

Introduction to Multisim

Multisim is a powerful simulation software, a product of National Instruments (NI). It can be used to simulate electrical and electronic circuits. It is also linked to Ultiboard, another NI product for designing printed circuit boards (PCB). This walkthrough document provides a step by step introduction to Multisim, explaining its main features and capabilities.

Program Installation

You can install Multisim program on campus on any PC connected to the University network (MWS), or off campus on your own PC or laptop at home.

National Instruments Account

For installation on your own PC and some other services, like Multisim Live, you need to create an NI account. It is free and anyone can create an NI account. However, if you use your University email account and provide the University license code, you will have access to Multisim Education version. To create an NI account:

1. Visit NI website at <http://www.ni.com/en-gb.html>.
2. Click on log in icon on top right of the screen and **Log in**.
If you already have an NI account, select **My Account**.
3. On the log in page, click on **Create Account**.
4. On the create account page, enter your details and select your role as '**Student** or **Professor/Instructor**', whichever applies.
5. **Important:** You cannot install the software on your own PC and will not have premium access to some NI services, like Multisim Live, if you create the NI account with an email address other than the University email address.

You cannot install the software to your own PC and do not have premium access to some NI products, like Multisim Live, if you create the NI account with an email address other than the University email

Installation on MWS PCs

Multisim is already installed on the PCs in the departmental labs and the PC centres within the Department. It might have been installed on PCs in other PC centres across the university by other users. Before attempting to install the program, please check if the program, **NI Multisim 14.2**, exists in the programs list. To install the program:

1. Open **Install University Applications** program.
2. From the applications list select **Multisim 14.2** and click on **Run**.
3. Follow the instructions during installation.

Installation on home PCs

Students can install Multisim on their own Windows based desktop or laptop PC, not connected to the University network (MWS). NI does not support other operating systems. There are some Windows emulator software for Mac users, however, they are not supported by the University and the Department will not responsible for any compatibility issues.

- Follow the instruction in the ‘[Installing NI Multisim Education Version on Home Computers](#)’ document.

If you had any problem during installation or registration, contact the Service Desk at the CSD at servicedesk@liverpool.ac.uk.

Getting Started

- From the programs list, select and run **NI Multisim 14.2**.

If the program is not installed, follow the installation procedure above.

The program graphical user interface (GUI) appears (Figure 1). The main area is the dot grid area in the middle, called the **workspace**, where you construct your circuit (schematic diagram). There is a menu bar and there are different toolbars, panels, and tabs around the workspace.

You do not need to be worried about these at the moment. You will know them gradually by walking through the activities in this document.

