

Introduction to Multisim 7

This tutorial is intended to get the student familiar with typical commands utilized in Multisim 11, formerly Electronics Workbench. Multisim is an electronic CAD software package that allows the user to build, test and trouble shoot circuits on the computer without running the risk of damaging lab components or equipment. It also provides a great working environment to try out and test your own ideals, which you are encouraged to do.

In this lab, you will be building a simple amplifier circuit using discrete components. Right now you may not know what all the components are that you will be using. That's ok, just get familiar with where things are located, Multisim commands, and navigation of the workspace.

Open Multisim by double clicking on your desktop icon, or by using the Start-Programs menu. You should see Figure 1.

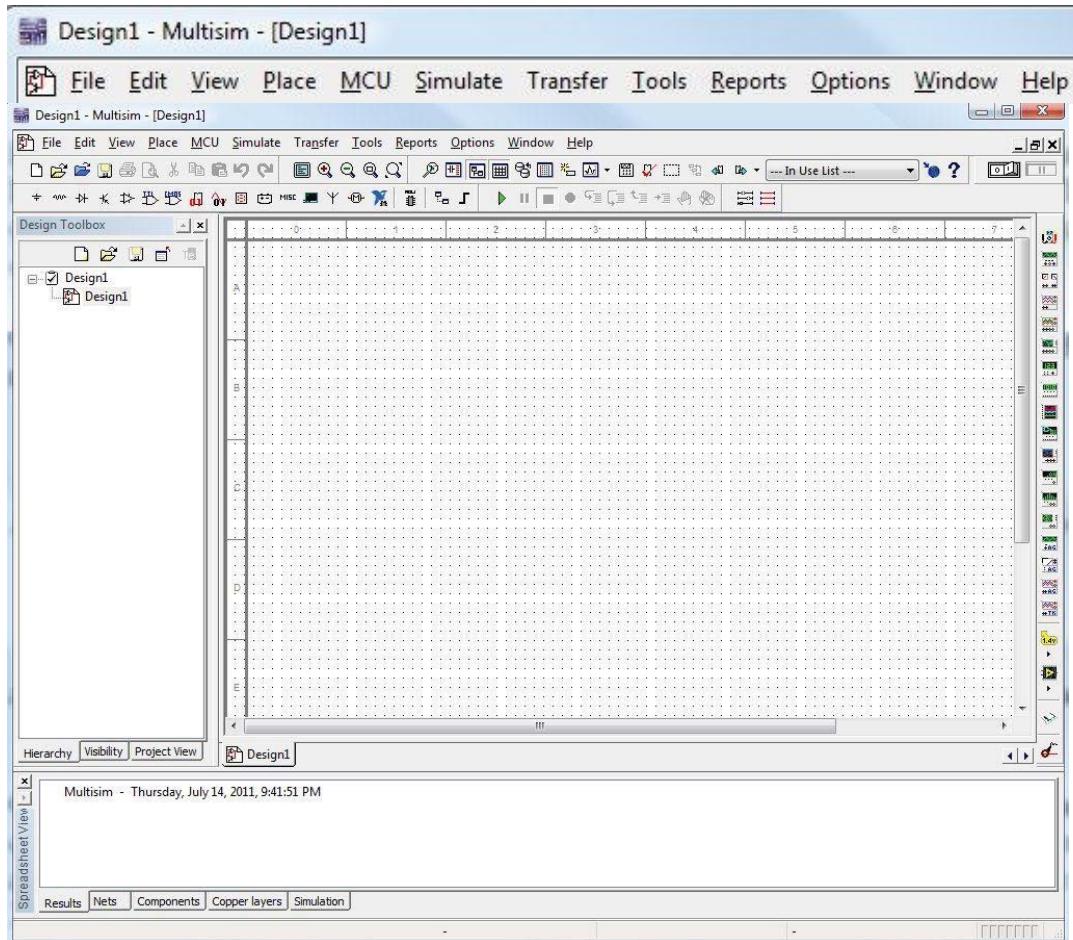


Figure 1

At the top left of the screen you will see the several menus available to you. Go ahead and click on each of these with your left mouse button. Take time to explore the menus.

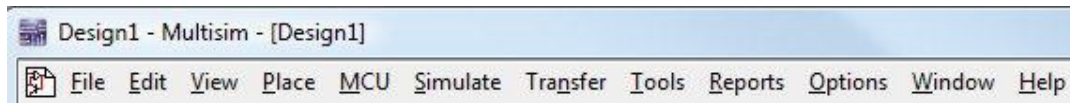


Figure 2

On the third top there are two component tool bars. Here you will find libraries of components. The user has two choices for placing parts, the left toolbar allows for adding specific parts and the right toolbar contains virtual components. See Figure 3

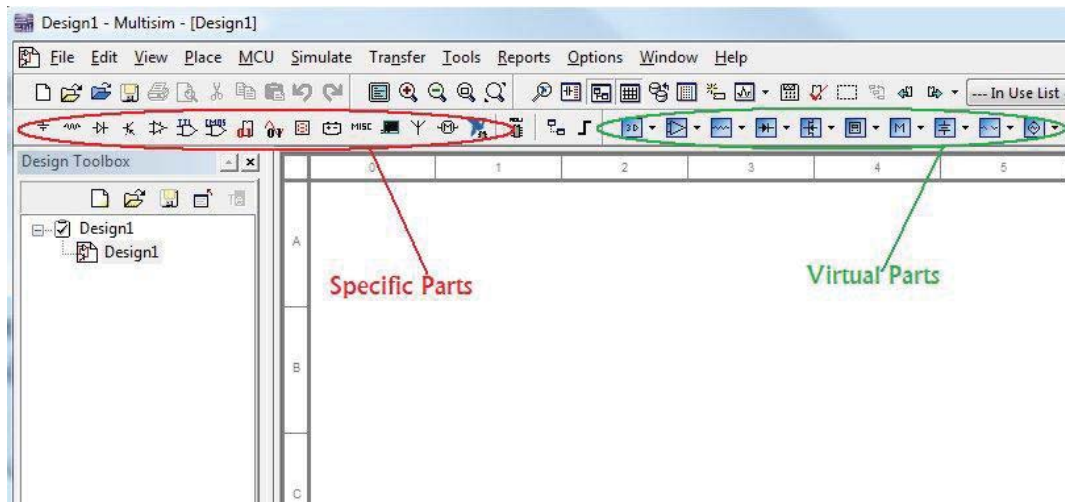


Figure 3

The choice of which toolbar to use depends upon the circuit. For a fast build and simulate you may use the virtual components since they are set by default to represent ideal component behavior. Once you have a design that works then you may wish to switch to the specific parts toolbar. This allows you to place parts available from distributors which behave more closely to the actual part you would use to build the circuit. Click on the second from the left, squiggly symbol, component button. A window should open as shown in Figure 4.

