Multisim Tutorial
on Principles of Electrical Engineering II Lab Experiment 2:
Frequency Response of Filters

By Gang Liu

4.1 Low Pass Filter
Schematic is below:

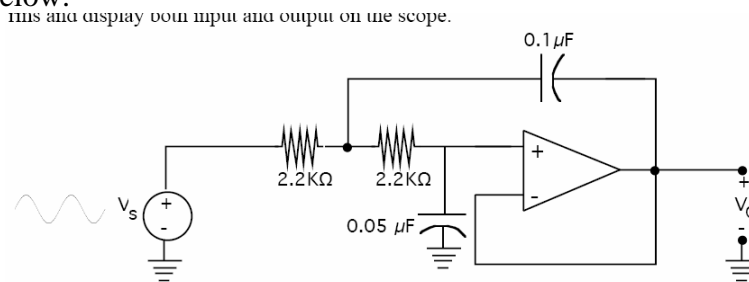
Fig. 7   A Second Order Low Pass Filter

Schematic Capture

_____

Complete the following step to launch Multisim:

1. Select **Start»All Programs»National Instruments»Circuit Design Suite 10.1»Multisim 10.1**. A blank file opens on the workspace called Circuit1.

Complete the following steps to save the file with a new name:

1. Select **File»Save As** to display a standard Windows Save dialog.
2. Navigate to the location where you wish the file to reside, enter 'PEEII_2_LPF' as the filename, and click the **Save** button.

Complete the following steps to start placing components:

1. Open PEEII_2_LPF.ms10 as described above.
2. Select **Place»Component** to display the **Select a Component** browser, navigate to the 741 OP AMP display as shown below and click **OK**. The component appears as a "ghost" on the cursor.
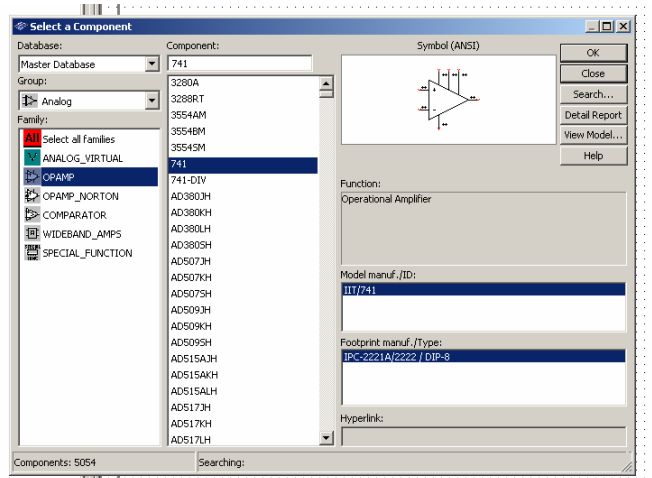
3. Move the cursor to the middle-right of the workspace and left-click to place the component. Note that the Reference Designator for this component is "U1."
4. Place the remaining components in the LPF area as shown below.
Note: since there is no '-VCC' available in Multisim, battery is used instead.

Complete the following steps to wire the circuit:

1. Click on a pin on a component to start the connection (your pointer turns into a crosshair) and move the mouse. A wire appears, attached to your cursor.
2. Click on a pin on the second component to finish the connection. Multisim automatically places the wire, which conveniently snaps to an appropriate configuration, as shown below. This feature saves a great deal of time when wiring large circuits.
3. You can also control the flow of the wire by clicking on points as you move the mouse. Each click "fixes" the wire to that point.
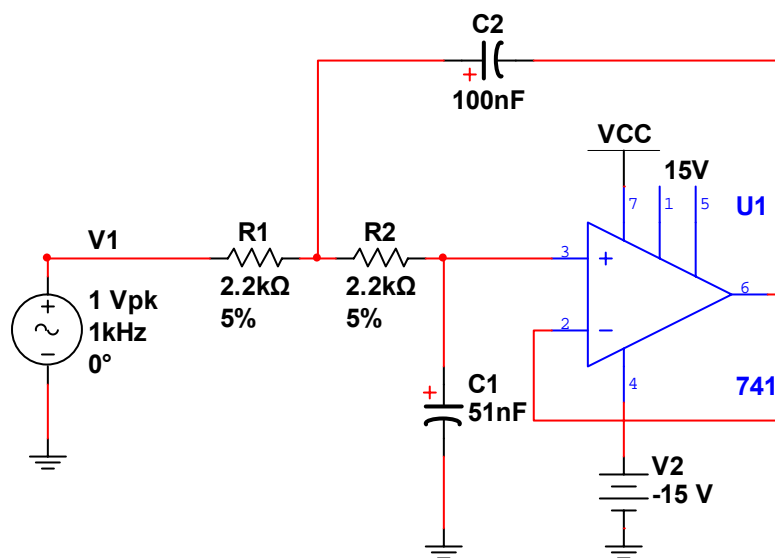
Simulation

