Introduction to Multisim: Learn to Capture, Simulate, and Layout in Less Than 30 Minutes

Publish Date: Apr 27, 2018

Overview

NI Multisim is a powerful schematic capture and simulation environment that engineers, students, and professors can use to simulate electronic circuits and prototype Printed Circuit Boards (PCBs). This article shows you how to capture, simulate, and lay out your first design in Multisim.

The example circuit in the article is an amplifier circuit. This non-inverting 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. Introduction

For this introductory example, you will simulate a standard non-inverting operational amplifier circuit (shown in Figure 1). The gain of this non-inverting amplifier is calculated by the expression Gain = 1 + R1/R2. Therefore, if R1 = R2, then the gain is equal to 2, which you will verify when you run interactive simulation in Multisim.
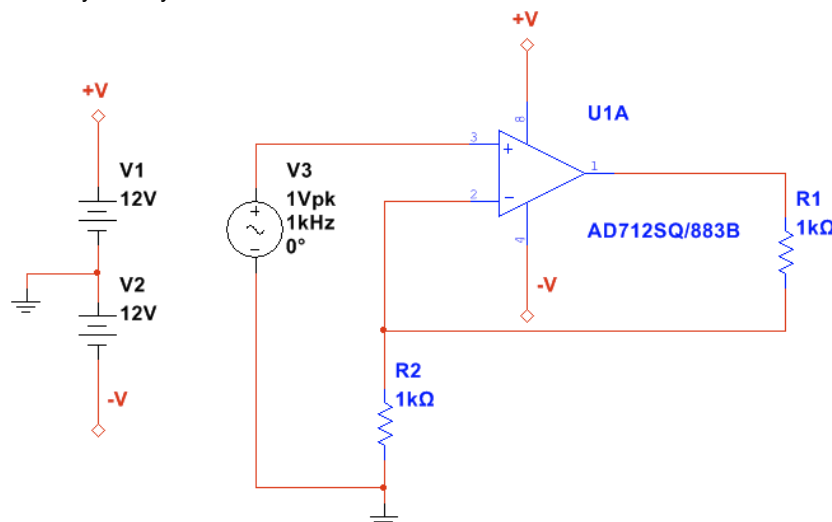

Figure 1. Non-inverting amplifier circuit.

2. Part A: Selecting Components

Begin by drawing your schematic in the Multisim environment.

1. Open Multisim by selecting **All Programs»National Instruments»Circuit Design Suite 13.0»Multisim 13.0**.
2. Select **Place»Component**. The **Select a Component** window appears (also known as the **Component Browser**), as shown in Figure 2.
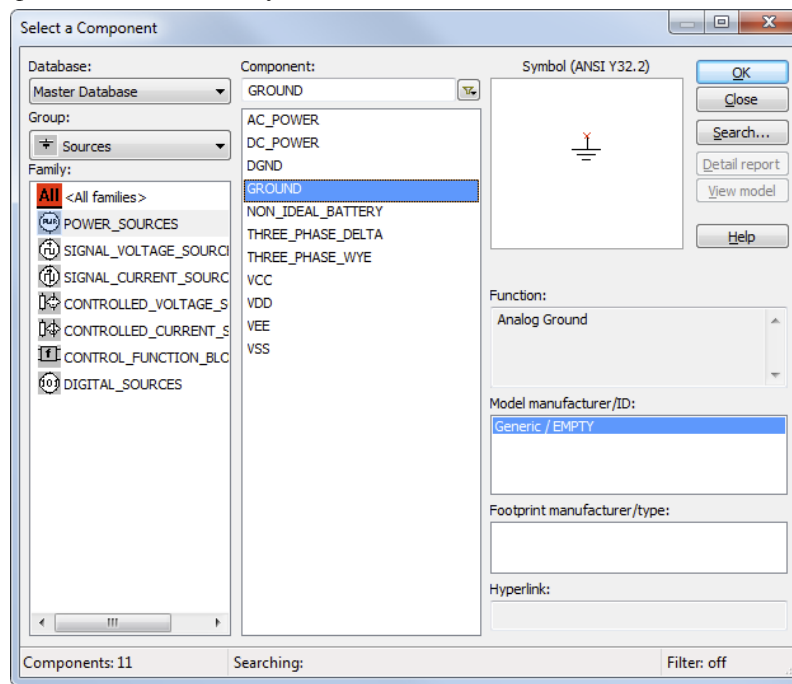
Figure 2. Select a Component window.

