Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu
EECS Department, UC Berkeley
Summer 2006
Version 1.00

EE100/EE42 MultiSim Tutorial

1. Introduction

The purpose of this document is to introduce the many features of MultiSim 8 from the perspective of EE100/EE42[1] (henceforth referred to as "EE100") course at the University of California, Berkeley. A student taking EE100 is expected to read and understand Sections 1 through 9 of this document half-way through the semester. Check the **Organization** (next) section to look at what students cover during the first half of a semester. The reference for this document is *MultiSim 8 Simulation and Capture User's Guide* (May 2005) from National Instruments.

EE100 is the introduction to electrical engineering class for non-EE majors. Most of the students in EE100 are engineering students who haven't used a circuit simulation tool. They need the concepts from EE100 for use in their capstone design course (like ME 102). Thus it is imperative these students be introduced to a robust and easy-to-use circuit simulation tool. It is not productive to teach these students the archaic SPICE[2] deck system. PSPICE is a good GUI front-end to SPICE, however PSPICE still relies on archaic concepts from SPICE (like Probe).

MultiSim is also a GUI front end to SPICE but it overcomes the difficulties mentioned above and also provides extra features like the breadboard tool. We have been using MultSim in our labs over the last year. It has been a big success, however there is not enough documentation explaining the subtle features of MultiSim. The aim of this document is to provide such an explanation so beginners can easy simulate complex circuits. Thus the document explains only the Capture and Simulation environment in MultiSim. For explanation of other features (like Printed Circuit Board design with UltiBoard) check the National Instruments website. In addition, MultiSim's help files are very useful. If you are stuck and can't find the answer in this document, please refer to MultiSim's help.

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.

2. Organization

This document is organized as shown below.

1. Introduction
2. Organization

[1] Website: http://inst.eecs.berkeley.edu/~ee100

[2] SPICE: *S*imulation *P*rogram with *I*ntegrated *C*ircuit *E*mphasis. This standard circuit simulation tool was developed at the University of California, Berkeley in the 1970s under the guidance of the late Dr. Donald Pederson.

3. Intended Audience and Conventions Used in this Document

This document assumes you know the basics of using the Windows Operating System. It also assumes that you are using MultiSim to check your circuits from EE100, hence it assumes you have a basic idea of how the circuit works. As a general rule, you should know how your circuit works so you can figure out if your simulation makes sense or not. As for the conventions in this document:

- Arial Black 10 pt. Font refers to actions you perform on the computer. Example: Left-click on the Options menu item.
- If possible, screen shots will be used for clarity.

4. The Capture and Simulate Environment

1. First, you need to log into a machine in the lab in order to use MultSim. Ask the TA in-charge for login information. You may be able to use MultiSim by logging into a remote computer. Ask your professor for more information on this.
2. Once you have access to MultiSim, Double-click with the left mouse button (henceforth referred to as Double-click) on the 

icon to start MultSim. 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.

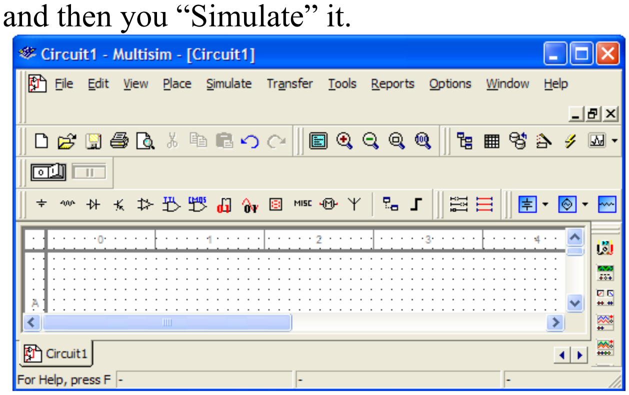


Figure 1. MultiSim startup screen

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

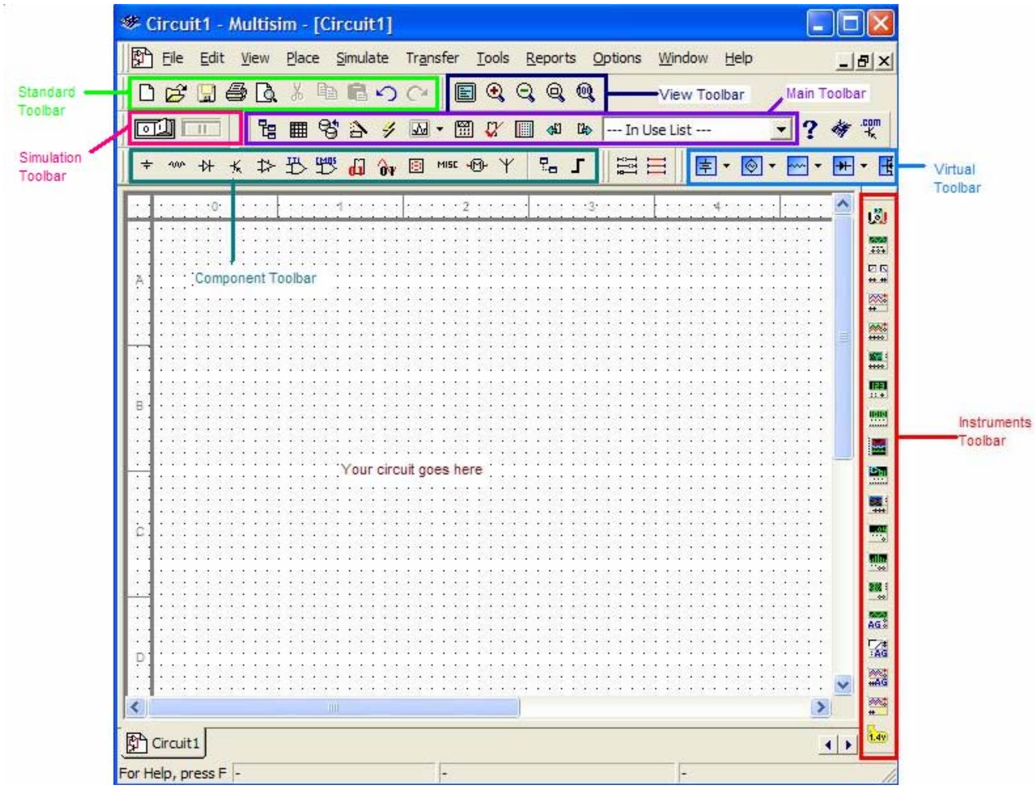


Figure 2. The most important components in the MultiSim workspace

The purpose of each toolbar will become clear as you move through this document[3]. 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.

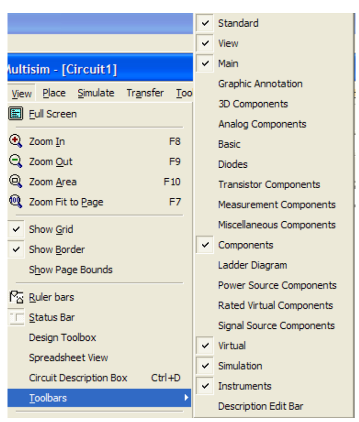


Figure 3. Viewing the toolbars

[3] If you want more information, refer to the *MultiSim 8 Simulation and Capture User's Guide*, Chapter 3 User Interface. This document is available on the EE100 homepage under handouts.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

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).

5. First Example: Simple DC Analysis in MultiSim

Lets construct the simple circuit shown in figure 4.

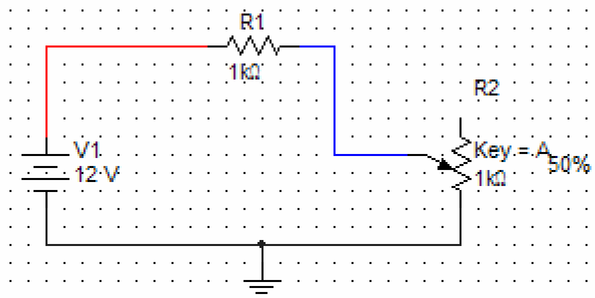
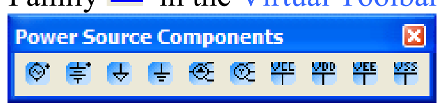


Figure 4. A simple series circuit constructed in MultSim

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.