This is the Basic components library.

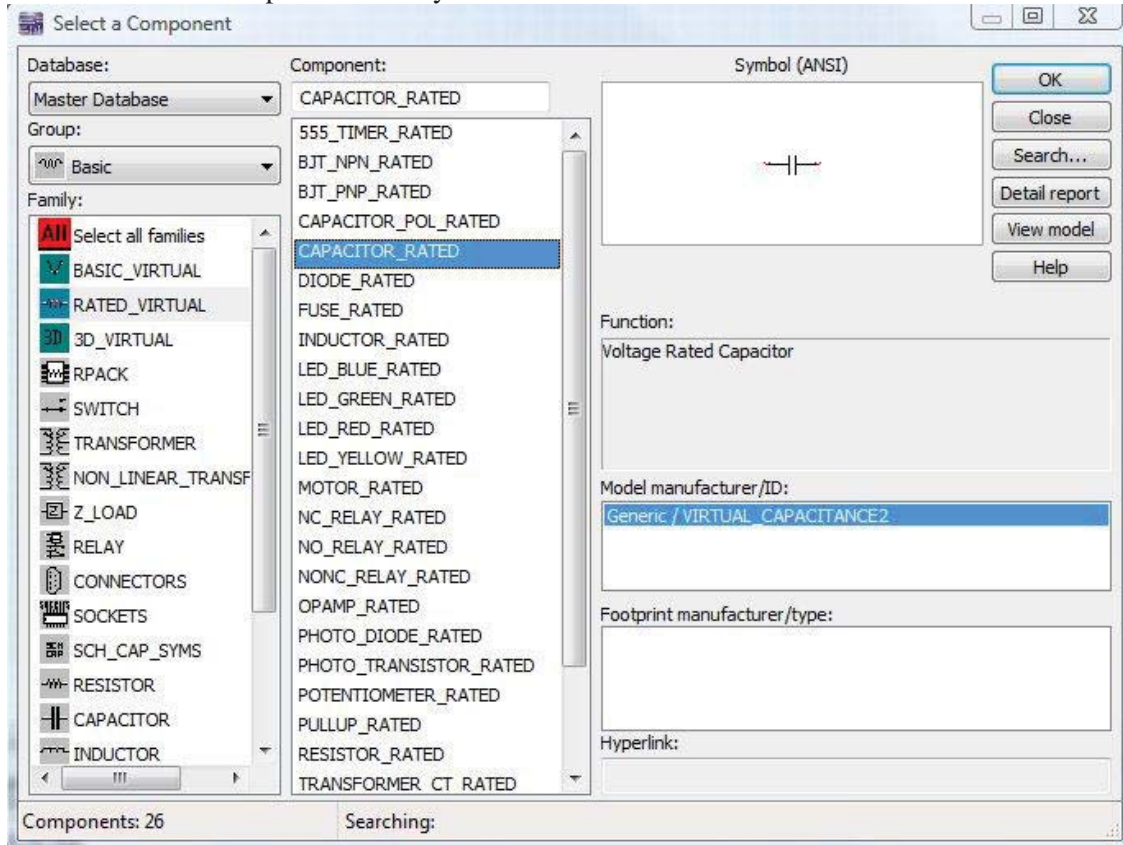


Figure 4

Notice the number of parts available to choose from. From this screen you can change groups by dropping down the group menu. Then you can select other parts. Go ahead and change groups and browse the component libraries. Notice the different manufactures and part numbers. As stated before this is the specific component tool bin.

Close the “Select a Component” dialog box. Now click on the resistor symbol on the Virtual Toolbar, third button. You should see the virtual “Basic Components” toolbar open as shown in Figure 5.

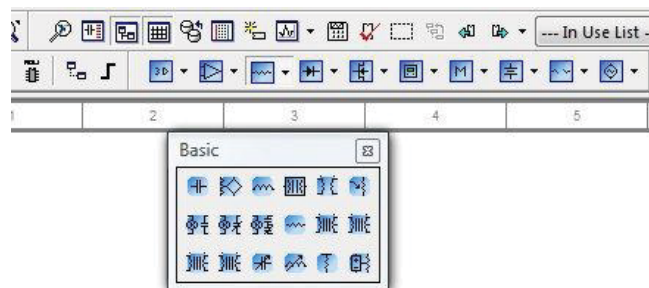


Figure 5

This toolbar is a simple layout of some commonly used circuit components. You can see the definition or name of the parts by placing your mouse cursor over the icon, try it. For most simulations in this class the virtual toolbar provides the simplest interface and it is

the toolbar we will use for the rest of this lab session. After you become more acquainted with Multisim, I would recommend you start using the specific toolbar. It has expanded choices and will simulate the circuit closer to what you would expect when building the real circuit.

On the right hand side of your workspace you will find the Instrument Toolbar. As shown in Figure 6.

