

# Introduction to Multisim:

## Learn to Capture and Simulate in Less Than 30 Minutes

### Overview

NI Multisim is an easy-to-use schematic capture and simulation environment that engineers, students, and professors can use to define and simulate circuits. This article shows you how to capture and simulate a simple circuit in Multisim.

This tutorial takes less than 30 minutes to complete and consists of 50 short steps.

The example circuit in the article is an amplifier circuit. This noninverting operational amplifier configuration consists of one active component (the operational amplifier) and two passive resistor components that will be used to complete the feedback network to provide gain in this circuit.

### Table of Contents

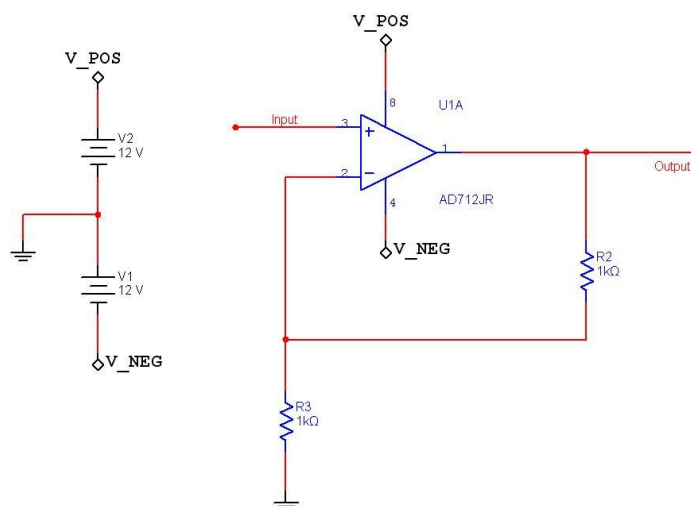
1. [Example Circuit](#)
2. [Step 1: Open Multisim](#)
3. [Step 2: Place Components](#)
4. [Step 3: Wire Components](#)
5. [Step 4: Place a Simulation Source](#)
6. [Step 5: Place Measurement Instruments](#)
7. [Step 6: Run a Simulation](#)

### Example Circuit

For this introductory example, simulate a standard noninverting operational amplifier circuit (pictured in Figure 1).

The gain of this noninverting amplifier is calculated simply with **Gain = 1 + R1/R2**.

Therefore if **R1 = R2** then the **gain is equal to 2**, which you can verify when you run an interactive simulation in NI Multisim software.



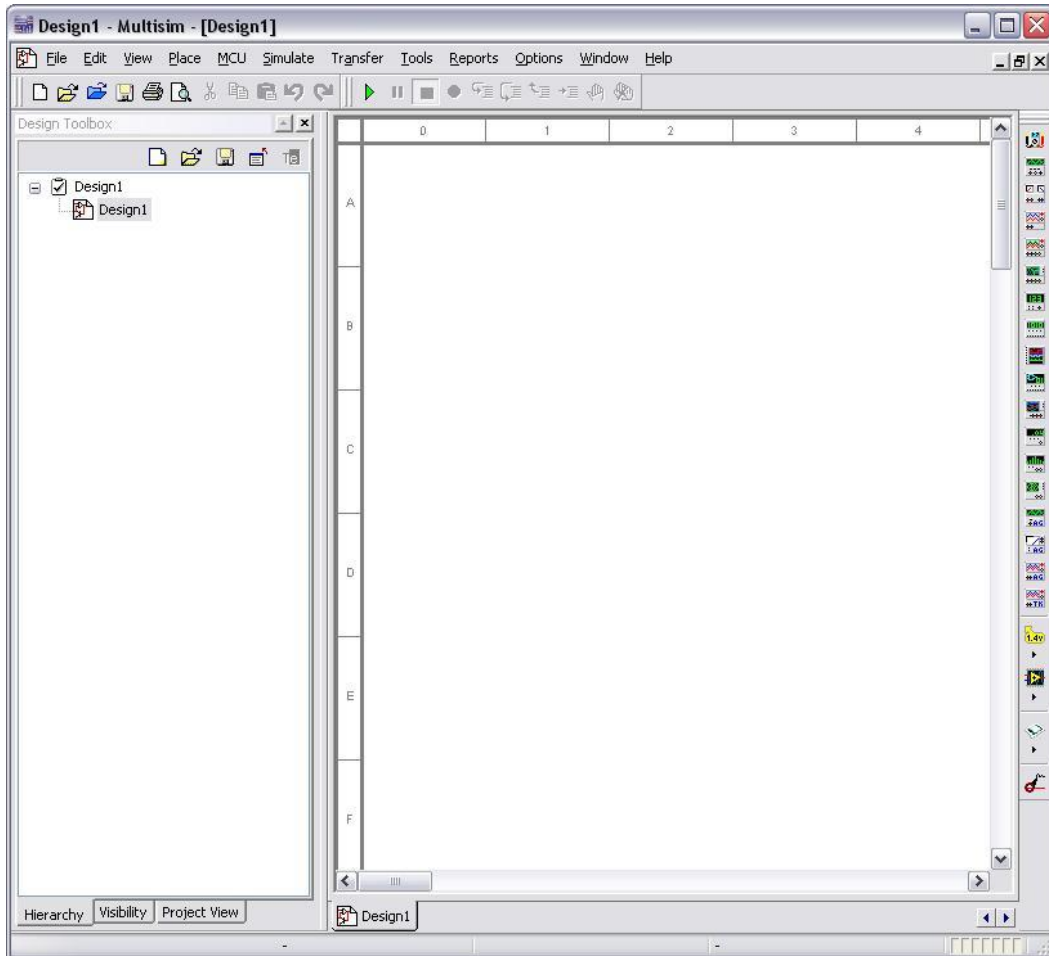
[\[+\] Enlarge Image](#)

