B. Muthuswamy and B. Boser
mbharat@cory.eecs.berkeley.edu

EECS Department, UC Berkeley
Summer 2007

SPICE / MultiSim Tutorial

1. Introduction

Cellular phones and computers are just two examples of some of today's extremely complex electronic systems. Such devices contain millions of circuit components, and simple trial and error is not an effective way of ensuring that the final product will work properly. As a result, designers often use circuit simulators to verify the performance of a circuit before fabrication.

The most popular component level circuit simulator available today is called SPICE (*S*imulation *P*rogram with *I*ntegrated *C*ircuit *E*mphasis), which was developed here at the University of California, Berkeley, in the 1970s under the guidance of Prof. Pederson. Today vendors offer many different versions of SPICE that differ mainly in the user interface but are internally very similar to the original "Berkeley SPICE". This tutorial introduces a version of SPICE called MultiSim.

Circuit simulation with SPICE (and MultiSim) involves two steps:

(1) Enter in the circuit schematic (with MultiSim's graphical user interface).
(2) Choose the type of analysis and run the simulation.

2. Organization of this Tutorial

Notes about this Tutorial:

- Before running a simulation, you should always have a general understanding of how your circuit works.
- In this document, `Boldface black` refers to actions you perform on the computer. Example: `Click` on the Options menu item.

3. MultiSim Environment

1. First, you need to log into a lab machine to use MultSim. Ask the TA in-charge for login information. You may also be able to use MultiSim by logging in remotely.

2. Once logged in, `Double-click` on the 

icon on the desktop. If a window appears with "Evaluation License" written in the middle of it, `click` the Evaluate button. After MultiSim finishes loading, you should see the screen shown below in Figure 1. This is called "Capture and Simulate" environment because you "Capture" your schematic by drawing it in MultiSim and then you "Simulate" it.

Figure 1 also shows the different parts of the MultiSim workspace; the location of the toolbars in your MultiSim window may be different.

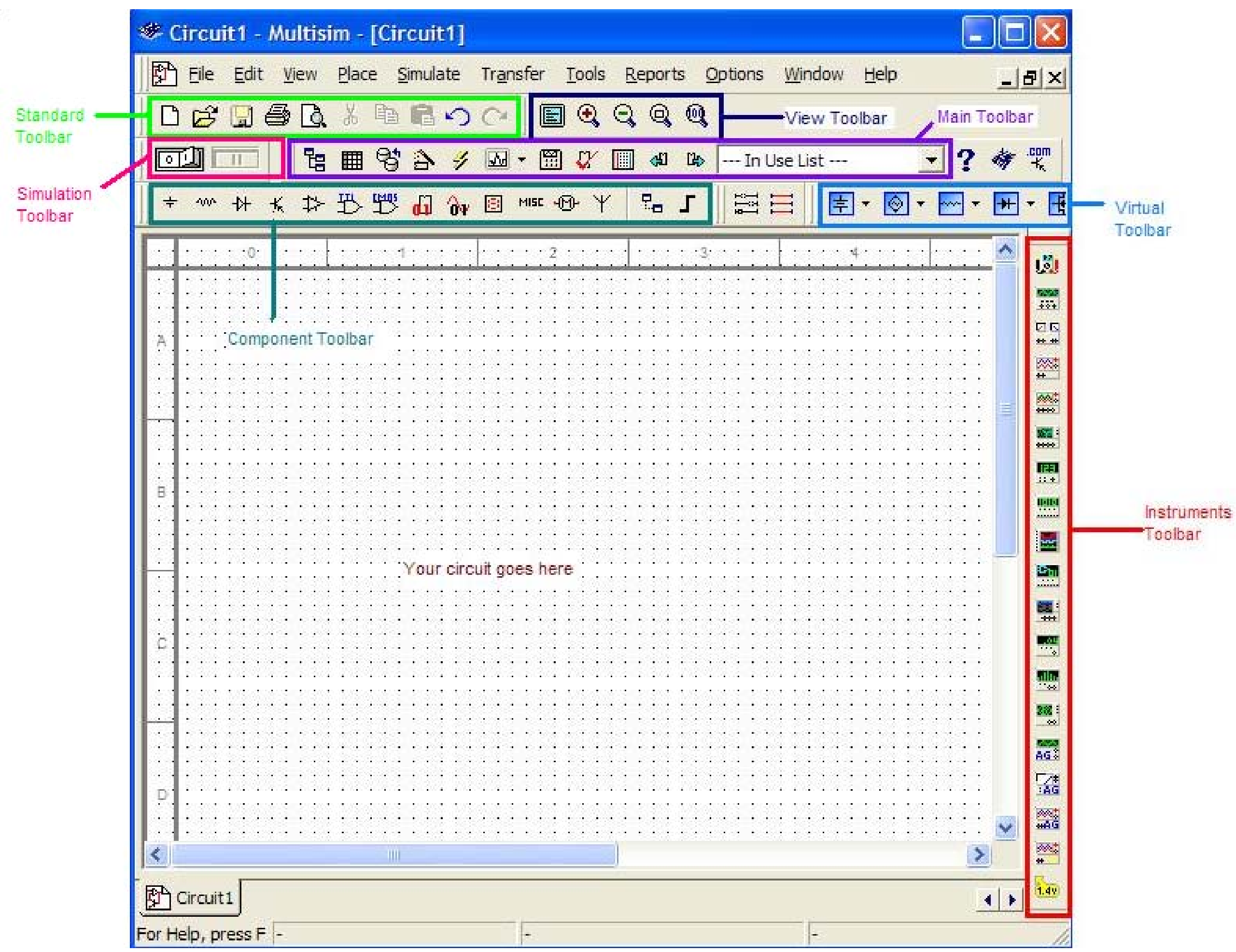


Figure 1 The most important components in the MultiSim workspace

The purpose of each toolbar will become clear as you move through this document. If you don't see the toolbars shown above, `click` on the View menu and go to Toolbars. Make sure that you at least have the toolbars shown in Figure 2 checked.

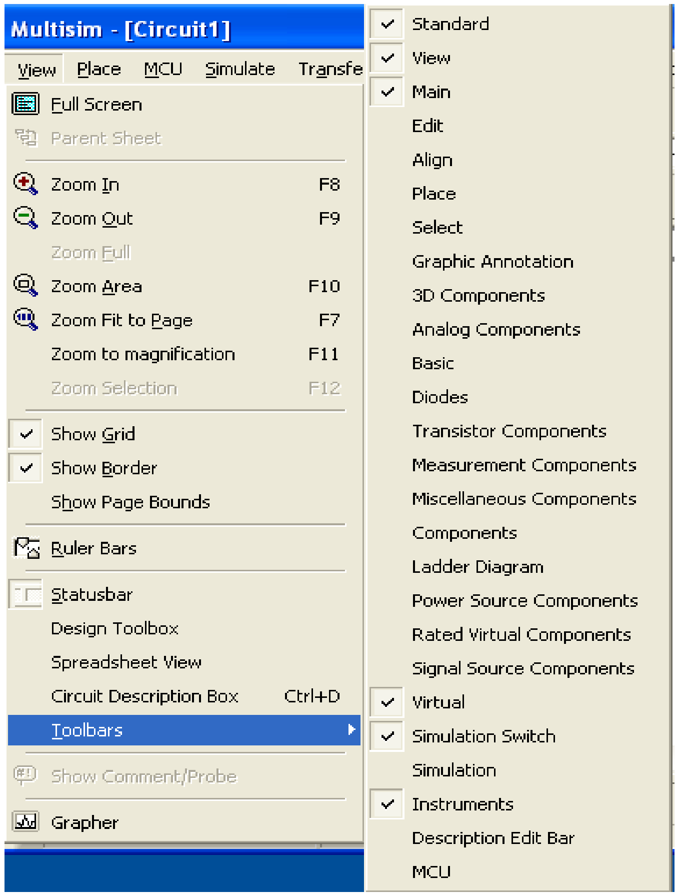


Figure 2 Viewing the toolbars

4. Schematic Capture (Entering a Simple Circuit)

To begin, let's construct the simple circuit shown below in **Error! Reference source not found.**. This circuit is composed of a voltage source (battery), a resistor, and a potentiometer (variable resistor).

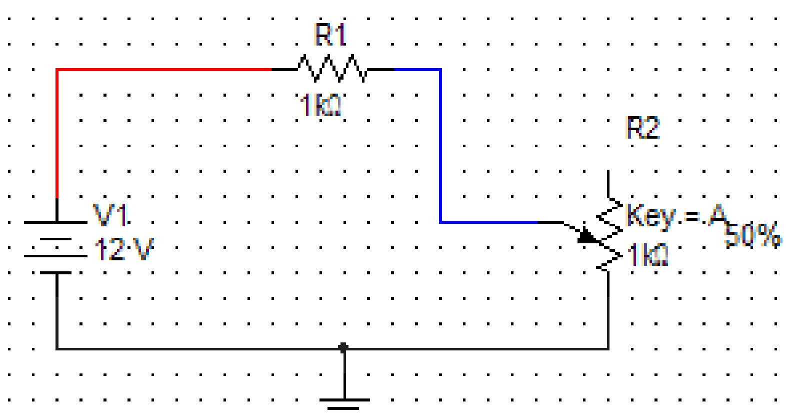


Figure 3 A simple circuit captured in MultiSim

I Adding the Voltage Source (Battery):

1. Click on the Power Source Family  in the Virtual Toolbar.
2. The Power Source Components 

will pop up.
3. Click on the DC Power Source icon  and then click on the workspace to place a battery. **Error! Reference source not found.** shows the result.