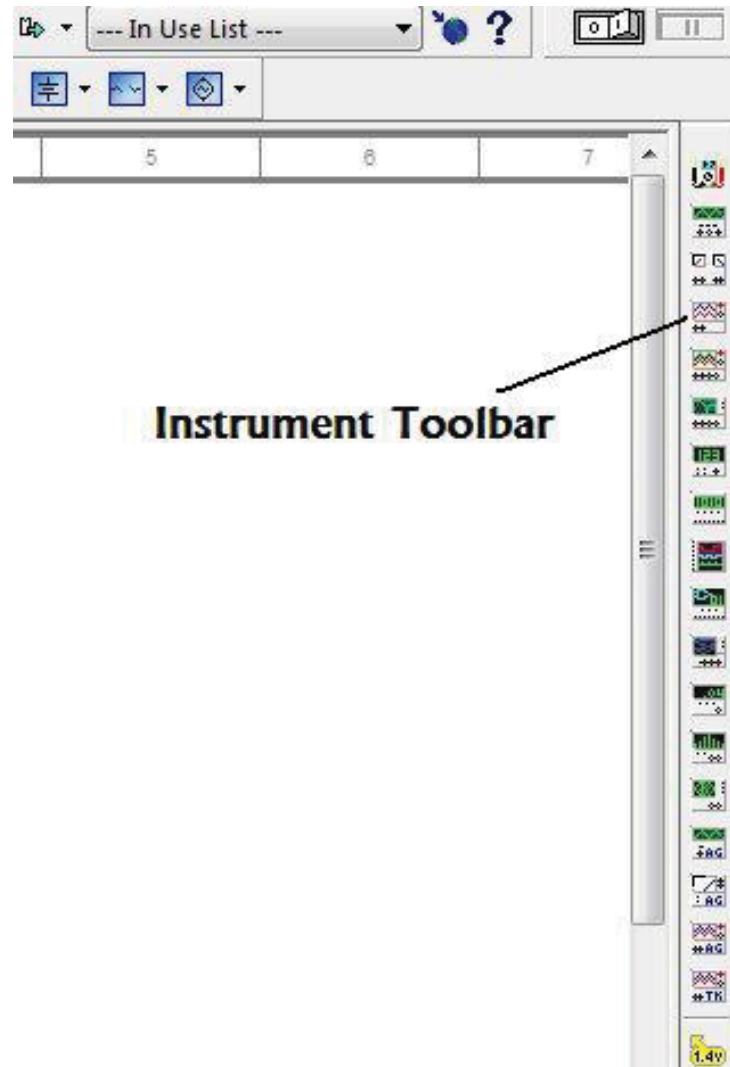


Figure 6

This is the location where the test instruments are kept, such as the Multimeter, Signal Generator, Oscilloscope, and so on. Place your mouse cursor over the instrument buttons to reveal the name of the instrument, just as you did for the virtual components. Go ahead and check them out.

As showed in Figure 7, the ON/OFF switch and Pause/Resume buttons are on the top right of the screen. You will use these shortly during circuit simulation. Note scroll bars are located on the side and bottom of the breadboard window for maneuvering around the workspace.

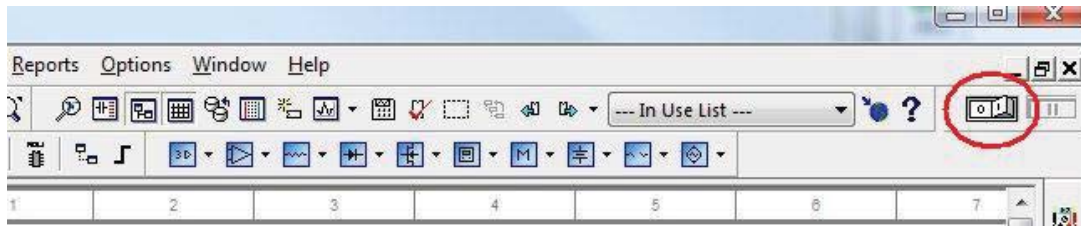


Figure 7

The rest of the lab will focus on learning Multisim by example; you will actually build and simulate a circuit. As you complete the activities try to focus on the steps required to complete them. Remember try to concentrate on learning the software at this point and do not get bogged down to the circuit details, you will have plenty of time for that later.

Building a Circuit

1. Click on the Resistor parts bin with your left mouse button. Left click on the resistor symbol and release. Move your mouse pointer to the workspace area. Then left click the mouse button. This places a resistor in your workspace. You should see Figure 8.

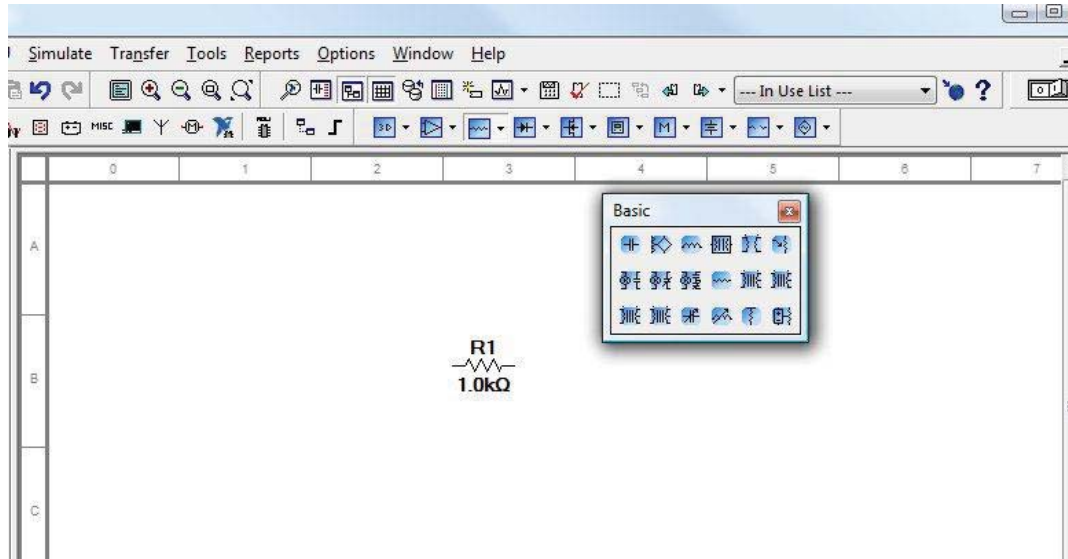


Figure 8

2. Now make four copies of the resistor. First, right click on the resistor, scroll down and select copy, and then left click on it. See Figure 9.

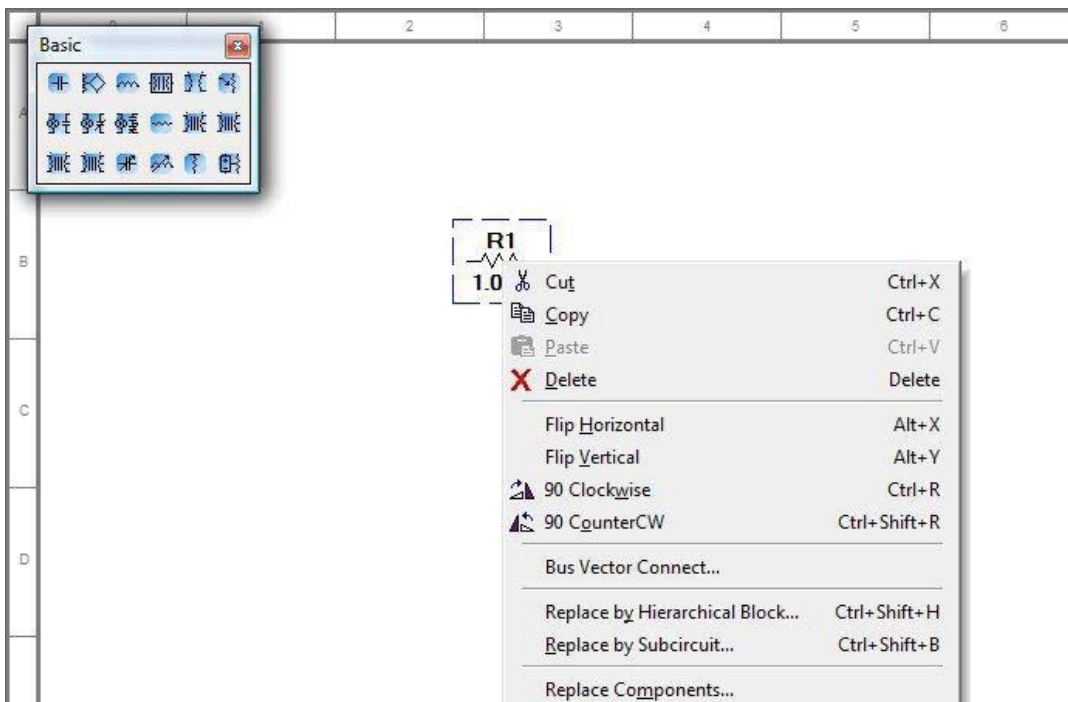


Figure 9

Now place the mouse pointer in an open space, in your work area, and right click. Next, scroll down select paste and left click on it. Click once more to place part. You should now have 2 resistors in your workspace. Repeat this procedure until you have a total of 5 resistors in your workspace. When you are finished you should see something similar to Figure 10.

