Lab 5 Circuit Simulation with Multisim

Laboratory Goals

- Introduce windows-based Multisim as a valuable design tool
- Create simple discrete circuits using Multisim
- Simulate output response for the designed circuits

Pre-lab reading

- Read the pre-lab introduction below
- Visit the National Instrument website (maker of **Multisim**)

Equipment needed

- Lab notebook, pen
- Workstation PC, with Multisim application

Parts needed

- No electronic parts are needed for this lab

Lab safety concerns

- There are no specific safety concerns for this lab

To start Multisim

Start ---> Programs ---> Electronics Workbench ---> Multisim 9 ---> Multisim 9

1. Multimism Design and Simulation for Circuit 1

(Refer to Figure 1 below)

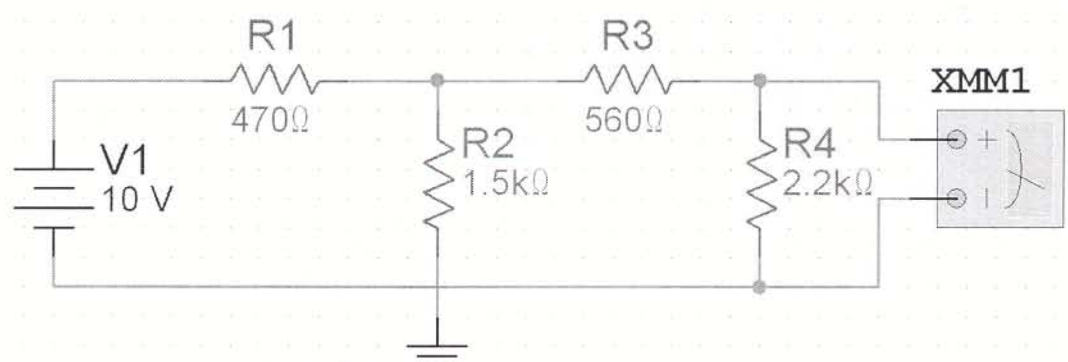
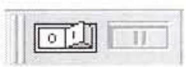


Figure 1 – Multisim Schematic of a Resistive Network

You will be following along these steps during this lab:

- ❑ Create a folder in **My Documents** for your Multisim designs to be stored. Use the last four digits of your student ID in the folder name (e.g., "206_1234")
- ❑ Open the Multisim application
- ❑ Choose the **Place** Menu→ **Component** (or use **CTRL+W**)
- ❑ Or just click right button in the circuit diagram and choose '**Place Component'**
- ❑ In the 'select a component', set 'Database' as **Master Database**, 'Group' as **Sources**, 'Family' as **POWER-SOURCE**, In **Component**, choose **DC_POWER**
- ❑ Click the **OK** button
- ❑ Move the **DC Power** to where you want it placed, then click the mouse button
- ❑ In the same database, set 'Group' as '**Basic**', set 'Family' as '**RESISTOR**', set 'filter' as '**Ω**'and '**ALL**', choose **470Ω_5%**
- ❑ Click the **OK** button
- ❑ Move the resistor **R1** to where you want it placed, then click the mouse button
- ❑ In the same database, set 'Filter' as '**kΩ**' and '**ALL**', choose **1.5kΩ_5%**
- ❑ Place resistor **R2** on the schematic with a mouse click
- ❑ Place **R2** on the sheet and rotate it 90 Clockwise (CTRL+R)
- ❑ In the 'select a component', set 'Group' as '**Basic**', set 'Family' as '**RESISTOR**', set 'filter' as '**Ω**'and '**ALL**', choose **560Ω_5%**
- ❑ Move the resistor **R3** to where you want it placed, then click the mouse button
- ❑ In the same database, set 'Filter' as '**kΩ**' and '**ALL**', choose **2.2kΩ_5%**
- ❑ Place **R4** on the sheet and rotate it 90 Clockwise (CTRL+R)
- ❑ In the 'select a component', set 'Database' as **Master Database**, 'Group' as **Sources**, 'Family' as **POWER-SOURCE**, In **Component**, choose **GROUND**
- ❑ Move the **GROUND** to where you want it placed, then click the mouse button
- ❑ Press the **CLOSE** button to stop working with the part (or right-click the mouse)
- ❑ Click  to place the Multimeter to the circuit