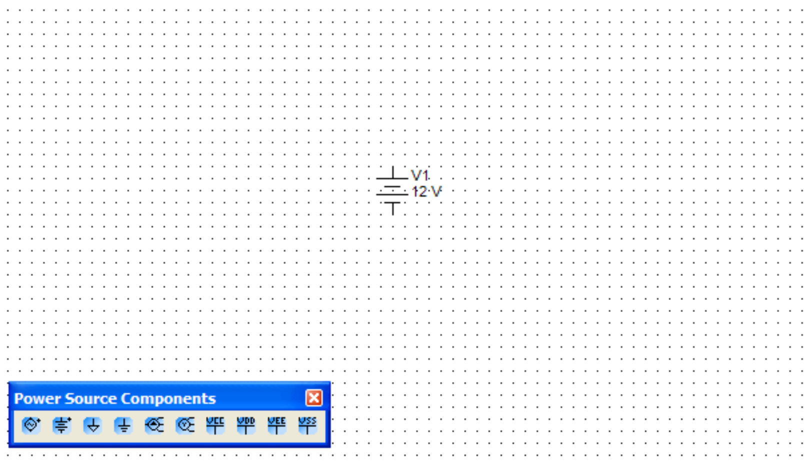


Figure 4 A DC power source in MultiSim

To change the value of the power source, Double-click the battery. This opens up the Power_Sources dialog box shown below. Make sure that the voltage is set to 12 V and then press OK.

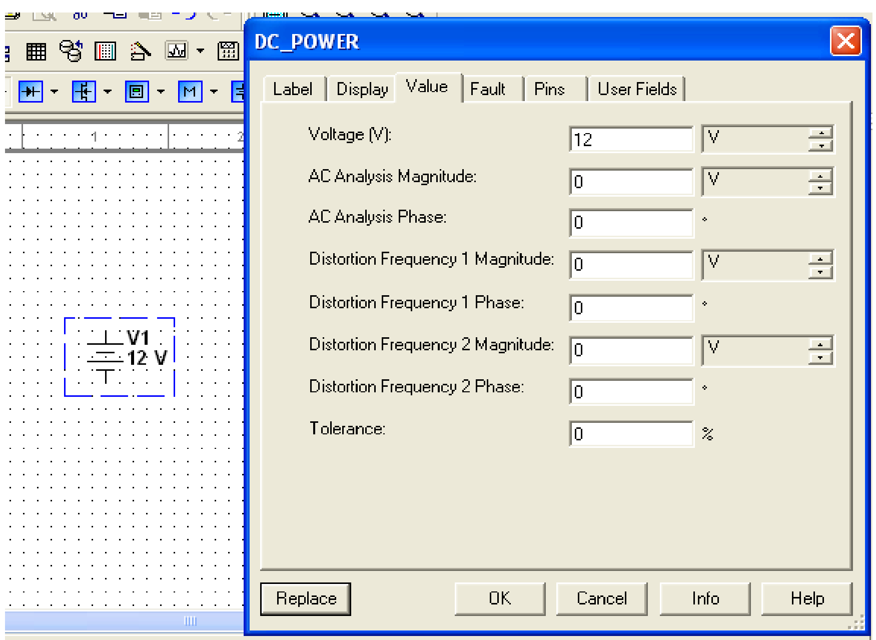
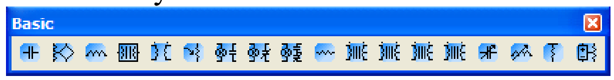
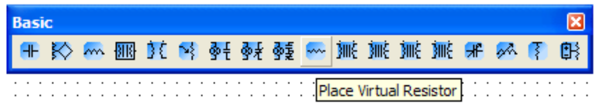


Figure 5 Power Sources Dialog box. Use this to change the value of the battery voltage.

II Adding the Resistor and the Potentiometer

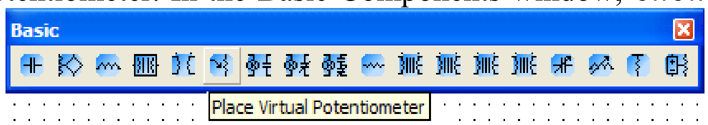
4. Click the Basic Components Family  in the Virtual Toolbar.
5. The Basic Components 

will pop up.



6. Click on the Virtual[1] Resistor tool and drag a resistor onto the workspace. Like the battery, you can double-click the resistor to change its properties.

7. Lastly, we must add the potentiometer. In the Basic Components window, click



on the Potentiometer Tool and drag a potentiometer onto the workspace. You can increase/decrease the resistance on the potentiometer by pressing 'a' / 'Shift+a' on the keyboard. The increase and decrease refers to the resistance between the middle leg and the bottom leg of the potentiometer. You can also double-click the potentiometer to open its properties and change the total resistance of the potentiometer or its increment / decrement value.

Error! Reference source not found. 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 kΩ: 500 Ω. If you press 'a', the resistance will increase by 5% (the resistance between the middle leg and the top leg will decrease by 5%). Again, you can Double-click the potentiometer to change the increment percentage. If you move your mouse over the potentiometer, you can also use the slider that appears to change its resistance.

---

[1] MultiSim distinguishes "Virtual" components from "Real" components. With real components, you place a part that has the actual shape of the real component, not a schematic symbol. You will see examples of this in Section 8.

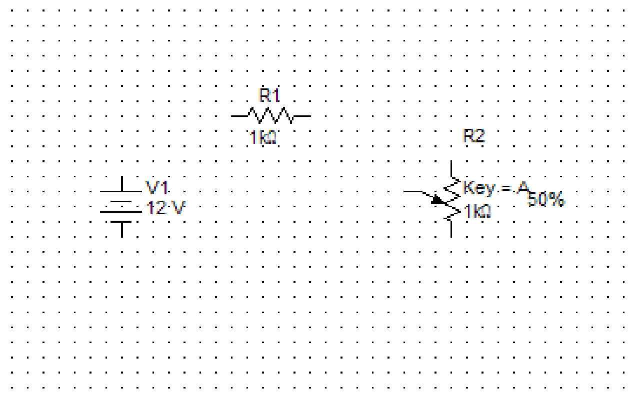
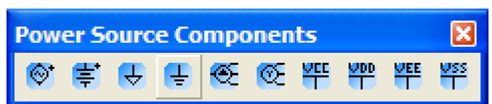
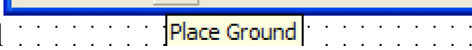


Figure 6 The circuit components are in place

III Adding the Ground

8. The final component to add is the ground. You cannot simulate the circuit without a ground, because 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, but for consistency, let's pick the node at the bottom of the circuit as ground.
9. 

   Click the Ground tool 

 in the Power Source Components menu. Drag the ground to the bottom of the circuit, the result is shown in **Error! Reference source not found.**.

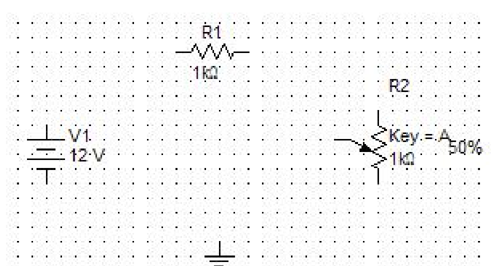


Figure 7 Circuit ready for wiring

10. Connections are placed by clicking on the terminal of the first component, moving the cursor the the target, and clicking again. **Error! Reference source not found.** shows the results of wiring the 12 V source to the 1 kΩ resistor.

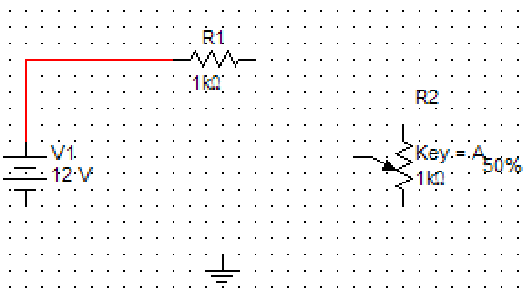


Figure 8 A wire connects the 12 V source to the 1 kΩ resistor

Complete the wiring as shown in **Error! Reference source not found.**. Make sure you connect the wire from the 1 kOhm resistor to the wiper of the potentiometer.

