A Concise Introduction to Multisim™

Adapted from

Computer Programming with Python™ and Multisim™ by James M. Fiore

Published by James M. Fiore via dissidents

For more information or feedback, contact:

James Fiore, Professor
Department of Physical Science, Engineering and Applied Technology
Mohawk Valley Community College
1101 Sherman Drive
Utica, NY 13501
jfiore@mvcc.edu

or via www.dissidents.com

"Without deviation, progress is not possible"

- Frank Zappa

1 Introduction to Multisim

Objective

The objective of this exercise is to become familiar with the Multisim™ electrical circuit simulation package in order to create simple schematics and perform basic simulations. The differences between virtual, real, and 3D components will be examined along with the use of virtual instruments to make simulated measurements. Version 14 of Multisim is presented here. Please note that the precise look of the windows, menus and dialog boxes may be slightly different from that pictured depending on the version of Multisim that is being used. Regardless of appearance, the functionality remains.

Procedure

After logging into the computer, open Multisim. If a desktop shortcut is not available Multisim may be accessed via the *Programs* menu under the *Electronics Workbench* menu item. This is a large program and may take a minute or two to load. Eventually, you will be greeted with something similar to the screen shown in Figure 1-1. As the toolbars are customizable, the precise look of the program may be a little different from that shown. In general, there are a series of toolbars along the top. These are used to select different components and editing or viewing functions. Multisim's schematic capture facility is object based, that is, you "draw" a circuit by selecting predefined objects such as resistors and transistors, and drag them onto the workspace. They are then wired together using the mouse. You can zoom into or out of the workspace using the mouse.

By default, along the left edge is the *Design Toolbox* browser window. This may be closed to create more working room for the schematic. At the bottom is the tabbed *Spreadsheet View* that shows simulation data, components, etc. Clicking on entries will highlight the corresponding elements on the schematic worksheet. Along the right edge is a vertical toolbar that contains virtual instruments.

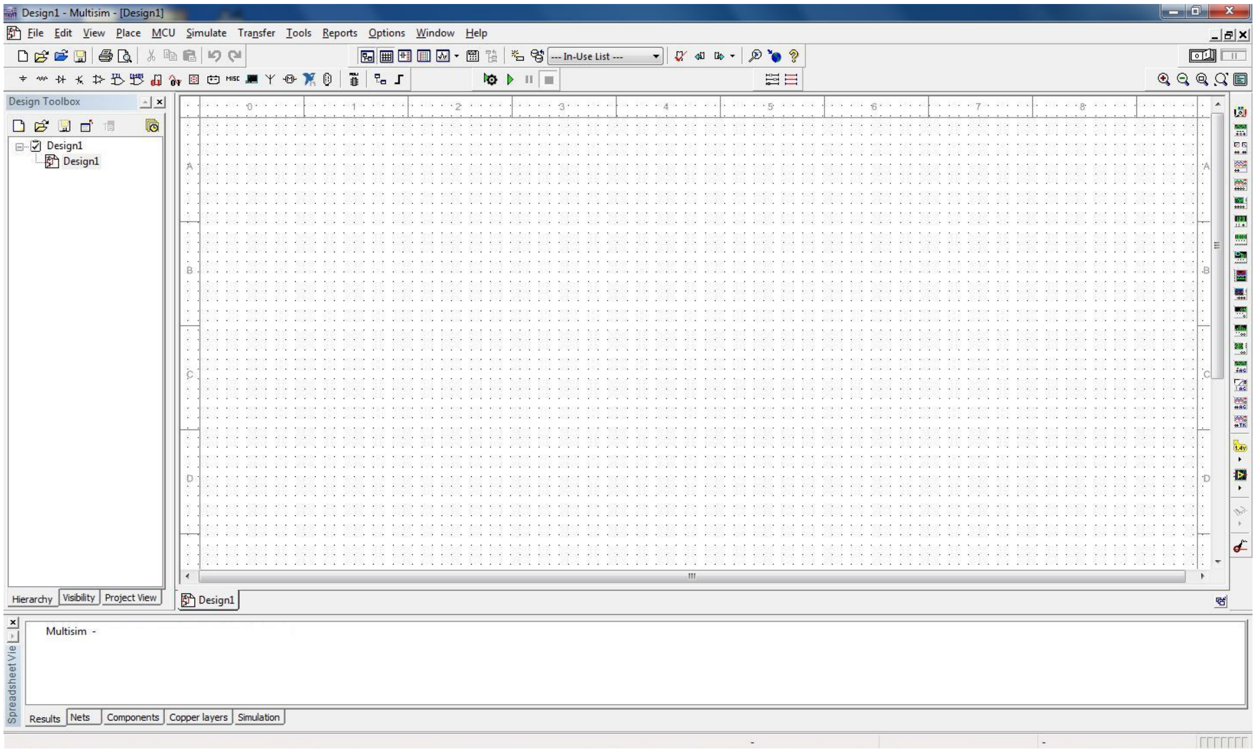

Figure 1-1

Multisim uses three different kinds of components to create schematics. They are virtual basic and rated components, real (or manufacturer's) components, and 3D virtual components. Virtual and real components use the standard industry schematic symbols. By default, components with physical footprints (most real components) are colored blue and components without a physical footprint (virtual components) are colored black. In contrast, 3D components look more like photographs of the actual component. Examples are shown in Figure 1-2. Although 3D components add a certain amount of color and false reality to a circuit, they are non-standard and not generally used for simulations. We shall not discuss them further.

The difference bewteen virtual and real components is that real components reflect items from a manufacturer's database. The items include physical parameters, such as size and pinouts, which are required for designing proper printed circuit boards. Also, real behavioral models for semiconductor devices such as op amps will be more accurate than the virtual models. Finally, the values of real passive components (resistors, capacitors and inductors) are limited to the nominal values specified by the manufacturer. In contrast, the values of virtual components can be set to almost anything, however, there are no corresponding physical data. As a consequence, if a PCB is needed, virtual components are not the appropriate choice. In practice, if the goal is to create a production circuit, real components will be used. If the goal is to simulate a lab exercise, virtual components will be used for the passives (rated resistors, capacitors and inductors) and reals will be used for the active components (transistors, diodes, op amps, etc.).