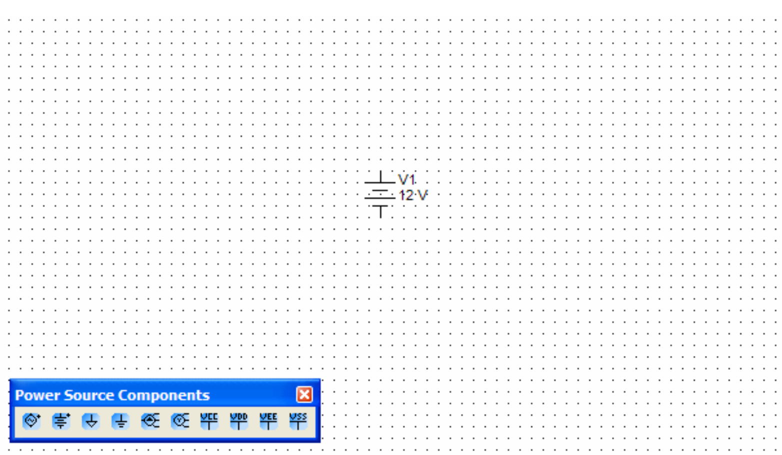


Figure 5. A DC power source in MultiSim

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.

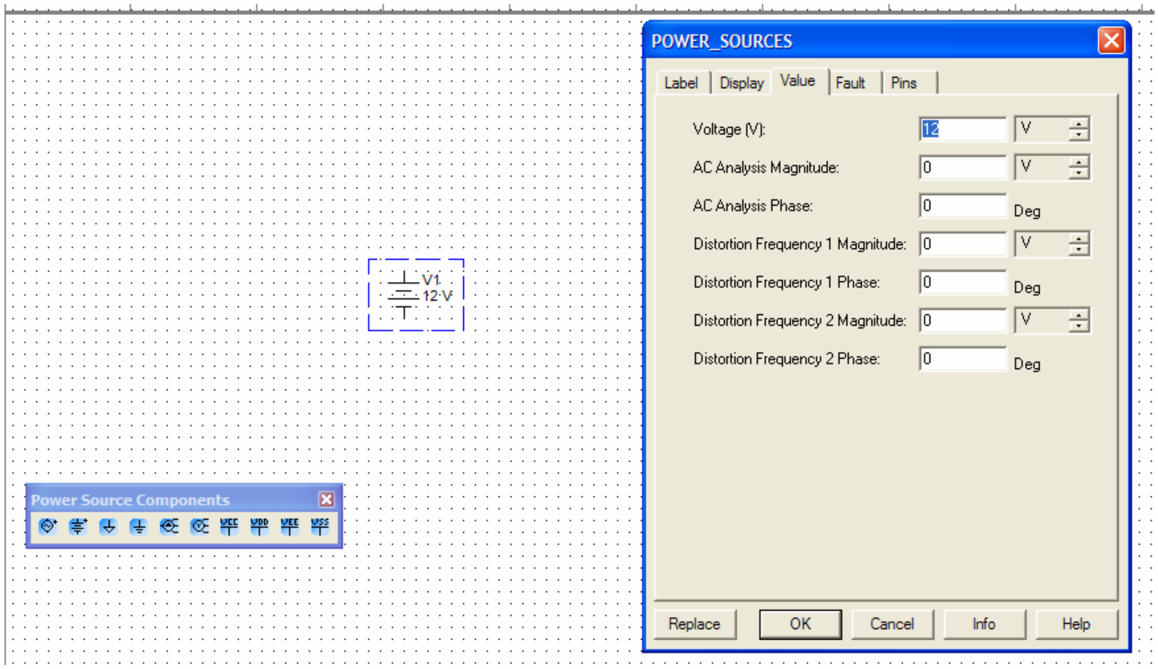
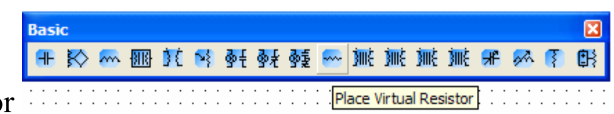
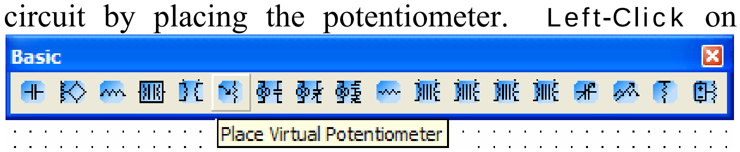


Figure 6. Power Sources Dialog box. Use this to change the value of the battery voltage.

4. Lets 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[4] 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. Lets 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.

[4] 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 10.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

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 kΩ: 500 Ω. 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.

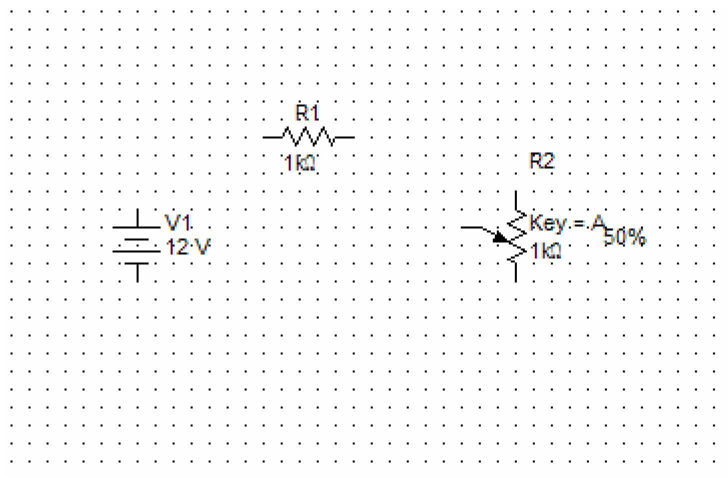
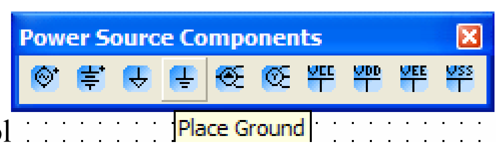


Figure 7. The circuit components are in place.

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.

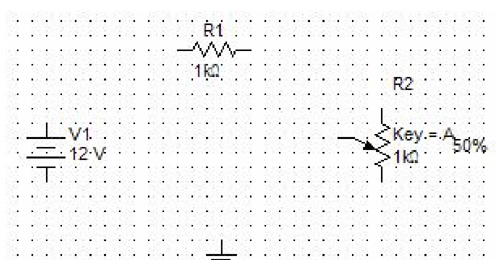


Figure 8. Circuit ready for wiring

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.

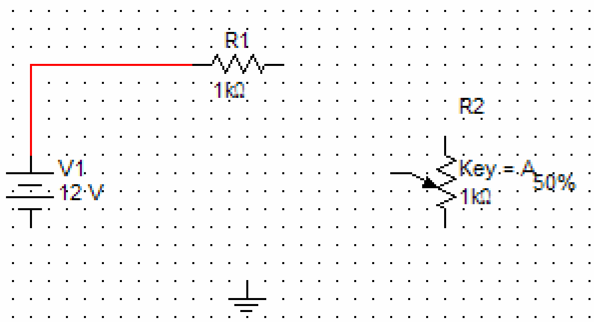


Figure 9. A wire connects the 12 V source to the 1 kΩ resistor.

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

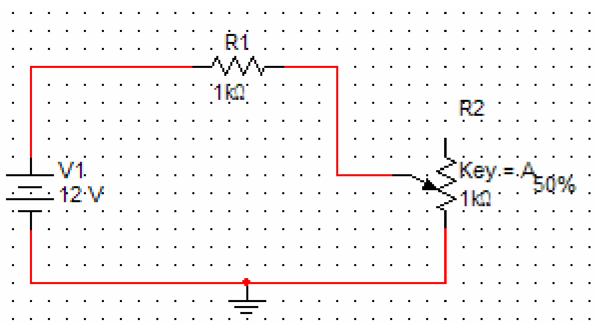


Figure 10. The circuit is complete

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.

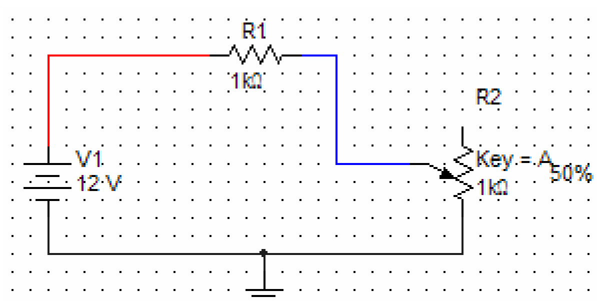


Figure 11. The circuit is ready for simulation

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.