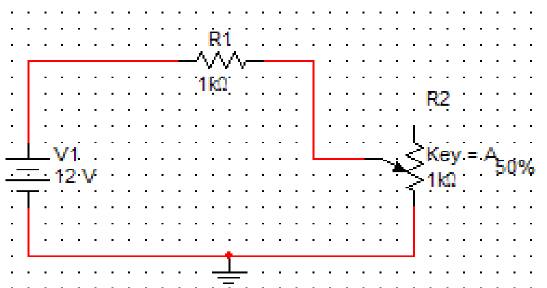


Figure 9 The wired circuit

To simplify debugging especially of larger circuits you can give the signals intuitive names, such as "Vin" and "Vout" and assign different colors to wires (e.g. red for power and black for ground). To do so right-click the wire and choose property. Type in the wire name and click the show box. Right-click the wire again and choose "Segment Color". Choose a color and press OK. Change the wire colors and add intuitive wire names as shown in Figure 10 below.

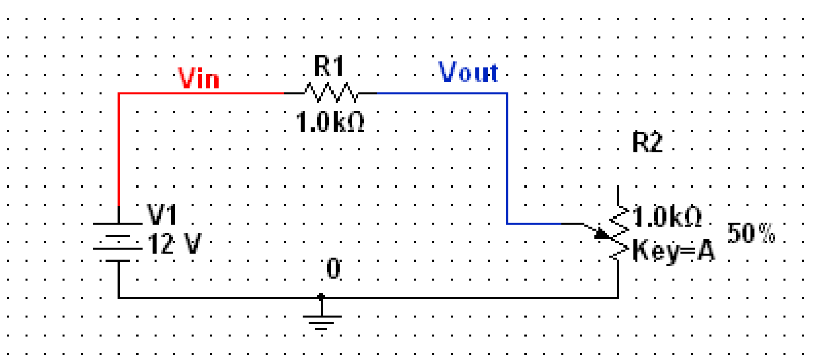


Figure 10 The circuit is ready for simulation

6. Running the Simulation and Using the Results Display

We are now ready to simulate our circuit. In the laboratory, you would now turn on the power use lab instruments, such as the multimeter and oscilloscope, to check voltages and currents in your circuit. Circuit simulators are much faster at finding the same result.

From the `Simulate` menu, choose `Analyses→DC Operating Point` .... The DC Operating Point Analysis window should pop up (**Error! Reference source not found.**). Next you choose circuit variables for analysis. V(vin) and V(vout) are the voltages and wire names you have given, and I(v1) is the current though the voltage supply[2]. Add all three variables to the panel on the right. More complex circuits have many more variables; in which case you would only choose the ones you are interested in.

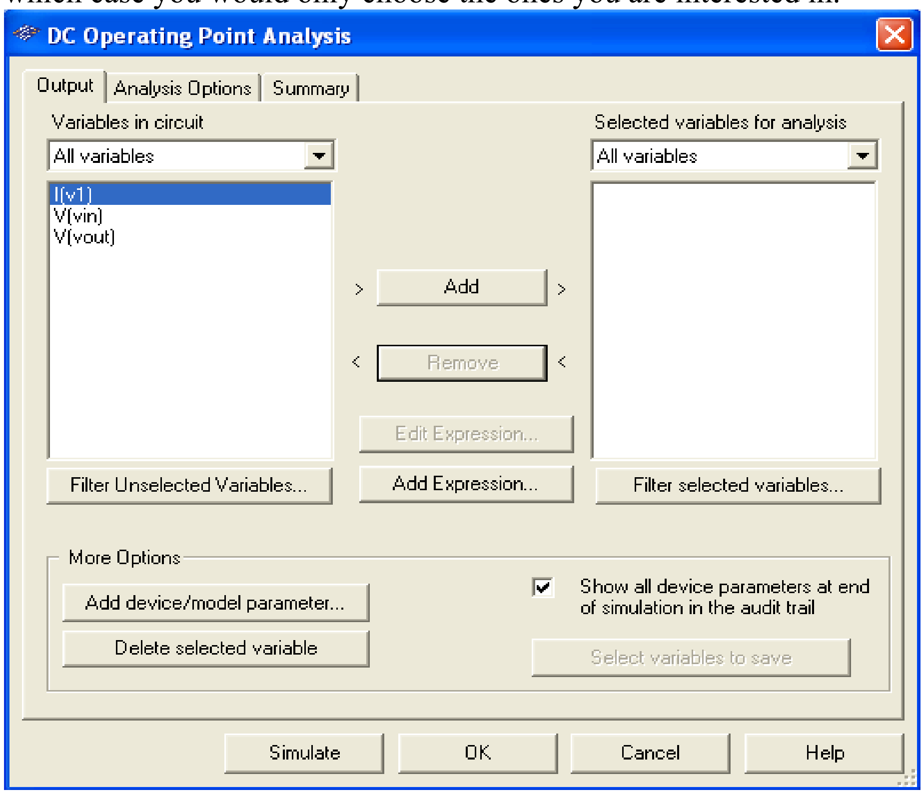


Figure 11 DC Operating Point Analysis Window

`Click` the "Simulate" button to perform the simulation. The window shown in **Error! Reference source not found.** appears with the results. With an input voltage of 12 V and the potentiometer set at 50% (500Ω), Vout is 4 V as expected. The current resulting from placing 12 V across a total resistance of 1.5kΩ is 8mA, which we can easily verify with Ohm's Law (V = IR). Since SPICE defines the current flowing into the positive terminal of the source as positive but the current actually runs in the opposite direction, it reports the result as a negative number. Play with the results by changing the resistance of the potentiometer by selecting it and pressing the "a" or "Shift-a" keys and rerunning the simulation.

---

[2] Voltage sources set to V=0 are the easiest way for determining branch current with SPICE (e.g. MultiSim).

B. Muthuswamy and B. Boser
mbharat@cory.eecs.berkeley.edu
EECS Department, UC Berkeley
Summer 2007

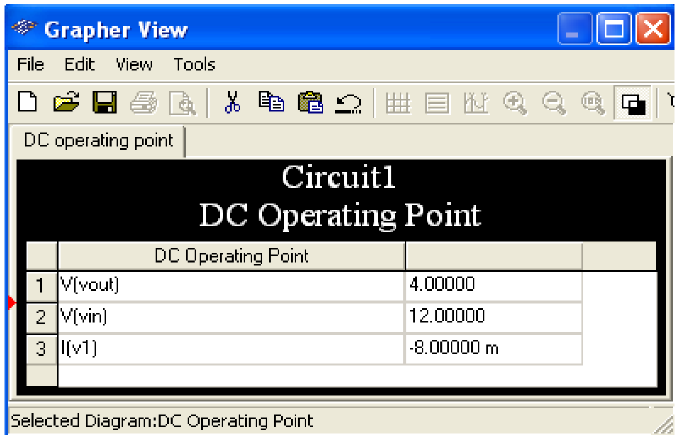


Figure 12 Simulation result for the DC operating point analysis

Analysis of Changing Signals

Most circuits deal with signals that are changing. SPICE offers three principal analyses for this purpose:

1. DC Sweep
2. Transient Analysis
3. AC Analysis

We will run each type of analysis on our sample circuit. These forms of analysis will appear throughout EE100 and other electronic circuit courses and laboratories.

DC Sweep

Choose `Simulate`→`Analyses`→`DC Sweep` and change the parameters so that they match the values in **Error! Reference source not found.**.

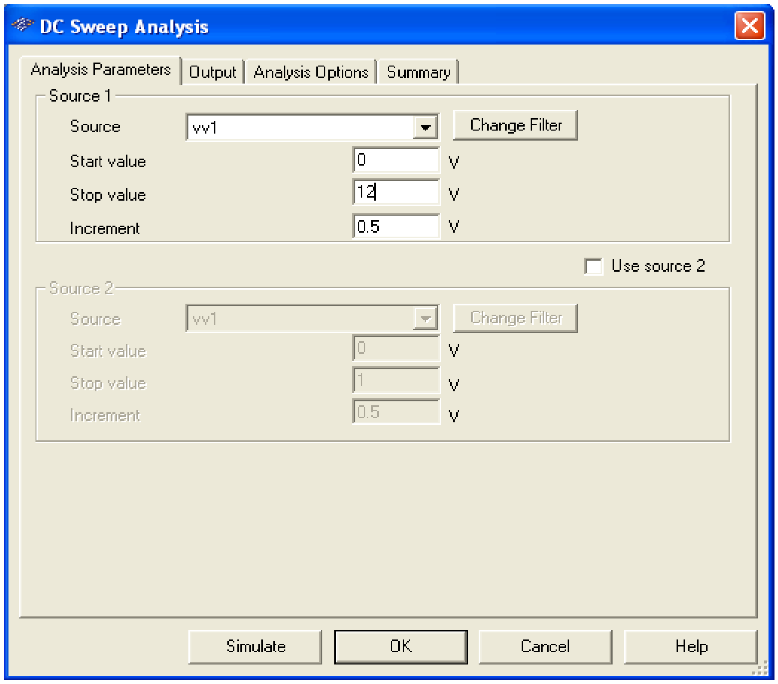


Figure 13 Analysis Parameters for DC sweep analysis

Then click on the Output tab and add V(vin) and V(vout) to the 'Selected Variables for Analysis' box as shown in Figure 14

