Tutorial #1: Introduction

The objective of this tutorial is to introduce students to the circuit simulator by building and simulating a simple circuit. The circuit simulator is a useful tool to view a circuit's behavior without having to hardwire real components (e.g. resistors, voltage sources, etc.). We will use some of the simulator's functions in this tutorial to simulate a simple circuit consisting of a voltage source and three resistors in series. Figure 1 briefly describes the main controls of the simulator. These will be discussed in more detail as we proceed with the tutorials.

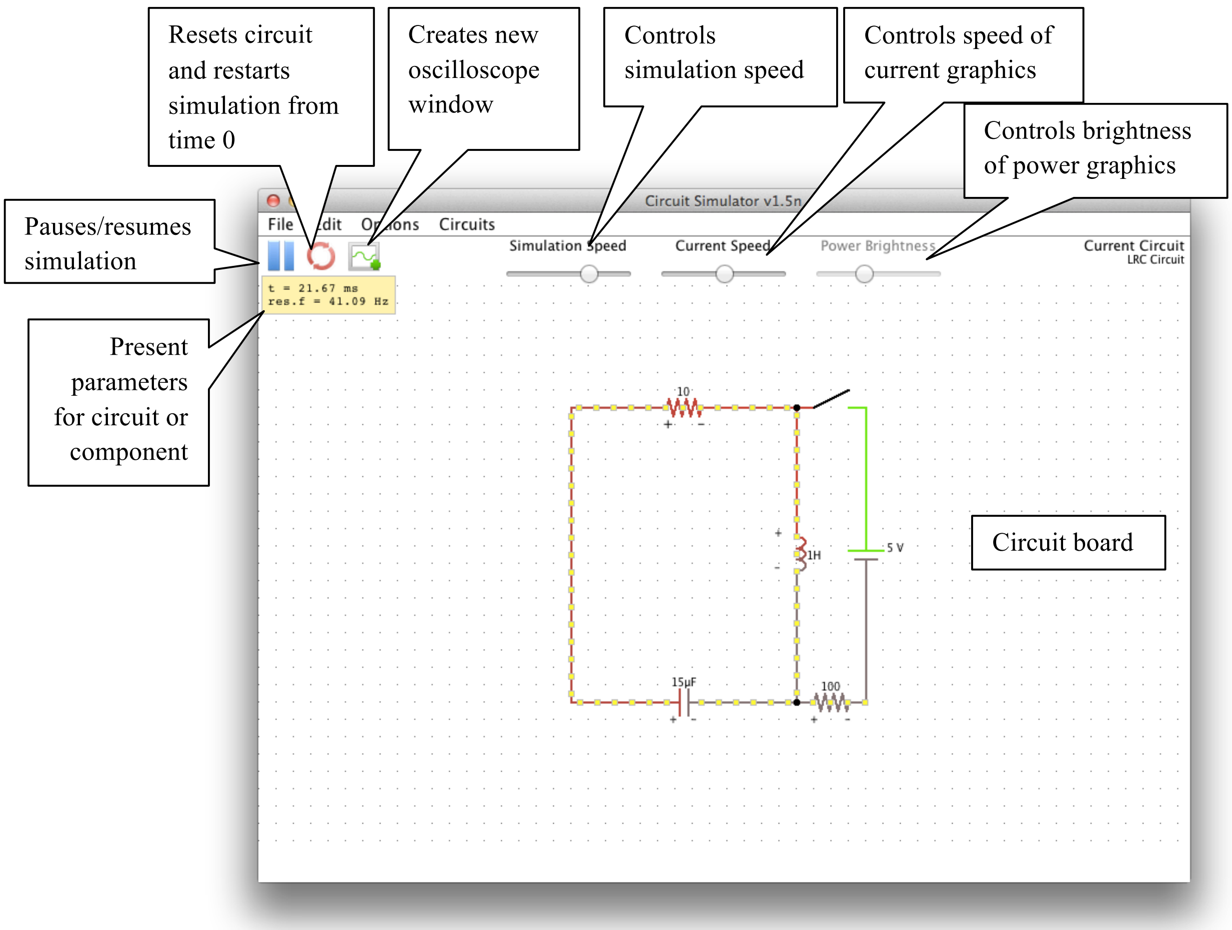