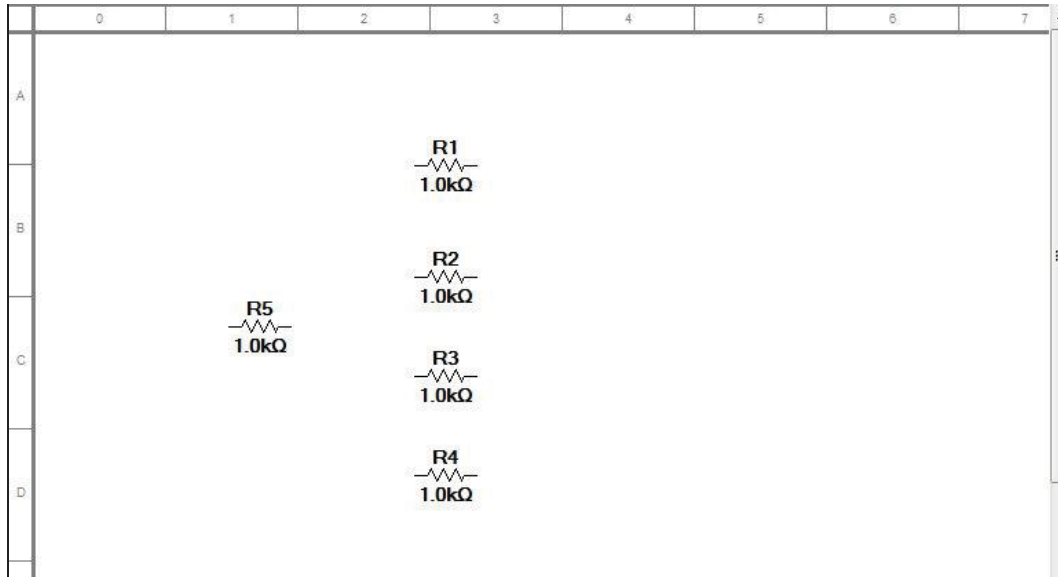


Figure 10

3. Now open the transistor parts bin, and drag out a NPN transistor. See Figure 11 for the location of the parts bin and schematic symbol for a NPN transistor.

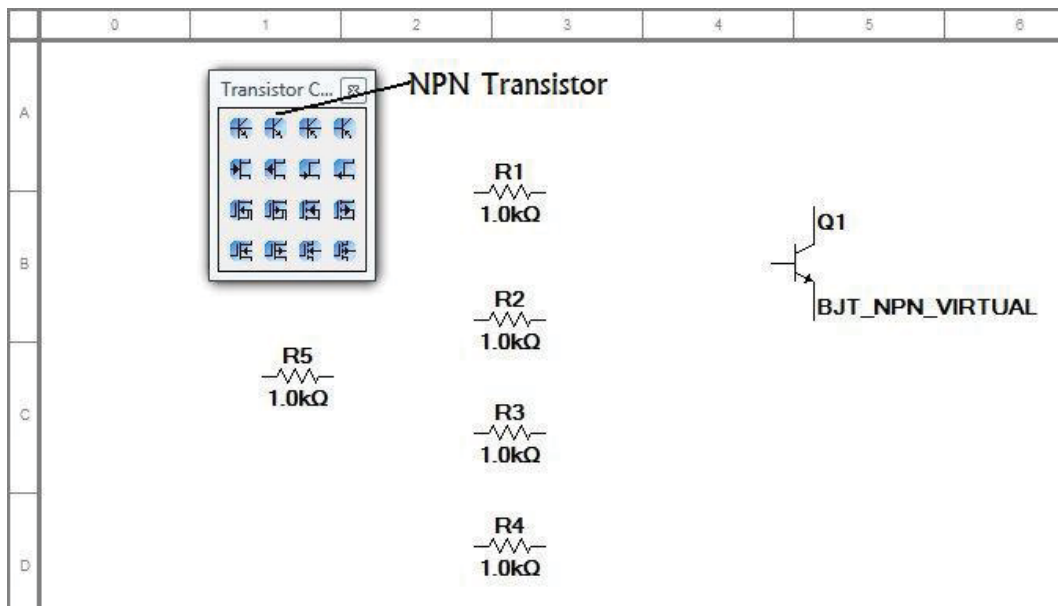


Figure 11

4. By now you should be getting the hang of placing components. Go ahead and place the following components on your workspace; 1-DC voltage source, 2-Capacitors, and 4-Ground Symbols. When you are finished your workspace should look like Figure 12.

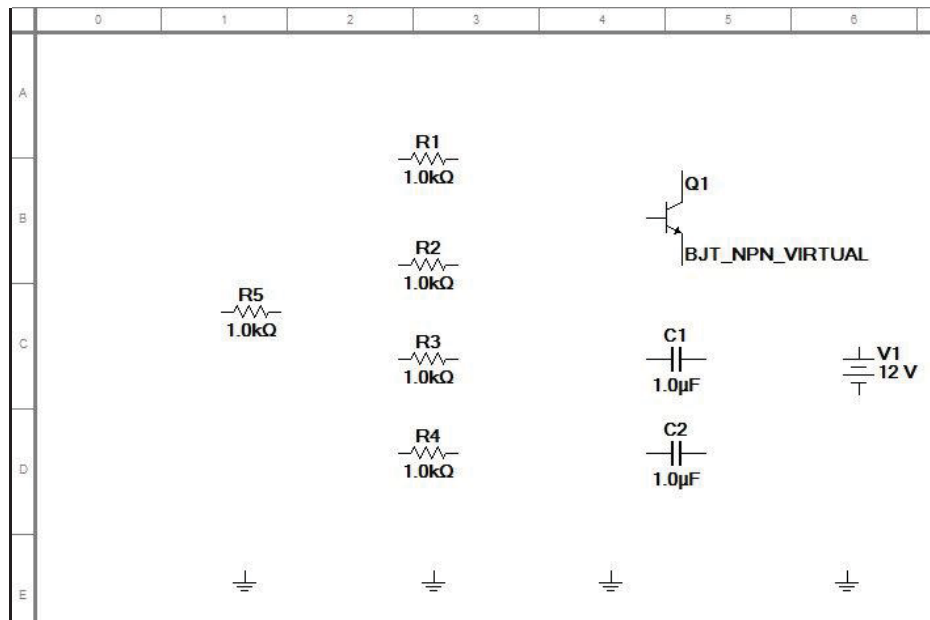


Figure 12

5. The components can be rotated by selecting a component and then using the rotate icon in the circuits tool bar. An alternative method is to use the mouse. Right click on one of your resistors, then scroll down to Rotate and left click on it. See Figure 13