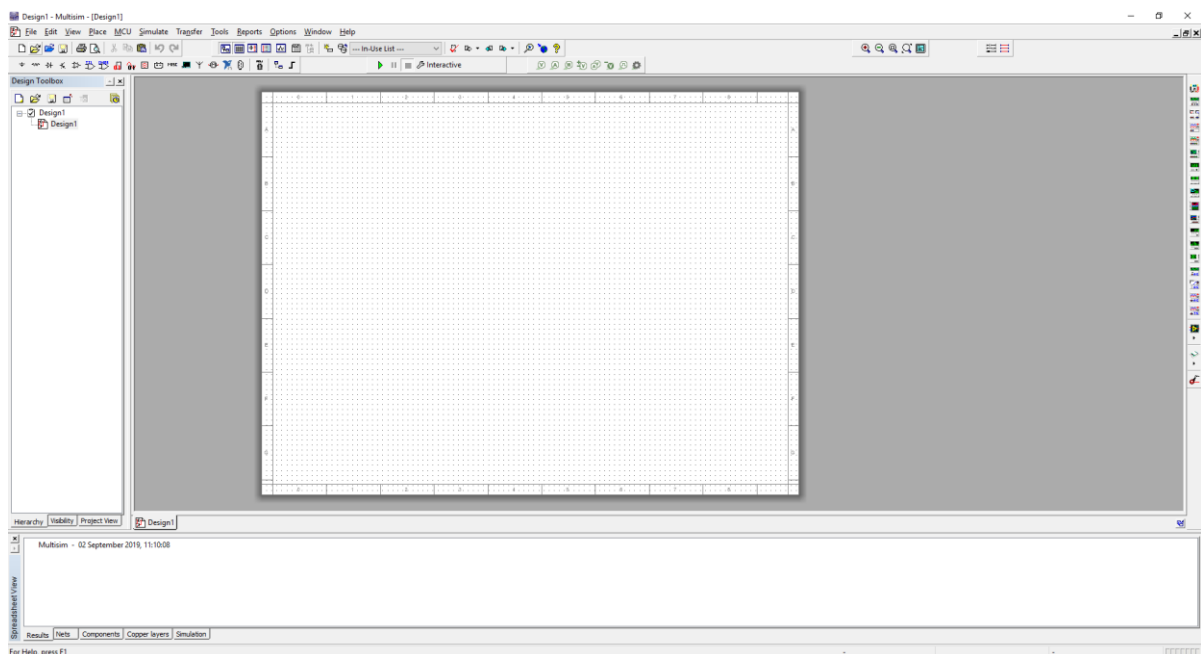


Figure 1. Multisim GUI.

Activity 1. A simple resistor circuit

In this activity you will build a simple circuit consisting of a few resistors and a battery. You will find the voltage across and the current in each resistor by simulation.

Building the circuit

1. On the top toolbar click on the battery icon on the left (or from the menu bar select **Place/Component**). The **Select a Component** window (also known as the Component Browser) appears (Figure 2).
2. From **Component** list select **DC_POWER** and click on **OK**.

If it is not shown on the list, from **Group:** select **Sources** and from **Family:** select **POWER_SOURCES**.

3. Place the battery somewhere in middle left of the workspace by clicking the mouse left button.

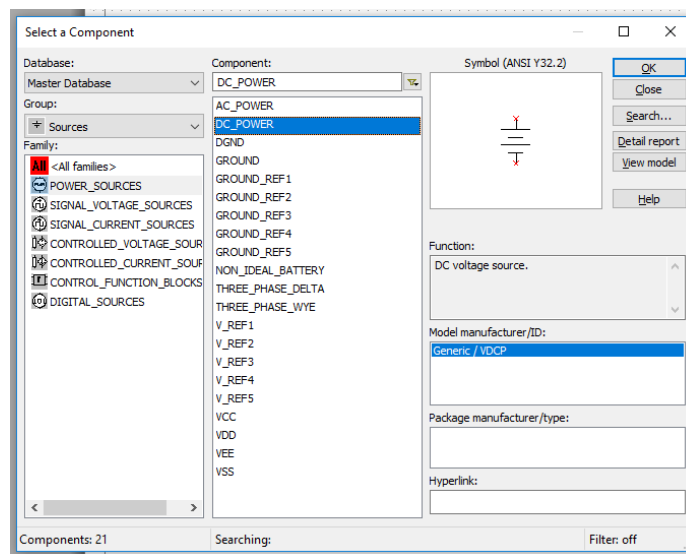




Figure 2. Select a Component window.

4. The **Select a Component** window opens again. Select **GROUND**  and place it below the battery.
5. Select **Group: Basic**, **Family: Resistor**, (or simply click on the resistor icon  from the top toolbar), and **Component: 1k**. Click on **OK**.
6. Place it on right side of the battery.
7. Repeat this step two more times and place the resistors on the right side of the previous ones. Close the **Select a Component** window.
8. The resistors have the reference designations of **R1**, **R2**, and **R3**, and the value of all is 1 k Ω .
9. Right click on the **R2** symbol. A dashed line square appears around the resistor symbol and a pop up window opens. Select **Rotate 90° clockwise** (or counter clockwise).
10. Left click on **R3** symbol. A dashed line square appears around the resistor symbol. Type **Ctrl+R** (this is a shortcut key for rotate and does the same operation as in the previous step).

Wiring

The components should be connected to each other by wires to complete the circuit.

1. Move the mouse cursor to the top terminal of the battery. The cursor changes to a crosshair shape. Click and move the cursor to the left terminal of **R1**. Release the mouse button and click again to connect the wire to the resistor terminal.
2. Repeat the step above to make all the connections, including connection to **GROUND** as in Figure 3.

3. Move the components by click and hold and dragging them around to make your circuit look better. You can also move the wires by selecting them (left click) and dragging the middle or the corner of the shape. You can zoom in or out using the mouse wheel or use a zoom option from the **View** menu.