Lets 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 12 shows the result.

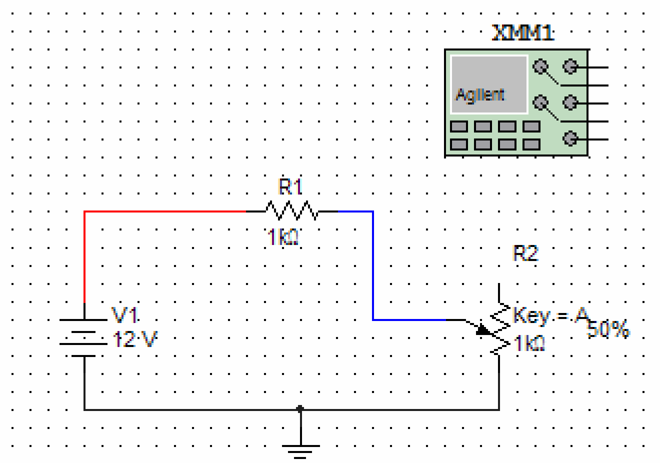


Figure 12. A multimeter placed on to workspace.

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

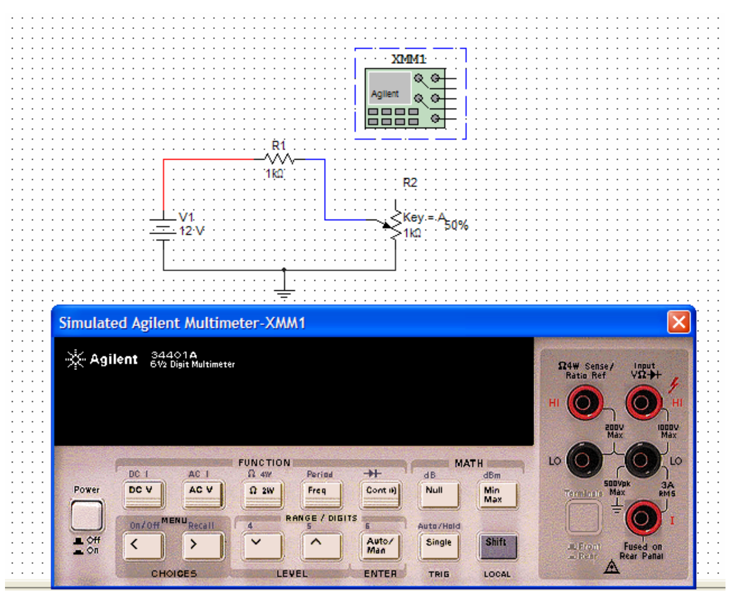


Figure 13. The Agilent 34401A Simulated Multimeter front panel

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

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 14 shows what you should get.

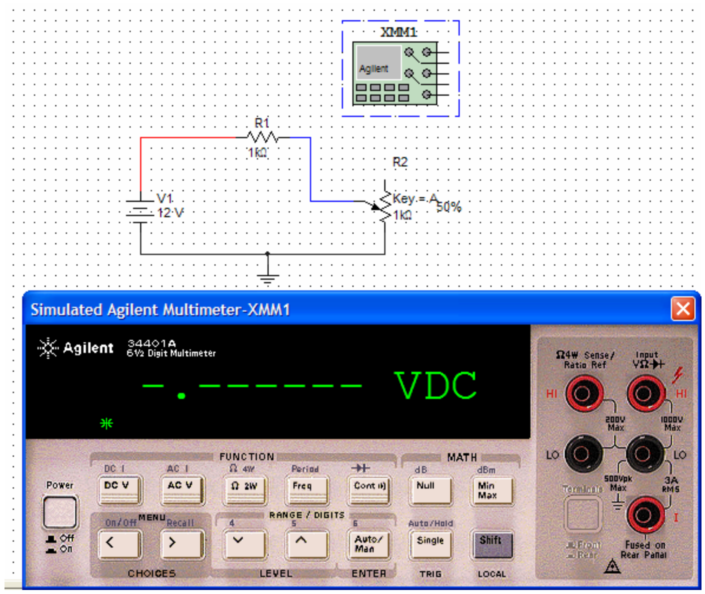


Figure 14. The Multimeter is set to the correct measurement mode

All that is left is to wire the multimeter terminals. Complete the wiring as shown in figure 15. Connect to the wires to the multimeter on the workspace. As you make the connections, MultiSim highlights the terminals on the frontpanel.

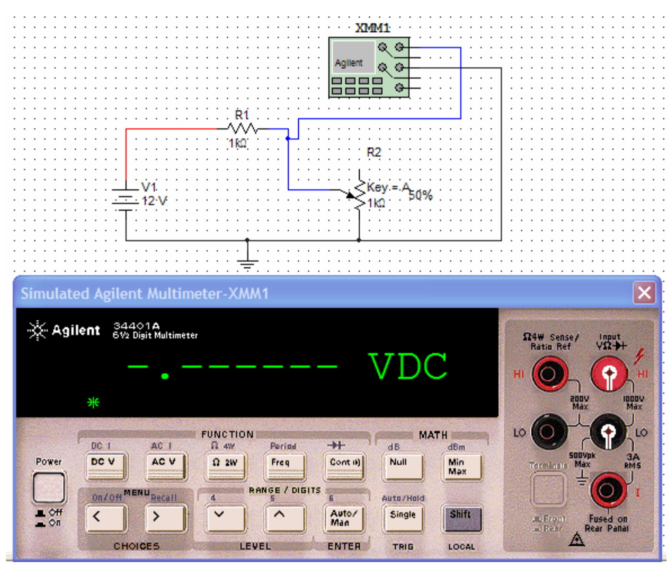


Figure 15. Ready for simulation

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

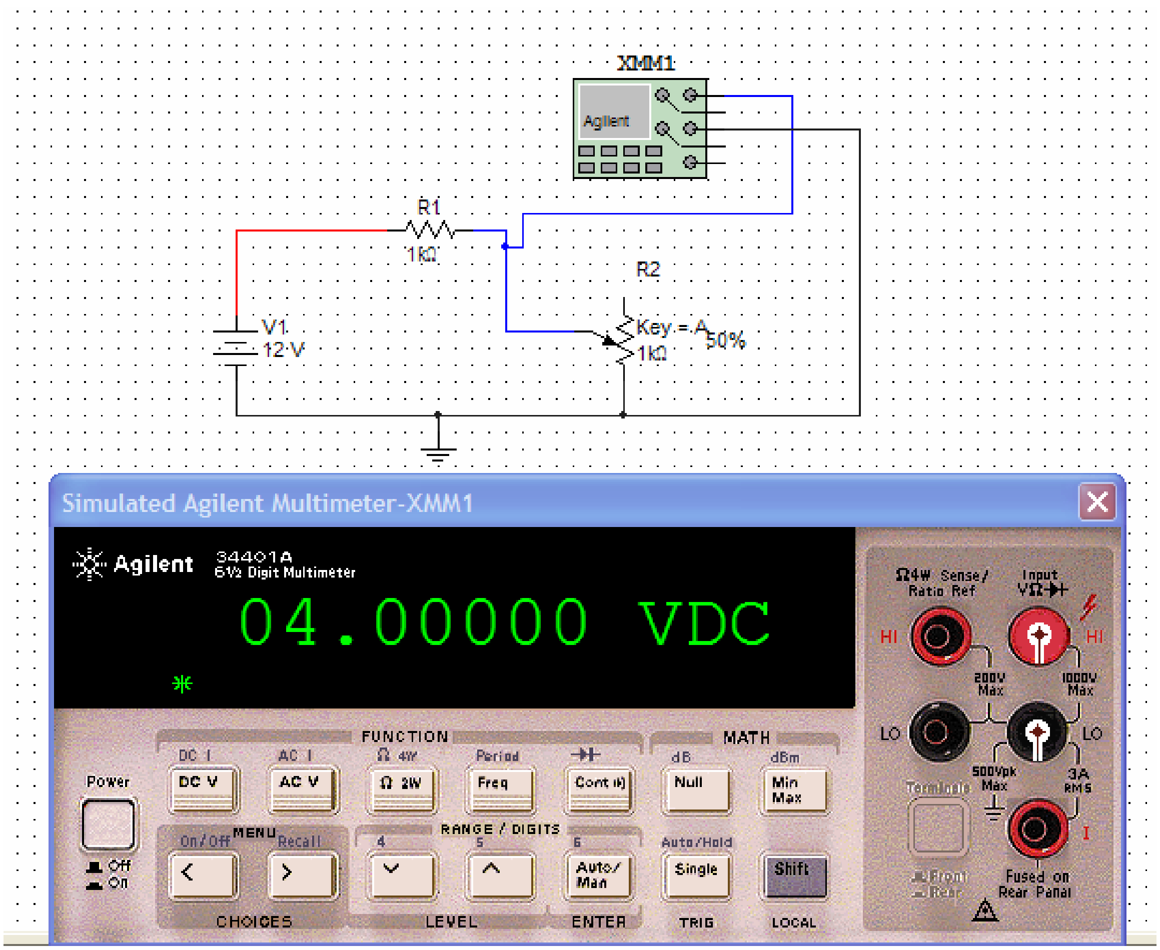


Figure 16. The simulation result

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 and circuit properties? Hint: Think about what the voltage divider formula when R1 = R2.