Figure 1: Circuit simulator

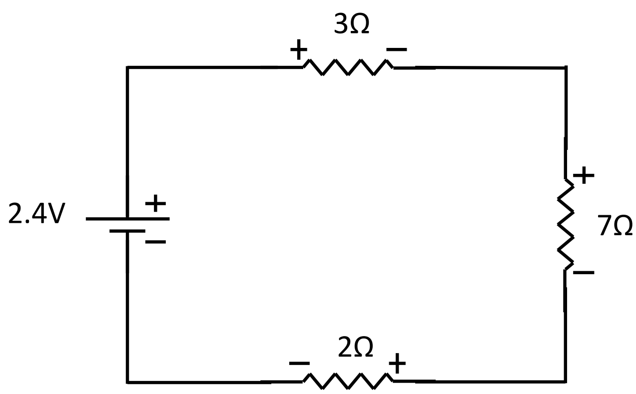

Figure 2: Circuit to be simulated

Step 1. Open the circuit simulator. You should see the simulator window (as shown in **Figure 1**) and another window titled “Oscilloscope”.

Step 2. Go to the “Circuits” menu and select “Blank Circuit” as shown in **Figure 3**.

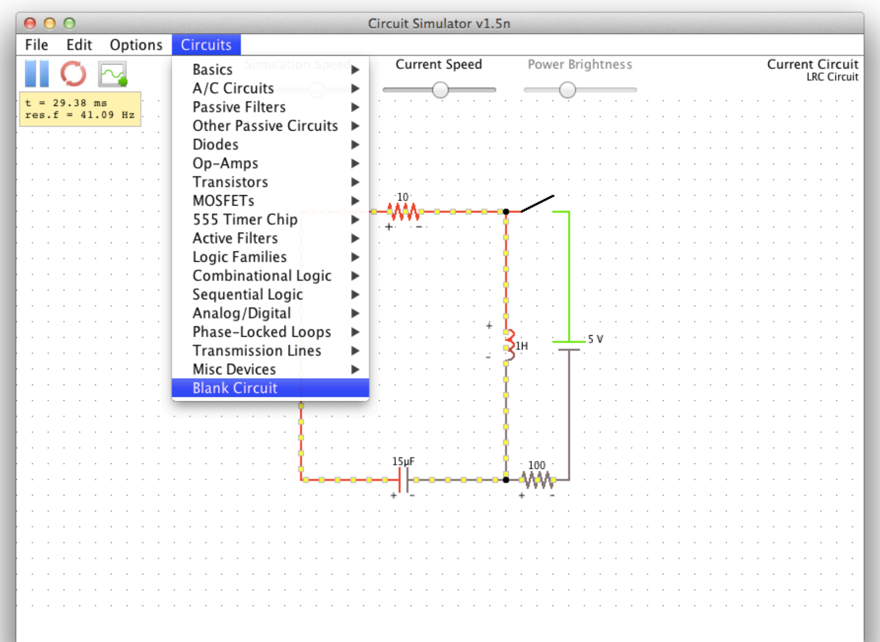

Figure 3: Blank circuit menu option

Step 3. Right click on the circuit board and select "Add Resistor (r)". Then, click anywhere in "circuit board" and drag a resistor as shown in **Figure 4**. As soon as you place the resistor (or any other component), the simulation will start.