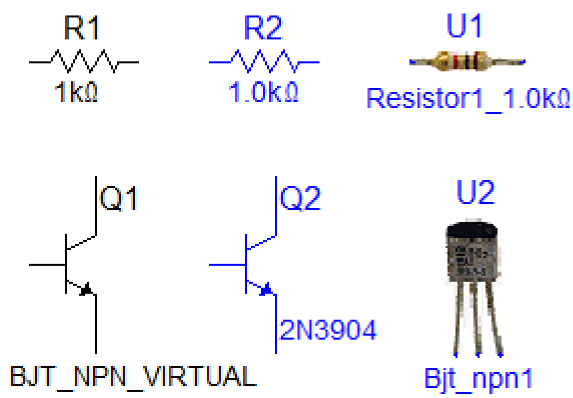

Figure 1-2

To illustrate the use and editing of components, drag a DC voltage source onto the workspace. It will show up with a default voltage value and label. Double click the symbol. A dialog box will pop up next to it as shown in Figure 1-3. (If the component toolbars are not shown on your version, use the *Place Component* menu instead.)

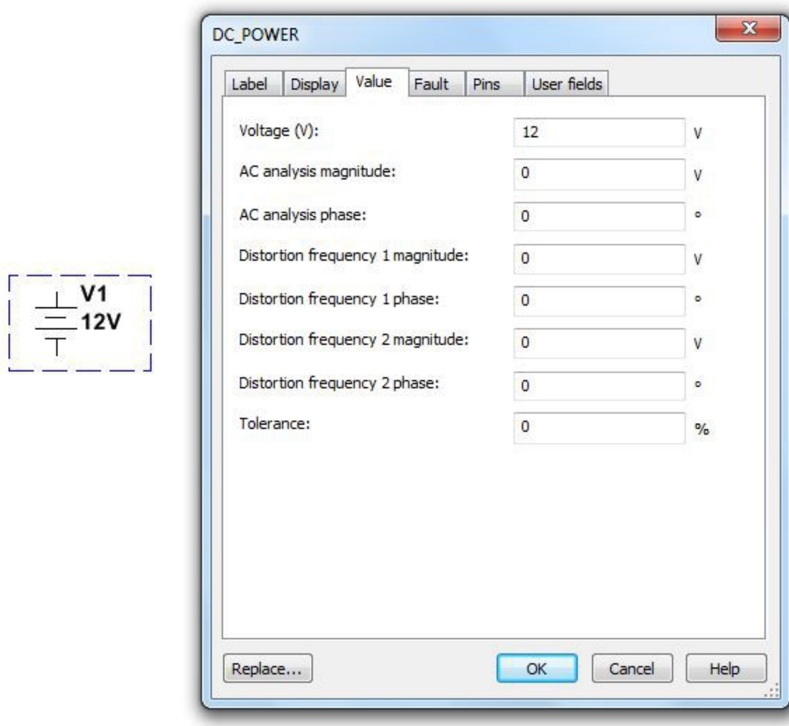

Figure 1-3

From this dialog you can change a variety of attributes, the most important of which are the voltage value and the name. Double clicking any component will bring up this settings dialog box although the precise items contained within it will vary from component to component. For example, for a resistor there will be a resistance setting instead of a voltage setting. If you need to remove a component, simply select it and hit the Delete key.

Editing the position and orientation of a component is straight forward. Once the item is selected (shown by a surrounding dashed line) it may be moved using either the mouse or the cursor keys. If you need to move a group of components, the mouse may be used to select several items by clicking and then dragging the mouse over them. Every component within the selection box will be highlighted and will move as a group. Also note that it is possible to select the text labels of components and just move them. This can be handy if a label becomes obscured by a wire.

Components may also be rotated and flipped. These commands can be accessed from the main menu, however, it is handy to remember certain keyboard shortcuts (such as CTRL-R). You can also customize the tool bars and add these commands as their own buttons for easy access. The *Customize* dialog is shown in Figure 1-4 and is accessed via the *Options* menu.

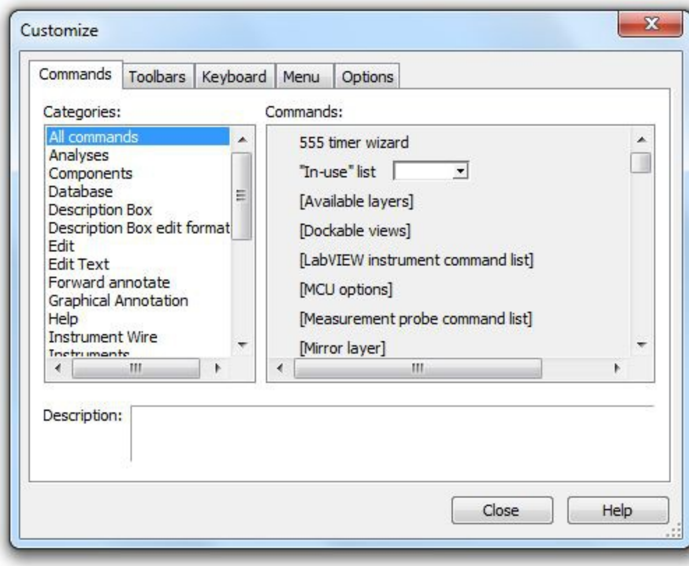

Figure 1-4

Along with editing the components and customizing the tool bars, you may also customize the look of the work space. Go to the *Options* menu and select *Sheet Properties*. From here you can select a variety of color schemes for the components and wiring. You can also select which component items (labels, values, etc.) will be displayed. Fonts may be altered as well. Be fore-warned, it is possible to spend a great deal of time trying to make the work space look pretty instead of doing truly productive work. Don't fall into this trap. Before we close this dialog, there is one important setting to note and that is the section labeled *Net Names*. For now leave it as it is. We shall revisit this in the future.