**Figure 1. Example Circuit**

## Step 1: Open Multisim

Begin by drawing your schematic in the Multisim environment.

1. Select **Start»All Programs»National Instruments»Circuit Design Suite 11.0»Multisim 11.0** to open Multisim.
2. Multisim opens showcasing the default capture and simulation environment.

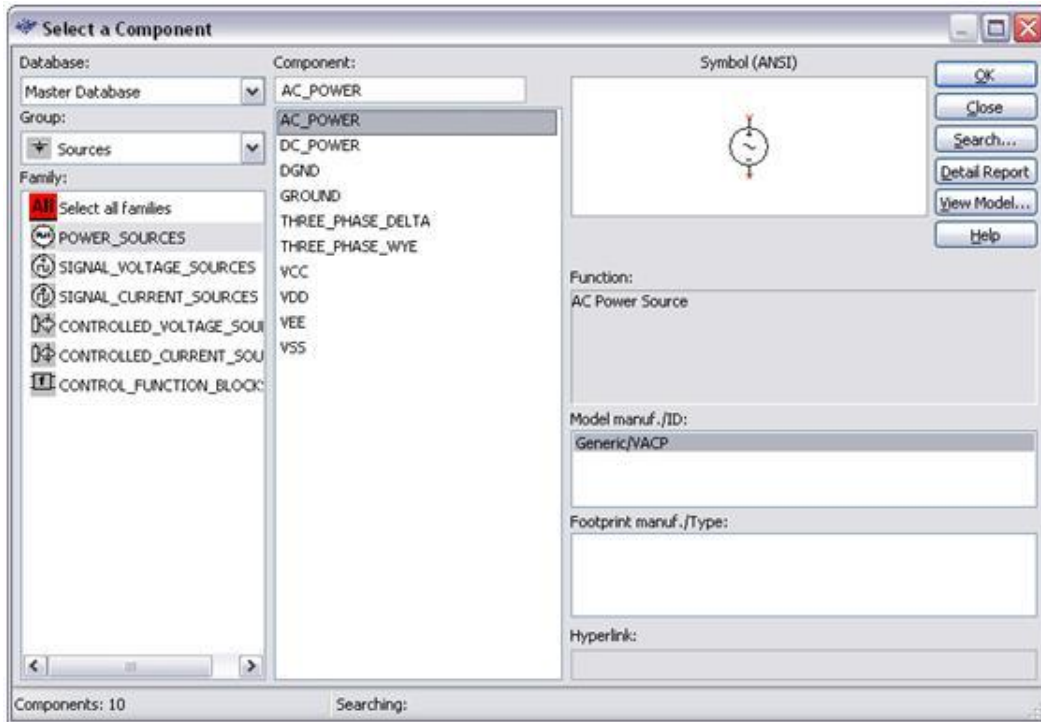


[\[+\] Enlarge Image](#)

**Figure 2. Multisim Environment**

There are two fundamental tasks in the design of a circuit: the placement of components and the wiring of these components to create a complete design.

