

Experiment 1

Electrical Circuits Simulation using Multisim Electronics Workbench: An Introduction

Simulation is a mathematical way of emulating the behavior of a circuit. With simulation, you can determine a circuit's performance without physically constructing the circuit or using actual test instruments. Multisim is a complete system design tool that offers a very large component database, schematic entry, full analog/digital SPICE simulation, etc. It also offers a single easy to use graphical interface for all design needs.

Introduction

Go to **Start**→**Programs**→**Multisim** and click on **Multisim**. This will open the main window as shown in Figure 1. In Figure 1 important toolbars and menu are labeled. In addition to toolbars shown in Figure 1, there maybe other toolbars appearing on your screen concentrate on the labeled items in Figure 1 at this time.

You can always open and close a toolbar from Main Menu. For example if you want to open or close (select/unselect) the Design Toolbar, select **View**→**Toolbars**→**Design**. If any toolbar is not appearing on your screen then use the above procedure to bring the toolbar.

Most of the analysis can be performed turning on-off the simulate switch. If the Simulation Switch shown in Figure 1 is not appearing on your screen then select **View**→**Show Simulate Switch** in the Main Menu. This will open the Simulation Switch.

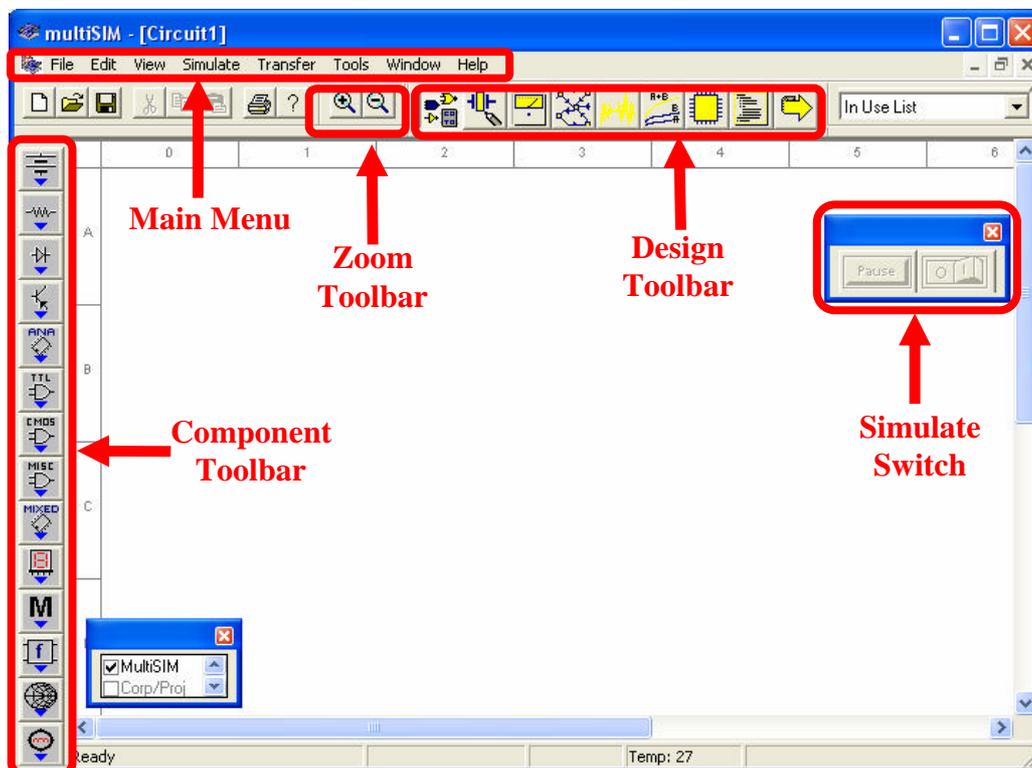


Figure 1: Main Window of Multisim Simulation Software

We will now try to learn about Multisim simulation techniques by solving a simple example.

Example

Build the circuit shown in Figure 2 using *Multisim Electronics Workbench*.

