle leg and bottom leg is 50% of 1 k $\Omega$ : 500  $\Omega$ . If you press A, you will notice that resistance will increase by 5% (the resistance between the middle leg and the top leg will decrease by 5%). Again, **Double-click** the potentiometer to change the increment percentage.



**Figure 7.** The circuit components are in place.

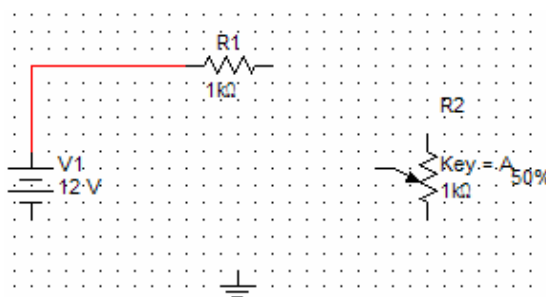
8. The final component to place is the ground. You cannot simulate the circuit without a ground. The reason for this is SPICE (the underlying simulation engine) uses nodal analysis to solve circuits. The first step in nodal analysis is to pick a ground node. It does not matter where we ground the circuit, for consistency lets pick the node at the bottom of the circuit as ground.

**Left-click** the Ground tool in the Power Source Components menu. Drag the ground to the bottom of the circuit, the result is shown in figure 8.



**Figure 8.** Circuit ready for wiring

9. To wire the circuit, simply **Left-click** at the starting node, drag the wire to the ending node and **Left-click** again. Figure 9 shows the results of wiring the 12 V source to the 1 kΩ resistor.



**Figure 9.** A wire connects the 12 V source to the 1 kΩ resistor.

Complete the wiring as shown in figure 10. Make sure you connect to the wiper of the potentiometer.

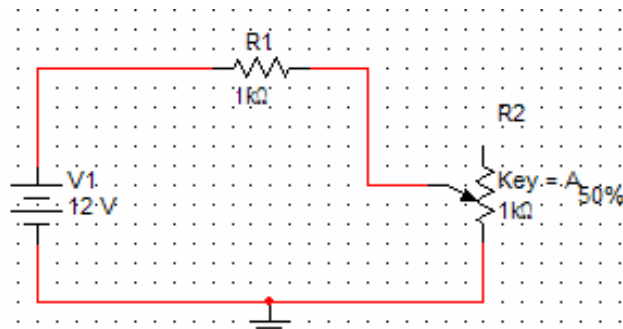


Figure 10. The circuit is complete

**Remark:** To make debugging easier in larger circuits, it would be instructive to change the wire colors. To do so, Left-click on the wire to select it and then Right-click to choose Wire color. Figure 11 shows the result. You should try to stick to electronics wire color conventions. For example: RED for power and BLACK for ground.

Before we can simulate the circuit, we need to add instruments so we can make measurements. One of the neat things about MultiSim is that it comes with a bunch of standard instruments. These instruments are the same (except for the scope) as on your lab bench. Hence your simulation environment is a step closer to your real lab environment.

Let's measure the voltage drop across the potentiometer. This will make an interesting exercise since you can see the voltage across the potentiometer change as you vary the potentiometer resistance.

10. **Left-click** the Agilent Multimeter from the Instruments Toolbar and drag the multimeter onto your workspace. Figure 11 shows the result.

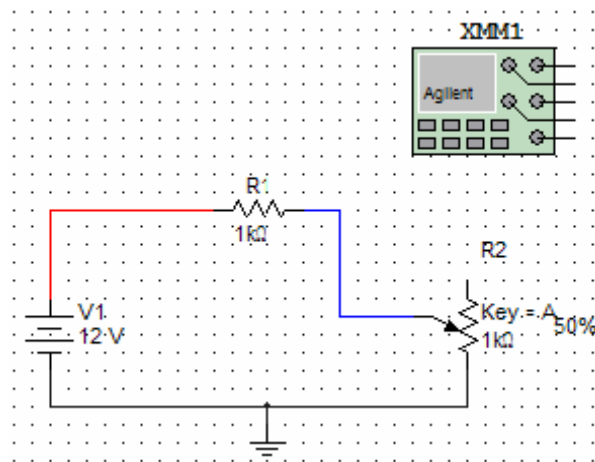


Figure 11. A multimeter placed on to workspace.

Now, **Double-click** on the multimeter to open up the instrument's front panel.