- Multisim is modeless; there is no need to toggle between wiring and part placement modes. As soon as your cursor is over a pin, Multisim knows you want to wire and the pointer changes to a crosshair.
- Place the mouse pointer on the right end of **R1** and click the mouse button. Drag the wire over to the left end of **R3**, then click again to connect the two parts.Finish wiring the components as shown in Figure
- Try deleting a wire by clicking on it, then clicking the **Delete** button
- Highlight any wire or component by clicking on it once
- Click and hold the mouse button on the selected part, then drag the part to where you want it. Release the mouse button
- Move any of the remaining parts, or wires until you are happy with the look of your schematic
- Click the **Increase Zoom (F8)** button to increase the size of your drawing
- Save the circuit schematic in your folder (give it a name you'll remember!)

- Click the **Simulate** button  (or press **F5)** to perform a DC analysis
- Double Click the **Multimeter Panel** to display node voltages
- To print your schematic:
  - Select the **Print** option from the **File** menu
  - Click **OK** to print from the print menu

2. Multimism Design and Simulation for Circuit 2

- Create the voltage-controlled voltage source shown in Figure 2 below.
- Note: In the 'select a component', set 'Database' as **Master Database**, 'Group' as **Sources**, 'Family' as **CONTROLLED-VOLTAGE-SOURCES**, In **Component**, choose **VOLTAGE-CONTROLLED-VOLTAGE-SOURCES**
- Double click on the **V1** label of the **VDC** icon. Label it "VBATT"
- Set **V1** to 10V
- Save the circuit schematic in your folder
- Press **F5** to simulate DC analysis

- Double Click the **Multimeter Panel** to display node voltages

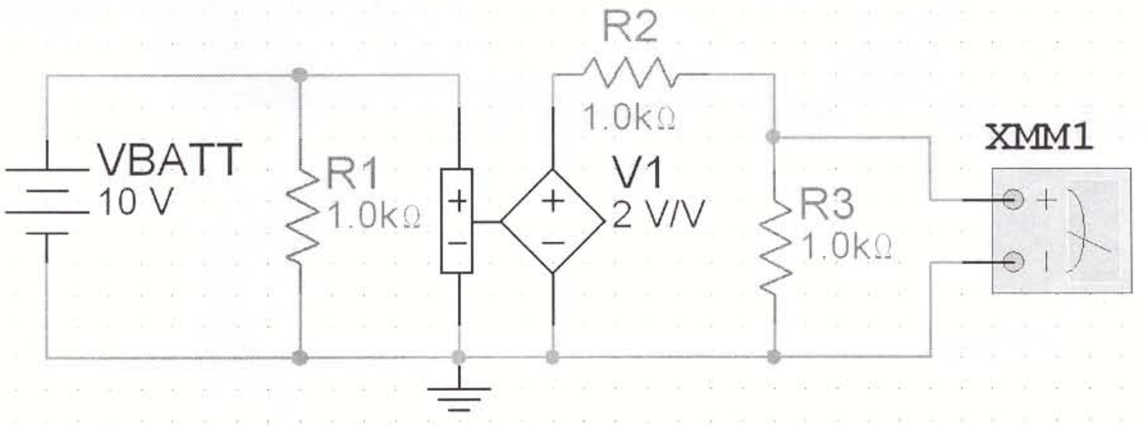


Figure 2 – Multisim Schematic of a Voltage-Controlled Voltage Source

3. Multimism Design and Simulation for Circuit 3

- Next, you will use what you have learned to create the inverting op-amp shown in Figure 3 below.
- Construction hints:
  - Click on the **New Schematic** button to start a new sheet
  - Add the components you need
  - In the 'select a component', set 'Database' as **Master Database**, 'Group' as **Analog**, 'Family' as **OPAMA**, In **Component**, choose **LM348N**
  - Click the **OK** button
  - Move the **LM348N** to where you want it placed, then click the mouse button
  - Set ground as **AGND** (Analog ground)
  - **R** (Make **R1** and **R3** = 1k, and **R2** =3**k**)
  - Add **Signal Generator**