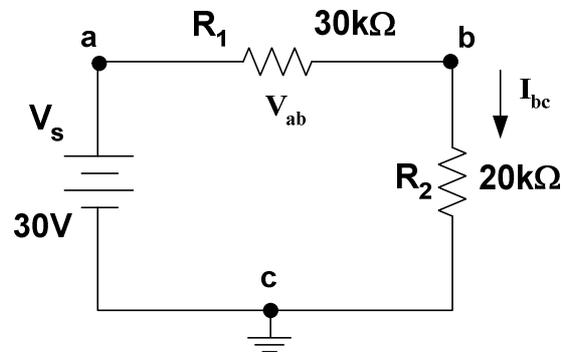


Figure 2: Circuit for Multisim Simulation

STEP A: Placing the Components

1. Place a Battery (DC Source)

a. Bring a dc source in the Multisim workspace:

Open the Multisim program if it is not open. In the Component Toolbar, select **Sources** icon (refer to Figure 1 to find the Component Toolbar). This will open another window with several types of dc sources and other components as shown below in Figure 3. Click on “DC Voltage Source” in this new window.

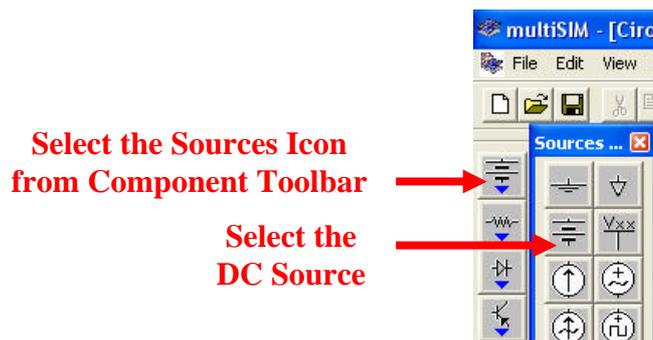
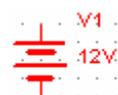


Figure 3: Selection of DC Voltage Source in Multisim

Now bring your cursor in the workspace area and notice the change in the shape of cursor to



Click at any point in the workspace. This will put the voltage source as



b. Change the value and name of voltage source:

Double click on the voltage source that you just placed in the workspace, a new window with the name **Battery** will appear, as shown in Figure 4. Select **Value** in the Battery menu, if it is not already selected. Change the value from 12 to 30. Keep the unit as Volts in this menu. Now select **Label** in this menu and change the **Reference ID** to Vs. Click on **OK**.

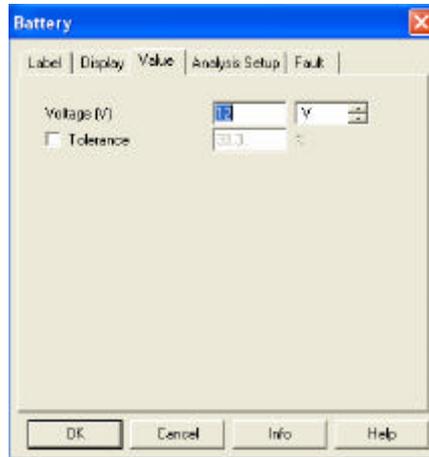


Figure 4: Battery Window for setup of DC voltage source

2. Place a Resistor:

a. Bring a resistor in the Multisim workspace:

In the Component toolbar, select **Basic** icon as shown. This will open another window with several basic components as shown below in Figure 5.

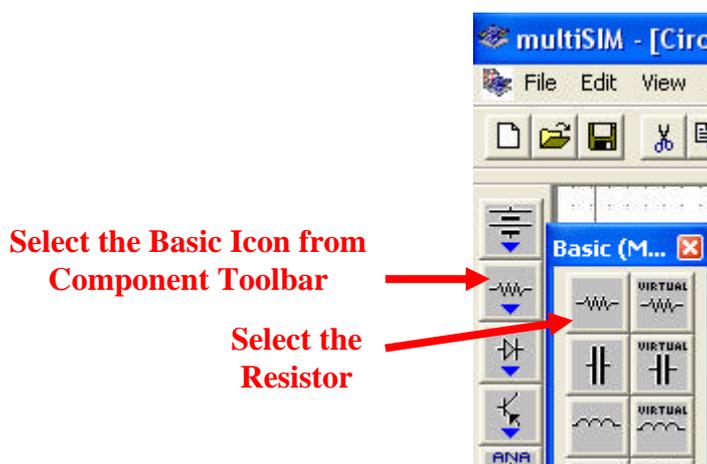


Figure 5: Selection of Resistor in Multisim

Click on “Resistor”, this will open the **Browser-Basic** window, as shown in Figure 6. Scroll through the **Component List**, select 30kohm, and click OK. The cursor shape will change again. Click in the workspace and this will put the resistor as,