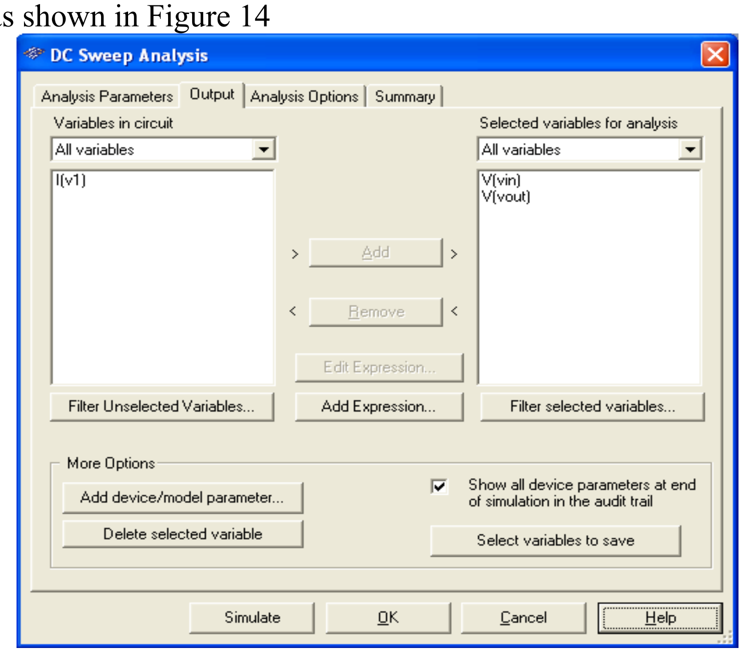


Figure 14 Analysis Parameters for DC sweep analysis

Finally, click the Simulate button to obtain the results shown in Figure 15.

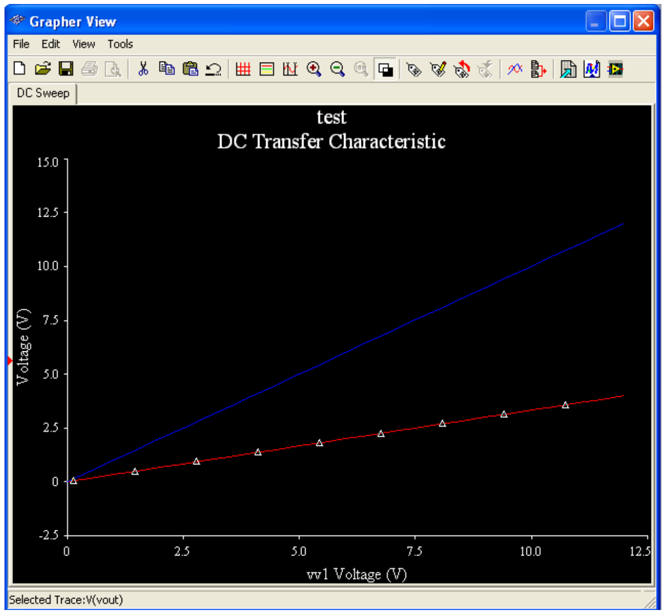


Figure 15 Result of DC sweep analysis

Can you find what percentage the potentiometer has been set to? Check your answer by running a simulation!

In the laboratory we can perform a DC analysis by repeatedly adjusting the input of the circuit and then measuring the output with the DMM. What a pain. Isn't MultiSim AWESOME!

Transient Analysis

Transient analysis evaluates how signals change over time. In the laboratory, you can examine this behavior with an oscilloscope. Figure 16 shows the circuit prepared for transient simulation.

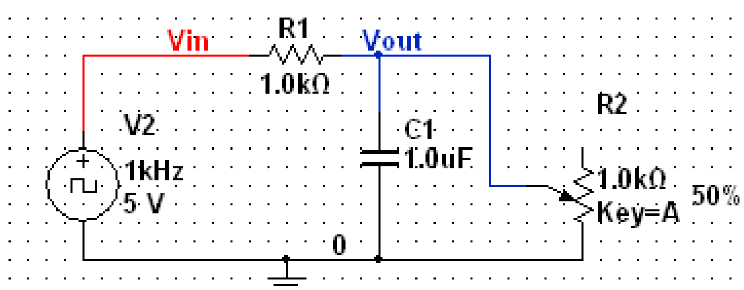
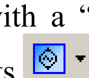


Figure 16 Modified circuit for transient analysis

The voltage source has been replaced with a "Clock Voltage Source" (which can be found under the Signal Source Components  button as shown in Figure 17).

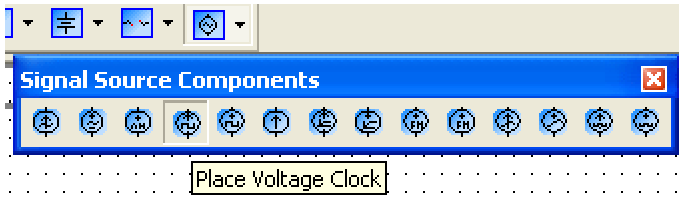
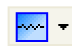


Figure 17 Adding the Clock Voltage Source

Double-click the clock voltage source and make sure that the voltage is set to 5 V and that the frequency is set to 1 kHz.

The capacitor C1 (which can be found under the Basic  button) has been added to make the circuit more interesting for transient analysis. You can rotate the capacitor by Right-clicking it and selecting '90° Clockwise' or '90° CounterCW'.

Next, choose Simulate→Analyses→Transient Analysis... and use the parameters from Figure 18. Make sure that V(vin) and V(vout) are in the 'Selected variables for analysis' box under the Output tab. Click the Simulate button! The simulation results should match those shown in **Error! Reference source not found.**9.