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 you can save custom components to be shared with colleagues. 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 Browser** shows the component name, symbol, functional description, model, and footprint all in a single pop-up.

3. Select the **Source**s Group and highlight the **POWER_SOURCES** family.

4. Select the **GROUND** component (as shown in Figure 2).

5. Click **OK**. The **Select a Component** window temporarily closes and the ground symbol is 'ghosted' to the mouse pointer.

6. Move the mouse to the appropriate place on the workspace and left-click once to place the component. After placing the component, the **Select a Component** window will open again automatically.

7. Go to the **Sources** Group again and highlight the **POWER_SOURCES** Family (if not already highlighted from the previous selection).

8. Select the **DC_POWER** component.

9. Place the **DC_POWER** component on the schematic.

10. Repeat steps 7, 8 and 9 to place a second **DC_POWER** component.

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 **Select a Component** window keeps returning as a pop-up until you have completed placing your components. Close the window to return to the schematic entry window.

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

11. Select the **Analog** Group and the **OPAMP** family.

12. Type AD712 in the **Component** field.

13. Select the **AD712SQ/883B** component, as shown in the next figure:
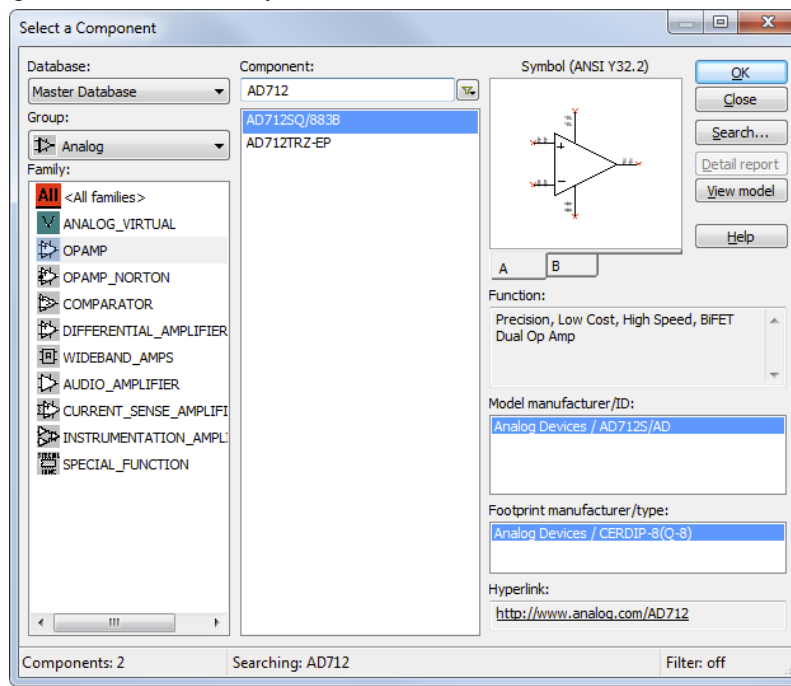
Figure 3. Selecting the operational amplifier.

Note that this component is a multisection component, as shown by the A and B tabs.

14. Place section **A** of the **AD712SQ/883B** component on the workspace area.

15. Return to the **Select a Component** window.

16. Select the **Basic** Group, **Resistor** Family.

17. Select a 1 kΩ resistor. In the **Footprint manufacturer/type** field, select **IPC-2221A/2222/RES1300-700X250**.

18. Place the resistor.

Note: you can rotate a component before placement by using the <Ctrl+R> shortcut on your keyboard when the component is ghosted to the mouse pointer.