1. Select **Simulate»Instruments»Oscilloscope** to place the oscilloscope on the workspace. Wire the instrument as shown in step 4.
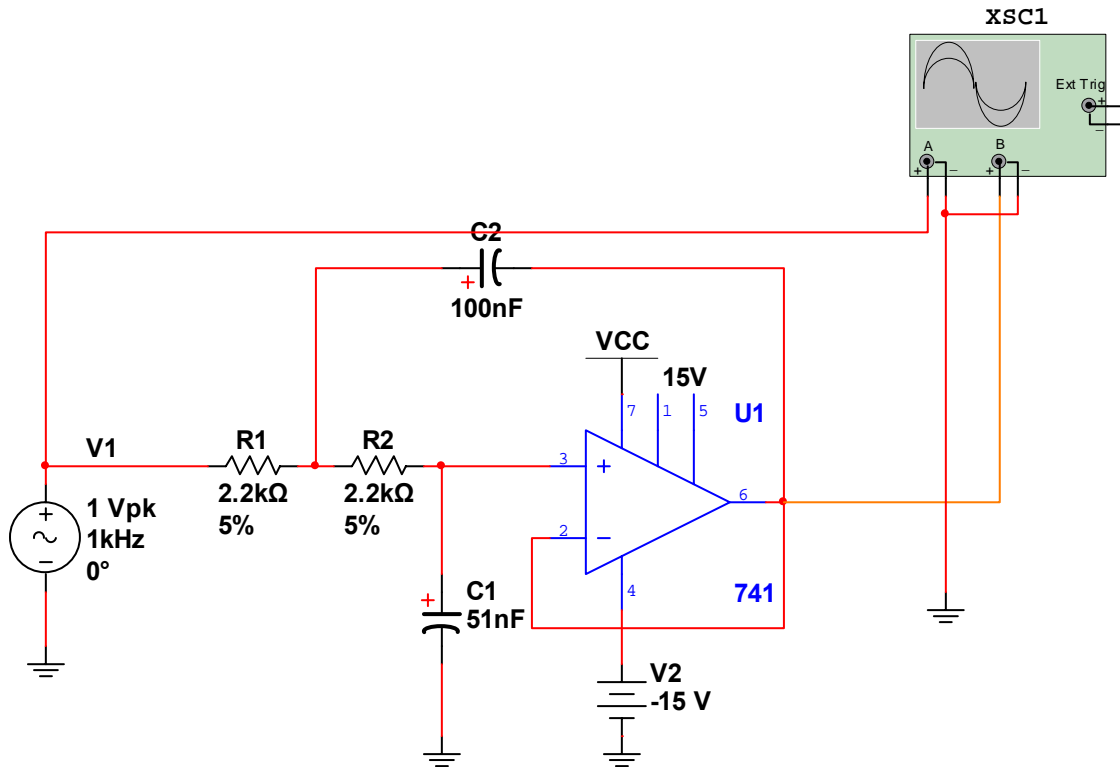
3. Double-click on the scope's icon to show the instrument face. Select **Simulate»Run**. The output of the OP_AMP appears on the scope.
4. Adjust the Timebase to 1mS/Div and Channel A and B's Scale to 1V/Div, Y position to -1. You will see the following displayed on the scope.
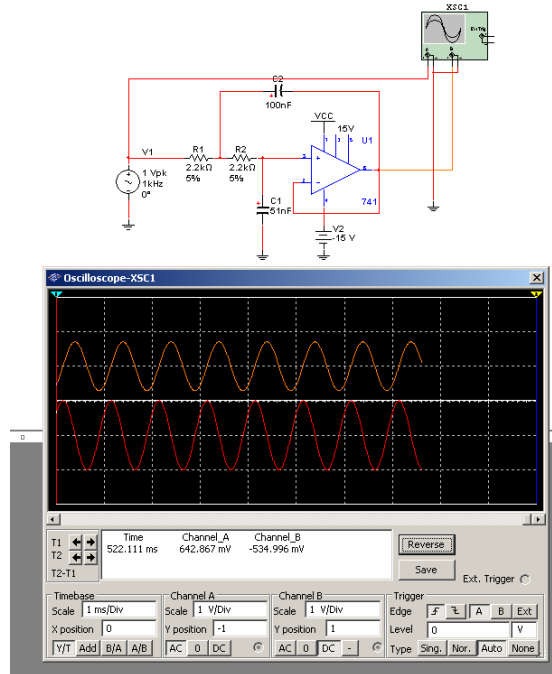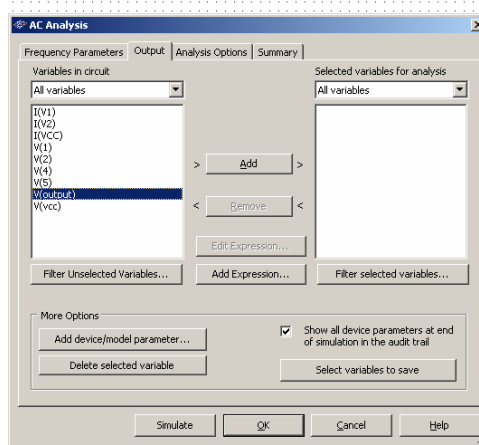
Analysis

_____

In this section, you will use **AC Analysis** to verify the frequency response of the amplifier.

