

## Introduction to Circuit Analysis Laboratory

# Lab Experiment 4

## *MultiSim Electronic Simulation Software*

---

### Part 4.1 - Introduction to multisim

Circuit simulation software allows us to predict circuit behavior by modeling and simulating an electronic circuit. It is used to find errors and make corrections to the circuit before we even build or manufacture the circuit under study.

Many circuit simulation tools are based on SPICE which is an acronym for Simulation Program with Integrated Circuit Emphasis. SPICE is a general-purpose circuit simulation program for DC, AC and transient analyses. Circuits may contain resistors, capacitors, inductors, independent voltage and current sources, as well as switches, semiconductor diodes, and BJTs, JFETs, Transistors. SPICE was originally developed at the Electrical Engineering and Computer Science Department of the University of California at Berkeley. PSpice is a free version of this program.

There are a variety simulation software packages available including PSpice, Circuit Maker and MultiSim. Today we will look at MultiSim.

Multisim is a schematic capture and simulation application that assists you in carrying out the major steps in the circuit design flow. Multisim can be used for both analog and digital circuits and also includes mixed analog/digital simulation capability, and microcontroller co-simulation. Simulating the circuits before building them, catches errors early in the design flow, saving time and money. The Multisim's user interface and its main elements can be seen in Figure 4.1