19. Repeat steps 16, 17 and 18 to place another 1 kΩ resistor.

20. Select the **Sources** Group, **SIGNAL_VOLTAGE_SOURCES** Family, and place the **AC_VOLTAGE** component. At this point, your schematic should look something like the following figure:
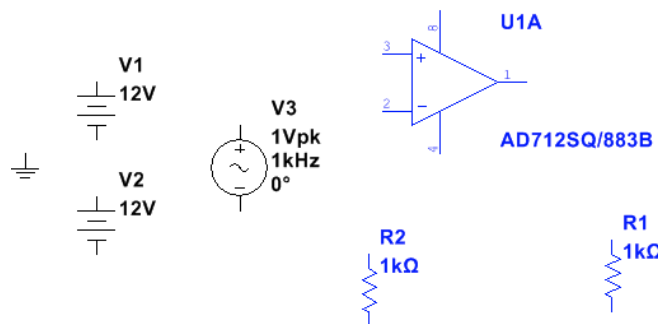

Figure 4. Components placed on the workspace area.

3. Part B: Wiring the Schematic

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

1. Begin wiring by moving the mouse pointer close to a pin of a component. The mouse appears as a crosshair rather than the default mouse pointer.

2. Place an initial wire junction by clicking on the pin/terminal of the part (in this case, the output pin of the opamp).

3. 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.

4. Create a copy of the ground symbol using Copy <Ctrl+C> and Paste <Ctrl+V>.

5. Complete the wiring as shown in Figure 5. Do not worry about the labeled numbers on the wires (also called nets).
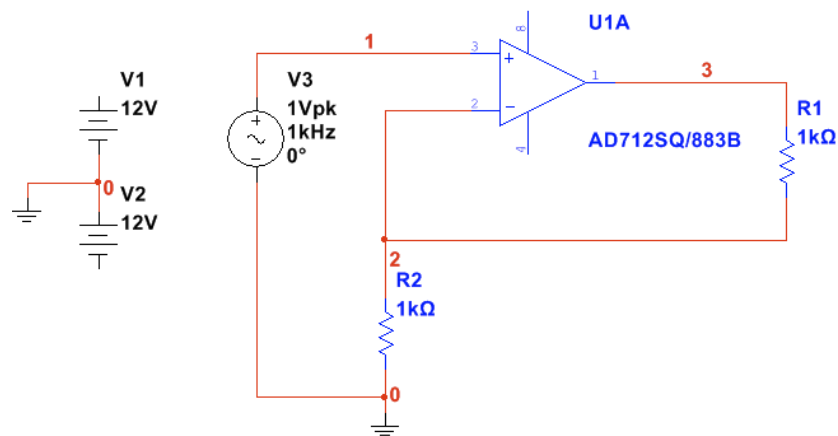
Figure 5. Wiring the schematic.

The last key step is to connect the power supply terminals to the positive and negative power rails of the opamp via a virtual connection using On-page connectors.

6. Select **Place»Connectors»On-page** connector and connect it to the positive terminal of the **V1** power supply. The **On-page Connector** window will open.

7. Enter **+V** in the **Connector name** field and click **OK**.

8. Select another On-page connector and connect it to terminal 8 of the opamp. The **On-page Connector** window will open again.

9. Select the **+V** connector in the **Available connectors** list and click **OK**. The positive terminal of the **V1** DC power supply is now connected to pin 8 of the opamp via a virtual connection.

10. Repeat steps 6 to 9 to connect the negative terminal of **V2** to pin 4 of the opamp. Name the On-page connector **–V**. The schematic should now look like the following figure:
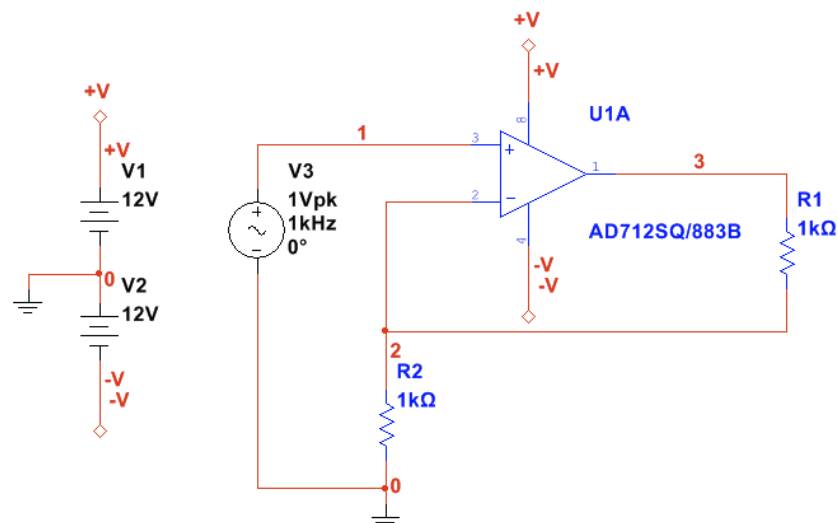