Moving the components and wires around and how your circuit looks like do not affect how they are connected and hence do not have any effect on the simulation. However, it is a good practice to draw your circuit tidy and well organized to increase the readability, particularly for large circuits.

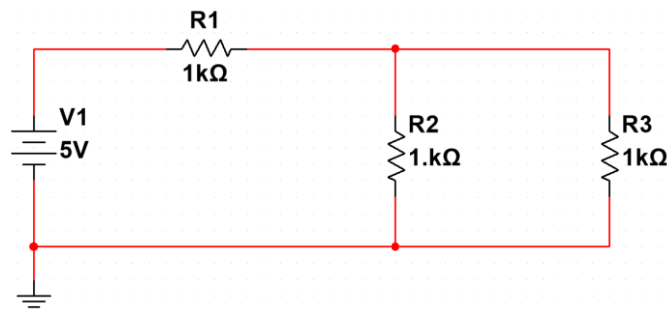





Figure 3. The resistor circuit.


4. Double click on the battery (**DC_POWER**), the **DC_POWER** window opens. From the **Value** tab, change the value to 5 V.
5. Double click on **R2** and from **Value** tab in **Resistor** window change the resistor value to 1.5 kΩ. Repeat this for **R3** and change its value to 3 kΩ.
6. Save your design in a folder you create on your M drive by clicking the save icon on the toolbar, or from **File** menu: **Save**.

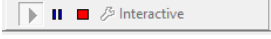


Using probes

Probes are used to show the voltage across or the current through the components. There is also a power probe which can calculate the power dissipated in or delivered by a component.

1. From the **Probes** toolbar at the top click on the **Voltage** probe icon.  Place it on the wire between **R2** and **R3**.
2. From the **Probes** toolbar click on the **Current** probe icon  and place it on the wire on the right side of **R1**.
3. In the yellow probe value boxes, there are some fields that you do not need for a dc simulation. To change the setting, click on the **Setup** icon on the **Probes** toolbar. 
4. In the **Probe Setting** window, on the **Parameters** tab, select **Instantaneous only**. Now the probes information boxes show only one item.

Running the simulation

1. On the **Simulation** toolbar click on the green **Play** icon  (or from the **Simulate** menu click on **Run**).
2. The probe values are shown in the yellow probe value boxes.
3. While the simulation is running you can move the probes to other places and check their current or voltage values. You can also add new probes.


- Click on the red **Stop** icon on the **Simulation** toolbar  (or from **Simulate** menu click on **Stop**).
- Place two more current probes on **R2** and **R3** branches.
- Remove the current probe on **R3** branch and the voltage probe by selecting them and pressing Delete key on the keyboard (or by right click and selecting **Delete** from the pop up menu).
- Place a voltage and current probe  above **R3**.
- Run the simulation. The probe shows the current in **R3** branch and the voltage between that point and the circuit 0 V reference point (**GROUND**).
- Sometimes we want to measure a voltage between two points where none of them is **GROUND**. We can use the differential voltage probe which measures the voltage between a point and another reference point, not **GROUND**.
- Stop the simulation. Click on the **Differential voltage** probe icon  and place it on the left side of **R1**. After placing the probe, a negative sign appears at the probe cursor. Move the cursor to the right side of **R1** and place it. The probes measure the voltage across **R1**, between the voltage probe (with positive sign) and its own reference (with negative sign).
- Run the simulation and check the values shown by each probe.
- Stop the simulation.
- Save your design.

It is a good practice to save your design from time to time.

Activity 2. RC Circuit

In the previous activity, we used dc analysis. In this activity we use ac analysis on a simple RC circuit.

Building the circuit

- From the **File** toolbar click on **New** icon , or from **File** menu select **New**.
- In the **New Design** window, select **Blank** and click on **Create**.
- A new workspace opens. It is called **Design** followed by a number, for example **Design2**. (When you open the Multisim program, a blank design opens automatically.)
- From **Place** menu select **Component**. The **Select the Component** window opens.
- Place **AC_VOLTAGE** from **Group: Sources, Family: AC_VOLTAGE_SOURCES**.
- Then place **GROUND** (from **Sources, POWER_SOURCES**), a 1 k Ω resistor (**Basic/RESISTOR/1k**), and a 100 nF capacitor (**Basic/CAPACITOR/100n**).
- Close the **Select Component** window.
- Rotate the capacitor by 90°, arrange the components, and connect them as in Figure 4. (Use the methods you learned in Activity 1.)