In the next section, we will see how to use the Breadboard tool to visualize how the circuit will look on a breadboard. This tool is invaluable for large circuits (like your project) as it helps you plan the layout of the components.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu
EECS Department, UC Berkeley

6. Using the Breadboard Tool

We will use the breadboard tool to wire our simple DC circuit from section 5. First delete the multimeter from your circuit. Although you could wire the multimeter on the breadboard, it is inconvenient and unecessary. The circuit to be wired is shown in figure 17.

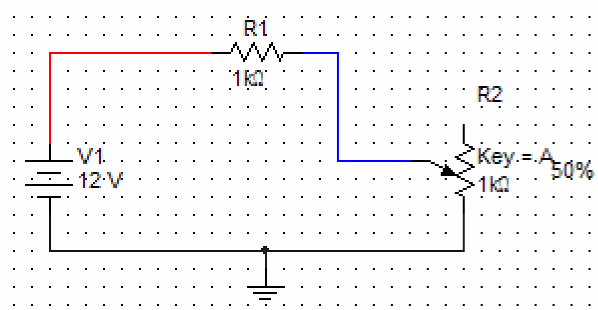


Figure 17. The Simple DC circuit from the previous section

1. `Left-click` on the Breadboard icon  in the **Main Toolbar** to open the Breadboard view. Figure 18 shows the breadboard view.

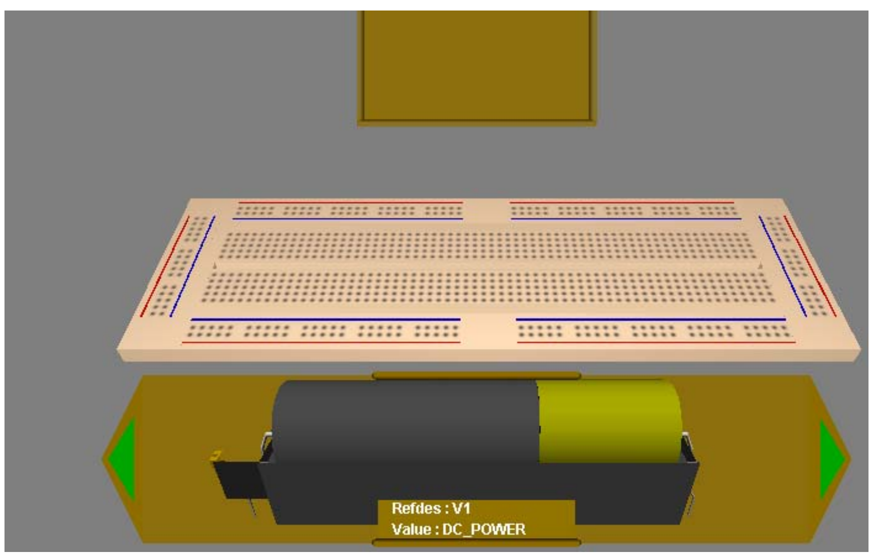


Figure 18. The Breadboard view in MultiSim

The tray at the bottom has all the circuit elements in your schematic, in this case you have a battery, a potentiometer and a resistor. You can change the size of your breadboard by `Left-clicking` Options and then selecting Breadboard Settings.

You can rotate your breadboard view by moving the mouse outside (or to the middle of) the breadboard until the cursor changes to a set of double arrows.

`Left-click` and drag to rotate the breadboard. If you move your mouse over any other area of the breadboard, you get a small wire pointer. You use this to place wires on the breadboard. `Left-click` one slot on the breadboard and drag a wire to another slot. Figure 19 shows a wire on the breadboard. Notice how MultiSim highlights which point on the breadboard you are wiring to. This makes wiring easy when you have a lot of components on the breadboard.

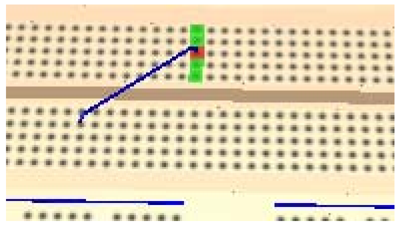


Figure 19. Breadboard wiring in MultiSim

To change the wire color, `Left-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. Lets start placing and wiring components on the breadboard. First we will place the components and then wire them.

2. `Left-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 Figure 20.

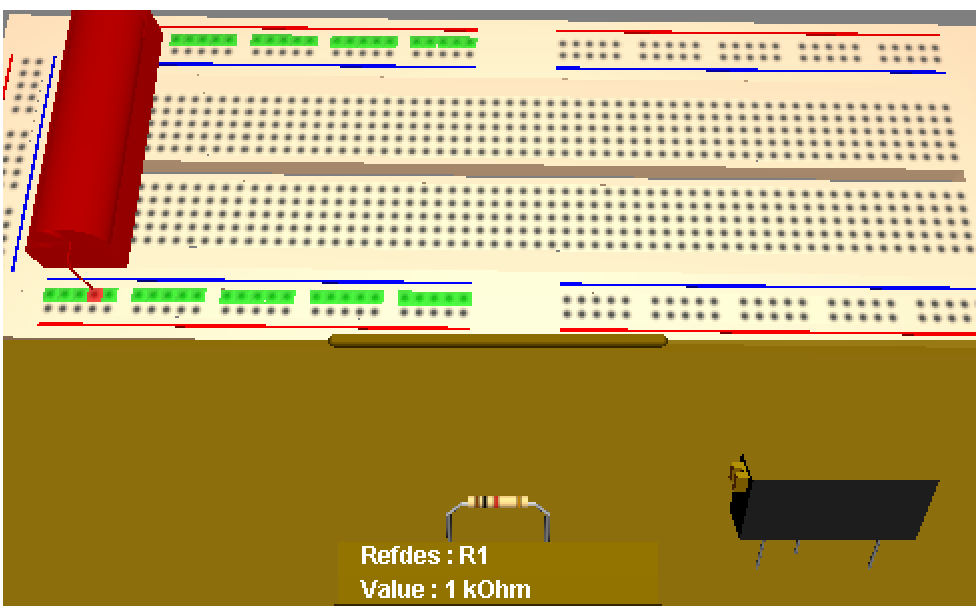
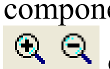


Figure 20. The battery is placed on the breadboard. We are using the outer connectors for the power, which is the convention followed when using a breadboard.

3. Place the resistor and potentiometer as shown in figure 21. Once you place all components, the tray at the bottom disappears. You can use the Zoom icons  on the top toolbar to get a better look at the components.

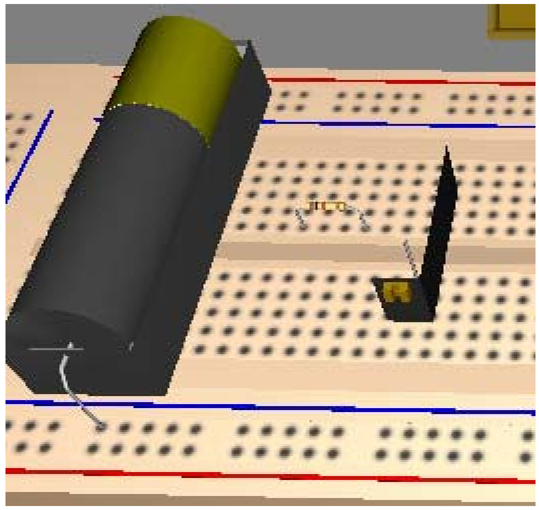


Figure 21. Components have been placed on the breadboard. Notice how the potentiometer is appearing transparent, this is to help you see the middle wiper of the potentiometer.