3. To select a component, go to **Place»Component ...**
4. The **Select a Component** dialog appears (also known as the Component Browser).



[\[+\] Enlarge Image](#)

**Figure 3.** *Select a Component Dialog*

The component browser organizes the database components into three logical levels. The **Master Database** contains all shipping components in a read-only format. The **Corporate Database** is where to save custom components to be shared with colleagues (via a network collection and so on). Finally, the **User Database** is where custom components are saved that can be used only by the specific designer.

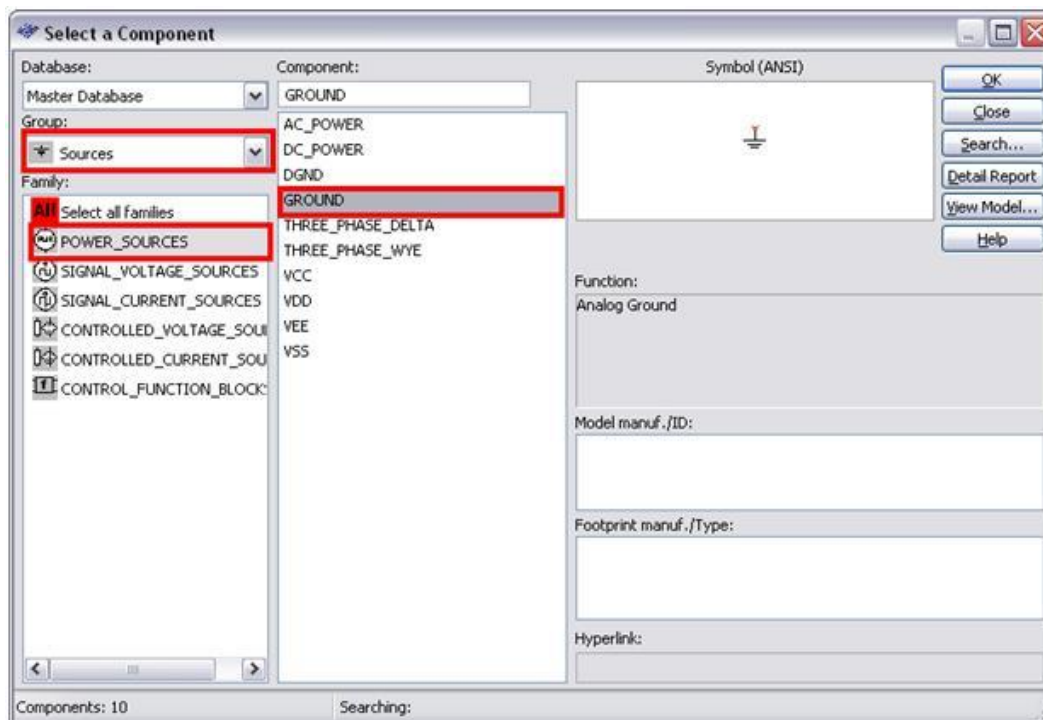
### Additional Points

- - The components (or parts) are organized into *Groups* and *Families* to intuitively and logically group common parts together and make searching easier and more effective.
  - The component selection box shows the **Component** name, **Symbol**, the functional description, **Model**, and **Footprint** all in a single popup.

## Step 2: Place Components

To invoke simulation, you need a power source and a ground somewhere in your circuit to correctly reference voltages and currents in your circuit simulation.

5. To place a Ground, go to the **Sources** group and highlight the **POWER\_SOURCES** family.
6. Highlight the GROUND component (as shown in Figure 4).
7. Click **OK**.
8. The component selection window temporarily closes and the ground symbol is "ghosted" to the mouse pointer.
9. Move the mouse to the appropriate place on the schematic and left-click once to place the component.



[\[+\] Enlarge Image](#)

**Figure 4.** *Placing a Ground Symbol*

To place a DC power supply:

10. Go to the Sources group again and highlight the POWER\_SOURCES family (if not already highlighted from the previous selection).
11. Select is called DC\_POWER symbol.
12. Place the DC Power source on the schematic.
13. Repeat steps 10, 11, and 12 to place a second **DC\_POWER** symbol.