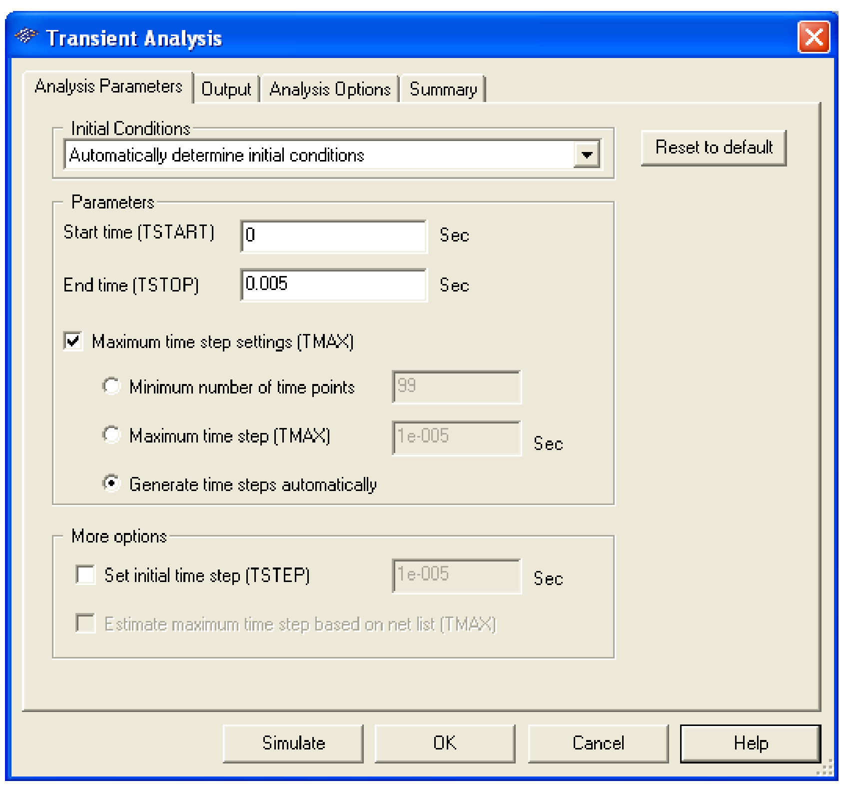


Figure 17 Transient analysis parameters

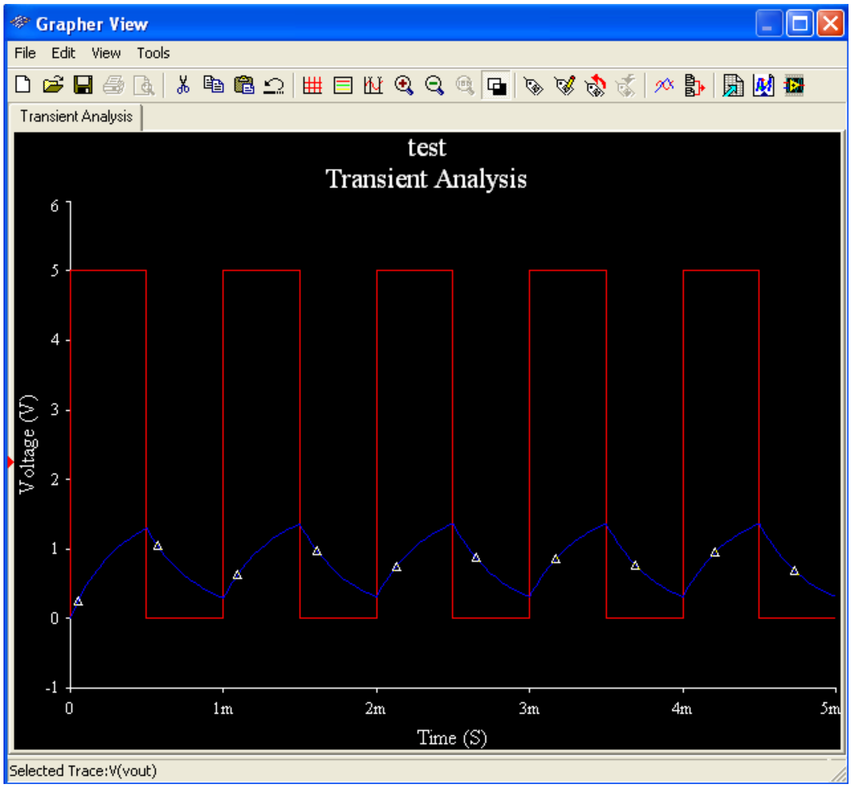


Figure 18 Transient analysis results

AC-Analysis

The AC-analysis directly computes the frequency response of a circuit and draws the Bode Plot (you may want to revisit this section later if Bode Plots have not been covered yet in the lecture). **Error! Reference source not found.**9 shows the sample circuit prepared for AC analysis. Notice that the clock voltage source has now been replaced by an AC voltage source. Like the clock voltage source, this can be found in the Signal Source Components Group.

Next, choose Simulate→Analyses→AC Analysis... and use the parameters from Figure 20. Make sure that only V(vout) are in the 'Selected variables for analysis' box under the Output tab. Click the Simulate button! The simulation results should match those shown in Figure 21.

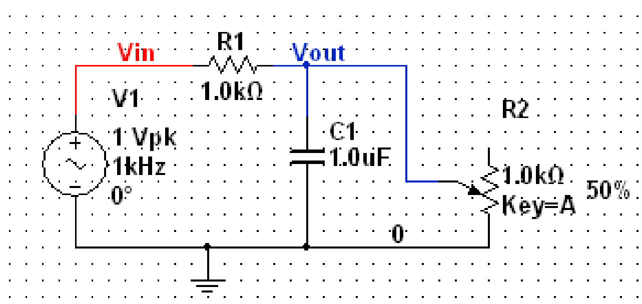


Figure 19 Circuit for AC analysis

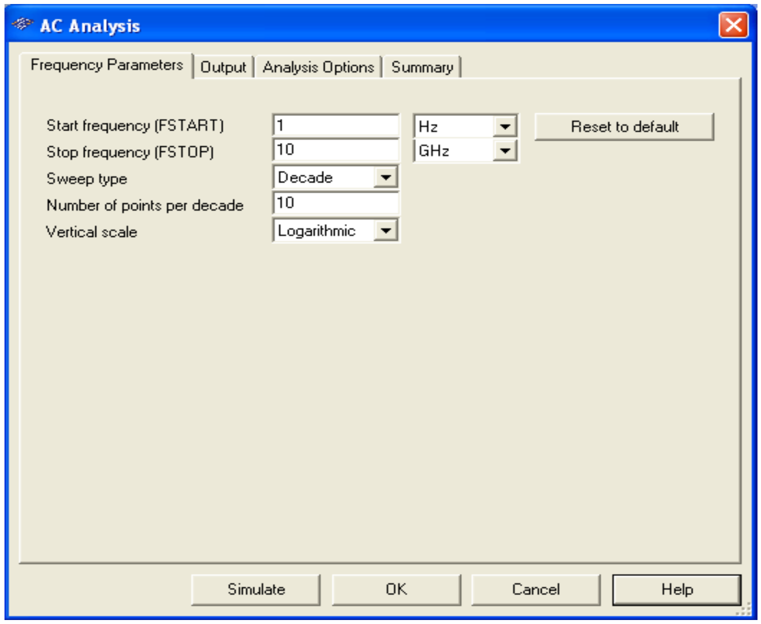


Figure 20 AC analysis simulation parameters

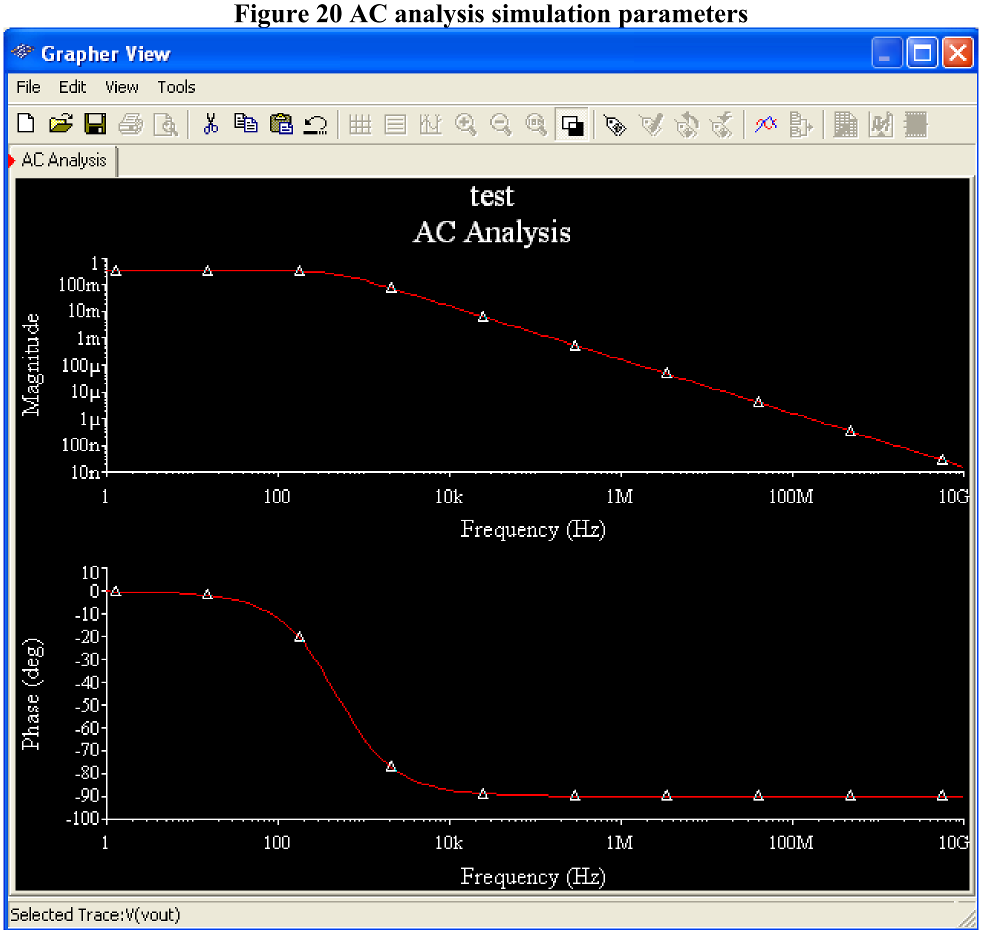


Figure 21 AC analysis simulation results for V_{out}

B. Muthuswamy and B. Boser
mbharat@cory.eecs.berkeley.edu
EECS Department, UC Berkeley
Summer 2007

II Alternative Ways of Circuit Simulation in MultiSim

7. Simulated Instruments

This section describes an alternative way to perform simulations with MultiSim that more closely resembles what you would do in the laboratory. Although this procedure may be more intuitive at first, similar to real lab work, it generally takes more time than the simulation approach described in Section 6.

This method involves using virtual instruments created in MultiSim that look and work just like those in the laboratory. To take any measurements from a simulation, we first need to add instruments. Hence your simulation environment is a step closer to your real lab environment.

Let's measure the voltage drop across the potentiometer.

1. First remove the capacitor and replace the AC voltage source with the original DC voltage source. Next, `Click` the Agilent Multimeter  in the Instruments Toolbar and drag the multimeter onto your workspace. **Error! Reference source not found.**2 shows the result.