4. Wire the components as shown in figure 22. Again it is prudent to follow the color convention you used on the schematic. Notice as you wire that MultiSim actually highlights the connection end-point.

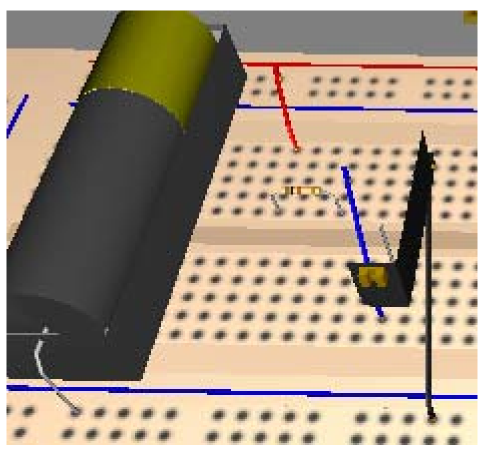


Figure 22. Wiring complete

5. One of the most powerful features of the MultiSim breadboard tool is the DRC (Design-Rules-Check) and Connectivity check. The Design-Rules (as the name implies) 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.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu
EECS Department, UC Berkeley

First lets run the DRC and Connectivity check. `Left-click` the Perform DRC and Connectivity check icon in the top toolbar. The status window at the bottom shows the results, refer to figure 23.


Multisim - 2006-06-08 16:55:32

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

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

Figure 23. Results from the DRC and Connectivity check

6. Lets introduce failure conditions for both checks on your breadboard. First, delete connection from the positive terminal of the battery (the red wire). The result is shown in figure 24.

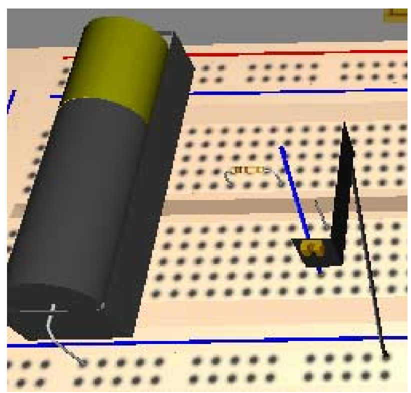


Figure 24. A Connectivity error has been introduced

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

7. Now lets introduce a design error. Rewire the positive terminal of the battery but short the 1 kΩ resistor. The result is shown in figure 25.

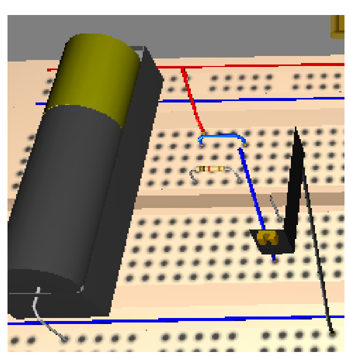


Figure 25. A Design error

If you rerun the DRC and Connectivity check, you will get a bunch of Design errors. Deleting the spurious wire removes the problem.

In this section you saw how you to use the breadboard tool to quickly wire your circuit on a breadboard. The main purpose of this tool is to give you an idea of component layout on the breadboard. Of course for a simple example like this, it is overkill. But what about more complicated circuits like your class project? In those cases, the breadboard tool is invaluable in planning your component layout.

Nevertheless, this simple circuit does introduce a very powerful concept. Notice we did not place a ground on the breadboard. MultiSim didn't complain as well. This rather subtle point is very important and hopefully clarifies the concept of a ground: its just a symbol on your circuit that indicates your reference node. A circuit need not have an explicit ground connection to Earth (unless you are dealing with very high voltages and need to provide a safe return path), many circuits do not have an explicit ground connection to Earth.

Now lets move on to more complicated circuits, starting with Dependent Sources.

7. Second Example: Dependent Sources

Dependent sources are in MultSim are easy. Here we will look at once kind of dependent source – the current controlled current source (CCCS). The other dependent sources work the same way. We will build the circuit shown below. The goal is to find V_o, but make sure you understand how to setup a dependent source.

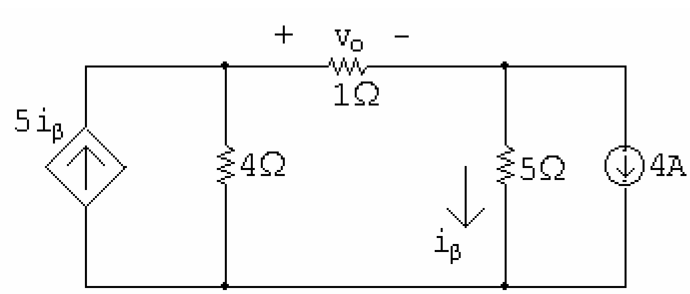


Figure 26. A circuit with a dependent source

1. Start MultiSim and place the non-dependent source components. The current source can be found in the Signal Sources family . Place a ground node at the bottom of the circuit and an Agilent Multimeter across the 1 Ω resistor. Figure 27 shows the result.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

EECS Department, UC Berkeley
Summer 2006
Version 1.00

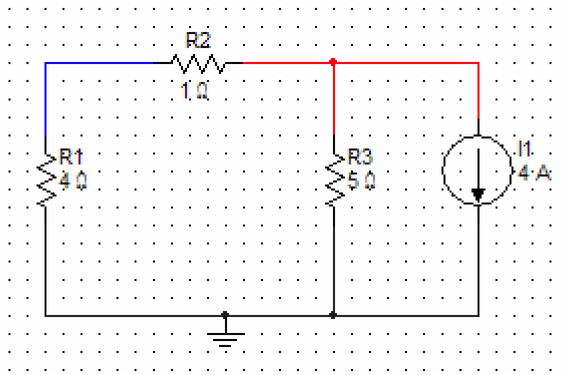


Figure 27. All we need to do is setup the dependent source

2. Left-Click on Place and choose Component... . The Select a Component window pops up as shown in Figure 28.

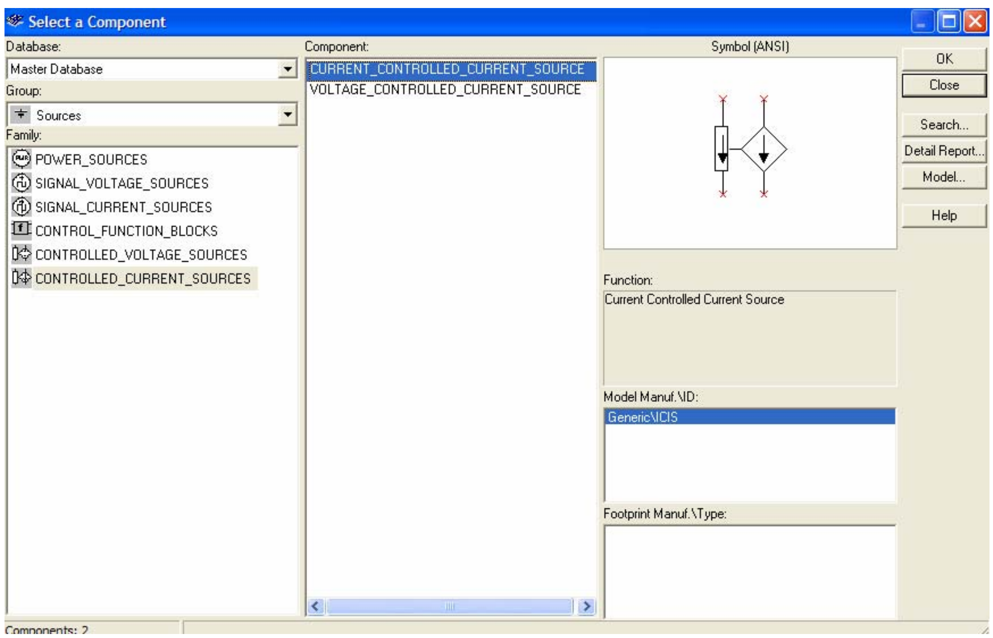


Figure 28. Dependent sources in MultiSim

Dependent sources can be found under the CONTROLLED_VOLTAGE_SOURCES and CONTROLLED_CURRENT_SOURCES in the Sources Component Group under the Master Database. Choose the correct source as shown in figure 28. Left-click the OK button to place the source in the workspace.