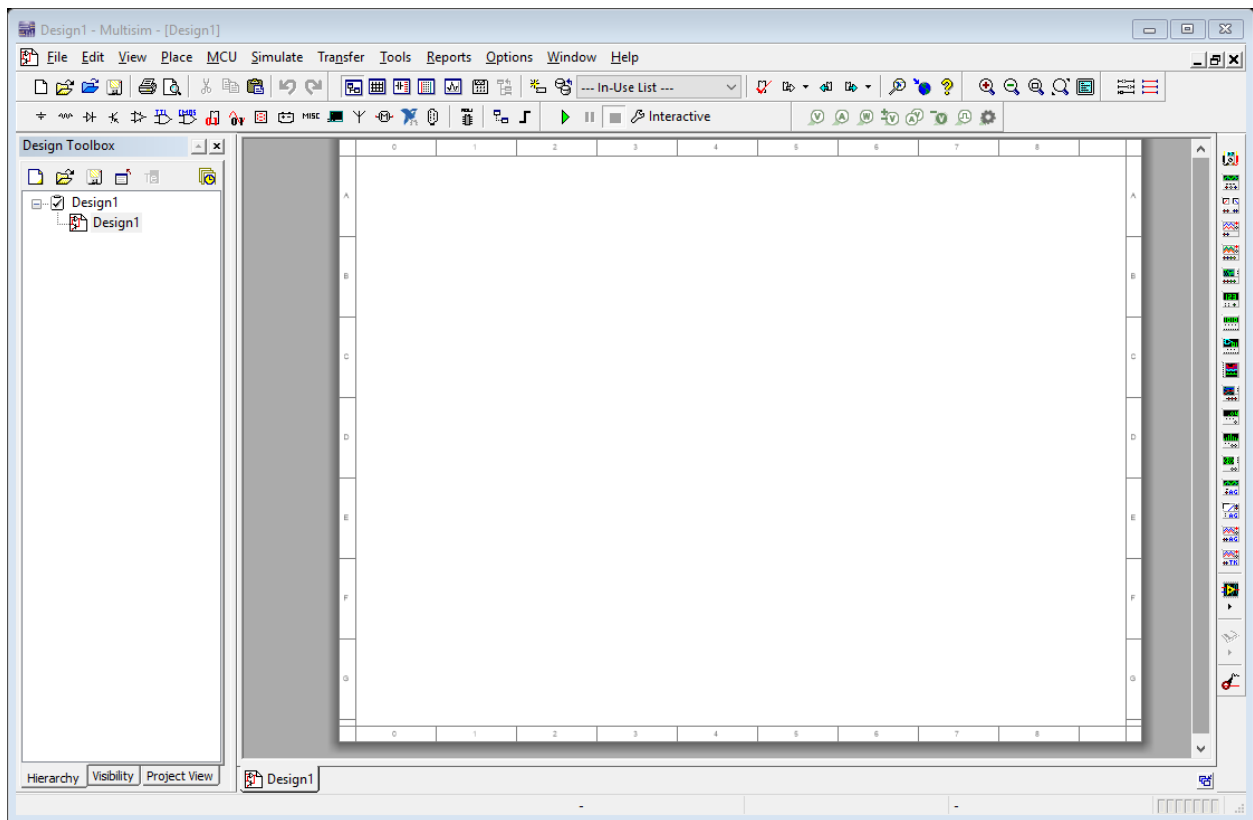


Figure 4.1 – Multisim Interface

#### 4.1.1. Multisim Interface

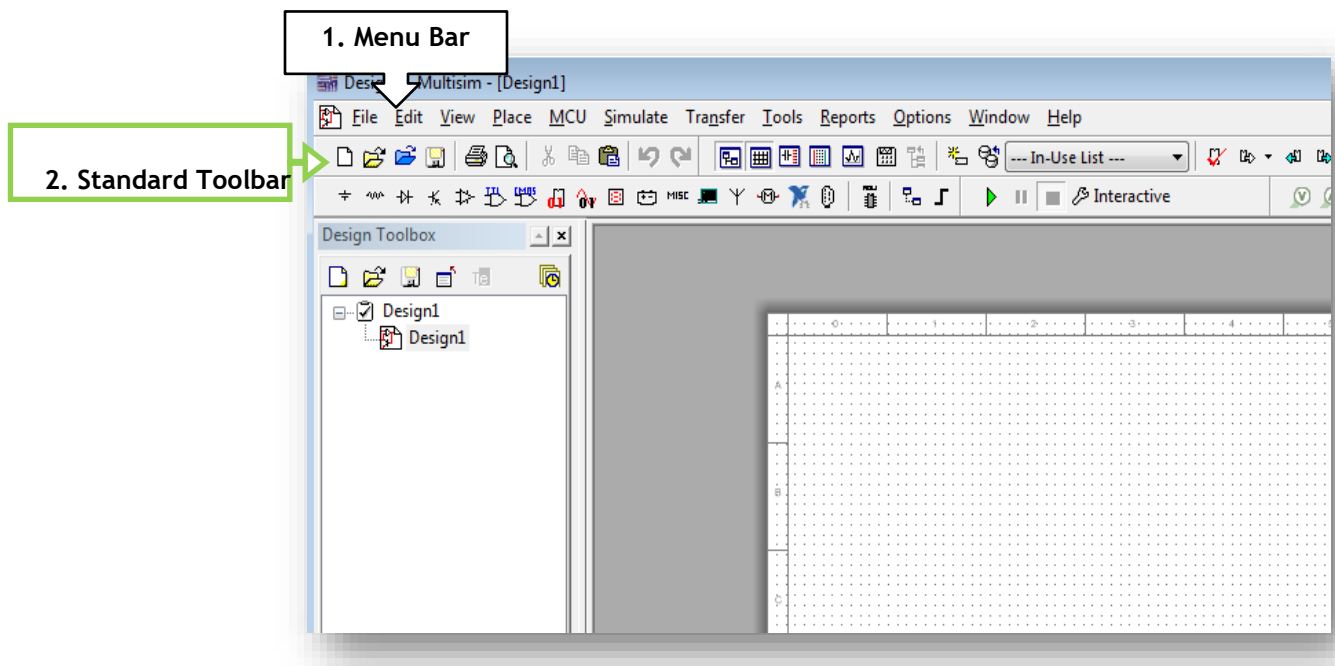
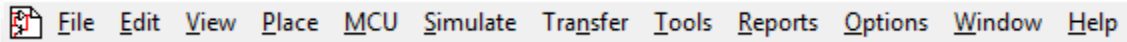


Figure 4.2 – Multisim Interface

## 1. Menu Bar

Menu bar contains the tabs or commands for all main functions: File, Edit, View, Place, MCU, Simulate, Transfer, Tools, Reports, Options, Window, and Help



## 2. Standard Toolbar

The standard toolbar contains buttons for commonly-performed functions: New, Open, Open Sample, Save, Print Circuit, Print Preview, Cut, Copy, Paste, Undo, Redo, Zoom In, Zoom Out, Zoom to Specific Area, Zoom Sheet, and Full Screen button



## 3. Component Toolbar

Component toolbar contains button that launches to the component browser of a selected Group: Source, Basic, Diode, Transistor, Analog, TTL (Transistor-Transistor-Logic), CMOS (Complementary metal-oxide-semiconductor), Mixed, Indicator, Power Component, Miscellaneous, Advance peripherals, RF, Electromechanical, Educational resources, and Connectors button



## 4. Simulation Toolbar

Simulation toolbar contains the buttons to run, pause, or stop the simulation of the circuit.