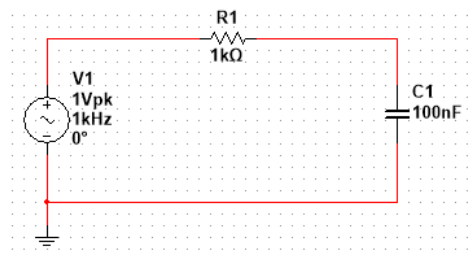


Figure 4. RC circuit.

- Save the design in a folder in your M drive. Give the design a suitable name.

Signal measurement

To measure the signals, you can add probes as in Activity 1. But for ac signals, it is better to use graphical tools like an oscilloscope.

1. From the **Instruments** toolbar on the right click on the **Oscilloscope** icon and place it above the circuit.
2. Connect **Channel A** to **V1** and **Channel B** to the point between **R1** and **C1** (Figure 5).
(You only need to connect the main leads shown by + sign. The connections shown by a – sign are internally connected to the **GROUND**.)
3. Start simulation by clicking on the **Play** button.
4. To view the signals, double click on the oscilloscope.
5. The **Oscilloscope-XCS1** window opens (Figure 6).

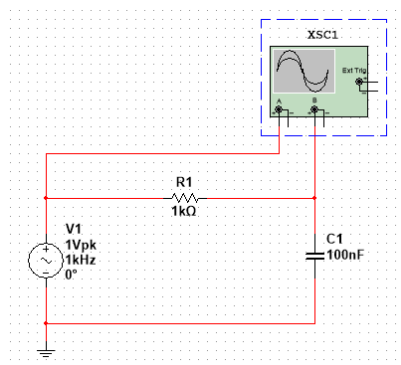


Figure 5. RC circuit and oscilloscope.

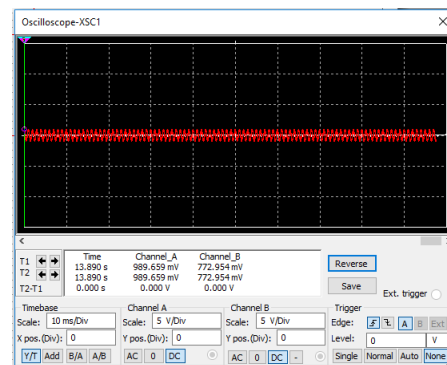


Figure 6. The oscilloscope window.

6. Change the Channel A and Channel B **Scales** to **500 mV/Div**.
7. Change the **Timebase Scale** to **500 us/Div** (us is used for μ s).
8. Click on **Single** to have more stable waveforms.
9. There are two line cursors on the left of the screen. Drag them to the middle and place them at two different positions. The values at the cursor positions are shown below the screen.

This oscilloscope is a very basic one with very limited functionality. We can add a more realistic oscilloscope.

10. Stop the simulation.
11. Close the oscilloscope window.
12. Select the oscilloscope **XSC1** and hit the Delete key on the keyboard. When the oscilloscope is removed, its connections will be removed too.
13. From the **Instruments** toolbar click on the **Tektronix Oscilloscope** icon and place it above the circuit.

This is 4-channel advanced oscilloscope with digital storage, similar to the oscilloscopes used in the lab.

14. Connect Channel 1 to **V1** and Channel 2 to the point between **R1** and **C1**.
15. Start simulation.
16. Double click on the oscilloscope. The **Tektronix Oscilloscope-XC1** window opens. It contains an interactive oscilloscope panel (Figure 7).
17. Click on **Power** button to turn the screen on.