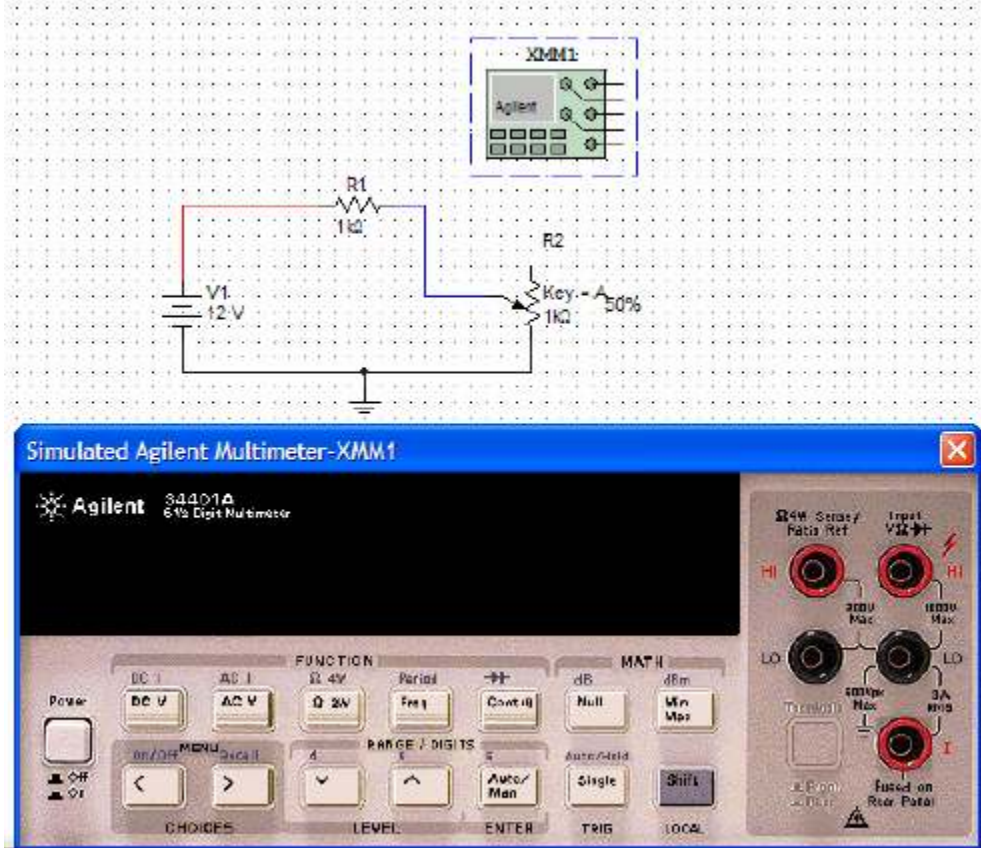


Figure 12. The Agilent 34401A Simulated Multimeter front panel

Notice how the simulated multimeter is the same as the one on your workbench! **Left-click** the button to turn on the instrument. You will be measuring DC voltage, so **Left-click** the button on the instrument. Figure 13 shows what you should get.