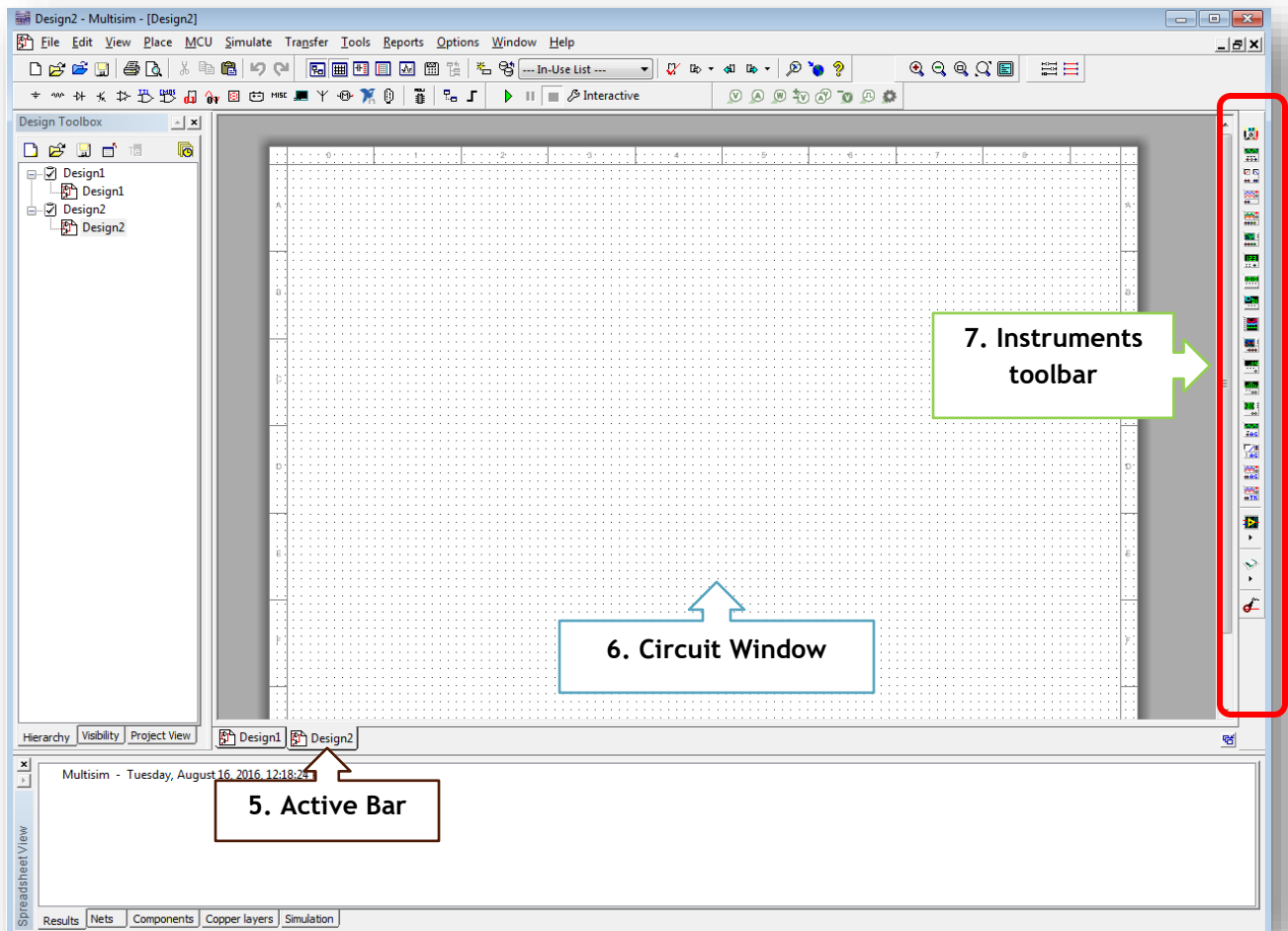


Figure 4.3 – Multisim lab equipment

### 5. Active Bar

Active bar shows the current workspace.

### 6. Circuit Window

Circuit window is the active workspace where the circuit is built.

### 7. Instruments Toolbar

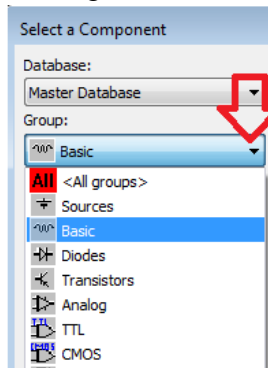
Instruments toolbar contains buttons that place a specific instrument on the workspace: Multimeter, Function generator, wattmeter, oscilloscope, four channel oscilloscope, bode platter, frequency counter, word generator, logic converter, logic analyzer, IV analyzer, distortion analyzer, spectrum analyzer, network analyzer, Agilent function generator, Agilent multimeter, Agilent oscilloscope, Tektronics oscilloscope, and LABView instructions.

### 4.1.2. Searching for components

The main components that are used for circuit analysis are located at Group **Sources** and **Basic**. There are three different ways to search for *Sources* and *Basic* components:

**Alternative 1:** Search for *Sources* and *Basic* components from Menu bar.

- In the Menu bar, select the tab **Place**
- From the Place list, select **Component...** a *Select a Component* window will appear.
- In the *Select a Component* window, you will see the *Group* selection on the left of your window.
- Click on the pointing down arrow and select the Group of components that you are looking for.



Each Group of component is organized by a Family of components, for example, when you select Group: *Sources* or *Basic*, the following Family components will show:

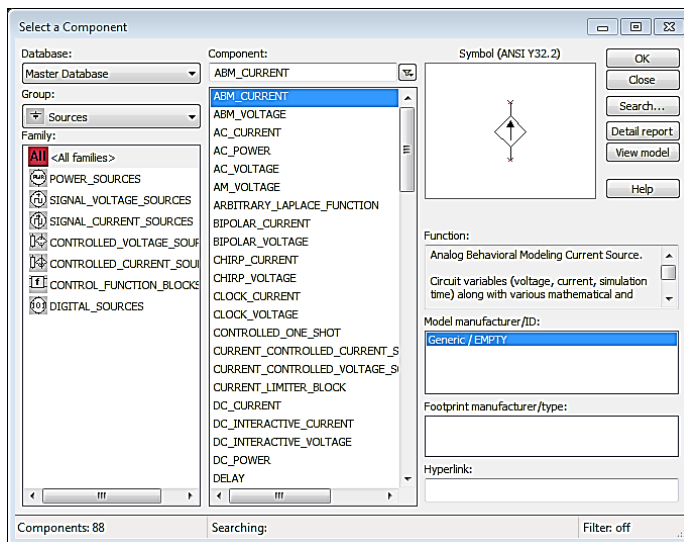


Figure 4.4 - Component list window: Group **Sources**

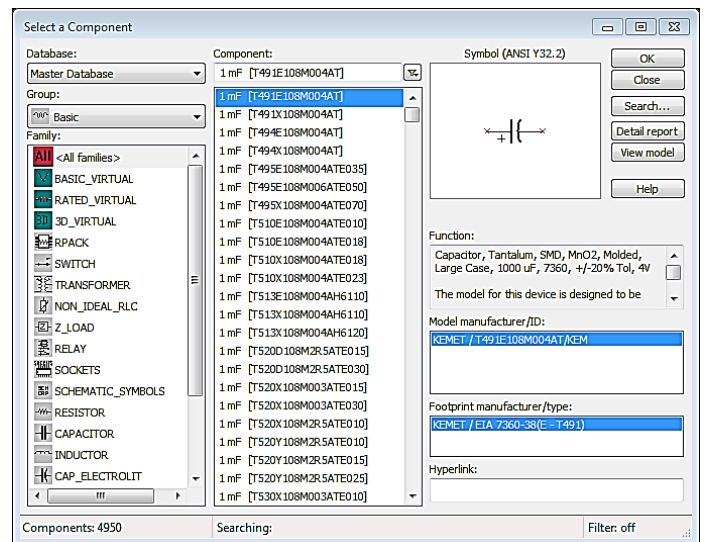


Figure 4.5 - Component list window: Group **Basic**

- Select the component that you need from the list of *Component*

**Alternative 2:** Search for *Sources* and *Basic* components from Component Bar:

- Move the cursor to the Basic or Sources icon

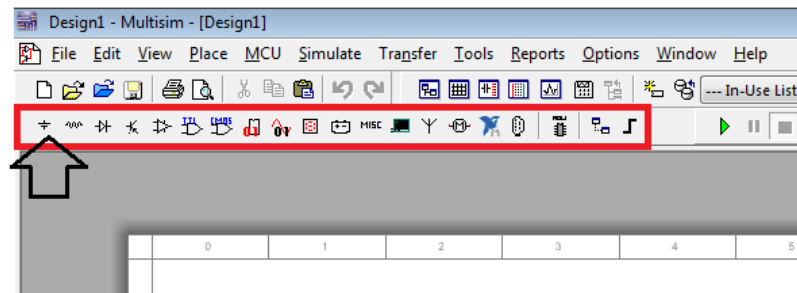


Figure 4.6 – Component Toolbar

- When you do so, the *Select a Component* window appears
- From the window, select Group: *Sources* or *Basic*

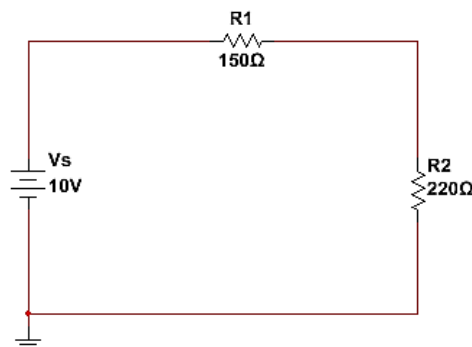
**Alternative 3:** Using combination key Ctrl + w

- From your keyboard, press the combination Ctrl + w
- When you do so, the *Select a Component* window will appear
- From the window, select Group: *Sources* or *Basic*

## Lab Experiment Procedure

### Exercise 4.1. Building a series resistive circuit

For today's lab, we will need to build a circuit as shown in Circuit 4.1 using Multisim. In Multisim, we will learn how to obtain components, make connections between components, use lab instrument, and measure the current through and voltage across an element.



Circuit 4.1 – Series resistive circuit

### *Open multisim and save the multisim file*

To save the multisim file:

- Open multisim → Click on *File* from the menu bar → select *Save As*
- Save the file as “LastName\_Lab4A.ms14”. Note: remember where the file is saved.

### *Placing components in a worksheet*

- Obtain the components from the following Group and Family, and also position them in their respective location in the workspace:
  - ✓ Ground → **Group:** Sources, **Family:** POWER\_SOURCES, **Component:** GROUND; **Location:** 3E
  - ✓ 10 V DC power source → **Group:** Sources, **Family:** POWER\_SOURCES, **Component:** DC\_POWER; **Location:** 3D.

By default, the voltage source is automatically set to 12 V. To change the value of the voltage source, double click on the voltage source to open the *DC Power* window. In the window, click on the *Value* tab and change the voltage to 10 V. The voltage source label can also be changed to Vs instead of V1. Check Figure 4.7.

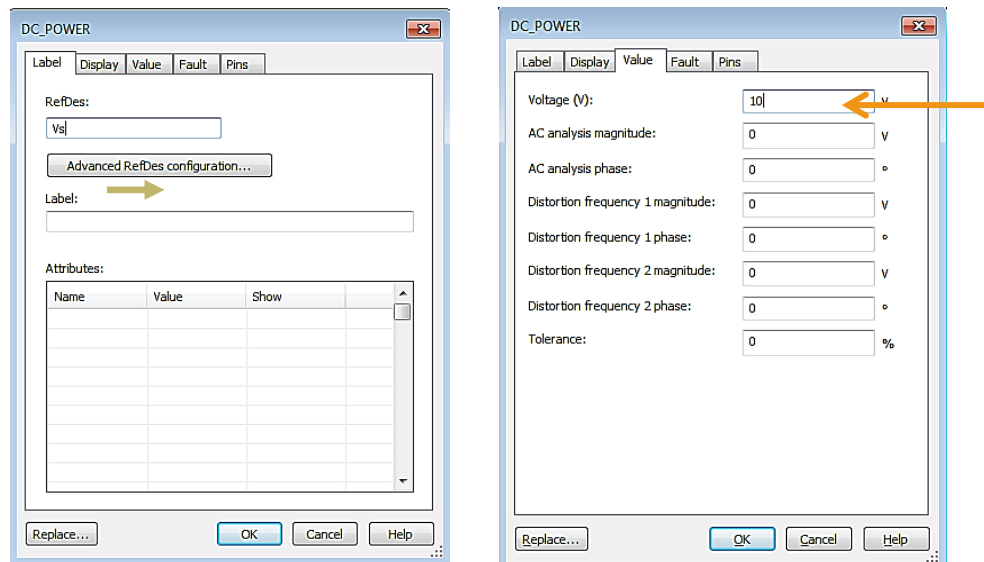


Figure 4.7 – DC Power window

- ✓ 150  $\Omega$  resistor → **Group:** Basic, **Family:** RESISTOR, **Component:** 150; **Location:** between 4C and 5C.
- ✓ 220  $\Omega$  resistor → **Group:** Basic, **Family:** RESISTOR, **Component:** 220; **Location:** 6D.