Tip To make your scroll through the Browser's **Component List** faster, simply type the first few characters of the component's name. For example, type 30k to move directly to the area of 30kohm list.

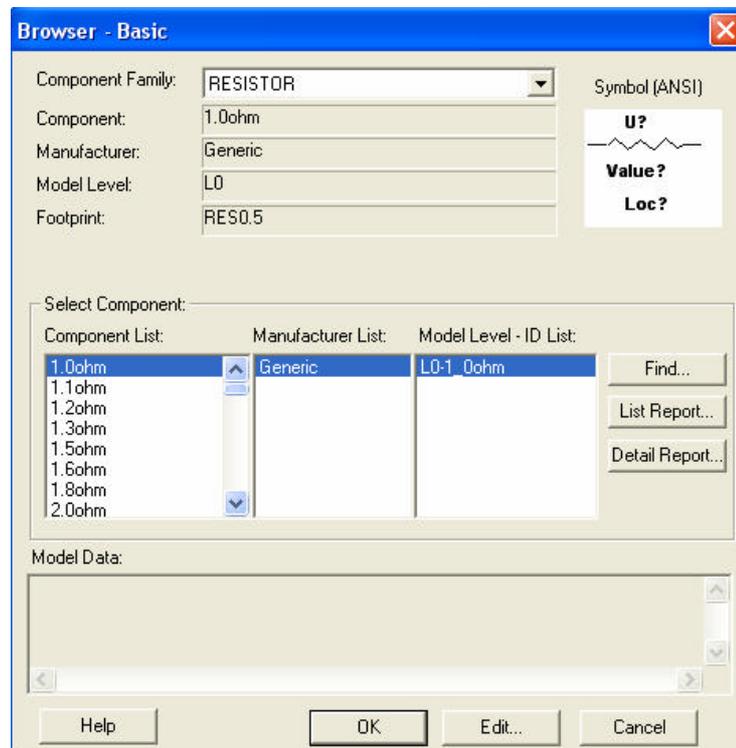


Figure 6: Setup window for Resistor values

b. Change the name of resistor:

Double click on the resistor, a new window with the name **Resistor** will open as shown in Figure 7. Select Label from the menu of this window. Change the **Reference ID** to R₁ (if it is not) and press OK. This will change the name of the resistor to R₁.



Figure 7: Battery Window for Label of Resistor

d. Add other resistor R_2

Place resistor R_2 of value $20\text{k}\Omega$ in the workspace through the same procedure.

e. Rotate the resistor:

Select resistor ' R_2 ' and press **Ctrl-R** to rotate the resistor or select **Edit→90 Clockwise** from the Main Menu. This will make the resistor vertical. Labels and values of all the components can be dragged individually. Drag the label ' R_2 ' and value ' 20kohm ' individually to put them at a proper place.

3. Place Ground:

In the Component Toolbar, select **Sources** icon. Now click on **Ground** icon in the new window as shown in Figure 8. Click in the workspace to put the Ground symbol as