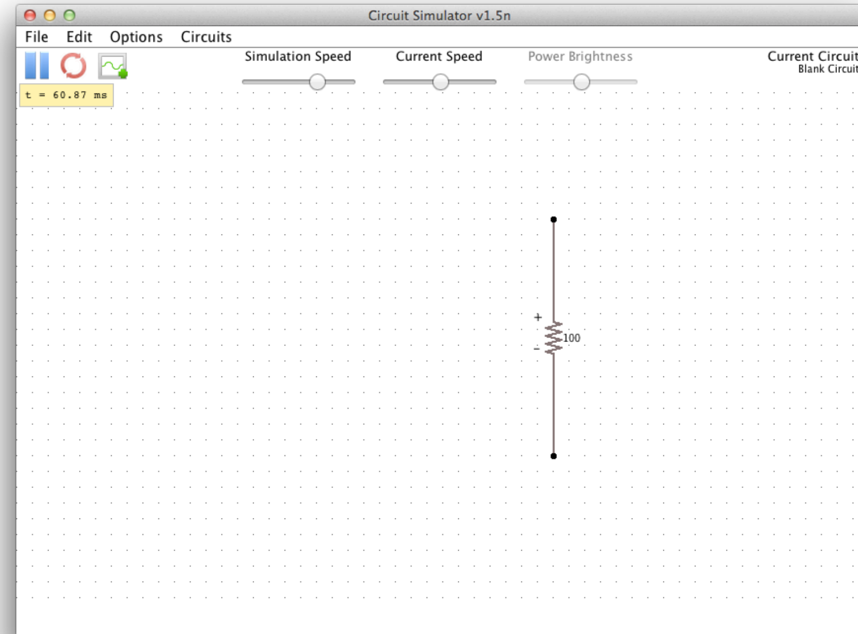

Figure 4: Circuit after placing first resistor

Step 4. The default resistance for a resistor is 100Ω. To change the resistance, right click on the resistor and select "Edit". Enter the new resistance (7Ω for this tutorial) in the dialog box and press "OK". Note that as you move the cursor over the resistor it changes color to cyan. This indicates that it is selected.

Note: The plus and minus signs on the resistor indicate the element's oscilloscope polarity. When the element's voltage or current is plotted in the oscilloscope, the sign of the waveform will follow the passive sign convention using the polarity as indicated. No matter which way the resistor is oriented, the circuit will operate the same. The polarity is only for reference. The need for this will become more apparent when dealing with AC circuits, where the current through an element will often change direction. When drawing an element on the circuit board, the first click will place the positive terminal and the second the negative.

Step 5. Repeat step 4 and 5 to place 3Ω and 2Ω resistors as shown in **Figure 5**.

Note: You can also use a wire to connect two resistors. To draw a wire, right click on the circuit board and select "Add Wire (w)". Then, click anywhere on the circuit board and drag to draw a wire.

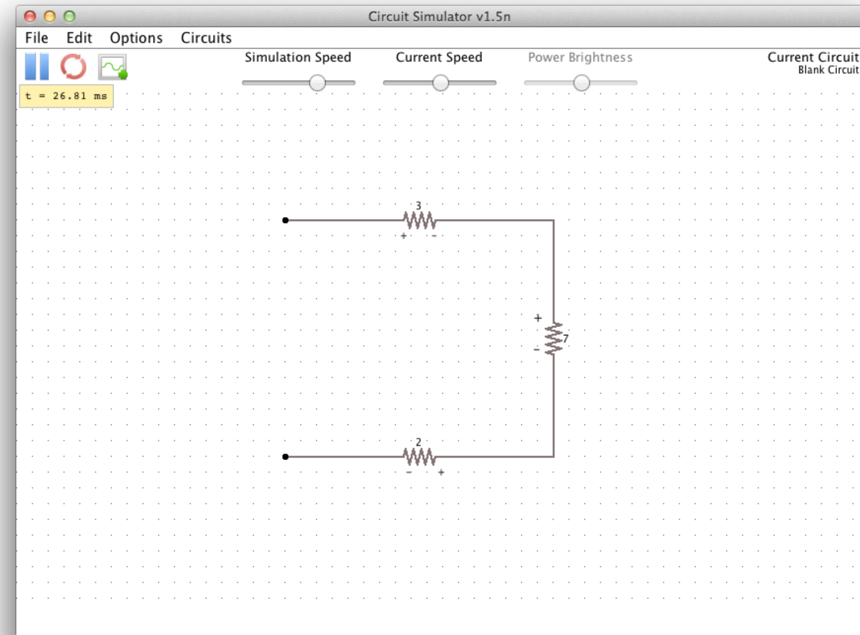

Figure 5: Circuit after placing resistors and changing resistances

Step 6. To add a voltage source, right click on the circuit board and select "Inputs/Outputs" then select "Add Voltage Source (2-terminal)". Click on the loose terminal of the 2Ω resistor and drag to place a voltage source as you placed the resistor in the step 4. The complete circuit should look like the one shown in **Figure 6**. As soon as the circuit is closed, current will start flowing through the resistors. Current flow is shown by the moving yellow squares. You can control their speed by adjusting the "Current Speed" slider given on the top. Note that changing the current speed is purely visual, it has no effect on the simulation.