Wire up the circuit as shown in figure 29. The input to the dependent source is the terminal on the left. Notice how the current through the 5 A branch is passing through the left terminal of the dependent source. This current is the independent variable i_β in figure 26. Thus it is passing through the controlling terminal in the dependent source. Next, we have a value of -5 for the current gain. This means we have 5 i_β amps of current going from bottom to top in the leftmost branch, exactly as shown in figure 27.

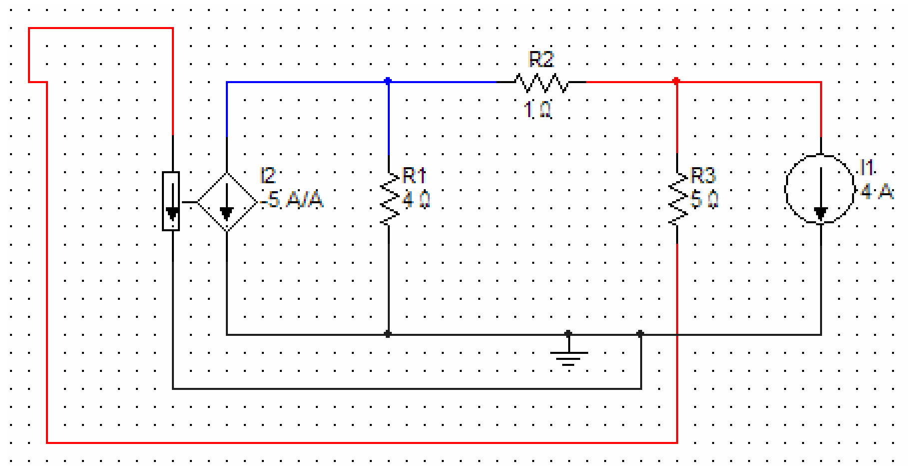


Figure 29. Complete the circuit with the dependent source

This completes the circuit. Add a multimeter across R2 to measure the voltage. Before running the simulation, you may want to solve the circuit by hand[5] to make sure your simulation is correct.

That wraps up the section on dependent sources. Using the knowledge gained from this section and the previous two, you can simulate a lot of circuits. You should try to practice simulating circuits from your textbook so you get used to MultiSim. In the next section we will simulate first-order circuits.

8. Third Example: Transient Analysis

Simulating circuits with capacitors or inductors (modelled by first-order differential equations[6]) is simple. It requires knowledge of two concepts: knowledge of switches and using the virtual oscilloscope. We will cover switches in MultiSim first.

There are four main kinds of switches you need to be familiar with: SPST (Single Pole Single Throw), SPDT (Single Pole Double Throw), DPST (Double Pole Single Through) and DPDT (Double Pole, Double Throw)[7]. Switch Terminology: A **Pole** is the number of switch contact sets. **Throw** is number of conducting positions, single or double. Table 1 shows the different kinds of switches in MultiSim. You can get to the Switches library by Left-Clicking Place and then choosing Component... . Go to the SUPPLEMENTARY_CONTACTS family in the Electro_Mechanical group (Masters database).

---

[5] The point of a simulation is to study your circuit before building it. In most cases, you cannot solve the circuit to be simulated by hand. However if you can solve it, by all means do so before simulating. It will only add to your knowledge.

[6] Although you can use the concepts in this section to simulate higher-order differential equations, you are not responible for such circuits in EE100.

[7] Reference: http://www.kpsec.freeuk.com/components/switch.htm#standard

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

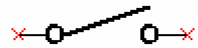
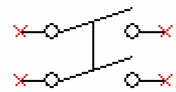
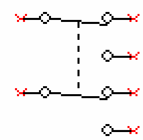
Switch Type	MultiSim Component
SPST	**SPST_NO_SB,** 
SPDT	There is no explicit MultiSim component, but you can design this with one of the other three types. Try it!
DPST	**DPST_2NO_SB,** 
DPDT	**DPDT_SB,** 

Table 1. The four different kinds of switches you will primarily use in EE100

As you know from your solving first order circuits, switches are used to put initial conditions on a capacitor or inductor. Construct the simplest possible first order circuit in MultiSim, shown in figure 30. You should be able to easily find the components you need.

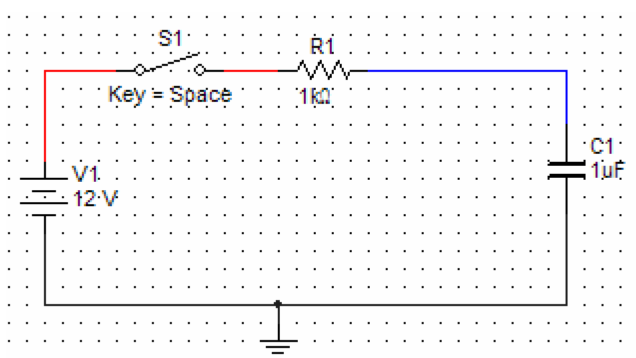


Figure 30. First order RC circuit

In order to visualize the transient behaviour of our circuit, we will use the Agilent Virtual Oscilloscope. `Left-click` the Agilent Oscillscope  and drag a scope onto the workspace. Connect one of the scope inputs to the circuit as shown in Figure 31.

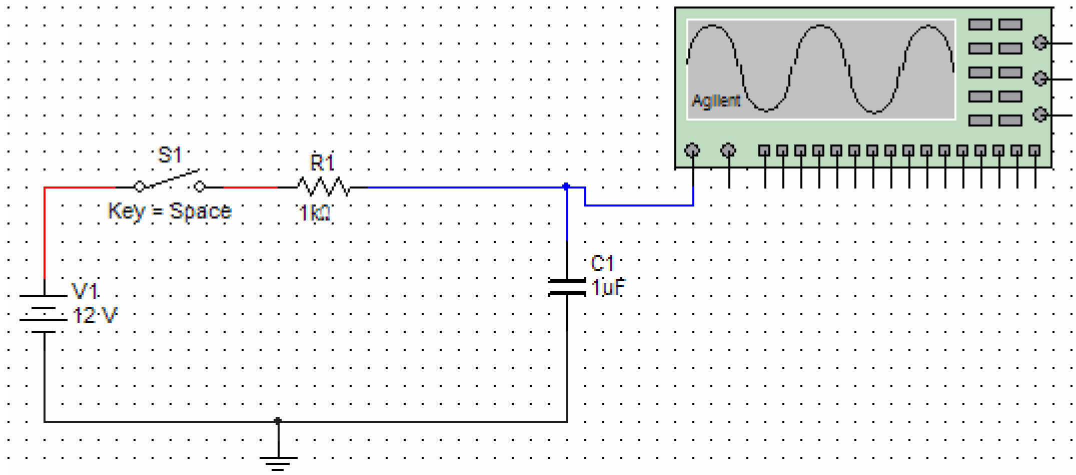


Figure 31. The Agilent Oscilloscope is placed in the circuit. You don't need to ground the scope if your circuit is properly grounded.

1. The first step is to configure the scope. Double click the scope to open up the front panel and `Left-click` the  button to turn on the scope, the result is shown in figure 32.