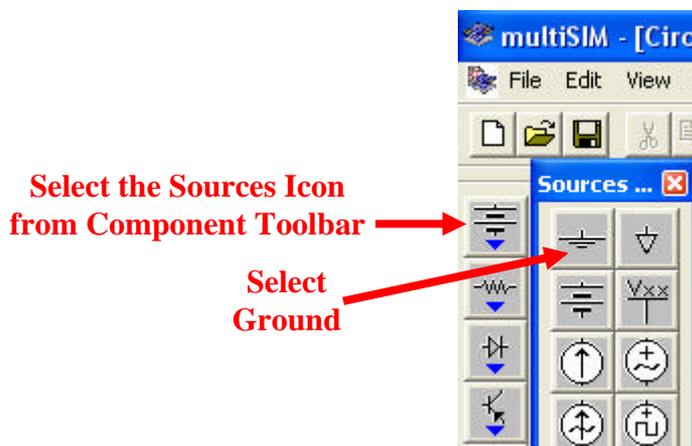


Figure 8: Selection of Ground in Multisim

STEP B: Connecting the Components

1. Arrange the components properly:

Arrange the components according to the circuit given in Figure 2. You can select and drag the component to any place in the workspace. Select the components and drag them one by one to proper places as shown in Figure 9.

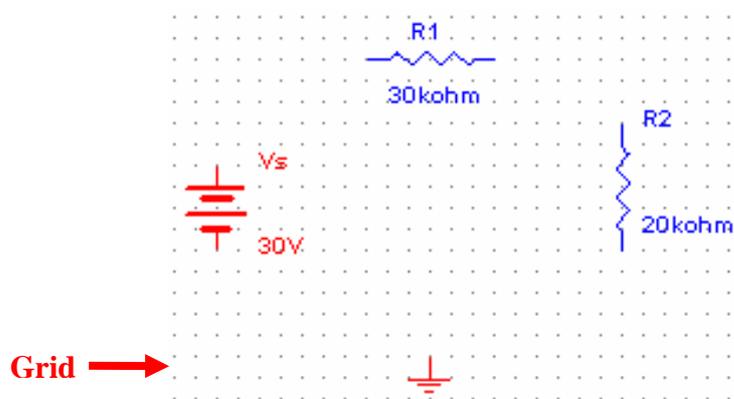


Figure 9: Arranging the Components in proper order

2. Show Grid in the workspace:

You may show grid for ease of drawing the connections. Select **View**→**Grid Visible** in the Main Menu if it is not visible.

3. Connect DC Voltage Source “Vs” to “R₁”:

Bring the cursor close to upper pin of “Vs”; cursor shape will change to a plus sign. Click and move a little upward. A wire appears, attached to the cursor. Click again at a small distance above the “Vs” source. Notice that the line will change direction. Control the flow of the wire by clicking on points as you drag. Each click fixes the wire to that point as shown in Figure 10. In this way, when the cursor reaches the pin of R₁ click again, this will connect “Vs” to “R₁” in a nice manner. Notice that a node number is automatically given.

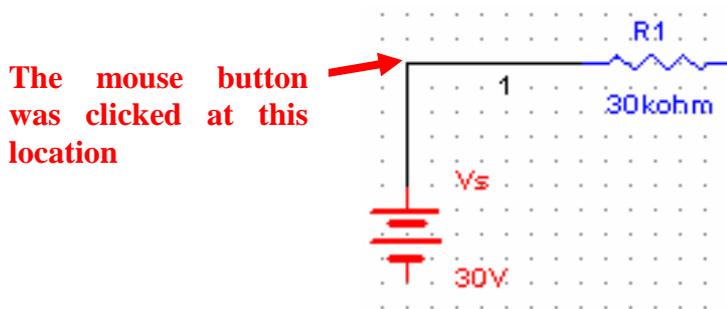


Figure 10: Manual connection of components

4. Connect “R₁” to “R₂”:

In the same way connect R₁ to R₂ through the same procedure.

5. Making use of Junction to connect Ground:

In the similar manner connect ground with V_s and R₂. Notice that a small black circle appears just above the ground, this is called junction. When 2 or more components are connected at one point, a junction is created. A junction can also be placed manually by pressing Ctrl+J or selecting **Edit**→**Place Junction**. This can be used to control the connection points manually. Also notice the ground node is automatically given node number 0. Do not alter it.

This completes the connection and the complete circuit is shown in Figure 11.