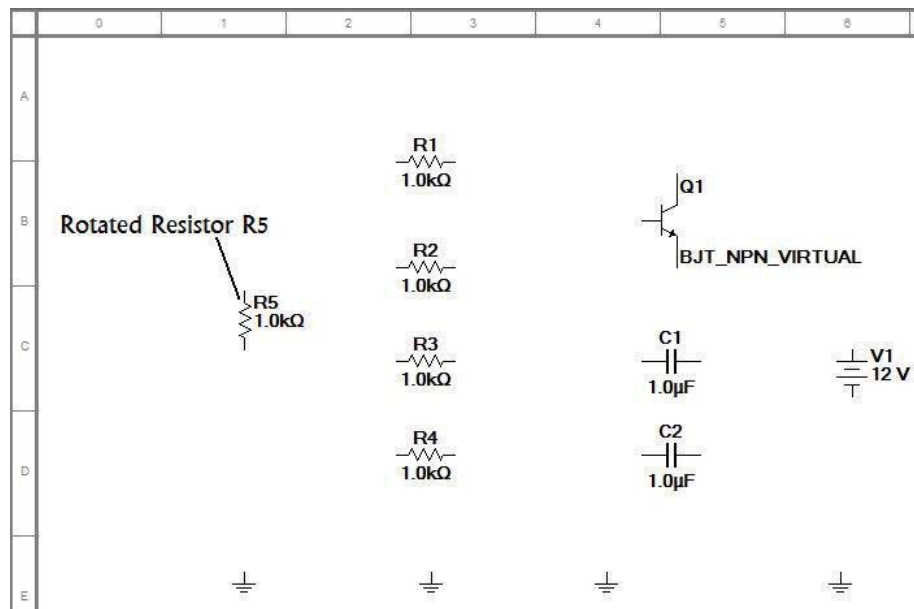


Figure 13

6. Rearrange your workspace to look like the workspace shown in Figure 14.

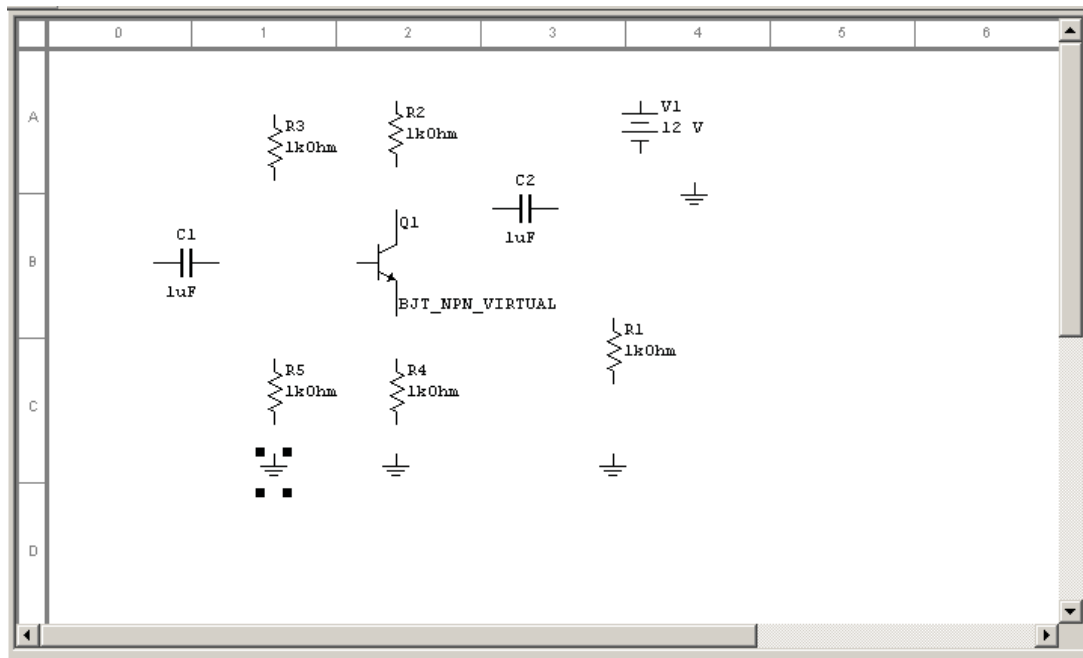


Figure 14

Next the component values must be selected, or changed from their default values, to be utilized in the amplifier circuit.