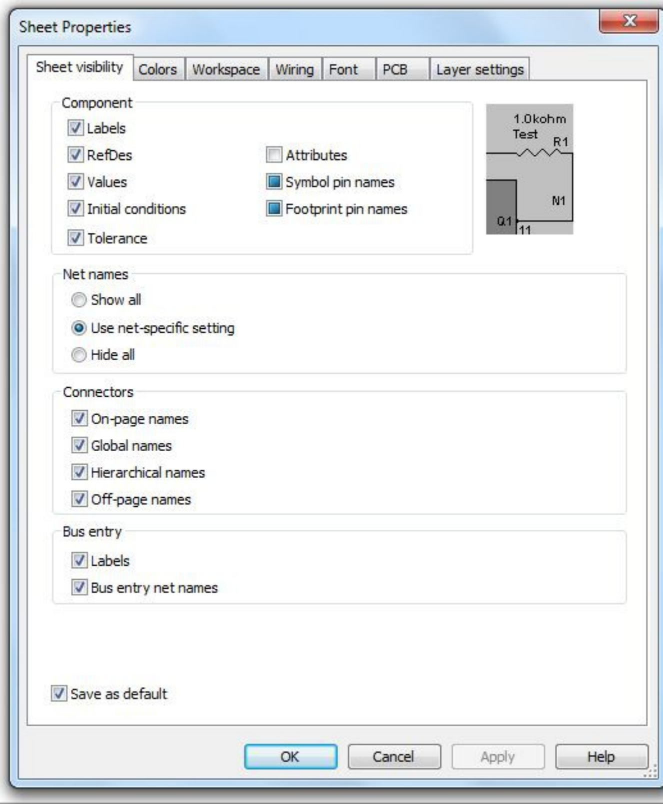

Figure 1-5

OK, let's create a circuit and perform a simulation. You should already have a DC voltage source on your work space. From the virtual (blue) components tool bars, select two resistors and an earth ground symbol (or alternately, use the *Place Component* menu). We shall make a series loop of the three elements with the negative end of the power supply at ground. One resistor will need to be rotated 90 degrees (one horizontal and one vertical). In order to wire the items together, simply click on the free lead of one component and move to the desired lead of another. While moving, Multisim will draw a ghost line. Clicking on the second component will create a proper wire (by default, colored red). Wires are always drawn along the horizontal and vertical with 90 degree bends, not directly from point to point. This is the proper way to draw a schematic in the vast majority of cases.

It is possible to click on the middle of a wire in order to tie multiple items together. A small node circle will be drawn at the connection point. To delete a wire, click on it to select it. You will see a set of small box "handles" around it. To remove the wire, simply hit the Delete key. Note that you can also move the wire with those handles if desired. In fact, it is possible to align wires at odd angles with these handles (again, this is not typical).

Note that if you move a component, Multisim will automatically move the wires along with it. You do not have to rewire it. Sometimes, especially if components are too close, the wire may be draw in an odd, looping form. The easiest solution is to simply separate the components a little more.

Once the components are in place and wired, double-click each component to set their values as shown in Figure 1-6. If you're using power rated virtual resistors, also increase their power rating to one watt, each. We shall then add two the two voltmeters. Please note that it is perfectly acceptable to change the component values immediately after dragging them onto the work space; you don't have to wait until they are wired together. It is suggested, though, that you decide upon a particular workflow and stick to it otherwise the chances of forgetting to change components from their default values increases.

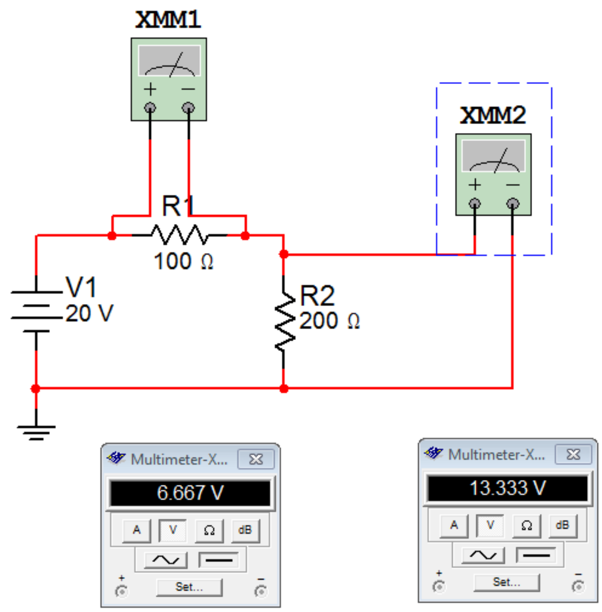

Figure 1-6

To add the voltmeters (XMM1 and XXM2 in Figure 1-6), select a virtual DMM from the top of the *Instruments* toolbar along the right edge of the screen. Drag this onto the work space and wire it across R1 as shown in Figure 1-6. Grab a second virtual DMM and repeat the process for R2. Now double click on the DMMs. Two small windows will pop up (they might overlap each other, if so, move one over). Select the V (voltage) button and the straight line (DC) button on each of them. The circuit is now ready to perform a simulation.

To start a simulation, you can select the the rocker switch located in the upper right corner of the menu area. This is the virtual on-off switch. Alternately, you can select *Run* from the *Simulate* menu or use the green *Run* button on the simulation toolbar. In a moment, you should see voltages appear on the two virtual DMMs. **In order to edit the circuit, the simulation must be turned off.** So, if we wish to add or delete components, we must remember to "power down" the circuit just as we would in a real lab. To be safe, do this now.

Multisim has a wide variety of virtual instruments. Some of them are fairly simple such as the DMM just used. Other instruments are virtual recreations of real-world test instruments. For example, from the Instruments toolbar select the Tektronix Oscilloscope. You may place this anywhere on your schematic. Now double click on the small icon for this device (XSC1). A rather ornate window opens which appears to be the front panel of a Tektronix TDS 2024 oscilloscope, very similar to the models we have in electrical labs, right down to subtle shadows around the knobs! See Figure 1-7. This has the advantage of immediacy (assuming you've used this type of oscilloscope before), however, it is not the ultimate way to perform a simulation. We will look at even more powerful and flexible ways of creating simulations in the next exercise. For now, you may wish to delete this new instrument from the work space and then save the existing schematic. Remember to always save to either your student account on the H drive or to a USB drive. Never save directly to the C drive.