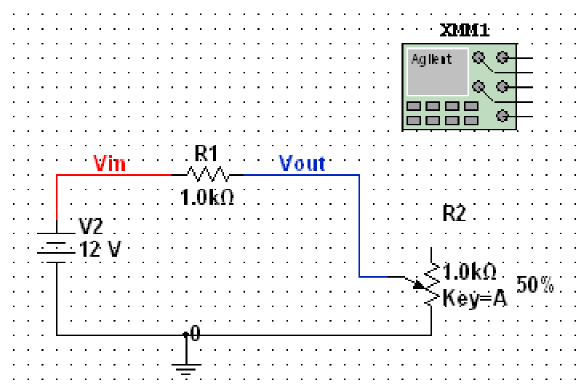


Figure 22 A multimeter placed on to workspace

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

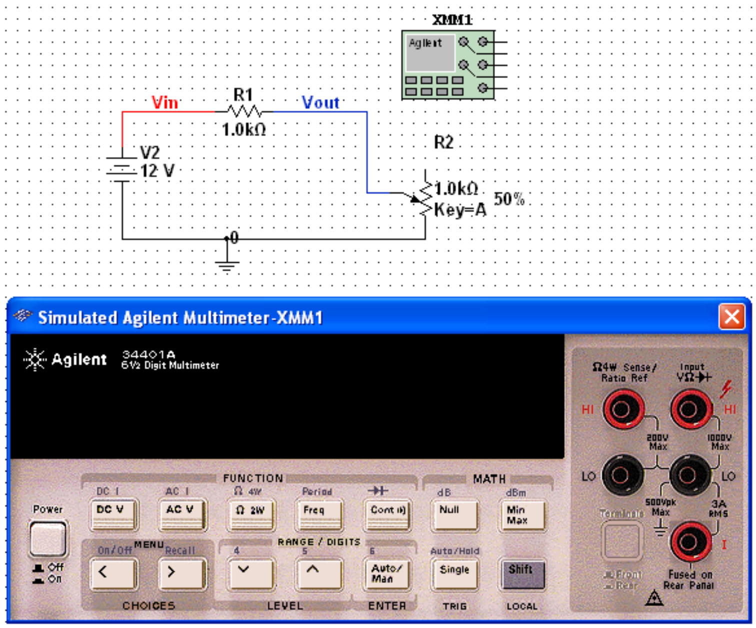
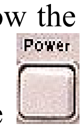
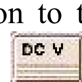


Figure 23 The Agilent 34401A Simulated Multimeter front panel

Notice how the simulated multimeter is the same as the one on your workbench! Click the  button to turn on the instrument. You will be measuring DC voltage, so Click the  button on the instrument. **Error! Reference source not found.**4 shows what you should get.

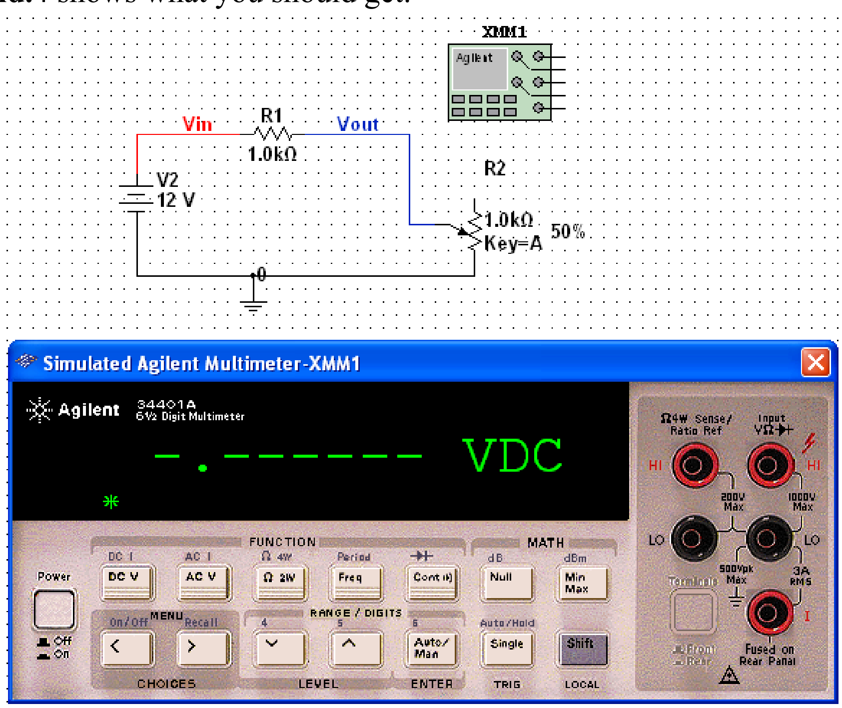


Figure 24 The Multimeter is set to the correct measurement mode

Lastly, draw wires from the multimeter terminals to the circuit as shown in **Error! Reference source not found.**5. As you make the connections, MultiSim highlights the terminals on the frontpanel.

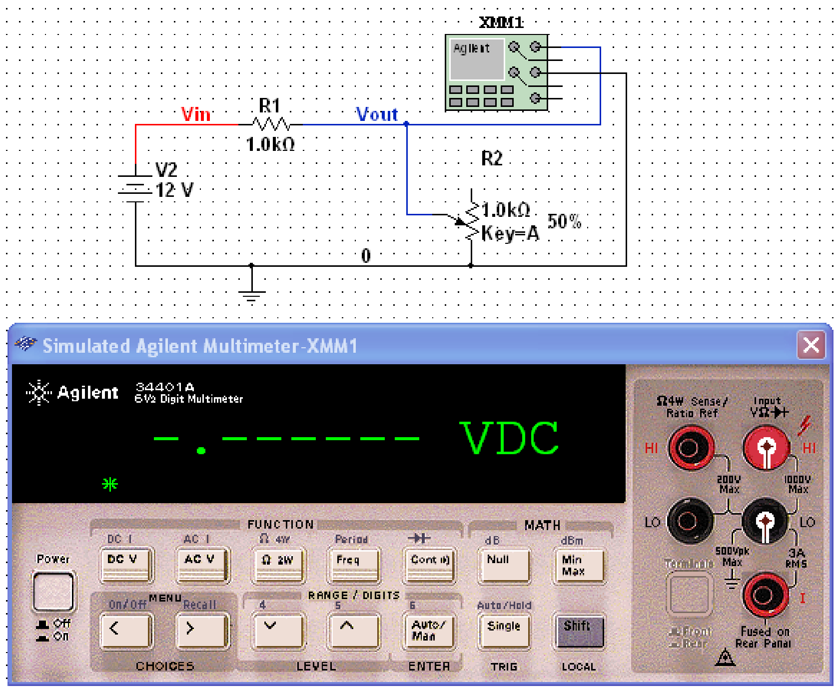
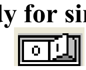


Figure 25 Ready for simulation

2. To simulate the circuit, `Click` the  button in the **Simulation Toolbar**. **Error! Reference source not found.**6 shows the result.

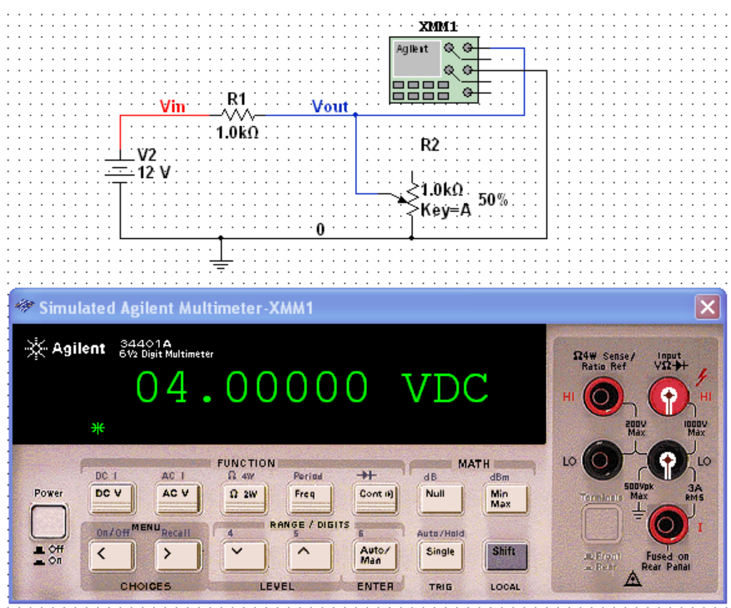


Figure 26 The simulation result

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? Hint: Think about what happens to the voltage divider formula when R1 = R2.

8. Using the Breadboard Tool

If you have trouble in the laboratory mapping circuit diagrams to the solderless breadboard, this section is for you. The breadboard tool allows you to see your circuit as if you had physically constructed it in lab. This tool is invaluable for large circuits (like your project) because it can help you plan an organized layout of the components before you actually build your circuit. Similar to using MultiSim's simulated instruments, though, this process tends to be time consuming. So once you are comfortable with schematic diagrams, you should probably forgo using the simulated breadboard tool.

We will try the breadboard tool with our simple DC circuit from Section 5. First, delete the multimeter from your circuit (be sure to turn of the simulation by pressing the  button before you try and delete the multimeter). Although you could wire the multimeter on the breadboard, it is inconvenient and unnecessary. The circuit to be wired should look like **Error! Reference source not found.**7.