7. Double click on a resistor. In the “Resistance” property box change the value from 1 to 47 kOhm. See Figure 15.

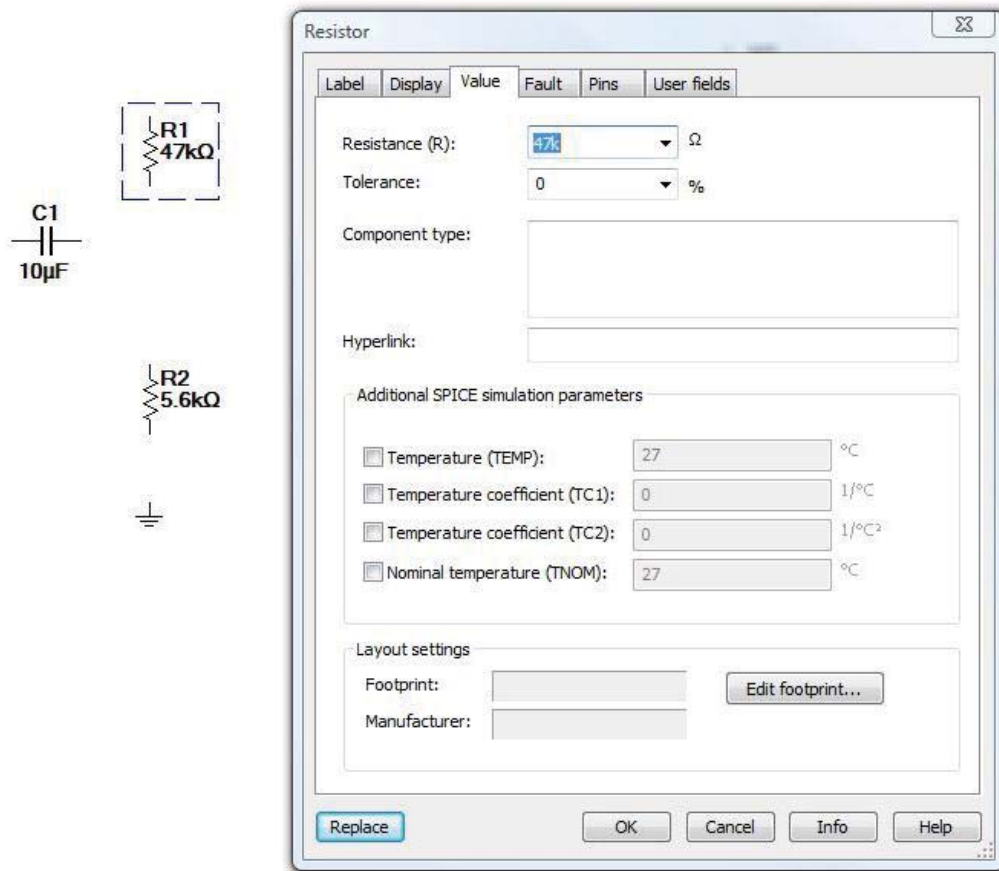


Figure 15

Next click on OK. You have just changed the value of the selected resistor from 1 (default) to 47 k-ohm.

-

[illegible]

Page 23

component and again depress and release. Try connecting your 5.6 k-ohm resistor to your transistor as shown in Figure 18.

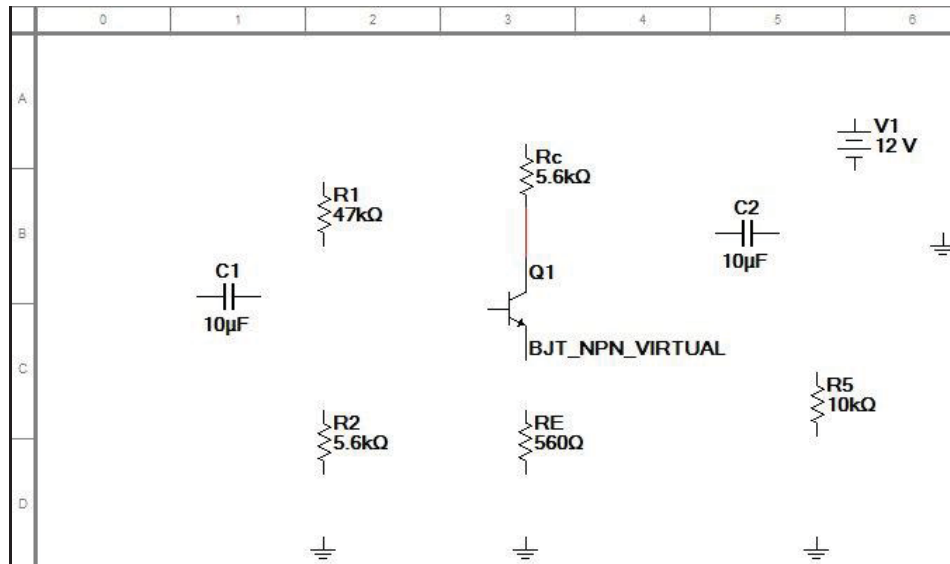


Figure 18

- Using the same procedure wire the rest of your circuit EXACTLY as shown in Figure 19.

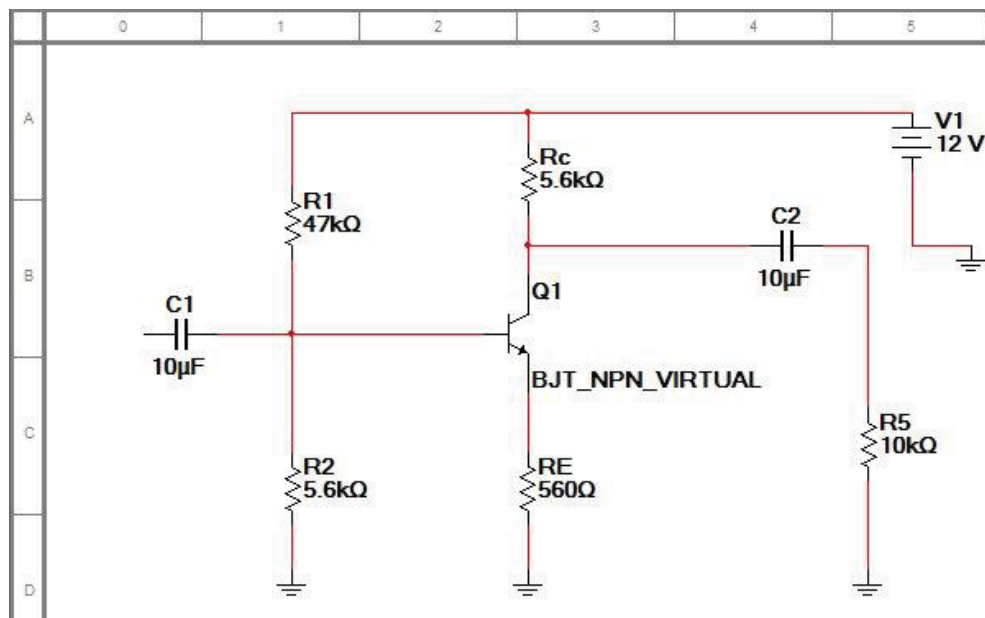


Figure 19

Congratulations, you have entered your first complete circuit schematic. Next, you will add instruments to your workspace that will allow you to simulate and monitor the behavior of your circuit.

10. Recall if you place the mouse pointer over an instrument, the Instrument's name appears. Try it. In the same fashion that you added components to your workspace, add the following instruments; 1-Function Generator and 1-Oscilloscope. When you are finished your workspace should look like Figure 22.