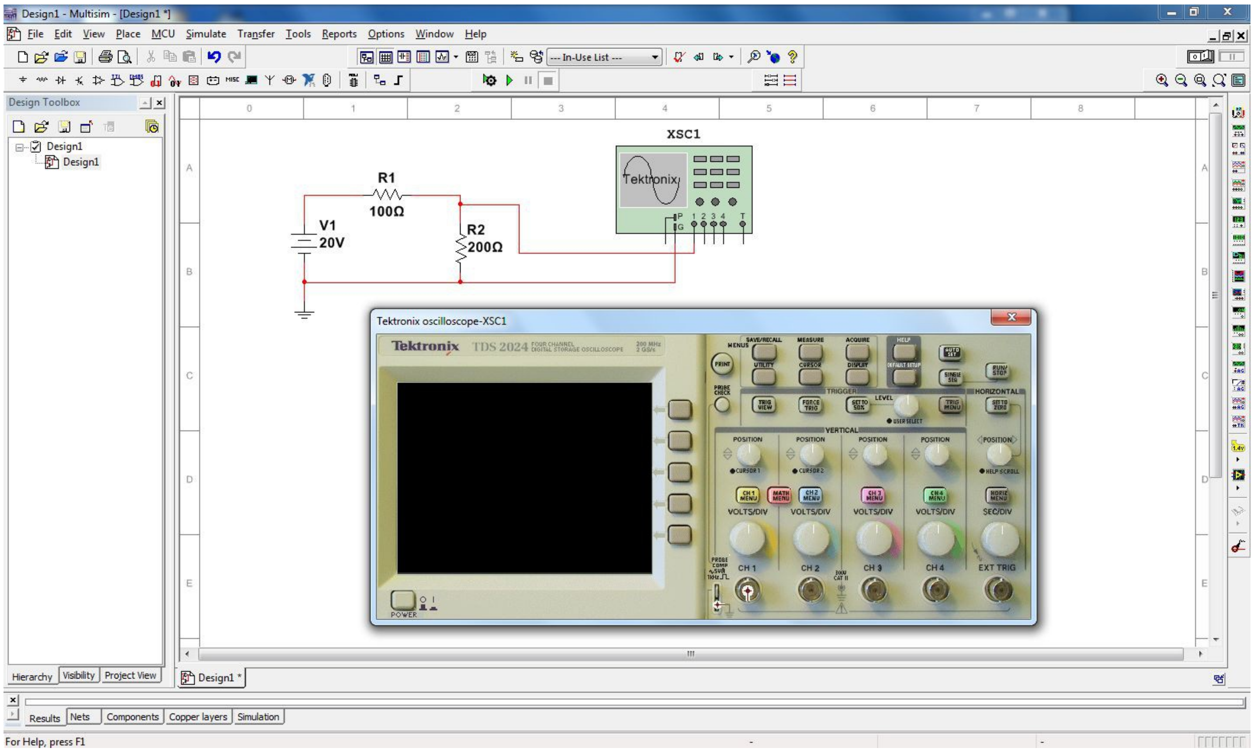

Figure 1-7

There are three very good "every day" uses for Multisim during your studies: First, it is a very handy tool for verifying lab results. That is, you can recreate a lab circuit, simulate it, and compare the simulation to both your theoretical calculations and lab measurements. Second, it is a handy tool for checking homework if you get stuck on a problem. Third, it is convenient for the creation of schematics, for example, for a lab report. An easy way to do this is to draw the circuit, capture the screen image (use the screen grabber utility or by pressing the Windows+Print Screen keys to copy the desktop into the clipboard) and then paste the image into your favorite image manipulation program. From there you can edit it as needed and then paste the modified image into your lab report.

At this point you may wish to experiment a bit by rewiring the circuit to measure current or to try building new circuits. As with any tool, continued practice will hone your skill. Once you are done and have saved the file (the extension should be .msX where X is the Multsim version number), close Multisim and then shut down the computer.

2 Multisim Extensions

Objective

The objective of this exercise is to become more familiar with the Multisim electrical circuit simulation package in order to use more generalized simulations via the *Grapher Window*.

Procedure

In previous work we have examined the basic functionality of Multisim, namely basic schematic capture functions such as component selection, placement, and parameter editing, along with simple simulations using virtual instruments such as a Digital Multimeter (DMM) to measure DC voltage. While virtual instruments are quick and easy to use, and offer some amount of familiarity, they are necessarily limited in other aspects. Some of the issues with virtual instruments include:

1. Limited measurements per unit, for example a single measurement for a DMM, requiring multiple units for multiple measurements.
2. The need to rewire the instruments (and hence the schematic) in order to take different readings.
3. Excessive amount of work space area obscured by the instrument(s) with accompanying clutter.
4. No convenient way of storing and recalling prior simulations.
5. No convenient means of exporting simulated measurement data to other programs.

To address these issues Multisim allows non-instrument simulations through the use of the *Grapher* window. The Grapher is a single, general purpose window that presents simulation data in both text and graphical form, as appropriate. Large amounts of data may be displayed simultaneously. The display itself is highly customizable (titles, axis ranges, colors, fonts, etc.). Each simulation is given its own sheet or tab and these may be saved for future reference. To top it off, nothing needs to be wired into the existing schematic and the Grapher may be minimized for greatest access to the schematic. The Grapher may be used for both DC and AC simulations, and includes interactive measurement tools for certain simulation types.

Because nothing is wired to the schematic, some means of identifying the measurement points is required. Multisim, like virtually all other simulation programs, does this through the use of numbered *nodes*. A node is simply a connection point in the circuit. Ground is always node number 0. The numbers increase as connections are made. Consequently, if the same circuit is entered by two different people who wire the components in a different order, the node numbers will not be the same between the two circuits. This is generally not a problem. As a side note, Multisim will not back-fill nodes that have been deleted. For example, if a circuit has been wired up to node ten and then node five is deleted, the next node to be wired will be numbered eleven, not five. Node number five will simply remain unused. Again, this is not a problem. It is important to note that, internally, Multisim always refers to connections via their node numbers, whether you use them or not.

When using virtual instruments, node numbers can add a certain amount of clutter to the schematic, consequently they are not shown by default. To show node numbers, select *Sheet Properties* from the *Options* menu. The dialog box of Figure 2-1 opens. This dialog box should be familiar as it was used in the prior exercise to set schematic colors and the like.

If the dialog box looks a little different from Figure 2-1, make sure that the *Sheet Visibility* tab is selected (leftmost tab). The center area is labeled Net Names. This controls whether or not node numbers will be visible on the schematic. To make node numbers visible, select *Show All*. Select *OK* to close the dialog box. We will now proceed to create a simple schematic.