Note that resistors by default are position horizontally. If the resistor needs to be rotated or flipped, right click on the resistor to open the resistor's properties. In the

properties, select *Rotate 90° clockwise*. Another alternative to rotate is by using combination keys *Ctrl + R*. Check Figure 4.8

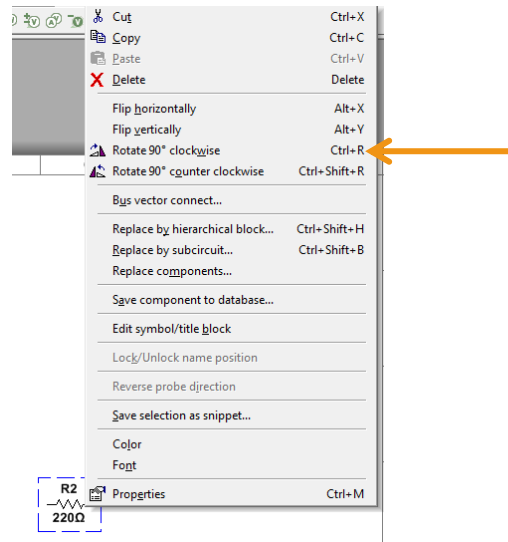


Figure 4.8 – Properties of a 220  $\Omega$  resistor

All components should be positioned in the workspace as shown in Figure 4.9.

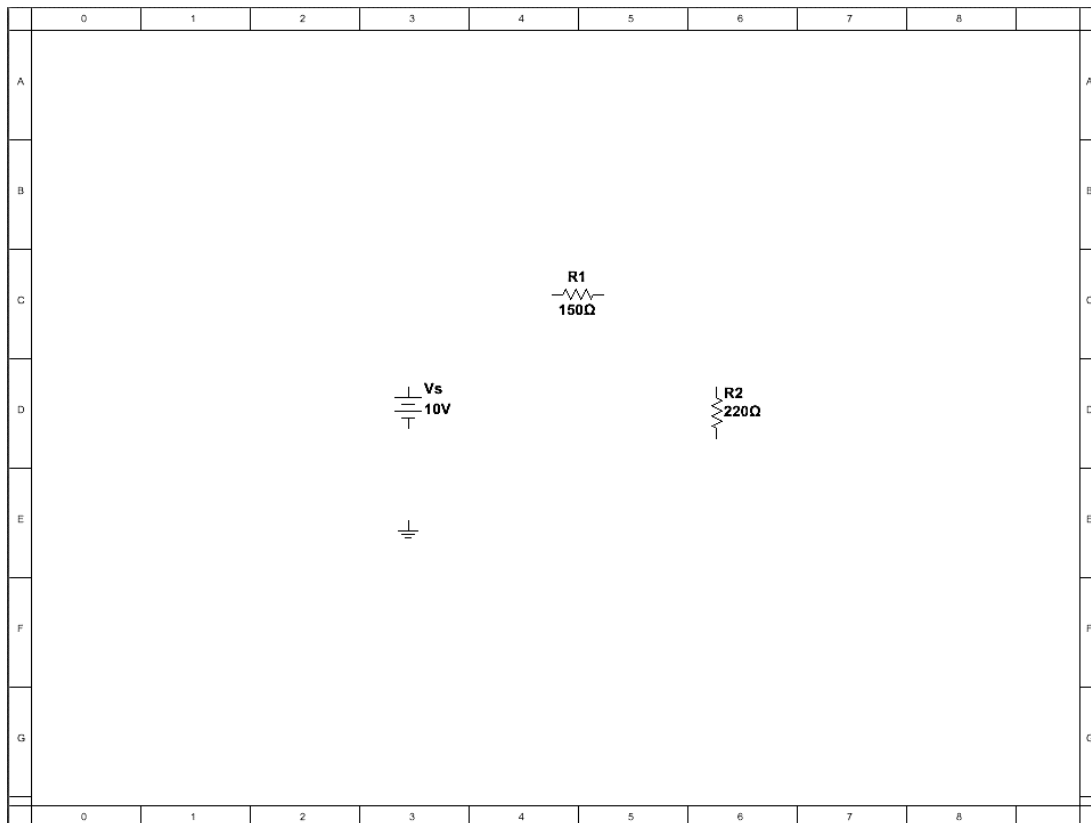


Figure 4.9 – Components position in a workspace

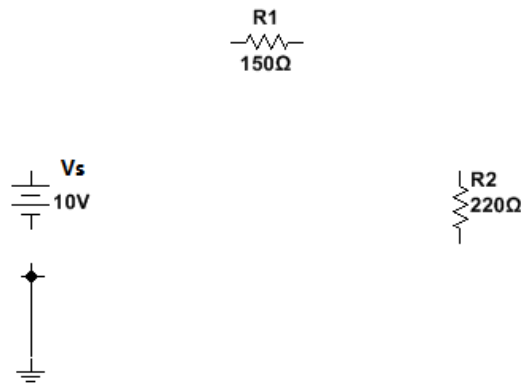


### ***Wiring the components***


All the components are placed. However they need to be connected or “wired together”.

To wire them together from the ground:

- Place the mouse cursor on the terminal of the *Ground* and click. Once it is clicked, a wire appears from the ground’s terminal. Check Figure 4.10



*Figure 4.10 – First connection of ground component.*

- Drag the wire to the negative terminal of the voltage source and click to make the connection.
- Click on the positive terminal of the voltage source, drag the wire to one terminal of  $150\Omega$  resistor, and click to make connection.
- Click the other terminal of  $150\Omega$  resistor, drag the wire to one terminal of  $220\Omega$  resistor, and click to make connection.
- Click the other terminal of  $220\Omega$  resistor, drag the wire to ground, and click to complete the circuit connection. The complete wired circuit should be as Circuit 4.1.
- Click on the Save icon  to save the work.