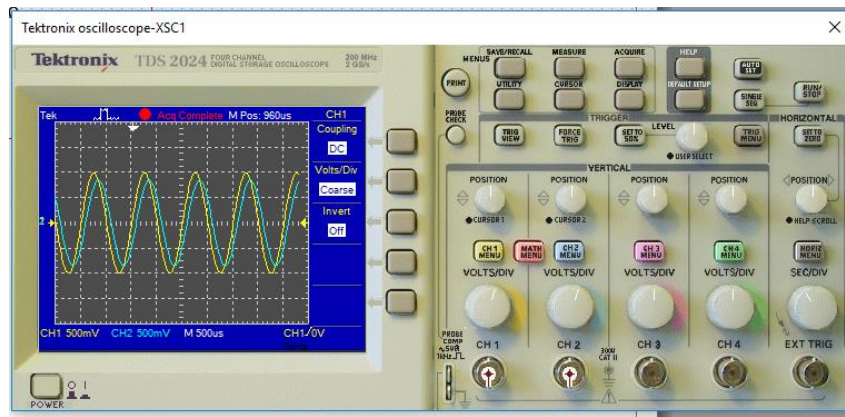


Figure 7. Tektronix oscilloscope.

18. Move the **VOLTS/DIV** knobs on **CH 1** and **CH 2** to set them at **500 mV**.

If **CH 2** trace is not shown, click on **CH 2 MENU** button to turn it on.

19. Move the **SEC/DIV** knob to set the timebase to **200 us** (μs).
20. To have more stable waveforms click on **SINGLE SEQ** button. You can retrieve the normal continuous mode by clicking on **RUN/STOP** button.
21. Play around with different knobs and buttons to see their functions.
22. Measure the amplitude of the signals and their phase difference.

If you have not worked with oscilloscopes before, you can use the measurement facilities of this oscilloscope. Click on the **MEASURE** button. Use the soft keys next to the screen to change the **Source** to **CH 1** and **Type** to **Pk-Pk** (peak to peak). Click on the lower soft key pointing to **Back**. Click on the fourth soft key from the top. Use the top soft keys to change the **Source** to **CH 2** and **Type** to **Pk-Pk**. Click on the **Back** soft key. The peak to peak values of the signals are now shown on the area next to the screen.

23. Stop the simulation.
24. Double click on **V1**. In **AC_VOLTAGE** window, change the frequency to **500 Hz**.
25. Start the simulation and adjust the timebase (**SEC/DIV**) of the oscilloscope to show a few cycles of the waveforms.
26. Observe the change in the amplitude and phase of the output signal.
27. Repeat the above to change the frequency to 100 Hz, 5 kHz, and 10 kHz, and 50 kHz.
28. See how the amplitude and phase of the output signal change with frequency.
29. Stop the simulation.

You can save your design with a different file name to be able to retrieve the original version. From File menu select **Save as** and save the design with a different name.

Using the function generator

To have a better control on the signal properties, we can replace the **AC_VOLTAGE** source with a function generator.

1. Delete **V1**.
2. From the **Instruments** toolbar on the right, click on the **Agilent function generator** icon and place it on the top left of the workspace, next to the resistor.
3. Connect the bottom lead to the resistor (Figure 8).

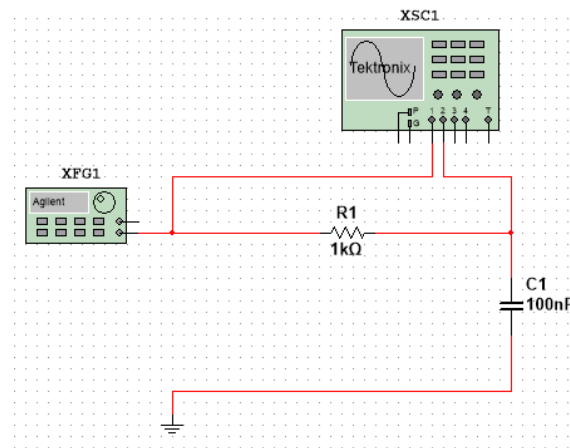


Figure 8. RC circuit with oscilloscope and function generator.