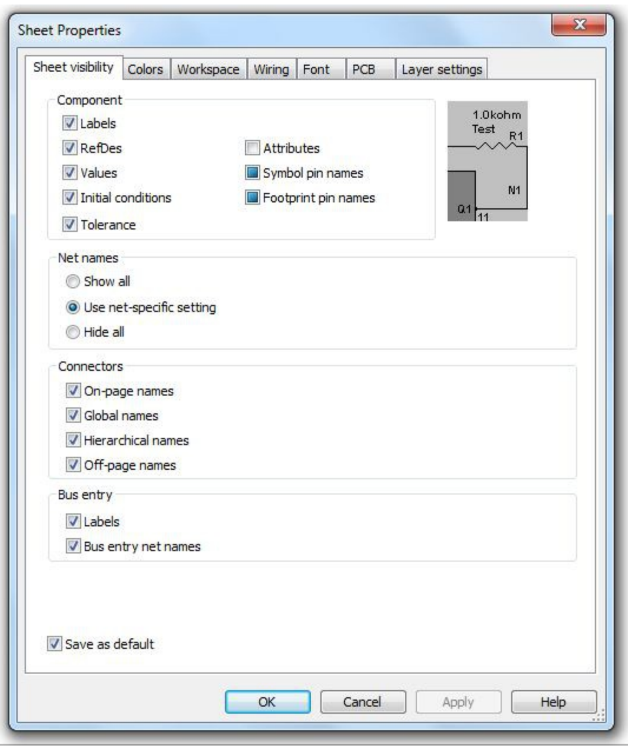

Figure 2-1

Using virtual components, drag a DC voltage source, an earth ground and two resistors onto the work space. Connect them in a series loop and edit the component values to 20 volts for the source and 4000 and 6000 ohms for the two resistors. Once completed, the circuit should look similar to Figure 2-2.

Note that ground is node zero. This is always the case. If the components were wired from left to right, the other two nodes will be one and two as shown. If the wiring steps were reversed, the node numbers will be reversed.

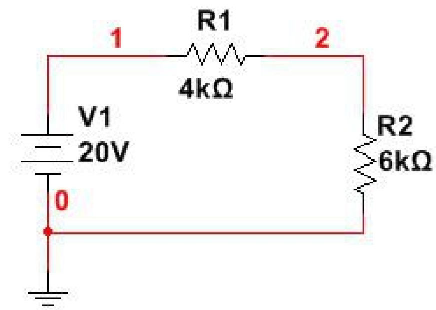

Figure 2-2

We shall perform a simulation to inspect a few DC voltages. Instead of wiring in virtual DMMs, we shall select the *Analyses and Simulation* item under the *Simulate* menu (see Figure 2-3).

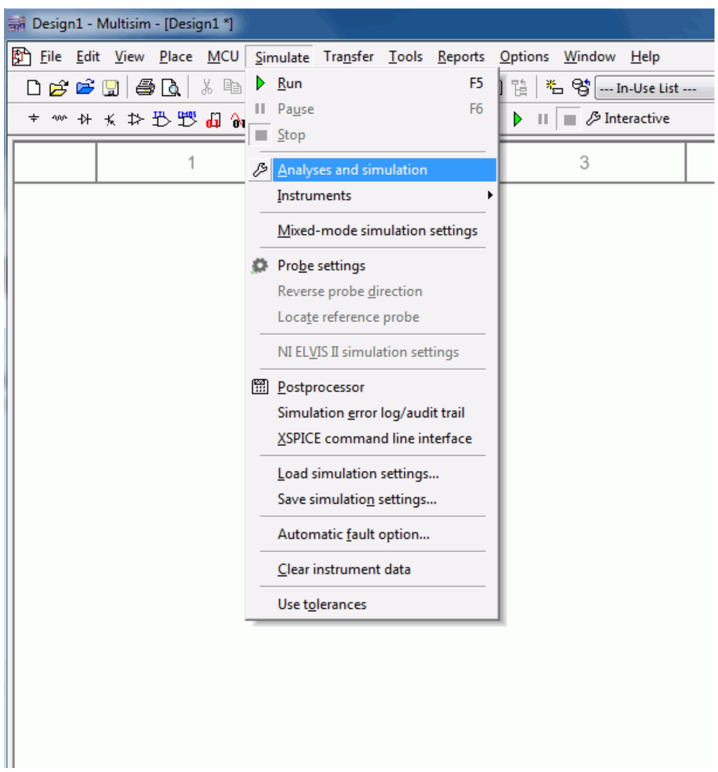

Figure 2-3

A dialog box will open (see Figure 2-4). There are well over a dozen analyses available. To find DC voltages or currents in our circuit, select *DC Operating Point* from the *Analysis* list. The dialog box will update and now contains two list areas. On the left will be a listing of available node voltages along with branch currents. The right side contains those items currently chosen for analysis. Node voltages will be shown with a V prefixed to the node number as in V(1). Branch currents will use an I prefix. In version 13 and earlier of Multisim the Analyses menu has a sub-menu of analysis choices. Simply select the analysis type from the sub-menu and then continue with the settings as described.

Select both nodes one and two on the input list with the mouse. To transfer them to the output list, select the Add button between the two lists. For now, this is all we need to concern ourselves with but it is worth mentioning that it is possible to create mathematical expressions of variables as well. For example, one node voltage could be subtracted from another in order to find the voltage across a single component or a group of components. Further, a node voltage could be squared and divided by a circuit resistance to determine the power dissipation.

Select the *Run* button at the bottom of the dialog to start the simulation.