Complete the following steps to perform an **AC Analysis** at the output of the OP_AMP:

1. Double-click on the wire that is attached to pin 6 of the opamp, and change the net name to OUTPUT in the **Net** dialog box.

2. Select **Simulate»Analyses»AC Analysis** and click on the **Output** tab.

3. Highlight V(analog_out) in the left column and click **Add**. V(output) moves to the right column.

4. Click **Simulate**. The results of the analysis appear in the **Grapher**.
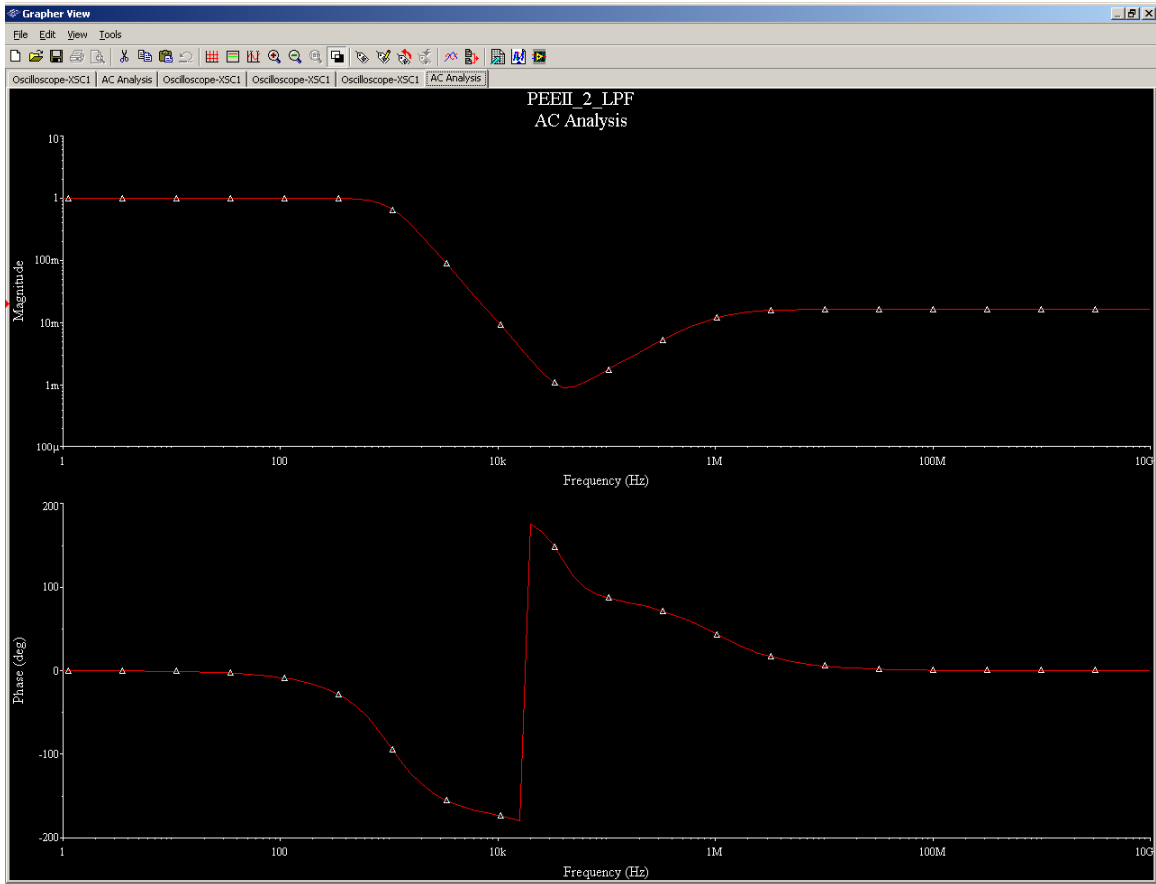
The Grapher
_____

The **Grapher** is a multi-purpose display tool that lets you view, adjust, save and export graphs and charts. It is used to display the results of all Multisim analyses in graphs and charts and a graph of traces for some instruments (for example, the results of the oscilloscope). Complete the following steps to view results of a simulation on the **Grapher**:
1. Run the simulation as described earlier.
2. Select **View»Grapher**.