### Additional Points

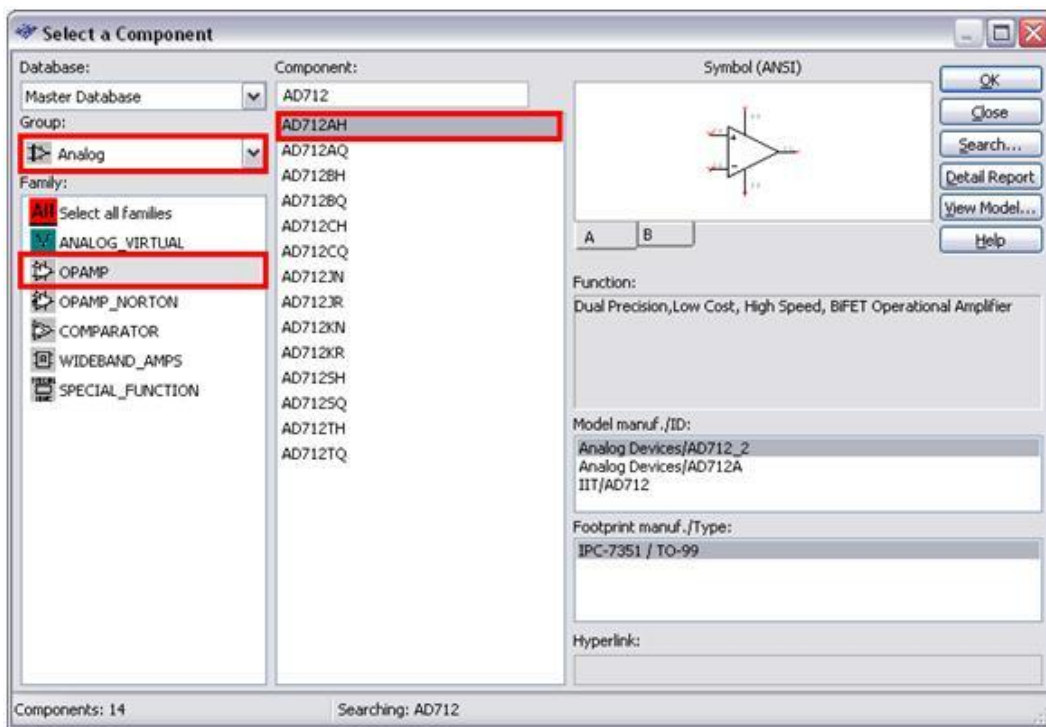
- - Without a power and ground your simulation cannot run.
  - If you need multiple components you can repeat the placement steps as shown or place one component and use copy (**Ctrl-C**) and paste (**Ctrl-V**) to place additional components as needed.
  - By default, the component selection box keeps returning as a pop-up until you have completed placing your components. Close the window to return to the schematic entry window (**Close button**). You can change this in the global preferences dialog box.

Now place the remaining circuit components using the techniques discussed in the previous steps.

14. Select **Place»Component**.
15. To place the **AD712** op-amp, select the Analog group and OPAMP family.
16. In the Component dialog box (circled in Figure 5), select type AD712 to filter the list of components in the family view.
17. Select any **AD712KR** component.

The difference in the various op-amps is due to the difference in associated land pattern (or footprint), which is an important consideration when physically implementing the design in the PCB layout environment of NI Ultiboard software.

18. Note that this component is a multisection component, as shown by the A and B tabs. This is indicative of the fact that in the single physical IC package. For this demo, it does not matter which one you select.
19. There are several SPICE models available for this component (these come from multiple vendors and highlight varying degrees of complexity when simulating the device performance).
20. Select the second model, the AD712\_2 model. This model is a more advanced model that takes into account all five terminals of the op-amp.
21. Place the component.