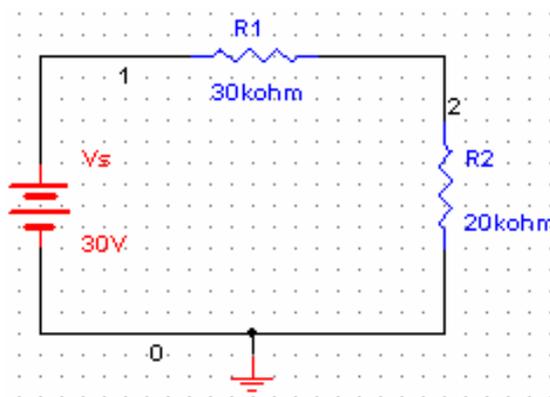


Figure 11: Complete Circuit in Multisim

6. Wire paths can be modified using drag points. Click on a wire. A number of drag points will appear on the wire as shown in Figure 12. Click any of these and drag to modify the shape. You can also add or remove drag points to give you even more control over the wire shape. To add or remove drag points, press CTRL and click on the location where you want the drag point added or removed.

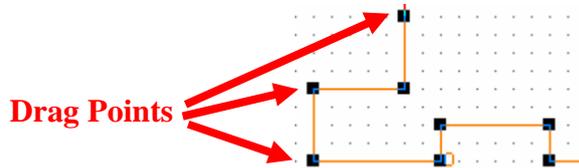


Figure 12: Drag points for connecting wire

STEP C: Placing Multimeter or Voltmeter in parallel to measure voltage.

1. To connect a Multimeter:

- a. Select **View** → **Toolbar** → **Instruments**. The Instruments toolbar will open as shown in Figure 13.

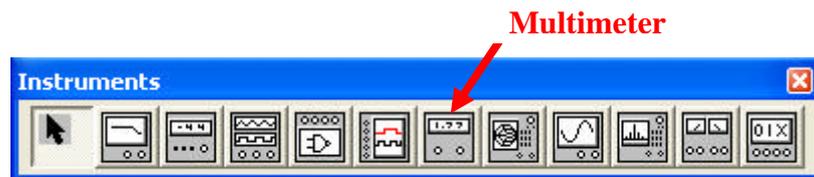


Figure 13: Instruments Toolbar

- b. Click on **Multimeter** icon. Now click in the workspace to place the Multimeter. Drag it and place it near resistor R_1 as shown in Figure 14. Make a connection from '+' terminal of Multimeter to the left pin of R_1 and from '-' terminal to right pin of R_1 . Note that reversal of + and - terminals will give opposite readings.

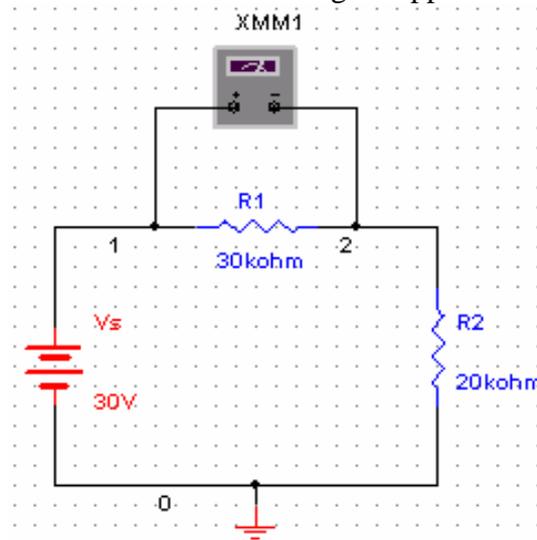


Figure 14: Multimeter connection for voltage measurement

c. Set the Multimeter to measure DC voltage:

Double click on Multimeter to open the properties window shown in Figure 15. Select 'V' to measure voltage. Select the DC wave shape. (Notice that the meter can also measure current 'A' and resistance ' Ω '. It can measure AC as well as DC values. Leave the window open for viewing the measurements.



Figure 15: Multimeter properties window

STEP D: Placing a Multimeter or Ammeter in series to measure current.

1. Place a second Multimeter in the workspace as we did in Step C. Remove the connection between R_1 and R_2 . Connect the '+' terminal of the Multimeter towards R_1 and the '-' terminal towards R_2 as shown in Figure 16.
2. **Set the Multimeter to measure current:**
Double click on this multimeter and select 'A' in the multimeter properties window. Set the wave shape to DC. If current flows from 3 to zero, the meter will read positive.