### **Exercises 4.2. Current Measurements in a resistive circuit**

To take current measurements in Multisim you need to “break the circuit” and add a DMM in line with the circuit along the component we intend to measure. The DMM is the uppermost item in the instrument panel. Check Figure 4.11

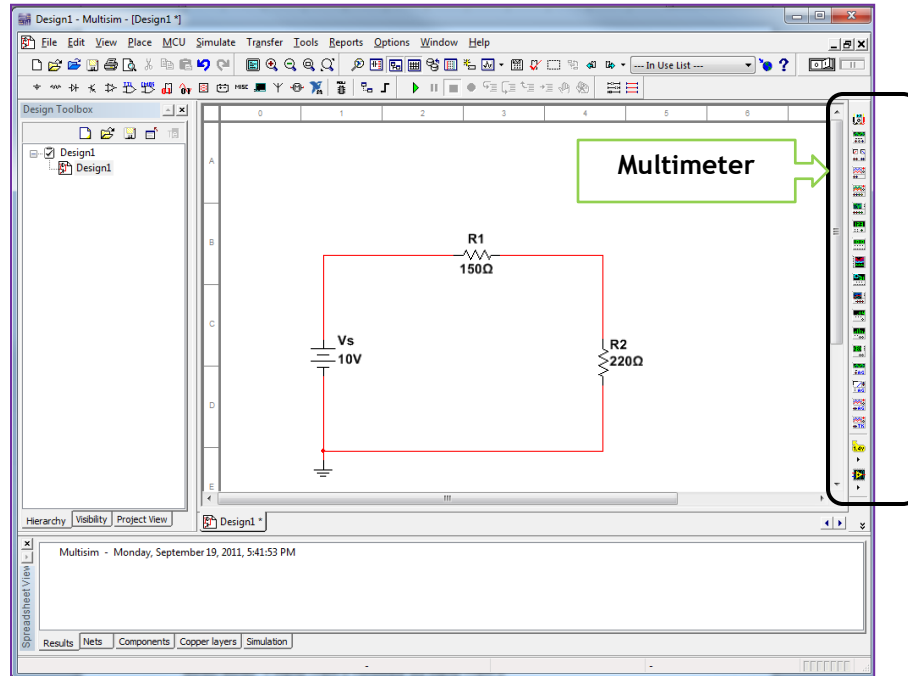


Figure 4.11 – Multimeter location in multisim

- Break the connection of one terminal of  $150\ \Omega$  resistor. Note: To break a connection, click on the wire and hit the *Delete* key.
- Obtain one multimeter from the instrument toolbar and connect the multimeter in between the break. Check Figure 4.12.

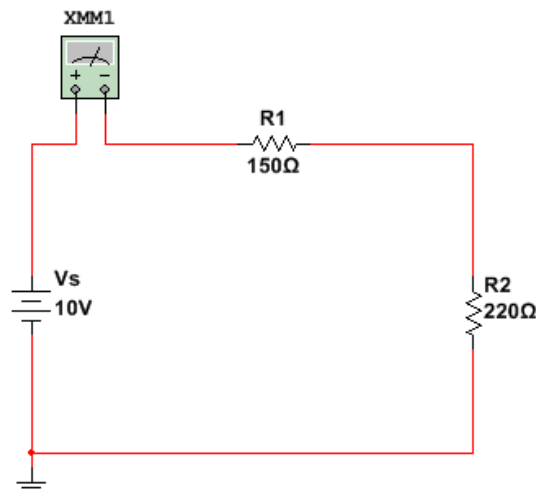


Figure 4.12 – Measuring the current through  $150\ \Omega$  resistor

- Break the connection of one terminal of  $220\ \Omega$  resistor.
- Obtain another multimeter from the instrument toolbar and connect the multimeter in between the break. Check Figure 4.13.

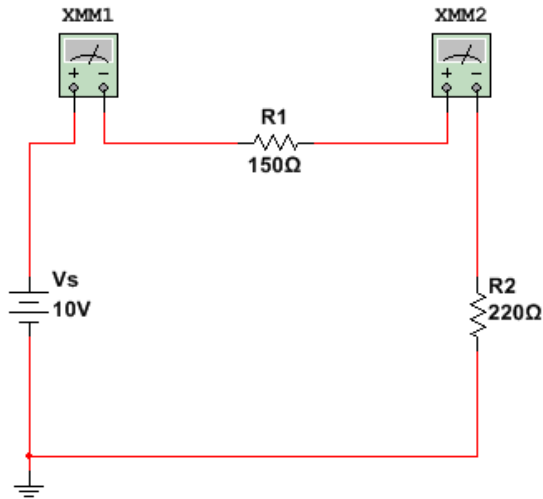



Figure 4.13 – Measuring the current through 150  $\Omega$  and 220  $\Omega$  resistors

- Run the simulation circuit by clicking the Run button  from the simulation toolbar.
- Double click on the multimeters to open the display window. Check Figure 4.14.