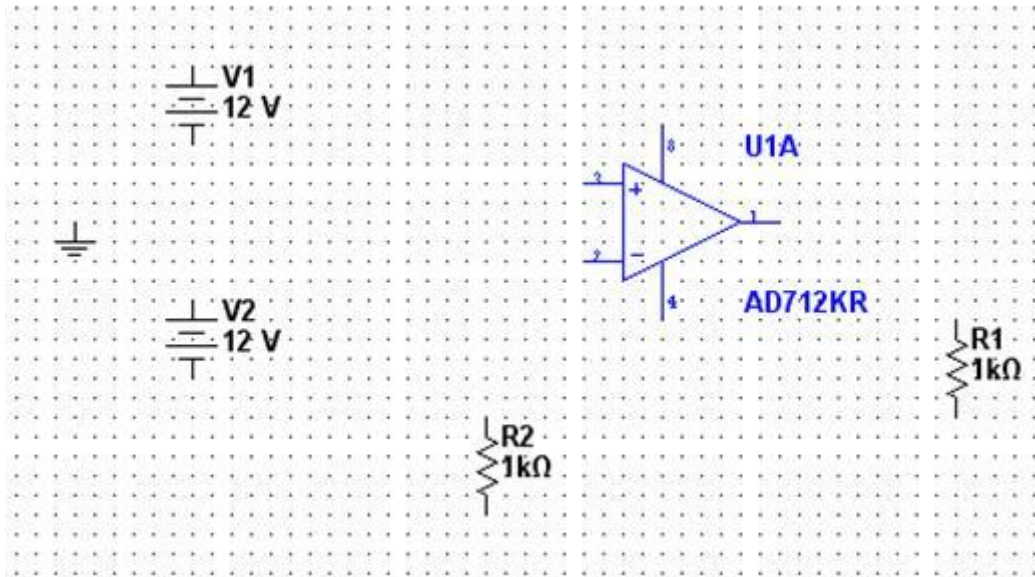
[\[+\] Enlarge Image](#)

**Figure 5.** Place an Op-Amp

Now place the resistors in the design.

22. Select the Basic group and the Resistor family.
23. In the Component field, type 1k to select a 1 k $\Omega$  resistor.
24. You can rotate a part before placement by using the <Ctrl-R> shortcut on the keyboard when the component is ghosted to the mouse. Once rotated, place the component.
25. Place another 1k resistor.

At this point, your schematic should look something like Figure 6:

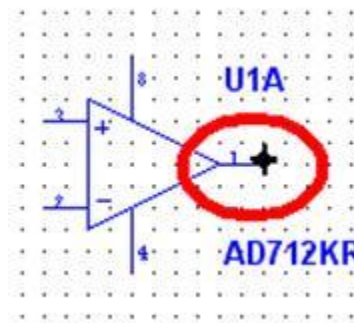


**Figure 6.** *Design So Far*

### Step 3: Wire Components

Multisim is a modeless wiring environment. This means that Multisim determines the functionality of the mouse tool by the position of the mouse. You do not have to return to the menu to select between placement, wiring, and editing tools.

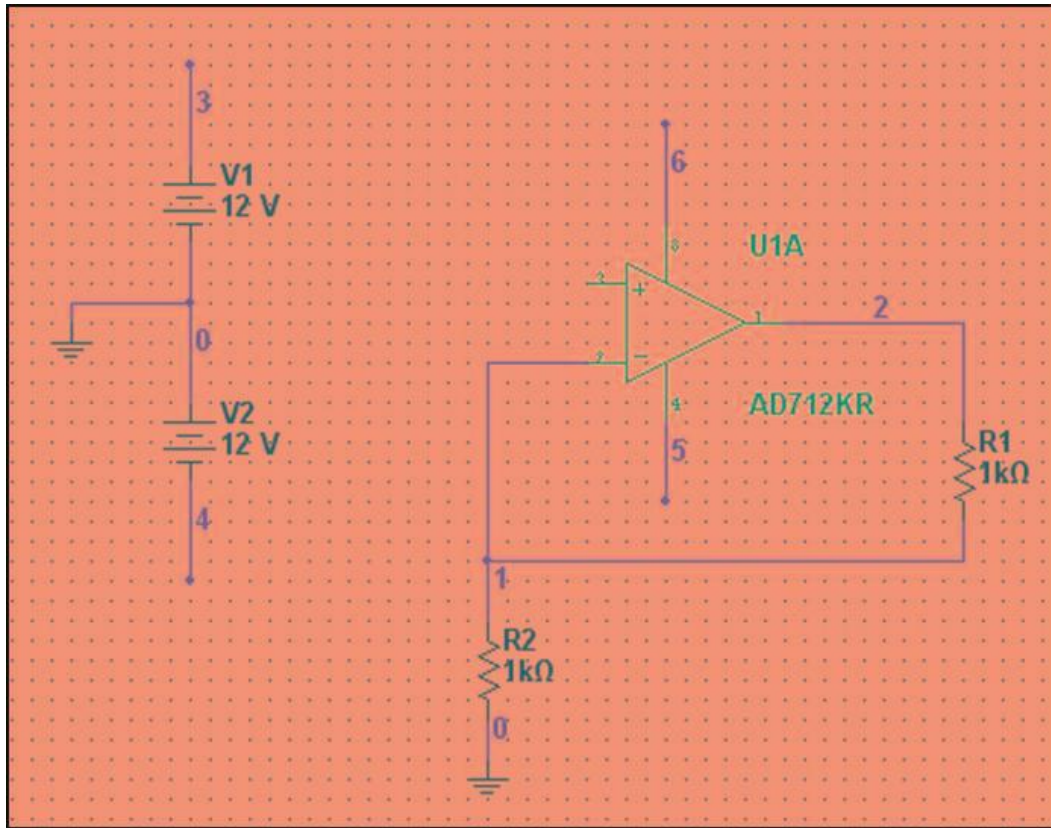
26. To begin wiring, move the mouse close to a pin of a component.
27. The mouse appears as a crosshair rather than the default Windows mouse.
28. Place an initial wire junction by clicking on the pin/terminal of the part (in this case, the output pin of the AD712KR).