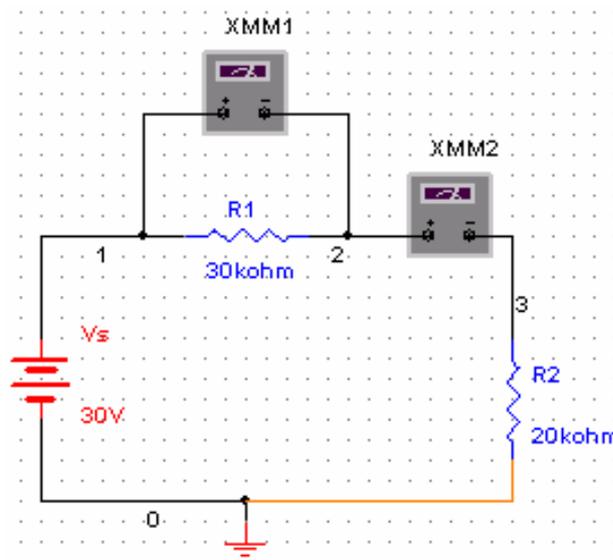


Figure 16: Multimeter connection for current measurement

STEP E: Simulate the circuit.

1. Save the file.

Select **File**→**Save**

2. Show the Simulate Switch, on the workspace.

Select **View**→**Simulate Switch**.

3. If the properties window is not open, double click the multimeters. Click to '1' position (ON) of the simulation switch to start simulation. Results will appear in the properties window of Multimeter. Compare your result with those in Figure 17 and show them to your instructor.

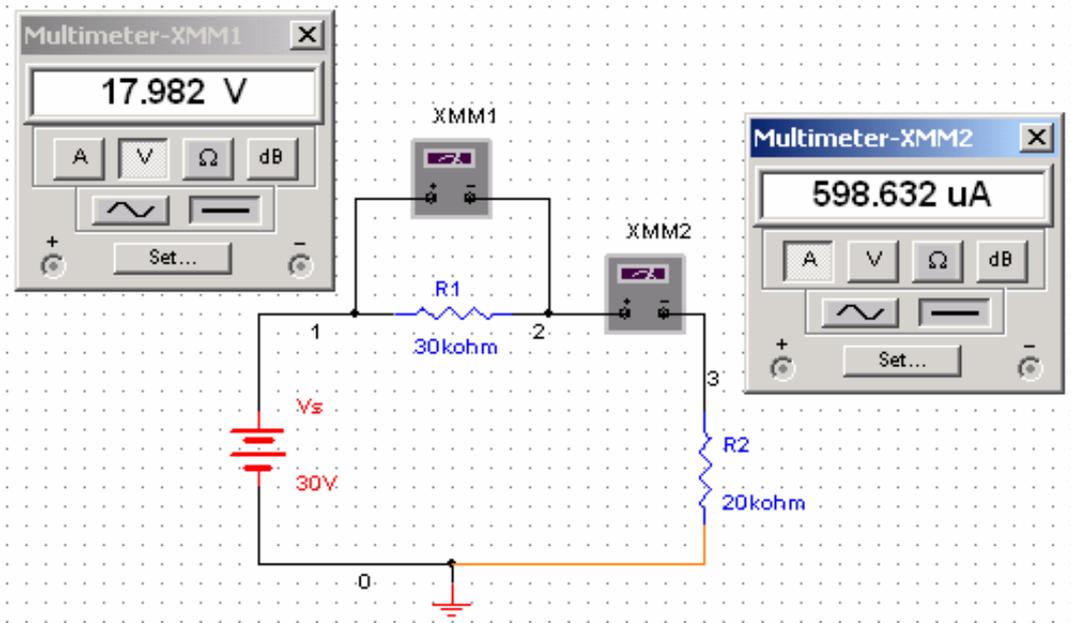


Figure 17: Simulation result

Exercise

1. Build the circuit of Figure 18 in *Multisim Electronics Workbench*.
2. Connect a voltmeter between nodes 'a' and 'b'.
3. Connect an ammeter for the measurement of I_{b0} .