Figure 6. Schematic with On-page connectors.

4. Part C: Simulating the Circuit

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 7. Instruments toolbar.

1. Select the **Oscilloscope** from the menu and place this onto the schematic.

2. Wire the Channel A and Channel B terminals of the Oscilloscope to both the input and output of the amplifier circuit.

3. Place a ground component and connect it to the negative terminals of the Oscilloscope.

4. Right-click the wire connected to Channel B and select **Segment color**.

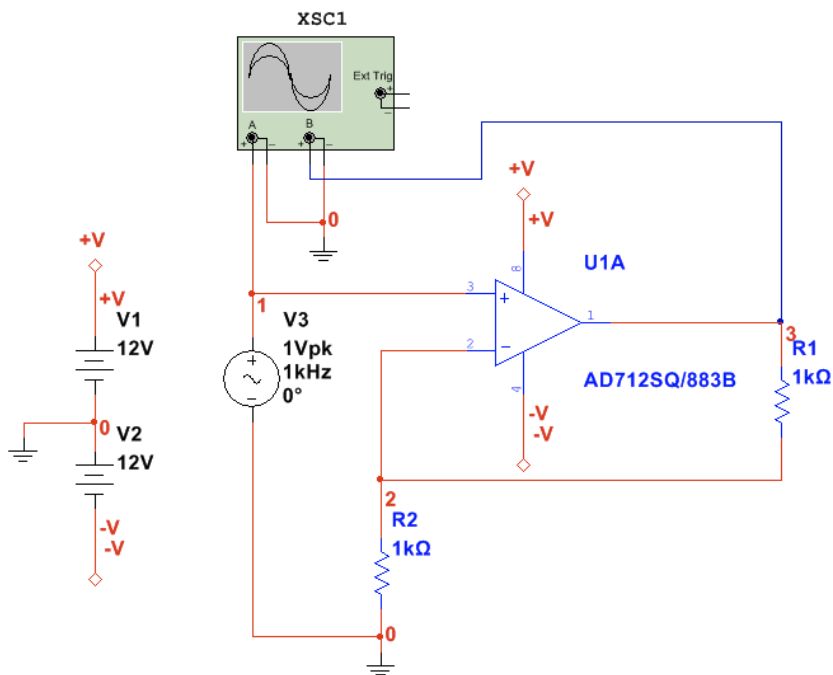5. Select a shade of blue and click the **OK** button. The schematic should look like Figure 8.

Figure 8. Connecting the Oscilloscope to the schematic.

6. Select **Simulate»Run** to start the simulation.

7. Double-click on the **Oscilloscope** to open its Front Panel and observe the simulation results (see Figure 9). As expected, the input signal is being amplified by a factor of 2.

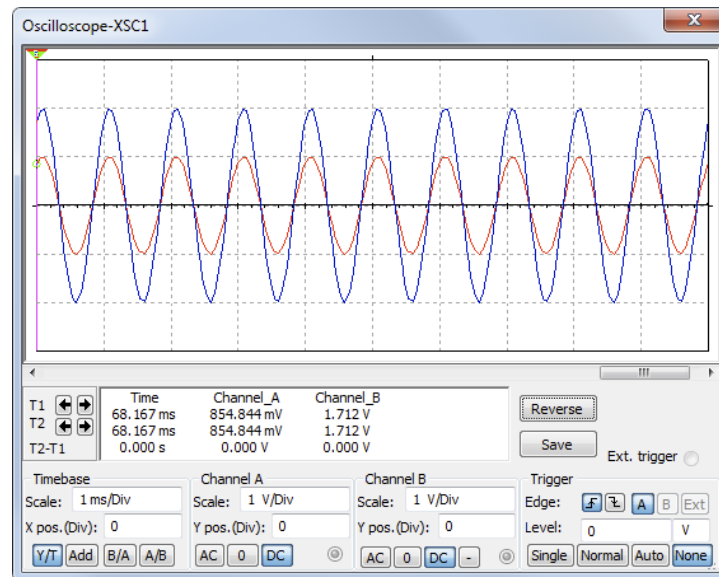8. Stop the simulation by pressing the red stop button in the simulation toolbar.