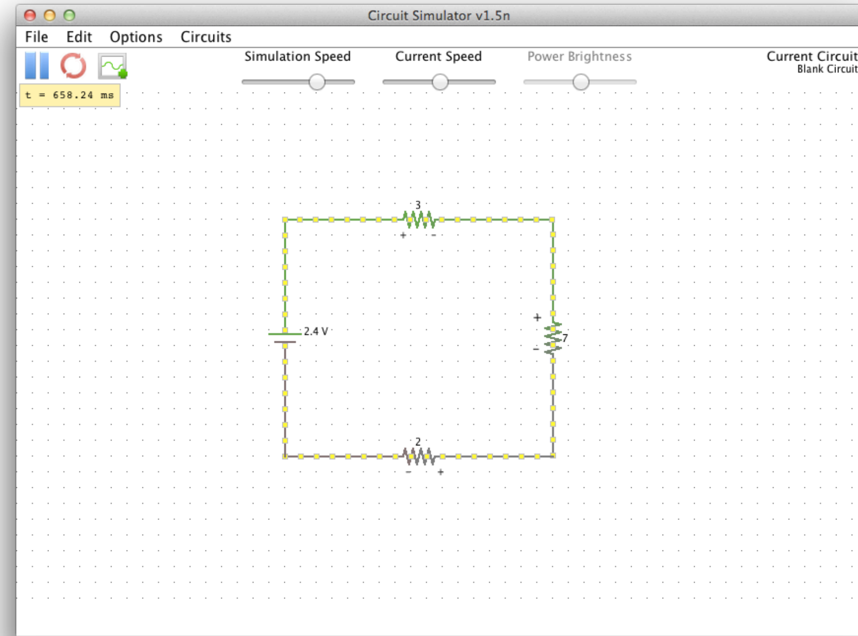

Figure 6: Circuit after placing voltage source

Step 7. The default value of the voltage source is 5V. To change this value, right click on the voltage source and select “Edit”. Enter the new value in the dialog box and press “OK”. You can also change the voltage source’s waveform from the drop down menu. For this tutorial, you should enter 2.4V and select a DC waveform.

Step 8. To measure the instantaneous current and voltage values of a component, move your cursor over it and the values will appear in the box at the top-left corner as shown in **Figure 7**. Note that the voltage difference (Vd) shown is an absolute value and independent of the element’s oscilloscope polarity.

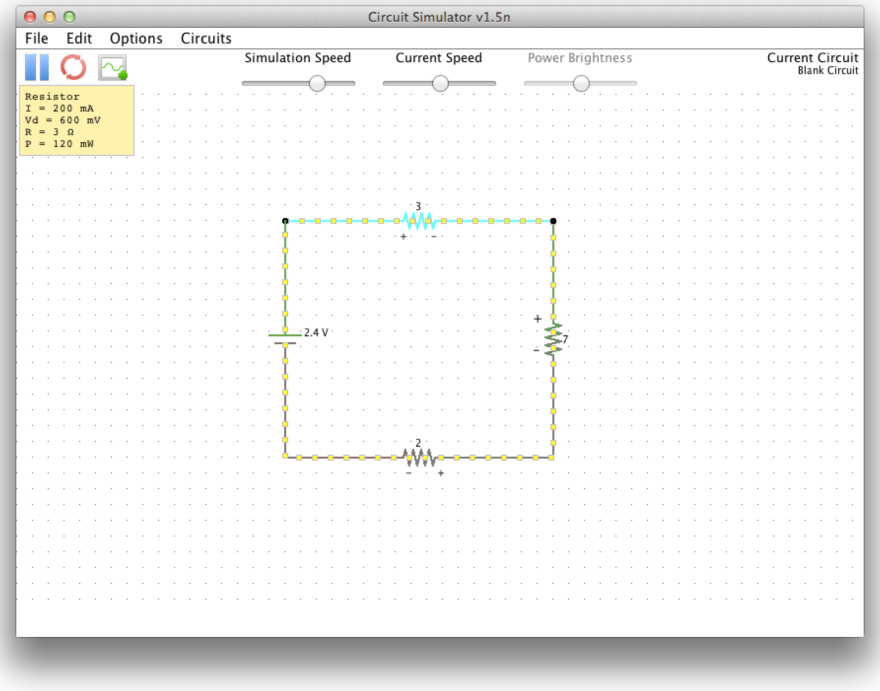

Figure 7: Instantaneous values for 3Ω resistor

Step 9. To save this circuit, go to the “File” menu, select “Save File, and enter the desired location and file name. Circuits are saved as plain text (.txt) files. You can reload a circuit by selecting “Open” in “File” menu.