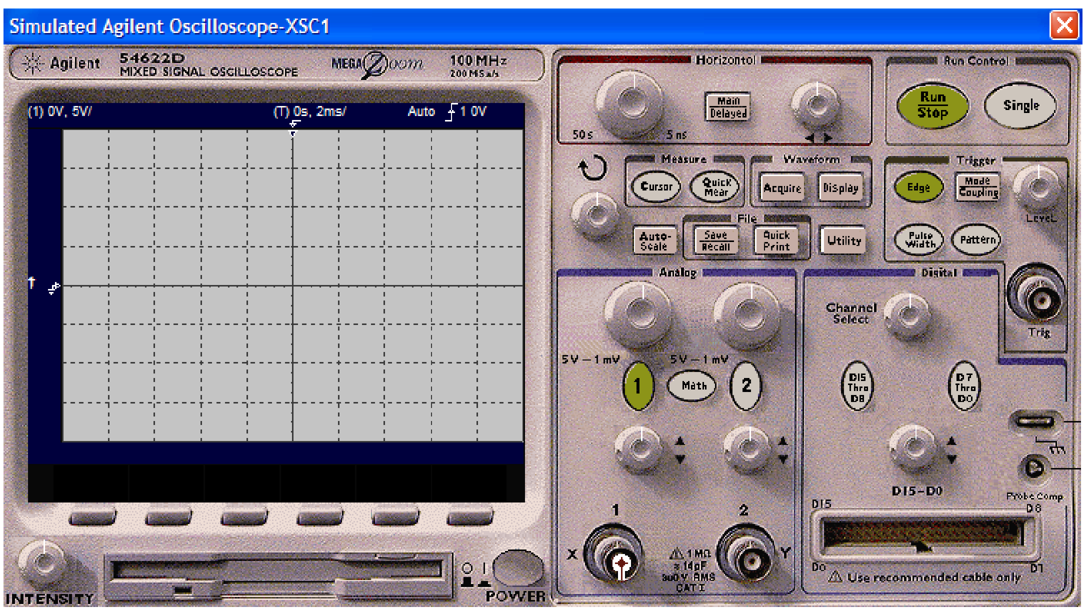
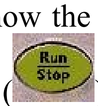
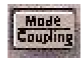


Figure 32. The Agilent Oscilloscope front panel

Notice how the appropriate source channel (), Trigger mode () and Run Control () are highlighted. All we need to do is set the correct trigger options so we trigger depending on the properties of our signal to be measured.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

2. Lets set the trigger mode first. `Left-click` the 

button. The result is shown in figure 33.

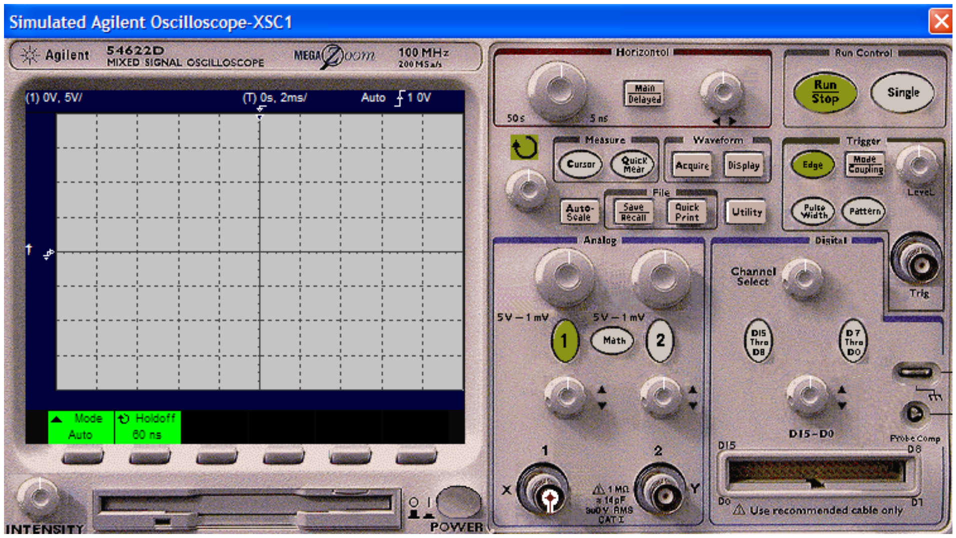
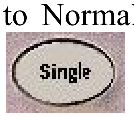


Figure 33. Setting up the Triggering mode

Set the mode to Normal by clicking on the softkey below the 

display. `Left-click` the 

button to turn on Single shot mode. `Left-click` and turn the Level knob 

clockwise to make sure you are trigger level is non-zero. Figure 34 shows the resulting front panel.

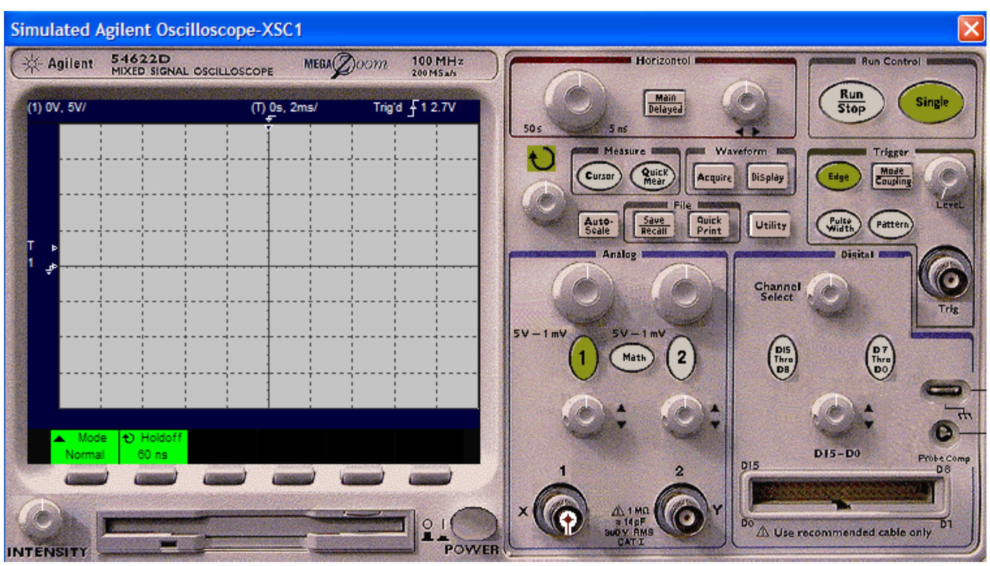


Figure 34. The scope setup for measurement

3. Press the  button to run the Simulation. Next, press the spacebar to close the switch and trigger the scope. Figure 35 shows the waveform.

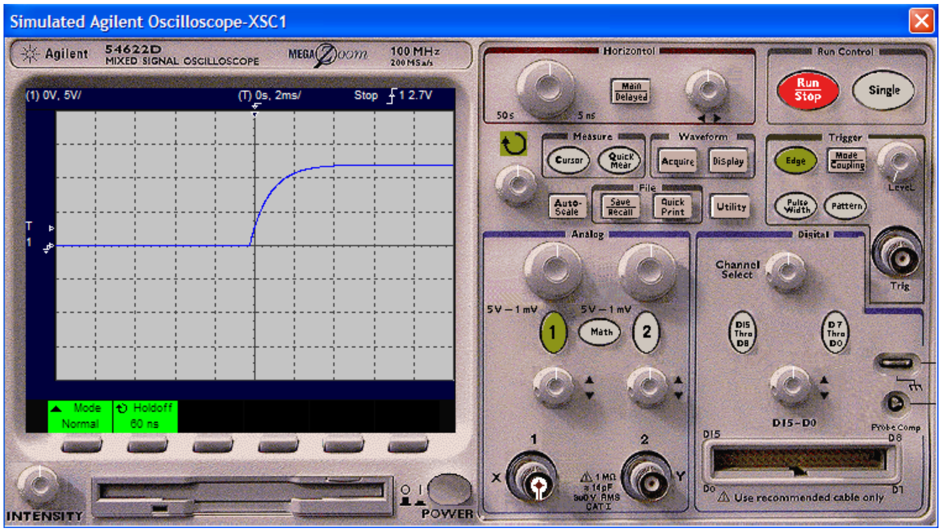
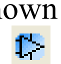


Figure 35. The voltage across the capacitor

This concludes our introductory look at transient analysis. This pretty much covers the topics in MultiSim that for EE100. Sections 9 through 11 just give you an example circuit and the location of the corresponding components in the database. Section 12 talks about different kinds of plots in MultiSim, but does not introduce anything new.

9. Fourth Example: Operational Amplifier Circuits

We will construct the circuit shown in figure 36 (R1 = 1 kΩ, R2 = 2 kΩ). The op-amp is the 5 terminal Virtual Op-amp  which can be found under the Analog Family in the Virtual Toolbar. The input to the circuit is from the Agilent function generator which can be found in the Instruments Toolbar. The input is a 500 mVpp sine wave, you just need to enable both channels in the scope. Figure 37 shows the output.