Figure 9. Simulation results.

5. Part D: Transferring to PCB Layout

We are now ready to transfer the Multisim design to Ultiboard for PCB layout. In preparation for this we need to take into consideration that sources (power, signal) and ground are virtual components and, therefore, they cannot be transferred to Ultiboard. Also, all components must include footprint information. It is a good practice to replace power sources and ground with connectors.

1. Remove **V1**, **V2**, **V3** and the **Oscilloscope** from the schematic. Do not remove the On-page connectors.

2. Open the **Component Browser**, and place the **282834-4** terminal block from the **Connectors** Group, **TERMINAL_BLOCKS** Family. This connector will be used to connect the power supplies (**+V**, **-V**).

3. Connect pin 1 of the connector to the **+V** On-page connector, pin 4 to the **–V** On-page connector, and pins 2 and 3 to ground, as shown in the next figure:
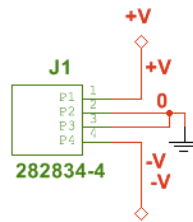
Figure 10. Connecting the terminal block.

4. Place another **282834-4** terminal block on the workspace. This connector will be used to connect the input and output signals.
5. Connect pin 1 of the connector to pin 3 (input) of the opamp.
6. Connect pin 2 of the connector to pin 1 (output) of the opamp.
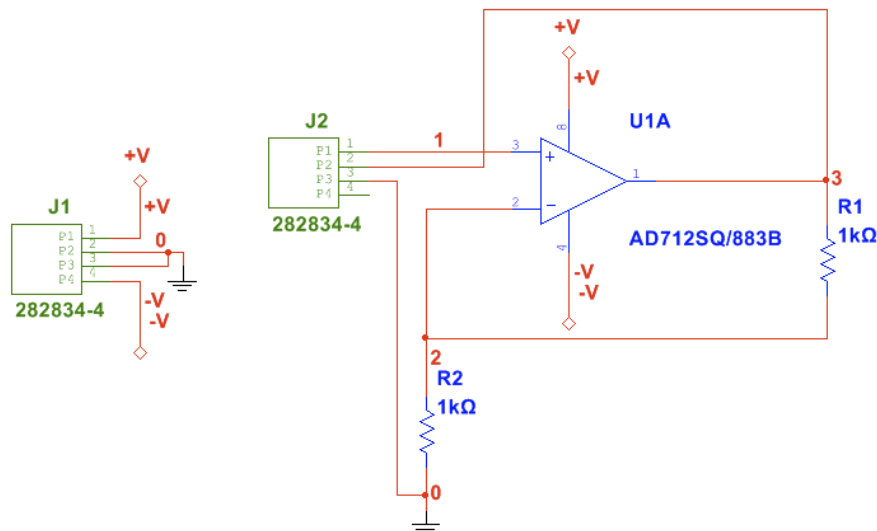7. Connect pin 3 of the connector to ground. The schematic will look like the next figure:

Figure 11. Schematic with terminal blocks.