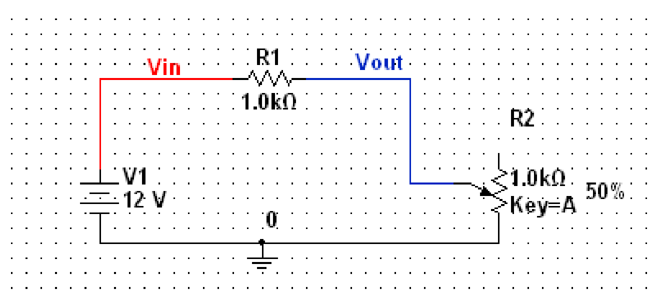


Figure 27 The simple DC circuit from the previous section

1. Click on the Breadboard icon  in the **Main Toolbar** to open the Breadboard view. **Error! Reference source not found.**8 shows the breadboard view.

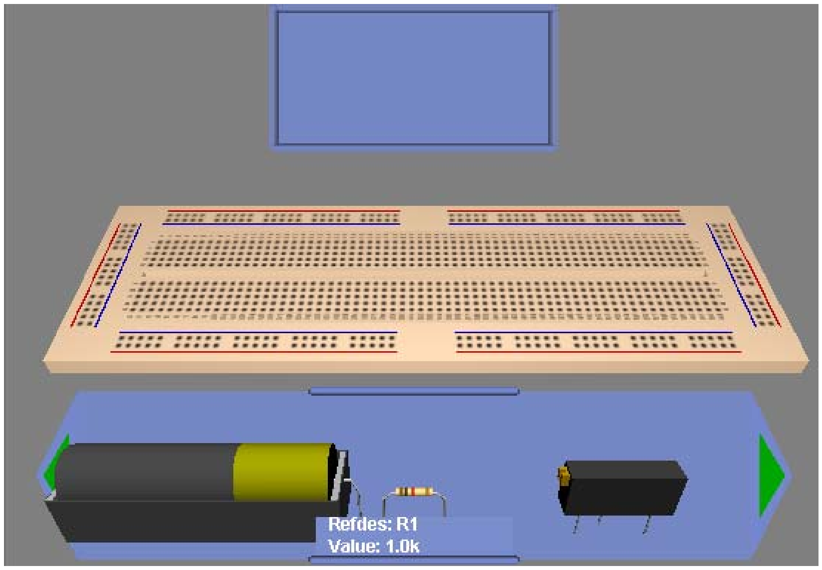


Figure 28 The Breadboard view in MultiSim

The tray at the bottom has all of the circuit elements in your schematic. For us, that includes a battery, a potentiometer and a resistor. You can change the size of your breadboard by clicking Options and then selecting Breadboard Settings.

You can rotate the breadboard by moving the cursor outside of the breadboard or to the middle of the breadboard until it changes to a set of double arrows. Click and drag the mouse to rotate the breadboard. If you hold the middle-mouse button, you can drag the mouse to move or translate across your breadboard. If you move the cursor over any other area of the breadboard, you get a small wire pointer. Use this tool to place wires on the breadboard. Click one slot on the breadboard and drag a wire to another slot. **Error! Reference source not found.**9 shows a wire on the breadboard. MultiSim highlights the point on the breadboard that you are wiring to, which makes wiring easier when you have a lot of components on the breadboard.

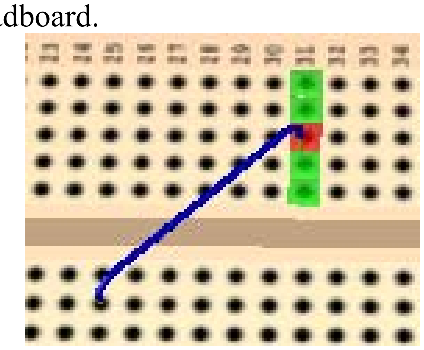


Figure 29 Breadboard wiring in MultiSim

To change the wire color, click the BreadBoard Wire Color  icon in the top toolbar. It is a good idea to stick to the wire colors you followed when wiring the schematic.

2. `Click` and drag the battery from the bottom tray to the breadboard. Use `Ctrl+R` to rotate the battery so the position is as shown in 30.

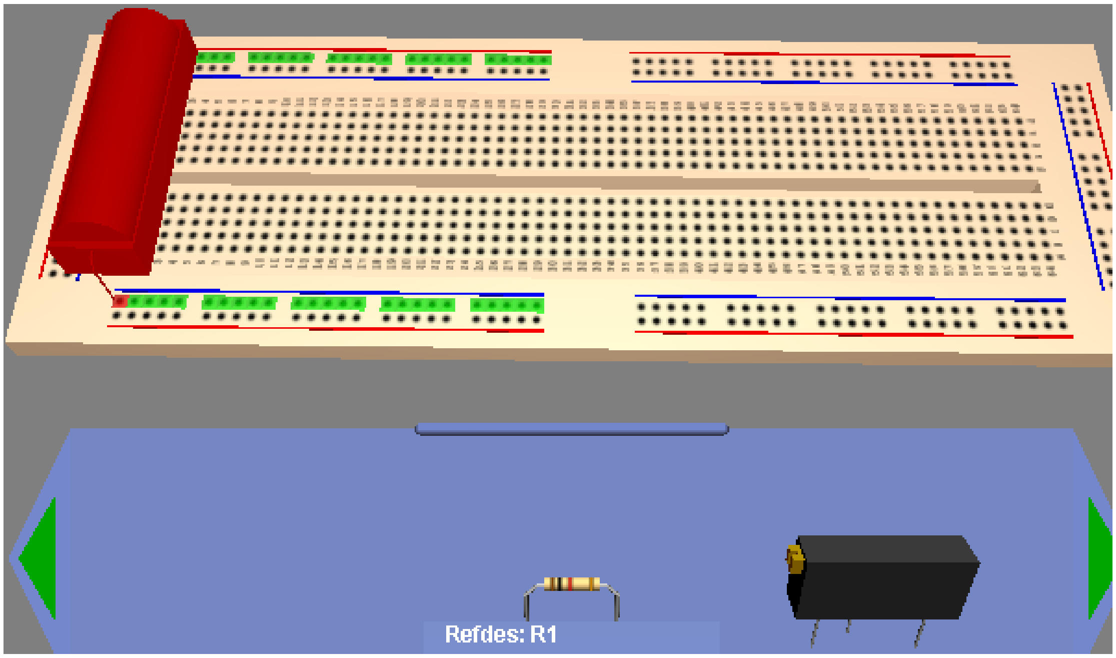


Figure 30 The battery is placed on the breadboard. We are using the outer connectors for the power, which is the convention followed when using a breadboard.

3. Place the resistor and potentiometer as shown in **Error! Reference source not found.**1. Once you place all of the components, the tray disappears. You can use the Zoom icons  on the top toolbar to get a better look at the components.

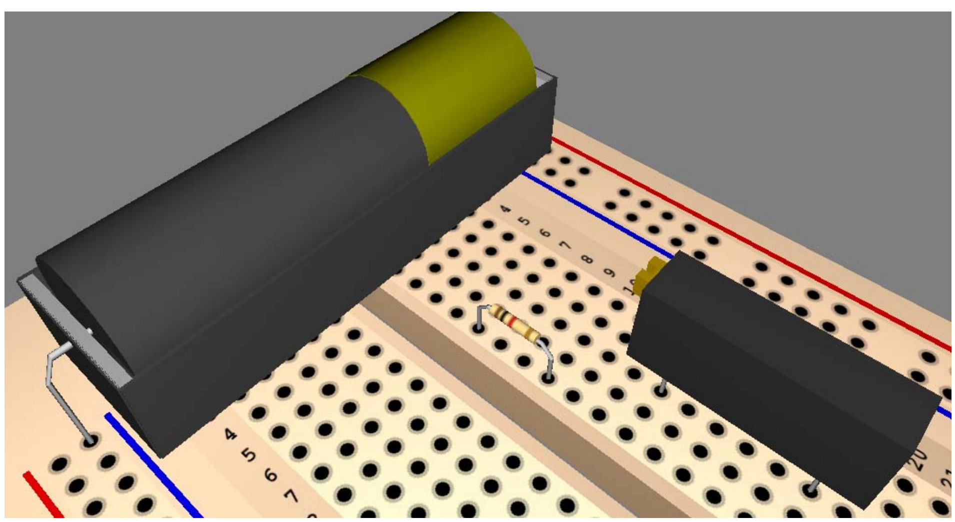


Figure 31 Components have been placed on the breadboard.

4. Wire the components as shown in **Error! Reference source not found.**2. Again it is prudent to follow the color convention you used on the schematic. As you wire, MultiSim actually highlights the connection end-point. Remember to draw the wire from the 1 kΩ resistor to the wiper (middle-leg) of the potentiometer.