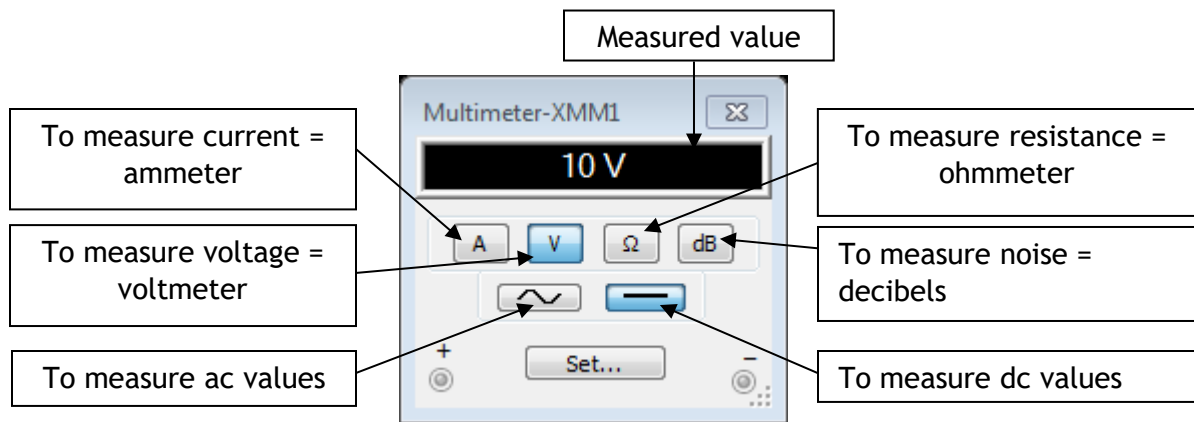




Figure 4.14 – Display window of a multimeter in multisim

- Since the circuit at Figure 4.13 is set to measure dc current, to do so, click on Ammeter  to measure the current flowing the circuit in a branch between the two nodes. Check Figure 4.15.
- Record the current through R1 and R2 in Table 4.1.
- Stop the simulation  from the simulation toolbar.
- Double click on the voltage source to change the voltage from 10 V to 16 V.

- Run the simulation again and record the current through R1 and R2 in Table 4.1.
- Stop the simulation, delete the connection of the nodes with the multimeter, and connect the components back as Circuit 4.1.

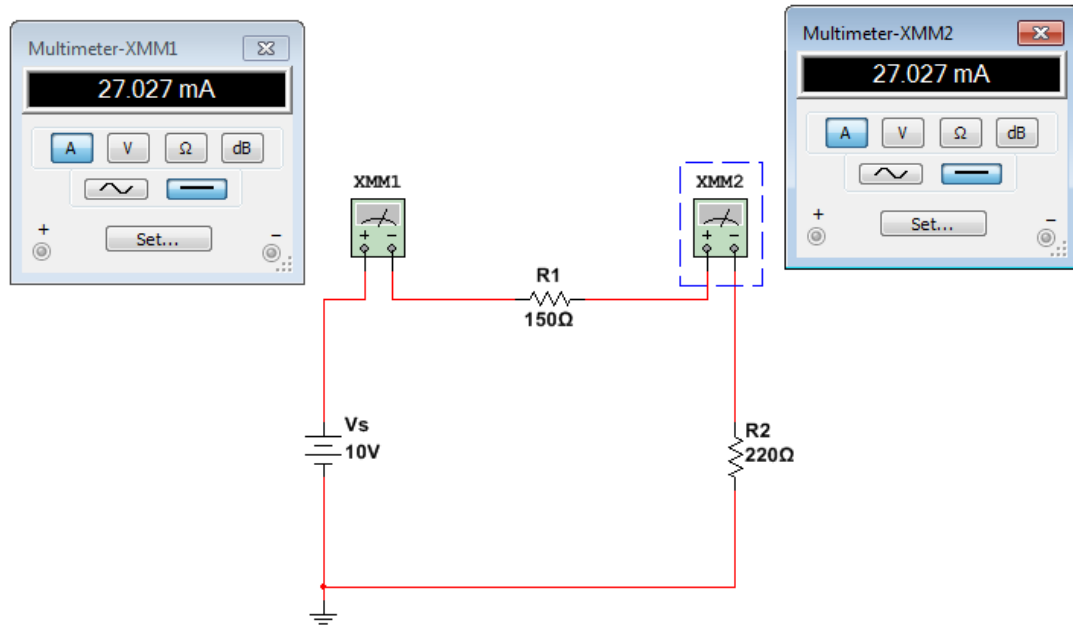


Figure 4.15 – To measure current through R1 and R2

$V_{\text{INPUT}}$	Current through R1, $I_{R1}$	Current through R2, $I_{R2}$
$V_S = 10 \text{ V}$		
$V_S = 16 \text{ V}$		
Table 4.1. Current measurements through R1 and R2		

### Exercises 4.3. Voltage measurements in a resistive circuit

To take voltage measurements, the multimeter has to be connected across the intended component to measure.

- Place the multimeter above or next to the component to be measured. Optional: rotate the 2<sup>nd</sup> multimeter to set it in parallel with R2.

- Attach the multimeter's probes in between two nodes of R1 and R2 respectively. Check Figure 4.16

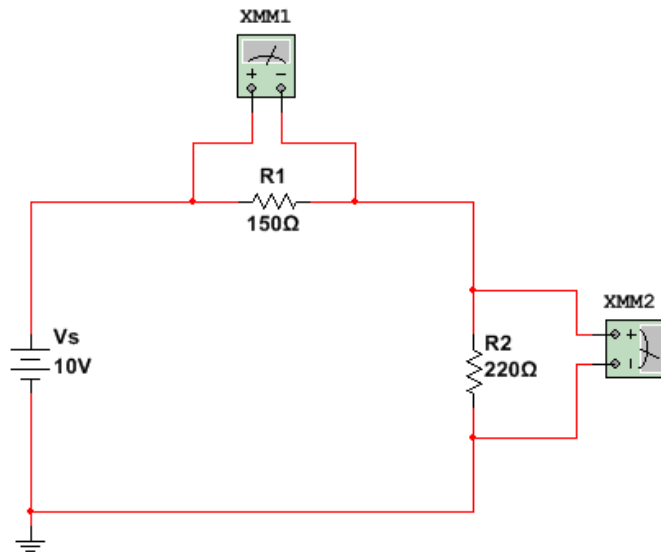



Figure 4.16 – To measure voltage across  $R1$  and  $R2$

- Run the simulation.
- Double click on the multimeters to display the measurement window.
- Since the circuit at Figure 4.16 is set to measure dc voltage, to do so, click on Voltmeter  to measure the voltage between two nodes. Check Figure 4.17.