8. Select **Transfer»Transfer to Ultiboard»Transfer to Ultiboard 13.0** and save the netlist file. Ultiboard will open automatically.
9. Click **OK** to accept all the actions listed in the **Import Netlist** window. Ultiboard will create a default board outline. Note that all the parts are placed outside of the board outline and the yellow lines (ratsnests) identifying the connections between pins, as shown in Figure 12.
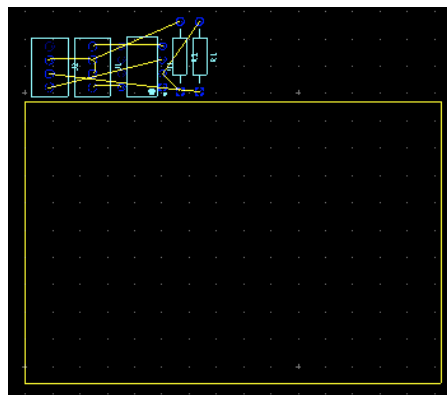

Figure 12. Default board outline and parts transferred from Multisim.

For this exercise we will use a 2x2 inch board. Follow these steps to resize the board outline.

10. Locate the **Design Toolbox** on the left side of the screen.
11. Select the **Layers** tab and double-click **Board Outline** to enable this layer, as shown below.
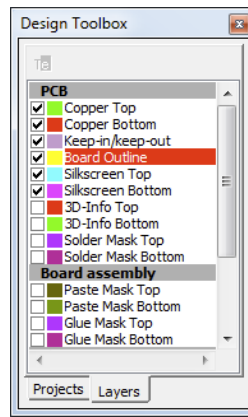
Figure 13. Design Toolbox.

The **Layers** tab of the **Design Toolbox** allows you to move between layers of your design and control the appearance of the layers.

12. Go to the toolbar area and locate the **Select** toolbar, referring to the following figure.

Figure 14. Select toolbar.

The **Select** toolbar contains the functions used to control selection filters. In other words, these filters control what can be selected by the mouse pointer.

13. Disable all the filters except **Enable selecting other objects**.

14. Double-click the board outline on the workspace area to open the **Rectangle Properties** window.

15. Select the **Rectangle** tab, change **Units** to **inch** and enter **2** for **Width** and **2** for **Height**.

16. Click **OK**.

6. Part E: Routing the Board

Place components inside the board.

1. Go to the **Select** toolbar and disable all the filters except **Enable selecting parts**.

2. Drag part **J2** and drop it inside the Board Outline. You can rotate parts by using the <Ctrl+R> shortcut.

3. Place the rest of the parts inside the Board Outline, use Figure 15 as guidance.
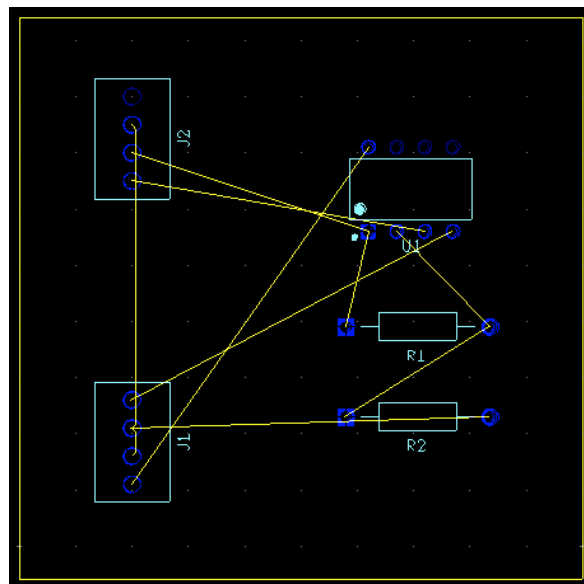

Figure 15. Parts placement.

For this exercise you will place traces on both the Copper Top and Copper Bottom layers.

4. Double-click the **Copper Top** layer in the **Design Toolbox**.

5. Select **Place»Line**.

6. Locate part **U1** (the opamp). Note that pin 1 needs to be connected to R1, as indicated by the ratsnest.

7. Click pin 1 of part **U1**, draw a line to **R1** and click its pin to finish the trace. Press **Esc** to exit the routing mode. The trace will look like the following figure:
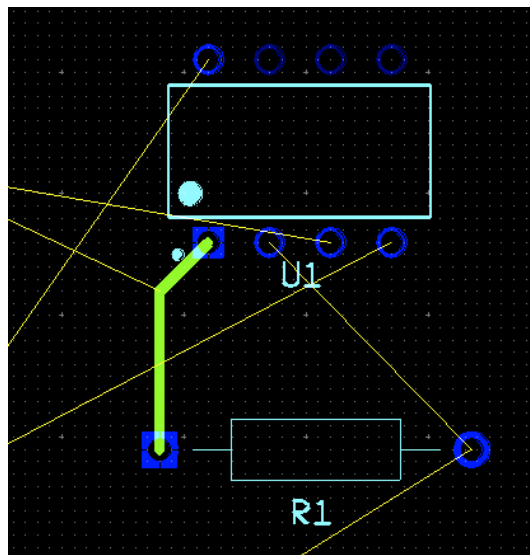
Figure 16. Placing a trace.