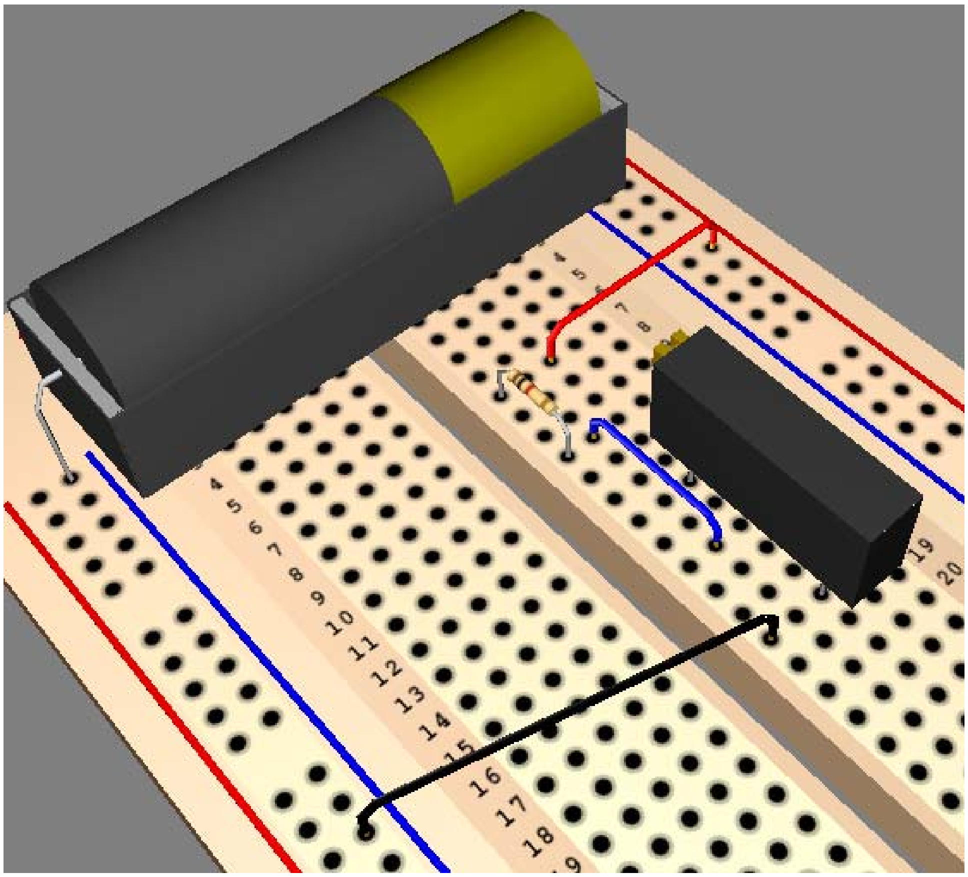
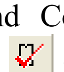


Figure 32 Wiring complete

5. A powerful feature of the MultiSim breadboard tool is the DRC (Design-Rules-Check) and Connectivity check. DRC checks if you have wires on the breadboard that are not on the schematic. Connectivity checks if your components are actually connected to each other.

   Let's run the DRC and Connectivity check. Click the Perform DRC and Connectivity check icon  in the top toolbar. The status window at the bottom shows the results, which should match **Error! Reference source not found.**3.

```
Multisim - 2007-06-28 20:18:44

---Design Rule Check---
---0 Design Rule Errors Found---

---Connectivity Check---
---0 Connectivity Errors Found---
```

B. Muthuswamy and B. Boser
mbharat@cory.eecs.berkeley.edu

Figure 33 Results from the DRC and Connectivity check

6. Let's introduce a connectivity error. Delete the wire connected to the positive terminal of the battery (the red wire). The result is shown in **Error! Reference source not found.**4.

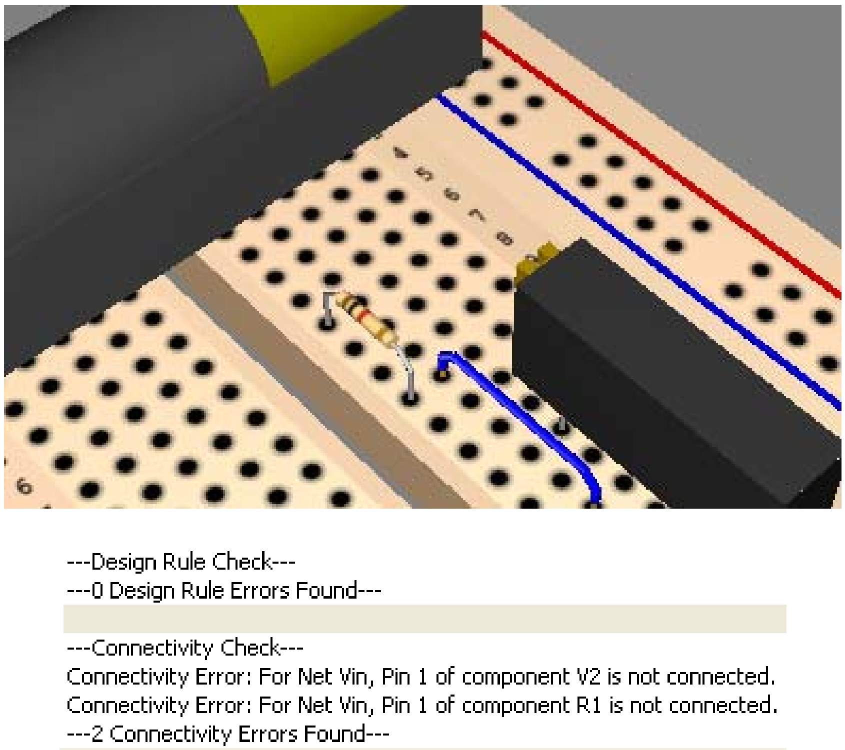
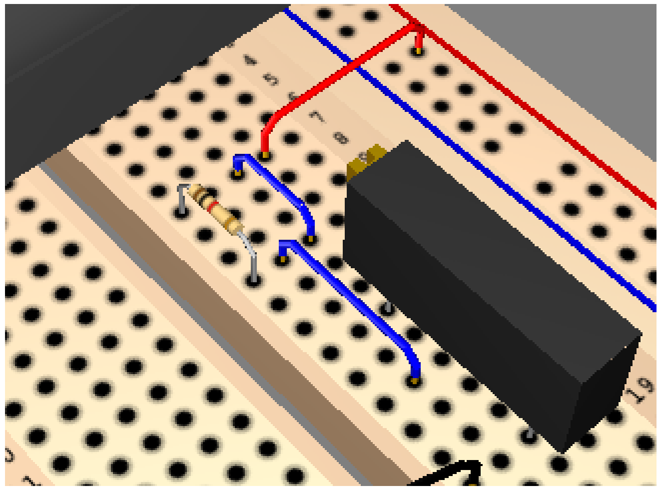


Figure 34 A Connectivity error has been introduced

If you rerun the DRC and Connectivity check, you should get a whole bunch of Connectivity errors.

7. Now let's introduce a design error. Rewire the positive terminal of the battery but short the 1 kΩ resistor by adding a wire in parallel with it. The result is shown in **Error! Reference source not found.**5.




```
---Design Rule Check---
Design Rule Error: Pin 1 of component R1 in net Vin is connected to pin 2 of component R1 in net Vout
Design Rule Error: Pin 1 of component R1 in net Vin is connected to pin 2 of component R2 in net Vout
Design Rule Error: Pin 2 of component R1 in net Vout is connected to pin 1 of component V2 in net Vin
Design Rule Error: Pin 1 of component V2 in net Vin is connected to pin 2 of component R2 in net Vout
---4 Design Rule Errors Found---

---Connectivity Check---
---0 Connectivity Errors Found---
```

Figure 35 A Design error

If you rerun the DRC and Connectivity check, you will get a bunch of Design errors. Delete the extra wire to remove the problem.

In this section you saw how you to use the breadboard tool to quickly wire your circuit on a virtual breadboard. The main purpose of this tool is to give you an idea of the component layout on the breadboard. For a simple example like this, using the tool is overkill. But for more complicated circuits like your class project, the breadboard tool may be invaluable in helping you plan out your component layout.

The Concept of a Ground

Nevertheless, this simple circuit does introduce a very powerful concept. Notice that we did not place a ground on the breadboard and no error occurred. Hopefully, this rather subtle point help clarifies the concept of a ground: it is just a symbol on your circuit that indicates your reference node. A circuit does not need to have an explicit ground connection to Earth (unless you are dealing with very high voltages and want to provide a safe return path). Many circuits do not have any explicit ground connection to Earth

8. Conclusion

This document has barely scratched the surface of MultSim, and there are many more powerful tools that are a part of this version of SPICE. Hopefully this document did give you a strong start in circuit simulation using MultiSim. The best way to learn is to experiment, don't be afraid to try out complicated circuits and MultiSim's new features.

Acknowledgements

Many thanks to Ferenc, Win, Daniel, Pete, Ming and Tho from the ESG (Electronic Support Group) at the University of California, Berkeley. Without their help it would not have been possible to write this document! Professor Dick White helped start the transition to MultiSim. National Instruments' generous donations help run the EE100 labs. Zach Nelson and Evan Robinson have been particularly helpful in this project.