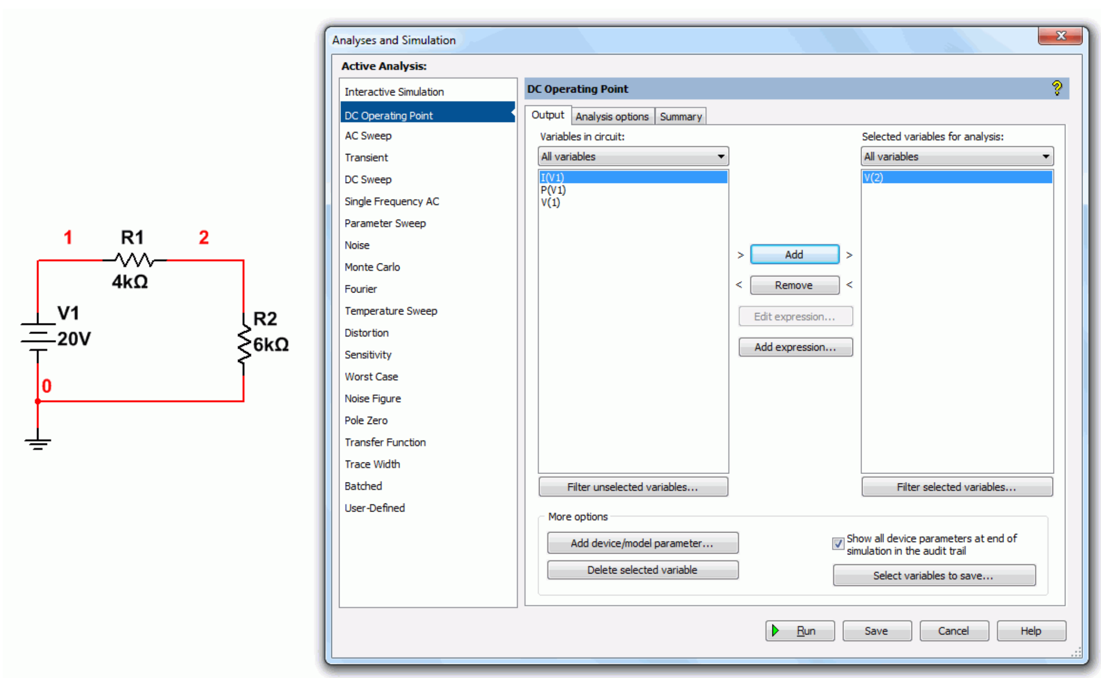

Figure 2-4

After a moment, the Grapher window will appear as shown in Figure 2-5. For this simulation, the Grapher simply lists the node number and the corresponding voltage in two adjacent columns. This is a nice, compact display, much nicer than the individual DMM windows, especially if a large number of voltages are needed.

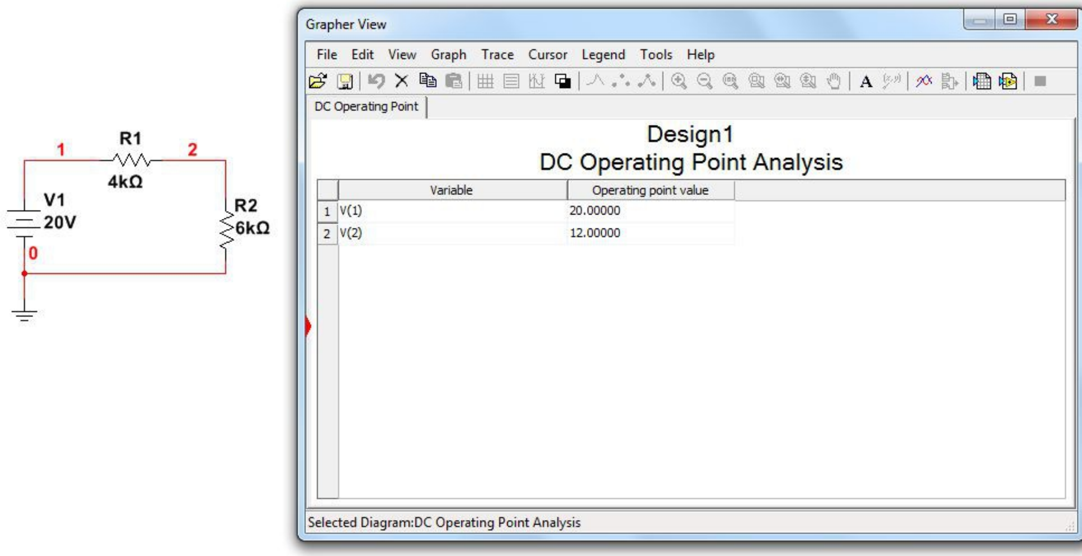

Figure 2-5

Along the top of the Grapher window is a toolbar area for saving, cutting, customizing, and so forth. As useful as this is, the Grapher comes into its own when used with AC simulations.

Close the Grapher and modify the circuit to replace the DC source with an AC signal source as shown in Figure 2-6. Make sure that the new source is set to 20 volts with a frequency of 1000 Hz (1 kHz).

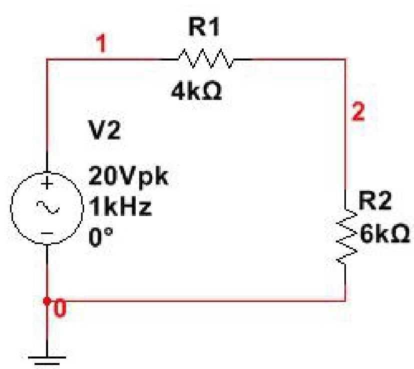

Figure 2-6

We are going to examine the node voltages once again, however, this time the voltages will vary over time. If we were to use virtual instruments for this we'd choose once of the virtual oscilloscopes. In this case, however, we shall choose *Transient Analysis* from the *Analyses and Simulation* menu. A dialog box will open like that of Figure 2-7.