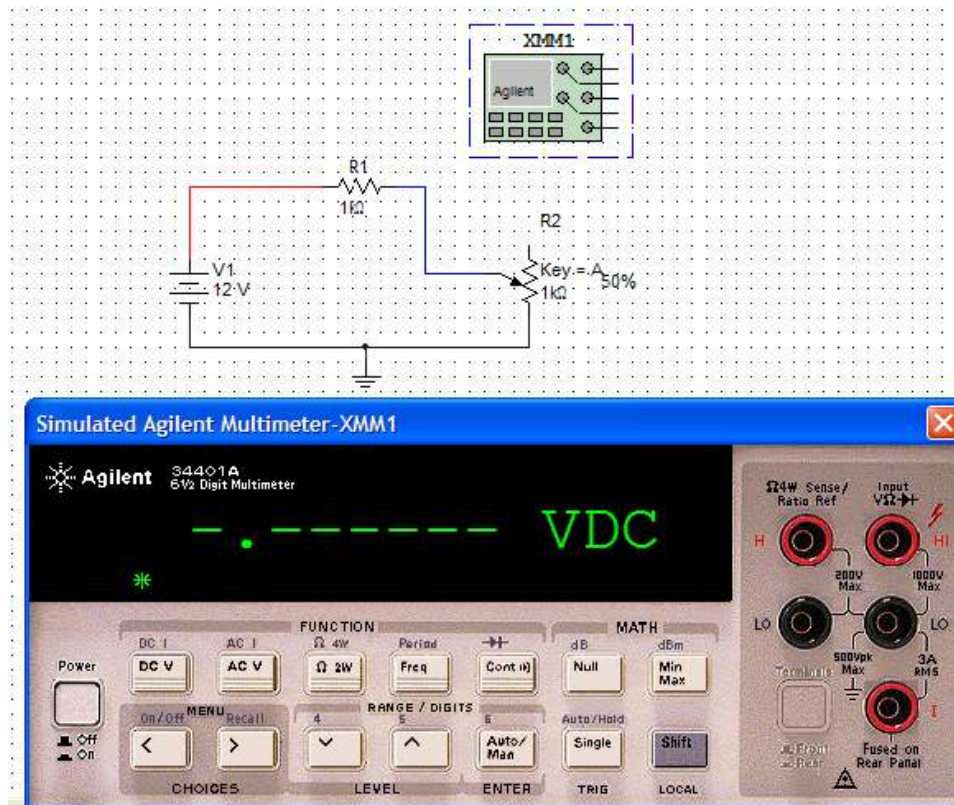


Figure 13. The Multimeter is set to the correct measurement mode

All that is left is to wire the multimeter terminals. Complete the wiring as shown in figure 14. Connect to the wires to the multimeter on the workspace. As you make the connections, MultiSim highlights the terminals on the frontpanel.

11. To simulate the circuit, **Left-click** the “Simulate” button in the Simulation Toolbar. Figure 15 shows the result.

One of the most powerful features of MultiSim is its interactive nature. Change the resistance of the potentiometer by pressing “A” or Shift+A and note how the multimeter readings change (you may need to wait a couple of seconds for the multimeter to register the change). Change the potentiometer resistance all the way to 1 kΩ (100%). What is the output voltage? Does this agree with your intuition and circuit properties? Hint: Think about what the voltage divider formula when  $R_1 = R_2$ .