8. Double-click the the **Copper Bottom** layer in the **Design Toolbox**.

9. Select **Place»Line**.

10. Click pin 2 of part **U1**, draw a line to **R1** and click its pin to finish the trace. Press **Esc** to exit the routing mode. Note that the color of the trace is red, which is the color configured for the **Copper Bottom** layer.

11. Finish placing traces for the rest of the connections. Your board should look like Figure 17.
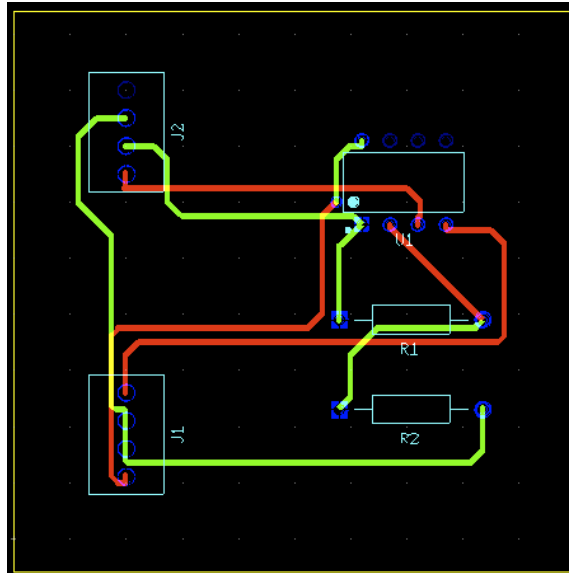

Figure 17. Routed board.

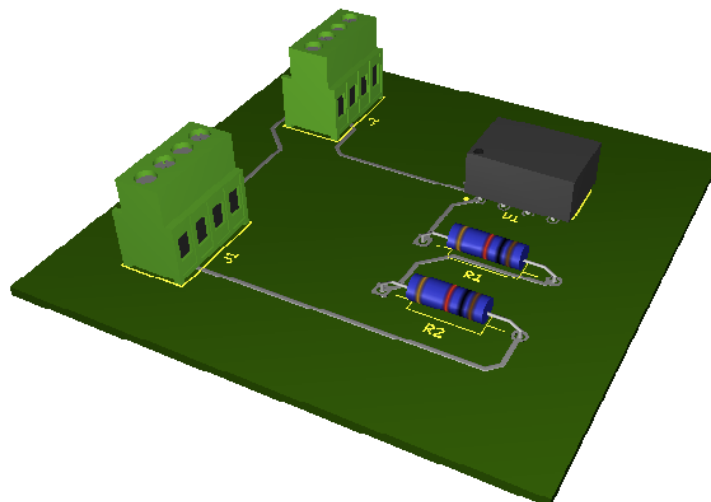12. Select **View»3D Preview** to open a 3D view of your design, as show below.

Figure 18. 3D Preview.

7. Additional Resources

NI Multisim and Ultiboard Technical Resources (http://www.ni.com/multisim/technical-resources/)

National Instruments Circuit Design Technical Library (http://www.ni.com/white-paper/5579/en/)

Customer Reviews

8 Reviews | Submit your review (http://sine.ni.com/apps/utf8/nidz_display_comments.create_comment?p_title=Introduction+to+Multisim%3A+Learn+to+Capture%2C+Simulate%2C+and+Layout+in+Less+Than+30+Minutes&p_doc_id=DZTUT-10710-EN&p_doc_type=DZTUT&p_url=http://www.ni.com/tutorial/10710/en&p_doc_url_id=10710)