- 4-channel scope 
- Save the circuit schematic in your folder
- Click the **Simulate** button (or press **F5)** to perform the circuit
- Change the frequency of the input signal and read the output signal from the scope.
- Select the **Print** option from the menu\
- Print the op-amp schematic

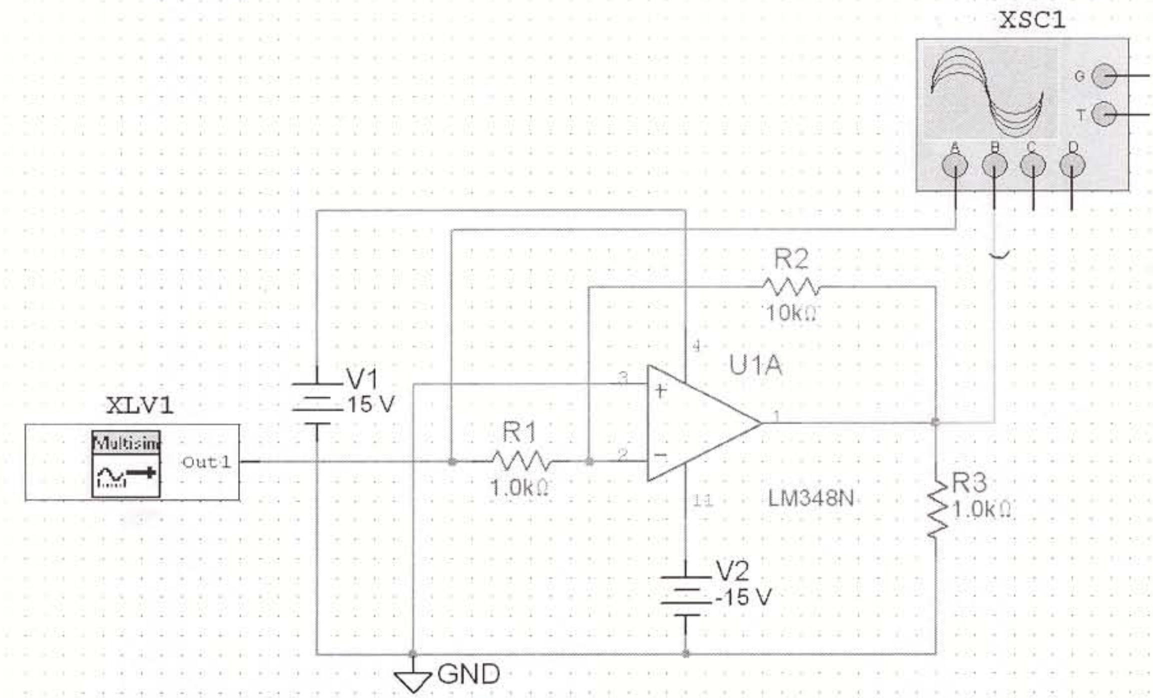


Figure 3 – Multimism Schematic of an Inverting Op Amp

4. Analysis

Write a brief summary report for this lab. Be sure to also include the following topics:

1) Compare the results from Lab 2 to the Multimism model you created in step 1 above. Record your findings in your lab notebook
2) Compare the results from Lab 4 to the Multimism model you created in step 3 above. Record your findings in your lab notebook
3) Explain any difficulties you had with this lab. (Please include suggestions to improve the lab, if you have them).

You can also do the RLC **Circuit** simulation as your after-lab task and you can get extra credits for it.