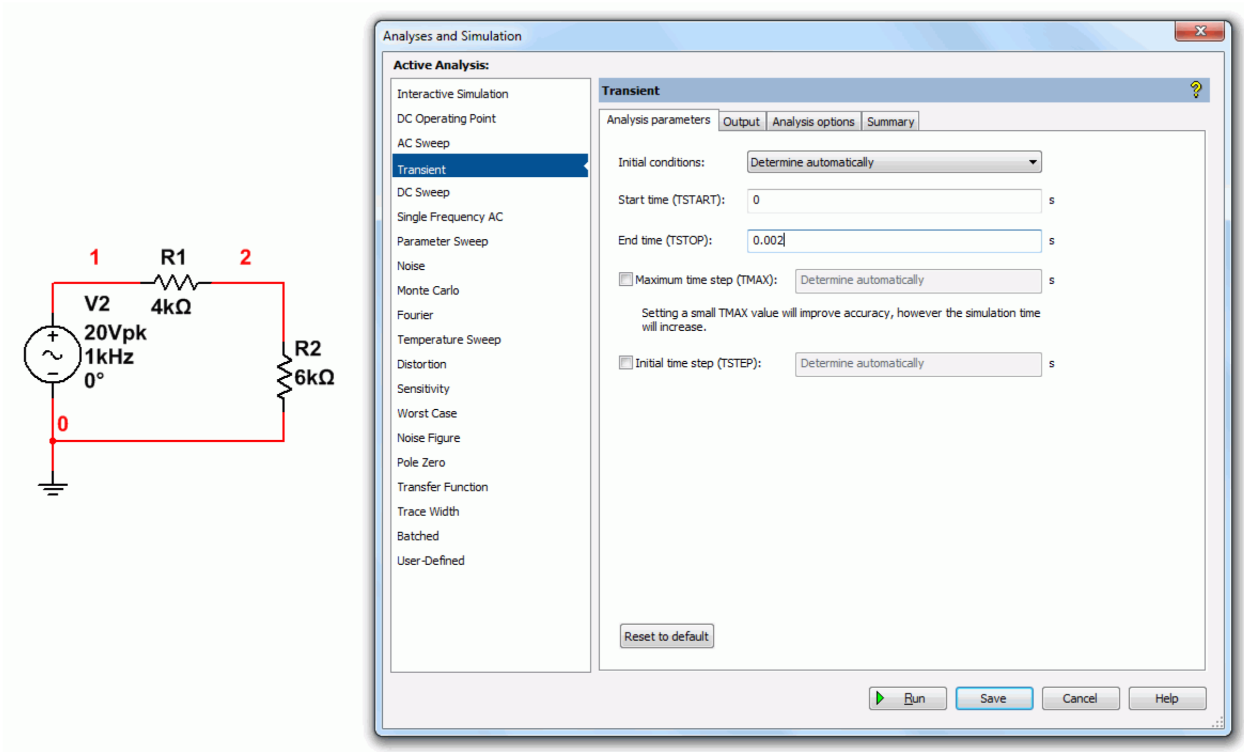

Figure 2-7

The first thing we need to do is select the range of times we wish to see. Set the *Start time* to 0 and the *End time* to two milliseconds (0.002 seconds). Now select the *Output* tab. This is the same tab you saw back in Figure 2-4. Make sure that voltage nodes one and two are in the output analysis (right side) list. Now select *Simulate*. The Grapher will reopen with a display similar to that of Figure 2-8.

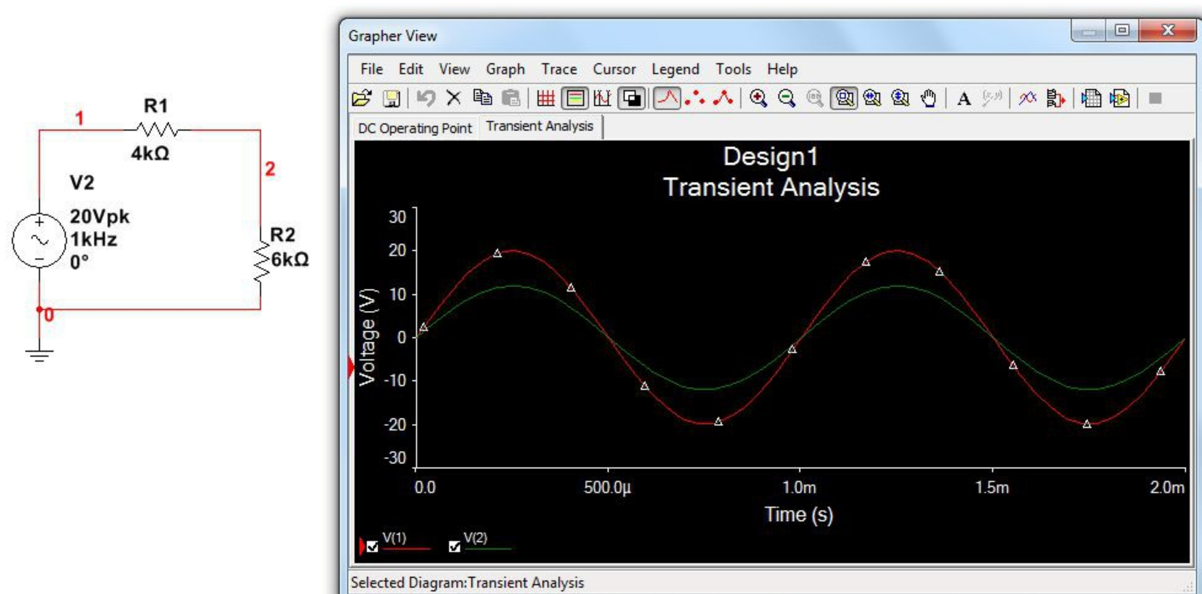

Figure 2-8

The red and green traces graph the voltages at nodes one and two as they change through time. Note that there are many ways to customize this display. For example, selecting the white/black double square toward the middle of the toolbar will toggle the background color and labels between black and white. The grid pattern will overlay a measurement grid and the set of horizontal lines will bring up a color-node number legend. You can also click on the labels and axes to change them.

One of the more useful items is the pair of interactive cursors. The button for this is between the *Legend* and the *Magnify* buttons. Selecting this will bring up a pair of vertical cursors which may be grabbed and moved with the mouse. A separate cursor window will open. This will list the coordinates of the cursors on the plot lines along with the differences between the points (that is, how far apart the points are in time and in voltage). It will also list maxima and minima along with other useful data. An example of the Grapher with many of these settings in place is shown in Figure 2-9.