**Figure 7.** *Wire an Op-Amp*

29. Complete the wire by moving the mouse to another terminal or just double-click to anchor the termination point of the wire to a floating location somewhere in the schematic window.
30. Copy the existing ground. Select the ground symbol and press <Ctrl-C> on your keyboard. To paste the ground, press <Ctrl-V> below the R2 resistor (as seen in Figure 8).
31. Complete the wiring as shown in Figure 8. Do not worry about the labeled numbers on the wires (also called nets).



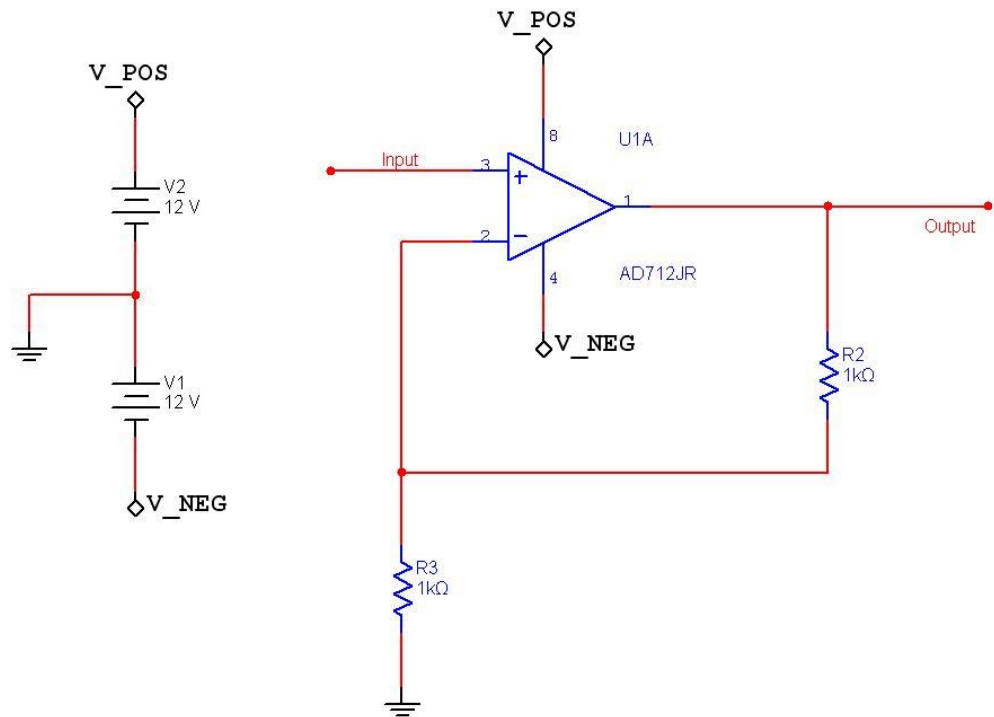


[\[+\] Enlarge Image](#)

**Figure 8.** *Wire the Components*

32. The last key step is to connect the wires (nets) for power supply terminals to the positive and negative power rails (DC Power Sources) via a virtual connection.
33. Select **Place»Connectors»On-Page Connector** and place at the first net originating from the first DC power source.
34. In the dialog box, enter the name **V\_POS**.
35. Select **Place»Connectors»On-Page Connector** and place at Pin 8 of the op-amp.
36. In the dialog box, select the **V\_POS** connector name. The virtual connection is made.
37. Repeat steps 33 to 34 for the net originating from the second DC power supply and name it **V\_NEG**.
38. Repeat steps 35 to 36 for Pin 4 on the op-amp.