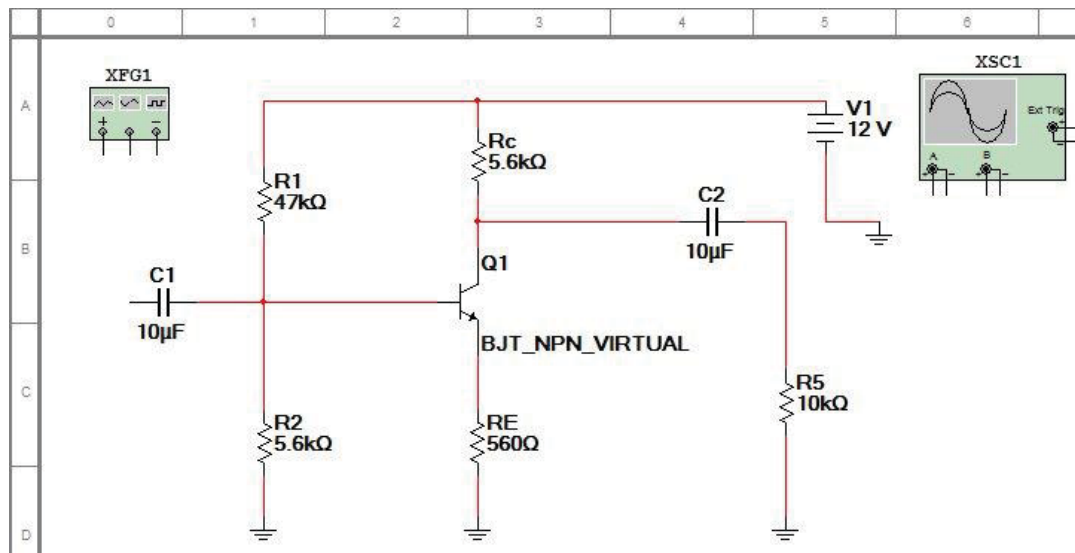


Figure 20

11. Now wire, connect, the Function Generator to the circuit input, at the 10 uF cap, and ground. Then wire the Oscilloscope to monitor both input and output signals. See Figure 21 for connection details.

Notice that channel A of the oscilloscope is connected to the input and channel B to the output. In addition, two ground symbols have been added to accommodate wiring.

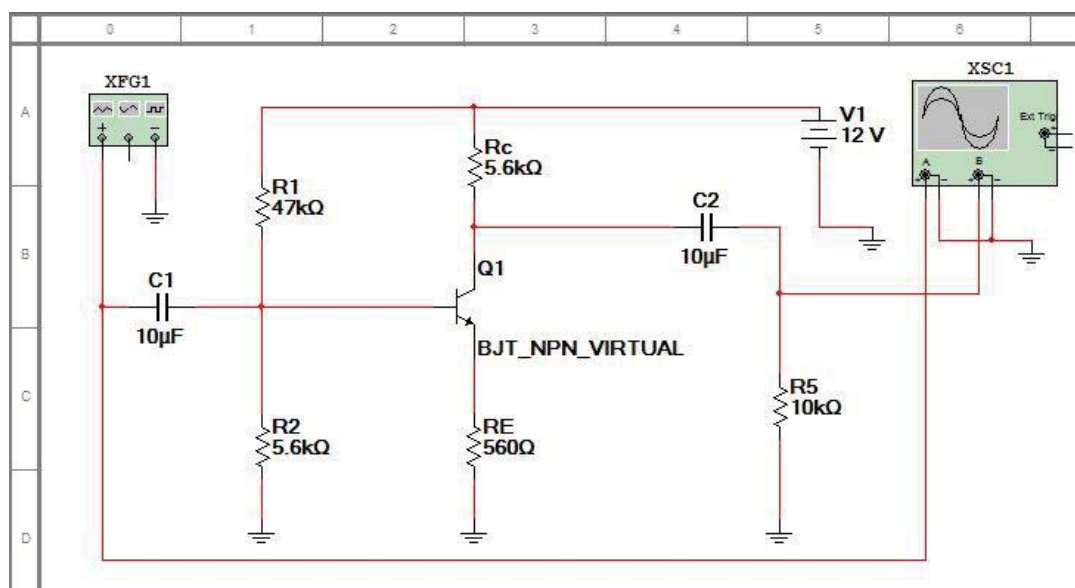


Figure 21

12. Using the left mouse button, double click on the Function Generator, then on the Oscilloscope. The instruments will open and you will see the expanded view.

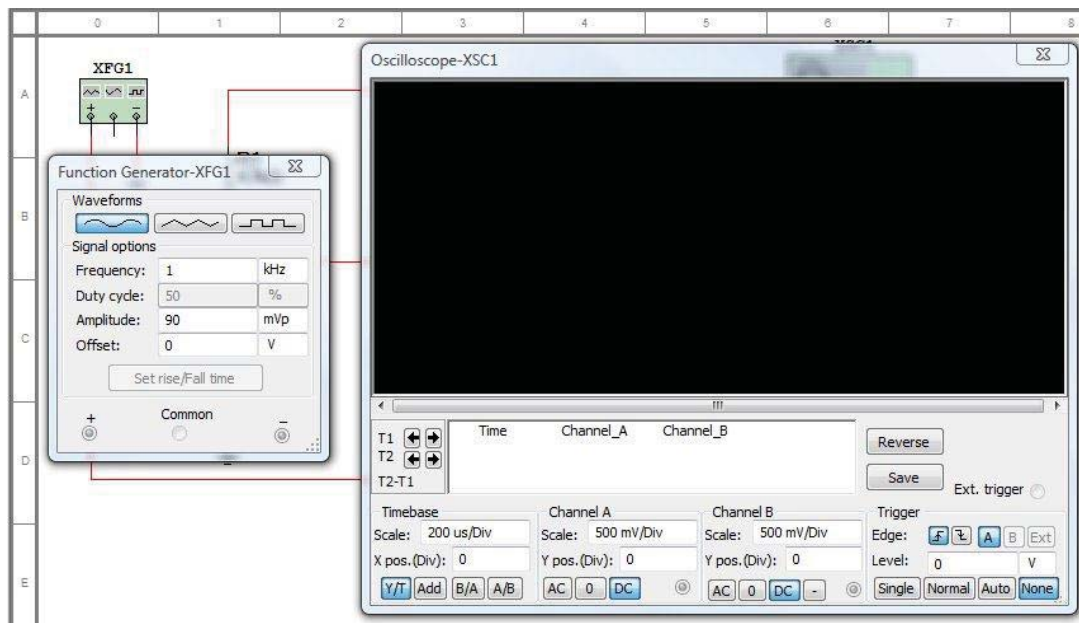


Figure 22

13. Now set up your instruments as shown in Figure 22. For your Function Generator, set Frequency to 1 kHz, Duty cycle to 50%, Amplitude to 90 mV, and Offset to 0. Set up your Oscilloscope as follows; for both channel A and B set the amplitudes to read 500 mV per division, the Y position to 0, and set inputs to AC. Next, set Trigger to auto and the Time base to 200us per division.

You are now ready to simulate your circuit. Double check your workspace to make sure it matches Figure 22.