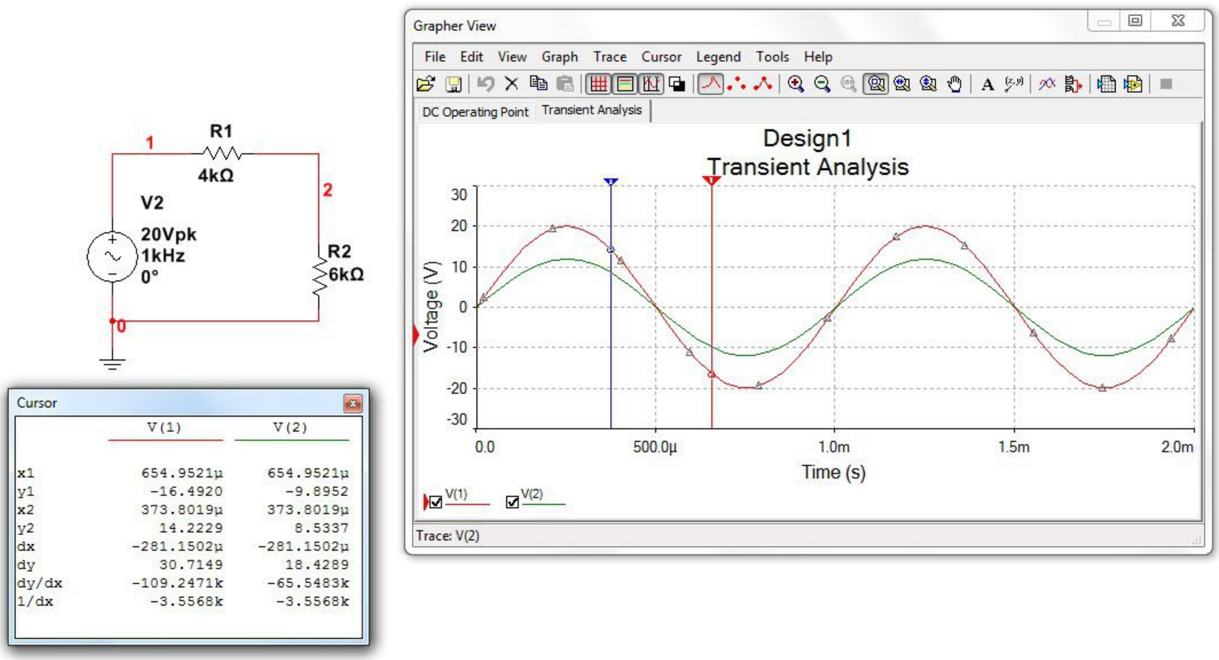

Figure 2-9

Finally, note that the original *DC Operating Point* tab is still there. That is, you can quickly recall the original DC analysis by just selecting this tab. It should be clear by now that while the Grapher takes a little more to set up, it is far more flexible and useful than simple virtual instruments.

Assignment

Recreate the transistor amplifier schematic shown in Figure 2-10. Try to make it as close to this drawing as possible. All parameters must be the same. Device designators must also be the same (after all, these will be cross referenced to the PCB layout and bill of materials). The only items to change are the date (use today's date) and the "Designed by" entry (insert your name).

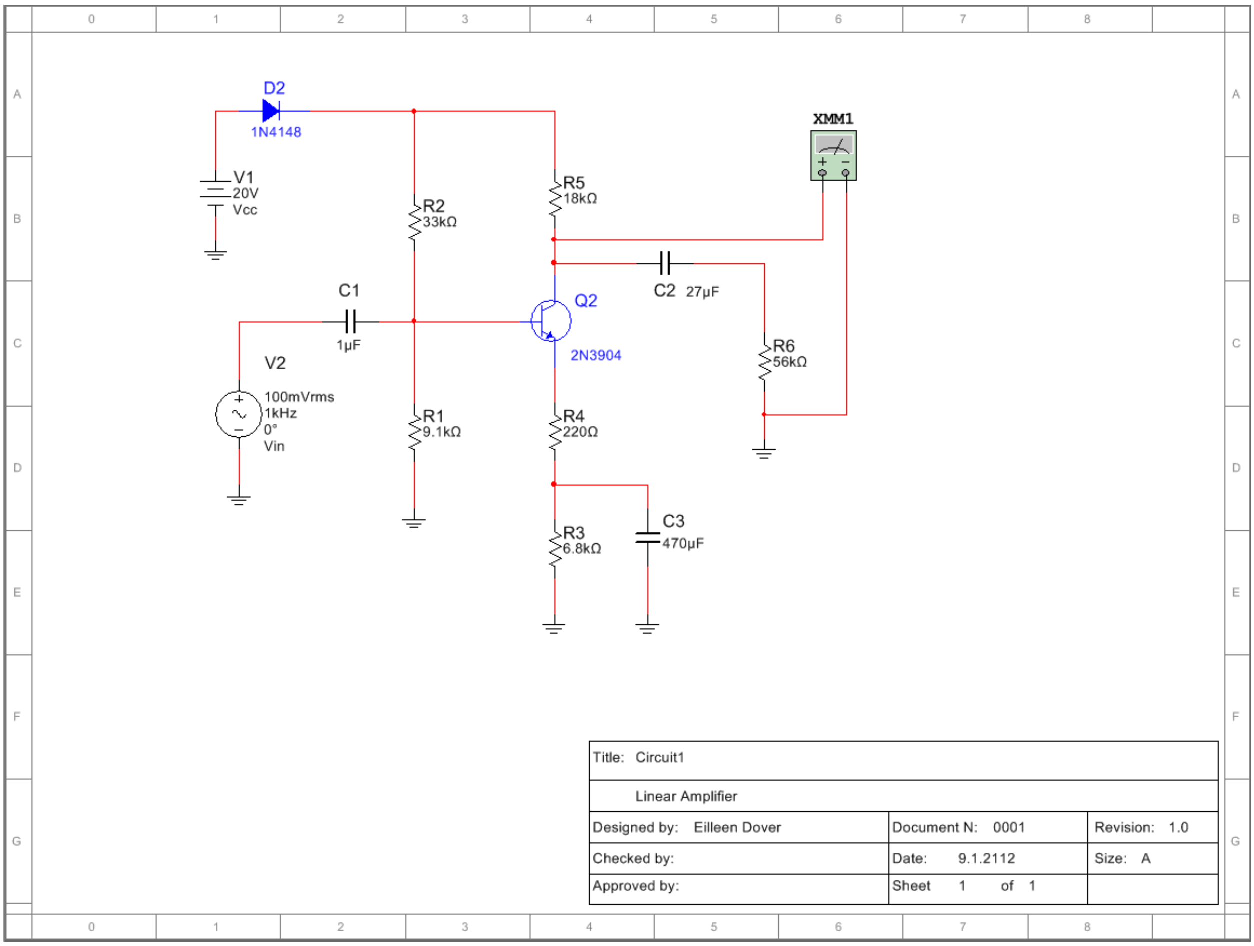

Figure 2-10