4. Start the simulation.
5. Double click on the function generator **XFG1**.
6. The **Agilent function generator – XFG1** window opens and shows the front panel of the function generator.
7. Click on the **Power** button to turn the function generator on.
8. The display should show the signal amplitude in V_{pp} . (If it shows something else click on the **Ampl** button.)
9. Using the right and left arrow keys on the panel, move the cursor (blinking digit) to the most left digit.
10. Using the up and down arrow keys change the value of the voltage to 2 V_{pp} . Alternatively, you can use the knob next to the display to change the amplitude.
11. Adjust the vertical scale (**VOLTS/DIV**) and timebase (**SEC/DIV**) on the oscilloscope to see a few cycles of the waveforms on the screen.
12. Click on the **Freq** button on the function generator panel.
13. Move the cursor to the most left digit using left and right arrow keys.
14. Use the knob or up and down arrow keys to change the frequency of the signal.
15. See the effect on the output signal on the oscilloscope. Adjust the oscilloscope scales if needed.

If the oscilloscope stops showing the waveforms, stop and start the simulations.

16. Save your design with a new file name.

Transient response

1. Change the signal frequency on the function generator to 500 Hz and set the oscilloscope timebase to 500 μ s.
2. Change the signal to square wave by clicking on the square wave button on the function generator panel.
3. Change the signal frequency and see the effect on the output signal. You might need to change the oscilloscope timebase to see the signal's detail.
4. In particular, see the effect on the output signal when the frequency is about 20 Hz and when the frequency is above 1.2 kHz.
5. Increase the frequency to above 20 kHz and see effect on the output signal. You need to change the vertical scale on **CH2** to see the output signal properly.
6. You can print the oscilloscope traces at any time by clicking on the **PRINT** button. In the **Print** window, you can also choose to save the print out as PDF and few other formats, or send it to One Note.
7. Stop the simulation.

Frequency response

The frequency response is plots of magnitude and phase of the transfer function (output divided by input) of a circuit or system. The magnitude is usually described in logarithmic scale (Bode plot) by decibels (dB). $1 \text{ dB} = 20 \log(|V_o|/|V_i|)$.

1. Open the original RC circuit you had saved before (Figure 4).
2. From the **Instruments** toolbar on the right click on **Bode Plotter** icon and place it above the circuit.
3. Connect the + lead of **IN** to the wire between **V1** and **R1** and the + lead of **OUT** to the wire between **R1** and **C1** (Figure 9).
4. Start the simulation.
5. Double click on the **Bode Plotter XBP1**.
6. In the **Bode Plotter – XBP1** window (Figure 10), change the initial frequency (**Horizontal/I**) to **1 Hz**, the final frequency (**Horizontal/F**) to **100 kHz**, and the initial amplitude to **- 50 dB**.
7. Drag the cursor on the left of the screen to any point you like. It will show the magnitude at each frequency on the panel below the screen. You can also move the cursors using the arrow buttons below the screen.

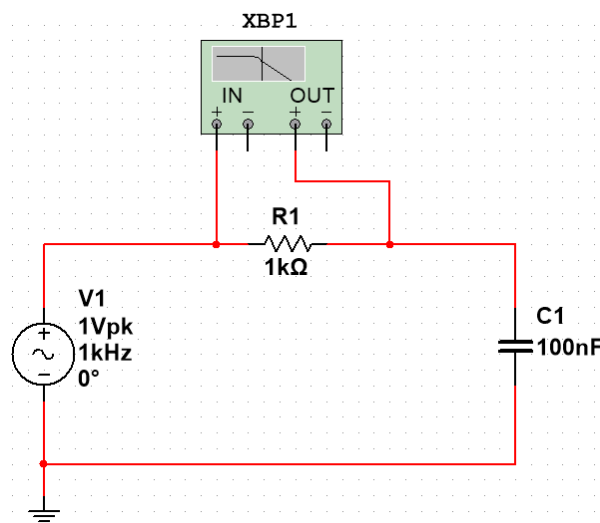


Figure 9. RC Circuit with Bode plotter.

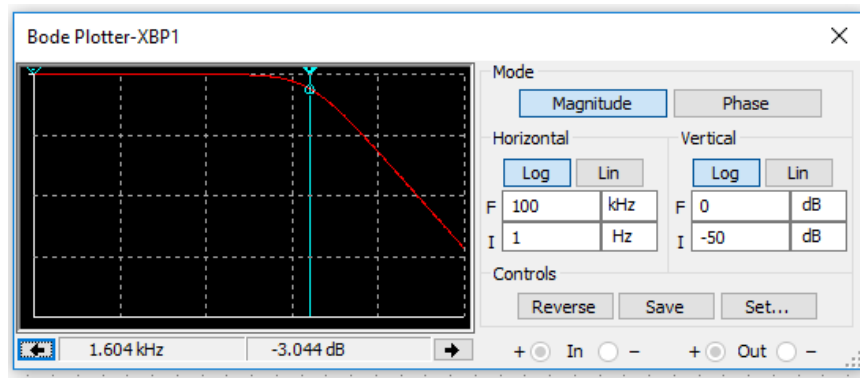


Figure 10. Bode plotter window.