14. Turn your circuit on by pointing to, and left clicking on, the ON/OFF switch in the upper left-hand corner of the screen. Make sure your instruments are zoomed open so that you can see the readings.
15. Your simulation will run until you stop it. After about 15 seconds, pause the simulation by clicking on the pause button next to the ON/OFF switch.

16. Now your screen should look similar to Figure 23 once you scroll backward using the scroll bar at the bottom of the Oscilloscope. Note you can push the reverse button on the scope to change the default colors. You can use the cursors to measure the amplitude of the signals. Right click on each of the cursor and explore the different options to set the cursors as shown in Figure 23.

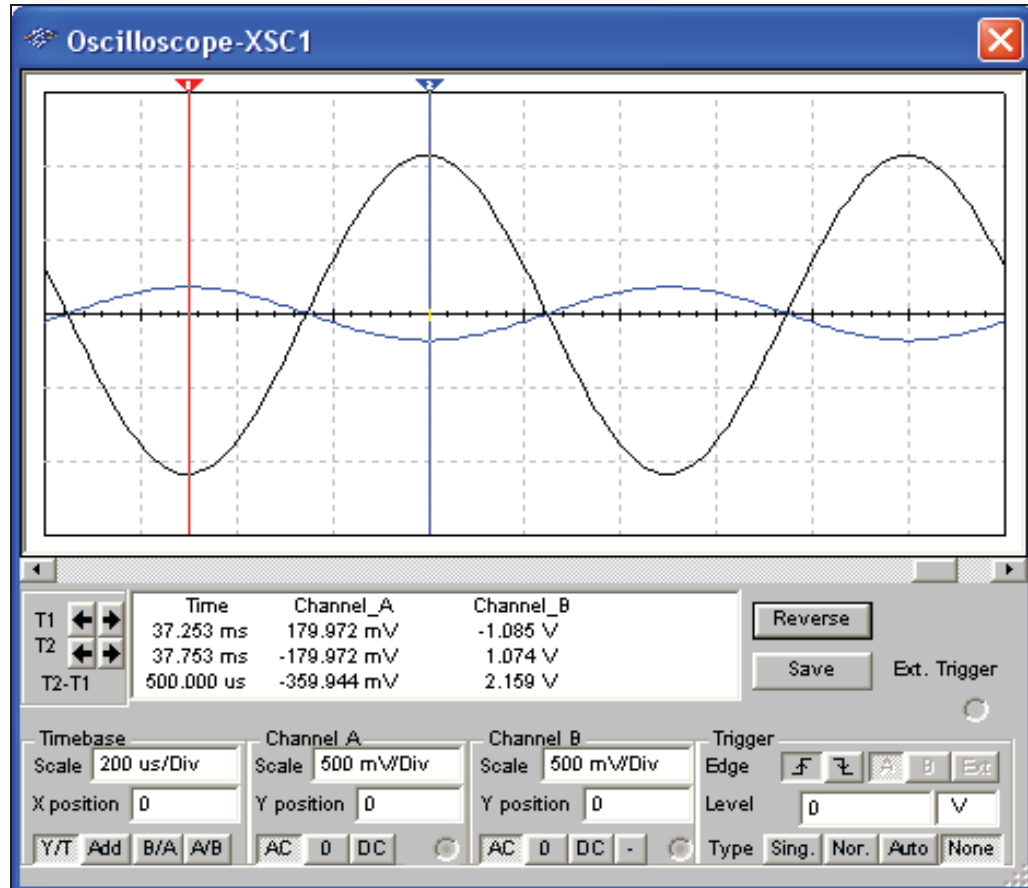


Figure 23

The screen shows the input and output waveforms for your amplifier circuit. Change some settings on your instruments and notice the effects. When you are finished you can save the circuit to your files disk by going to the File menu and clicking on Save. Make sure you are saving the file to your files disk and not to the hard drive of the computer. Next, go to the File menu and click on Print to print a hard copy of your circuit. Make sure when you are printing scope information, that you do not print the entire trace. This takes several sheets of paper. You can set up Multisim to only print a section of your data. Ask you instructor for help with the print options.

Exercise

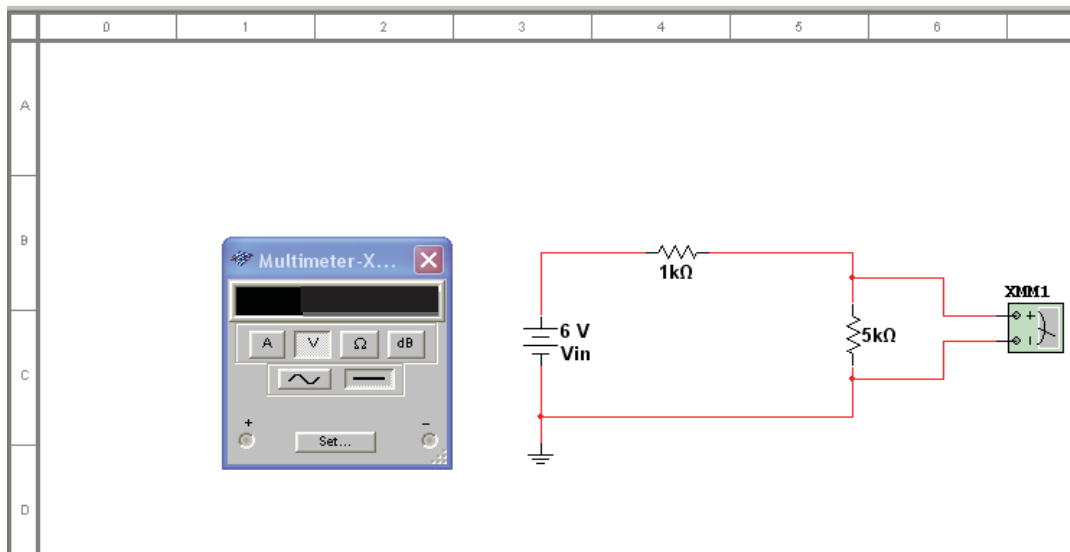


Figure 24

To illustrate the usefulness of a simulation package, open a new Multisim workspace and build the same voltage divider circuit of the exercise 1, page 12, in the Handling Measurement and Component Uncertainties section. Next connect a voltmeter at V(out) as shown in Figure 24.

Using the same voltage and resistor tolerances as before, Now vary the values of V(in) and the resistors to find the expected range for V(out). Compare your results to the results from the previous exercise.

References

Circuit Schematic

[1] Borris, John P “Linear Integrated Circuit Applications” Prentice-Hall, INC 1998.

Schematic Figures

[2] Produced using screen capture and Multisim Simulation and Capture Educational Edition software version 7.0