The circuit looks like Figure 9.



[\[+\] Enlarge Image](#)

**Figure 9.** Complete the Schematic

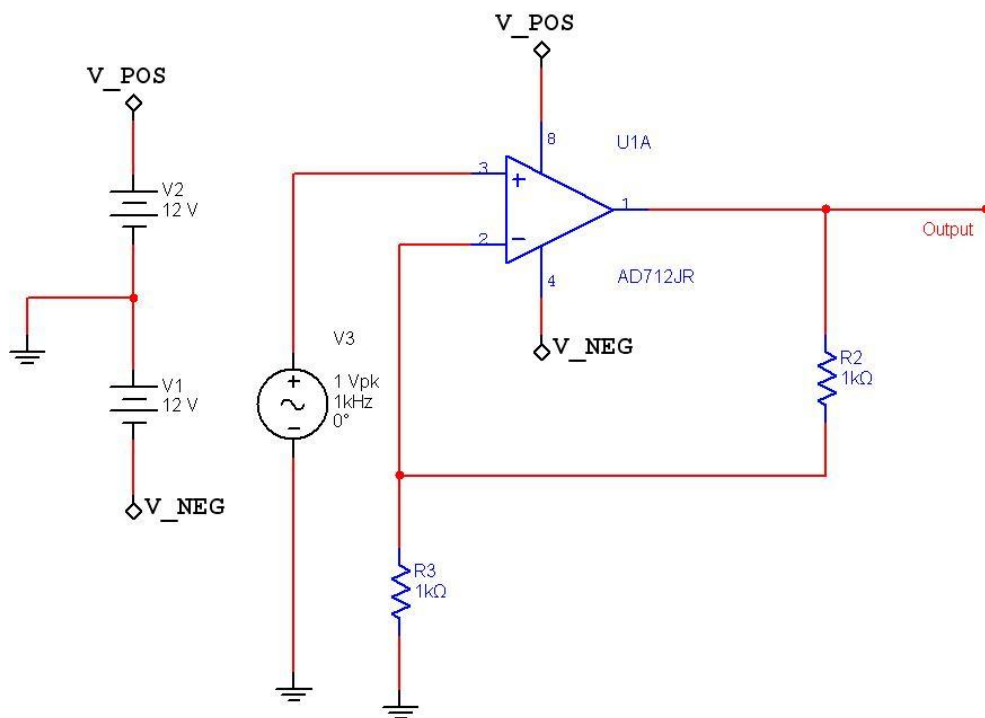
## Step 4: Place a Simulation Source

To use simulation in a Multisim schematic, you need to place and wire a signal source to the circuit to act as a stimulus.

You can choose from several signal sources including an LVM source based on LabVIEW. Place a basic AC voltage source for this demonstration.

39. To place an AC signal source, go to the **Sources** group and highlight the **SIGNAL\_VOLTAGE\_SOURCES** family.
40. Select the **AC\_VOLTAGE** component.
41. Place and wire the AC voltage source as shown in Figure 10.
42. Place and wire a Ground component to the AC source as shown in Figure 10.





[\[+\] Enlarge Image](#)