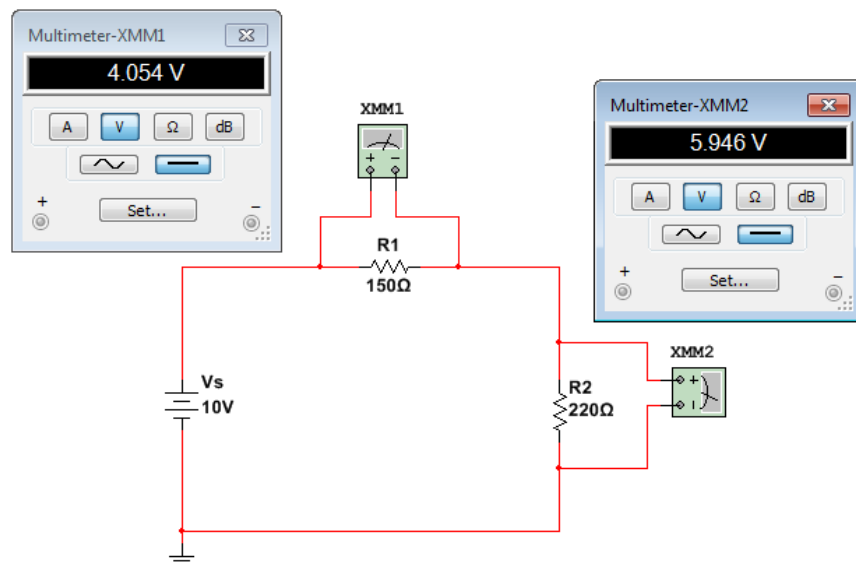


Figure 4.17 – Voltage measurement across  $R1$  and  $R2$

- Record the voltage across R1 and R2 in Table 4.2
- Stop the simulation.
- Double click on the voltage source to change the voltage from 10 V to 16 V.
- Run the simulation again and record the voltage across R1 and R2 in Table 4.2.
- Stop the simulation.

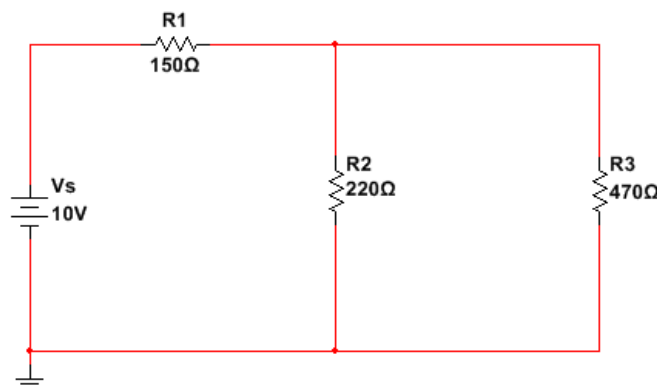
$V_{\text{INPUT}}$	Voltage across R1, $V_{R1}$	Voltage across R2, $V_{R2}$
$V_S = 10 \text{ V}$		
$V_S = 16 \text{ V}$		

*Table 4.2. Voltage measurements across R1 and R2*

#### Exercises 4.4 Building and measuring voltage and current in a resistive circuit

##### *To insert and modify a Title Block*

- Open a new workspace and save it as “Lab4\_LastName”
- Insert a Title block. The title block is located at **Place** tab → Title Block. Select the **DefaultV6.tb7** title block.
- Position the title block to the right-lower corner of the worksheet and click once to place the title block.
- Double click on the title block to open the Title Block window. The Title Block window is used to fill up information about the circuit schematic.
- Fill up the Title block with the following information:
  - Title:* Exercise 4.4 - Series-Parallel Resistive Circuit
  - Description:* Practice circuit to measure voltage and current through each resistor
  - Designed by:* Student’s name
  - Date:* Enter today’s date
- Click **Ok** to save the information in the Title Block.
- Build Circuit 4.2.



*Circuit 4.2 – Series-parallel resistive circuit*

- Obtain three multimeters and set the circuit to measure the current through each resistor.
- Run the simulation.
- Measure the current through each resistor and record measurement in Table 4.3.
- Stop the simulation.
- Change the voltage source to 16 V.
- Run the simulation and record the current through each resistor. Record measurement in Table 4.3.

$V_{\text{INPUT}}$	Current through R1, $I_{R1}$	Current through R2, $I_{R2}$	Current through R3, $I_{R3}$
$V_S = 10 \text{ V}$			
$V_S = 16 \text{ V}$			
<i>Table 4.3. Current measurements through R1, R2, and R3</i>			

- Stop the simulation, delete the connection of the nodes with the multimeter, and connect the components back as Circuit 4.2.
- Place each multimeter in parallel with each resistor. Optional: rotate the 2<sup>nd</sup> and 3<sup>rd</sup> multimeter to set it in parallel with R2 and R3 respectively.
- Run the simulation and record the voltage across R1, R2, and R3 in Table 4.4
- Stop the simulation
- Change the voltage source to 16 V.
- Run the simulation and record the voltage across each resistor. Record measurement in Table 4.4.

$V_{\text{INPUT}}$	Voltage across R1, $V_{R1}$	Voltage across R2, $V_{R2}$	Voltage across R3, $V_{R3}$
$V_S = 10 \text{ V}$			
$V_S = 16 \text{ V}$			
<i>Table 4.4. Voltage measurements across R1, R2, and R3</i>			

Save all your work in a portable memory and close multisim.

Student's Signature: \_\_\_\_\_ Lab instructor's signature \_\_\_\_\_ Date: \_\_\_\_\_

----- LAB EXPERIMENTS ENDS HERE, PROCEED WITH LAB REPORT -----