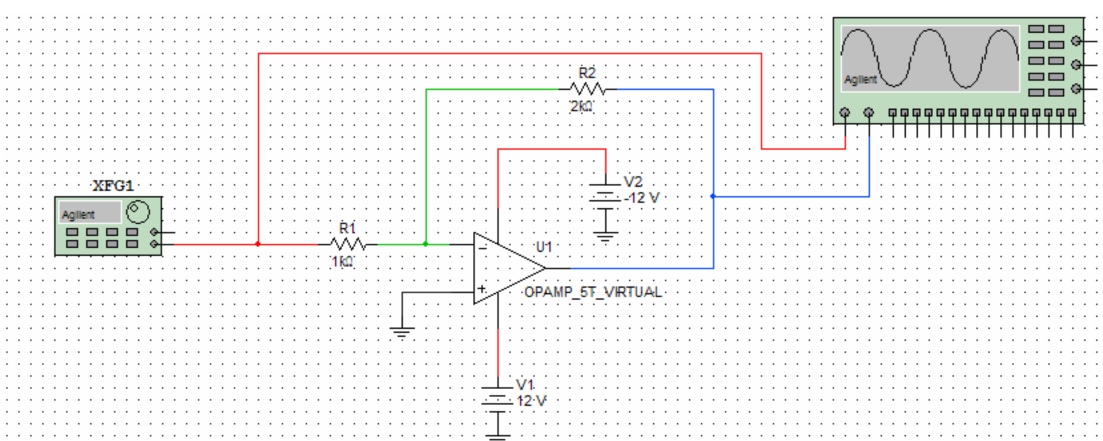


Figure 36. An op-amp inverting amplifier

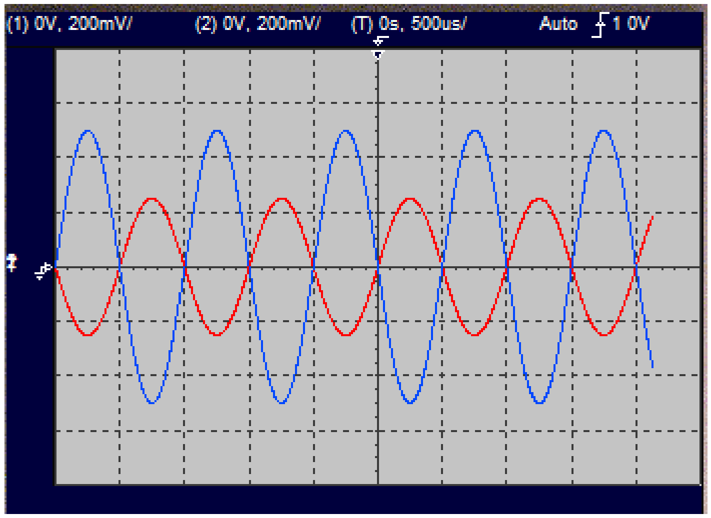


Figure 37. Output from the inverting amplifier showing a gain of -2

10. Fifth Example: Diodes

We will build the circuit shown in figure 38. The 1N4148 can be found in the Component database: `Left-Click` on Place and choose Component... . Choose the Diodes group and select DIODE family for the 1N4148. The 3D LED can be found under the  family in the Virtual Toolbar. The settings for the function generator is a 5 Vpp sine wave with frequency of 100 Hz. Run the simulation and notice the LED flashing. The output of the scope will be a rectified sine wave.

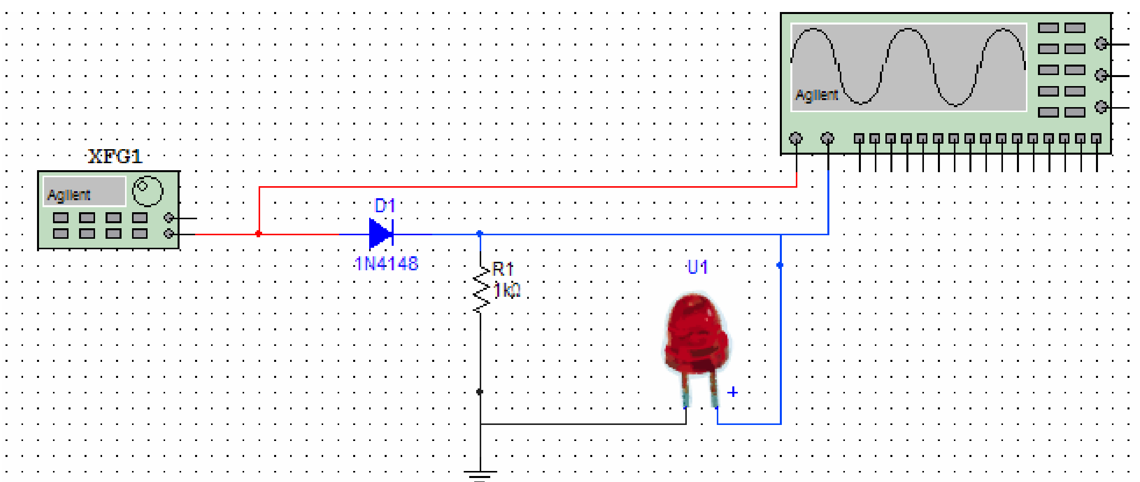


Figure 38. A diode half-rectifier, the output is taken across a 3D LED model.

Muthuswamy, Bharathwaj
mbharat@cory.eecs.berkeley.edu

11. Sixth Example: Transistors

Build the simple npn BJT circuit shown in figure 39. You can find the 2N2222A npn BJT in the component database: `Left-click` on Place and choose Component... . Choose the Transistors group and the BJT_NPN family for the 2N2222A. Can you calculate the XMM1 ammeter and the XMM2 voltmeter reading before running the simulation? (Hint: `Double-click` on the transistor and `Left-click` Edit Model for transistor parameters like β).

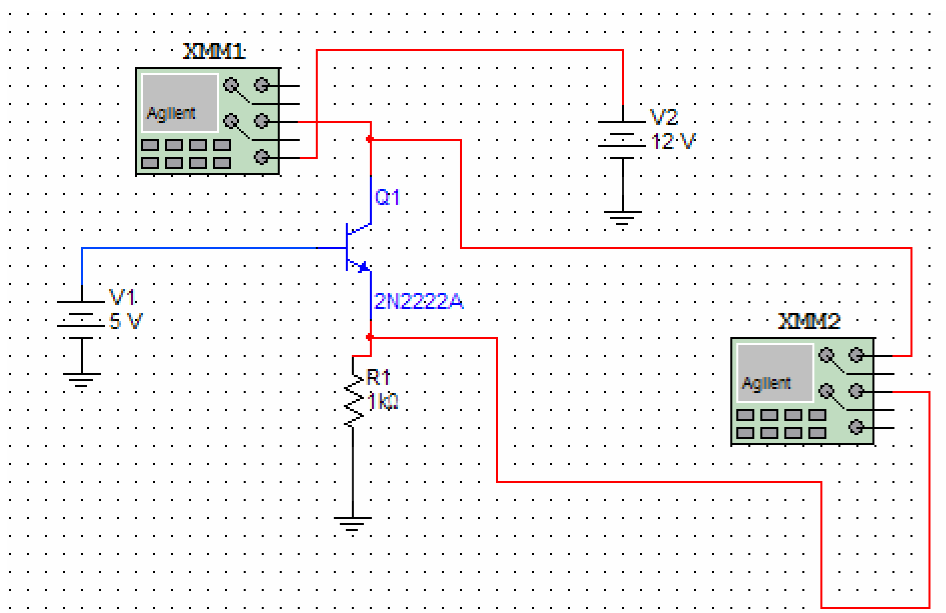


Figure 39. A simple npn BJT circuit

12. Seventh Example: Bode Plots

MultiSim has a convenient Bode Plotter tool in the Instruments Toolbar. Figure 40 shows a simple low-pass filter circuit with the Blode Plotter tool. The most important point while using the Bode Plotter: your input MUST be an AC source.

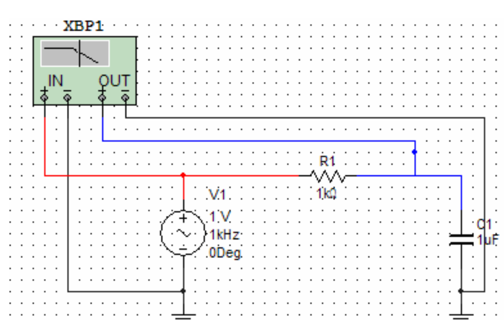


Figure 40. The Bode Plotter tool in MultiSim

13. Conclusion

This document has barely scratched the surface of MultSim. We haven't covered a whole bunch of topics like the CE BJT amplifier tool, the Filter Wizard and the whole PCB design suite in UltiBoard.

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 new features in MultiSim.