**Figure 10.** Place a Simulation Source

## Step 5: Place Measurement Instruments

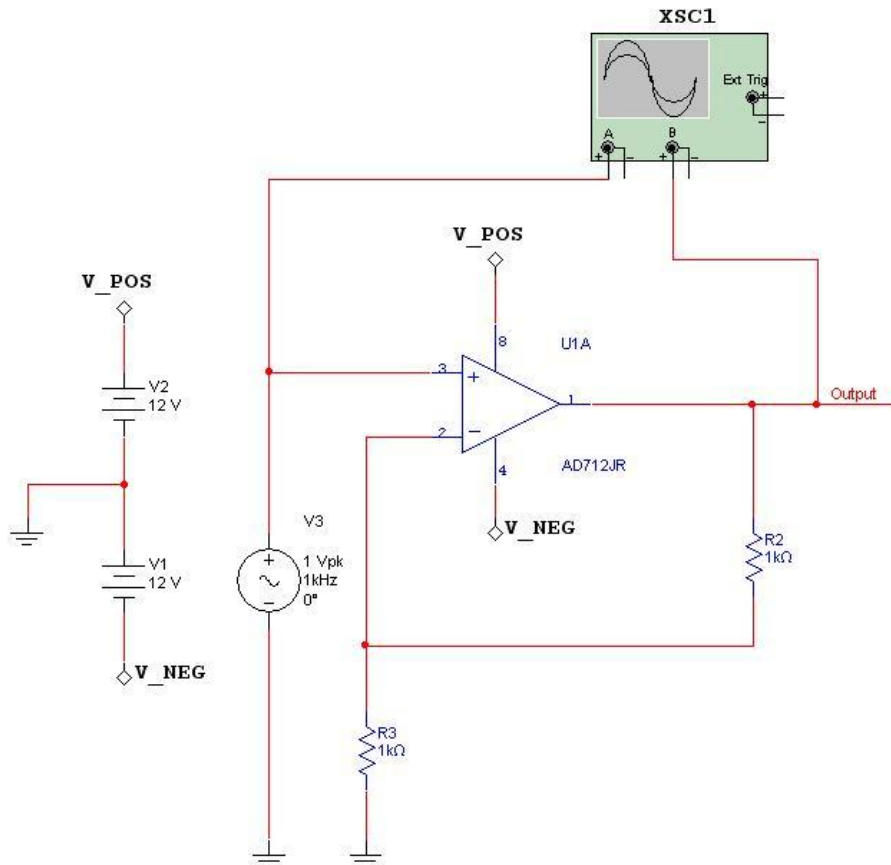
You are now ready to run an interactive Multisim simulation; however, you need a way to visualize the data. Multisim provides instruments to visualize the simulated measurements.

Instruments can be found on the **right** menu bar and are indicated by the following icons.



**Figure 11.** Instrument Toolbar

43. Select the oscilloscope instrument from the menu (fourth icon from the left) and place this onto the schematic as you would any other Multisim component.
44. Wire the Channel **A** and Channel **B** terminals of the oscilloscope to both the input and output of the amplifier circuit (as shown in Figure 12).
45. Change the color of the input channel net (connected to channel **A**) by right-clicking on the wire.
46. Select the **Color Segment** option.
47. Select a shade of **blue** and click on the **OK** button.



[\[+\] Enlarge Image](#)

**Figure 12.** Ready to Simulate

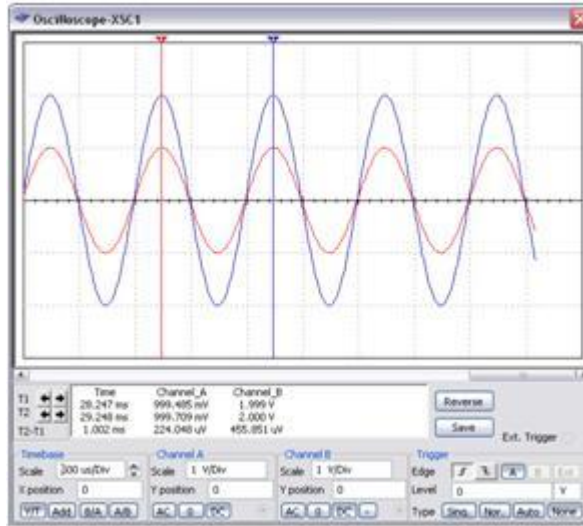
## Step 6: Run a Simulation

It is simple to begin running and visualizing an interactive simulation in Multisim.

48. Press the green run button on the simulation toolbar as seen in Figure 13.
49. Double-click on the oscilloscope (XSC1) on the schematic. You can now visualize the simulation measurements in the oscilloscope interface (Figure 14). Click on the **Reverse** button of the oscilloscope to change the background color from black to white.
50. You can now stop the simulation by pressing the red stop button in the simulation toolbar (Figure 13).



**Figure 13.** Simulation Toolbar



**Figure 14.** *Visualizing Simulation in the Oscilloscope*

You have just successfully built, simulated, and analyzed a circuit using Multisim.