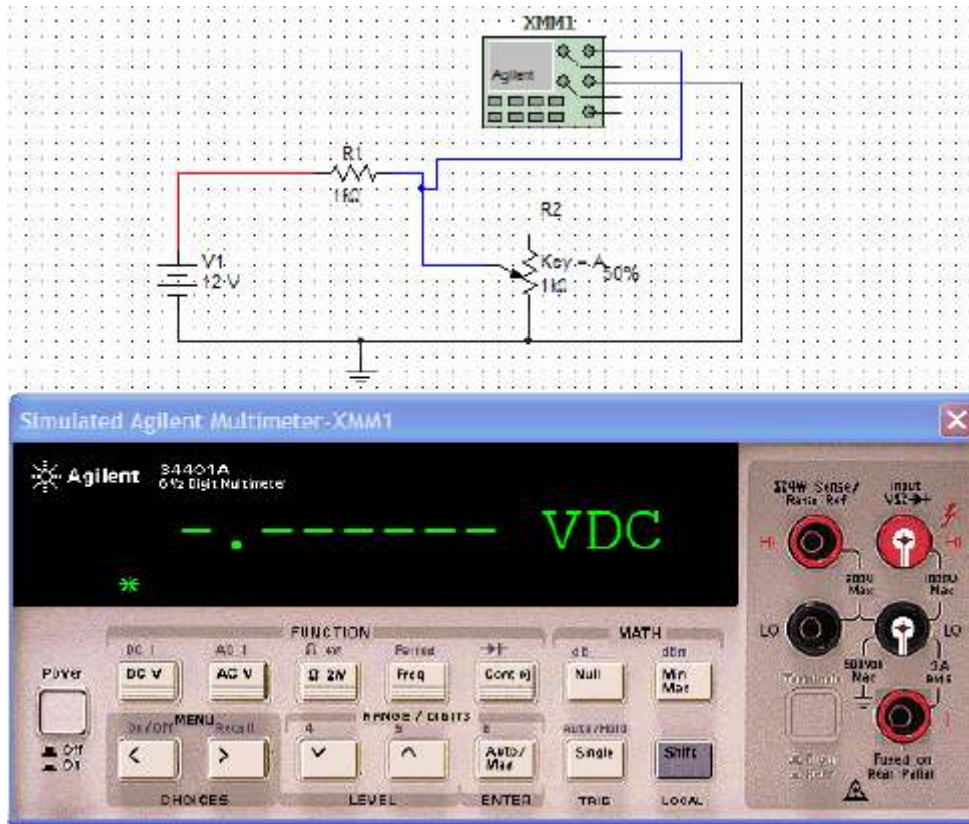


Figure 14. Ready for simulation

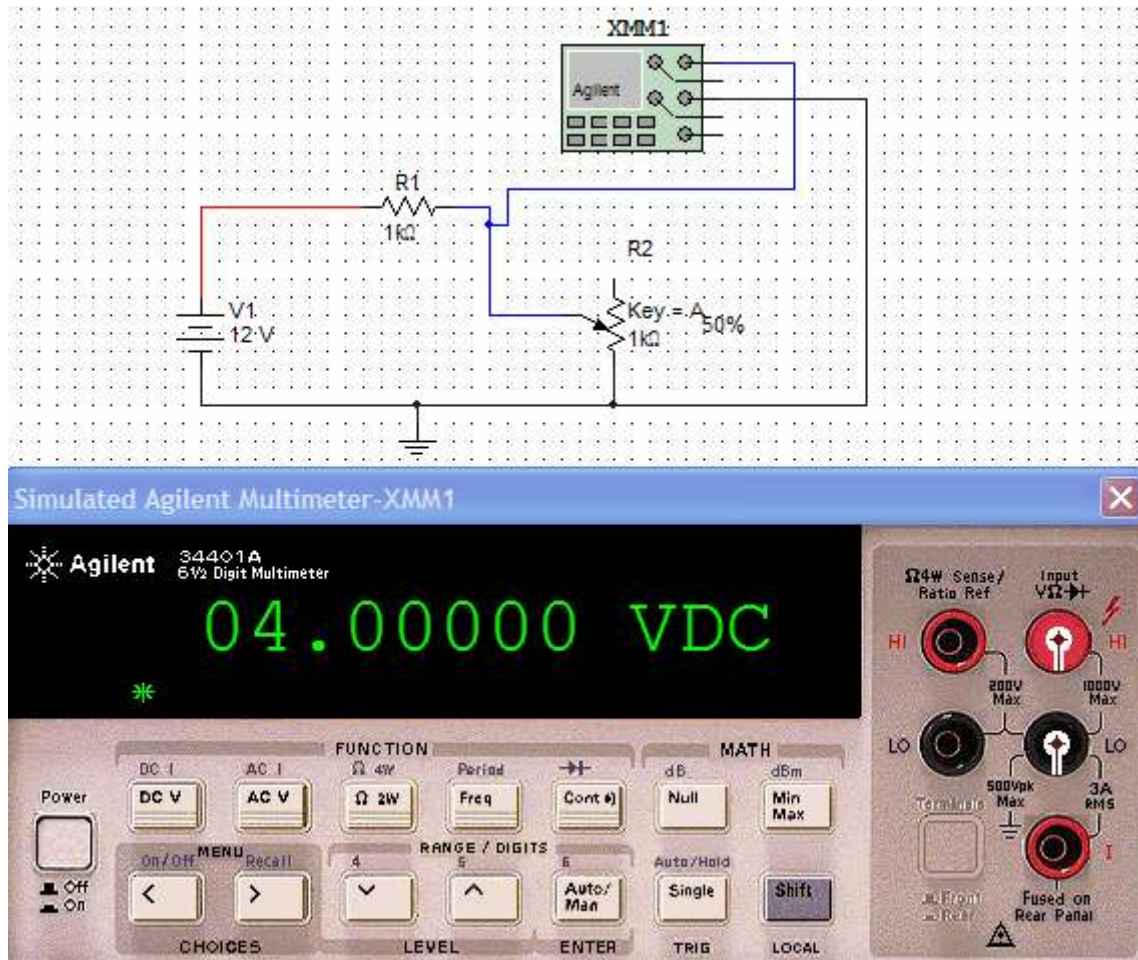


Figure 15. The simulation result

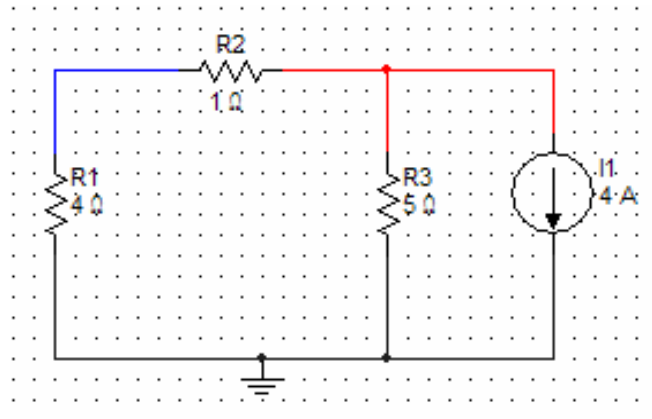
11. To simulate the circuit, **Left-click** the button in the **Simulation Toolbar**. Figure 16 shows the result.

One of the most powerful features of MultiSim is its interactive nature. Change the resistance of the potentiometer by pressing “A” or Shift+A and note how the multimeter readings change (you may need to wait a couple of seconds for the multimeter to register the change). Change the potentiometer resistance all the way to 1 kΩ (100%). What is the output voltage? Does this agree with your intuition and circuit properties? Hint: Think about what the voltage divider formula when  $R_1 = R_2$ .

## Examples

Simulate the following circuits:

a)



b)