8. Put the cursor at the point with magnitude of -3 dB and read the frequency.

The RC circuit is a lowpass filter (LPF) and the -3 dB point is called cut off frequency. Verify that the cut off frequency equals to $f_c = \frac{1}{2\pi R_1 C_1}$.

9. Click on the **Phase** tab.
10. Set the initial value of phase (**Vertical/I**) to **-90°** and the final value (**Vertical/F**) to **0°**.
11. Drag the cursor to the cut off frequency that you found from the magnitude plot.
12. Verify that at cut off frequency the phase is 45°.
13. Stop the simulation.
14. Double click on the resistor and change its value to 10 kΩ.
15. Run the simulation.
16. Click on the **Magnitude** tab.
17. Measure the cut off frequency (at -3 dB) and compare it with its previous value. Verify that it follows the cut off frequency equation.
18. Stop the simulation.
19. Double click on the capacitor and change its value to 10 nF.
20. Run the simulation and measure the cut off frequency. Compare it with the previous values.
21. Stop the simulation and save your design with a new name.

Activity 3. OpAmp circuit

In this activity, you will simulate a standard non-inverting operational amplifier (OpAmp).

Building the circuit

1. Open a new design.
2. On **Place** menu click on **Components** and from the **Select a Component** window place the following components on the workspace, as in Figure 11. Their exact position is not important at this stage. You will arrange them later.
 - From **Group: Analog, Family: OPAMP** select **AD712SQ/883B**. You can search the component by typing the first few letters of its name, for example AD721, in the component name box. There are two OpAmps in the package named as **A** and **B**. Select the component **A** and place it on the workspace.
 - From **Sources/POWER_SOURCES** place two instances of **DC_POWER**.
 - From **Sources/POWER_SOURCES** place three instances of **GROUND**.
 - From **Sources/SIGNAL_VOLTAGE_SOURCES** place **AC_VOLTAGE**.

- From **Basic/RESISTOR** select **1k** resistor. In the **Package manufacturer/type** field select **IPC-2221A/2222/RES1300-700X250**. Place the resistor on the workspace.

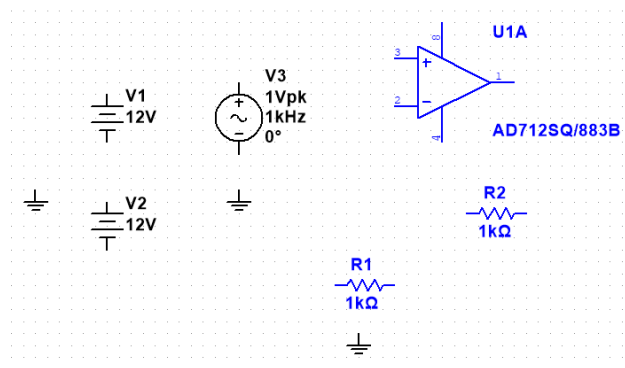


Figure 11. The components for a non-inverting amplifier.

3. Close the **Select a Component** window.
4. Select **R1** and type Ctrl+R.
5. Connect the components by wires as in Figure 12. Adjust the position of components if necessary.
6. From **Place** menu select **Connectors/On-page connector** and place it on the positive terminal of **V1**. The **On-page connector** window will open.
7. In the **Connector name** field, type **+V** and click on **OK**.
8. Select another **On-page connector** and place it at terminal 8 of **U1A**.
9. In the **On-page connector** window select **+V** from the list of **Available connectors**. In this way, terminal 8 of **U1A** is connected to the positive terminal of **V1**.
10. Repeat steps 11 to 14 to connect the negative terminal of **V2** to pin 4 of **U1A**. Name the **On-page connector** **-V**.
11. Save your design in a folder in your M drive.

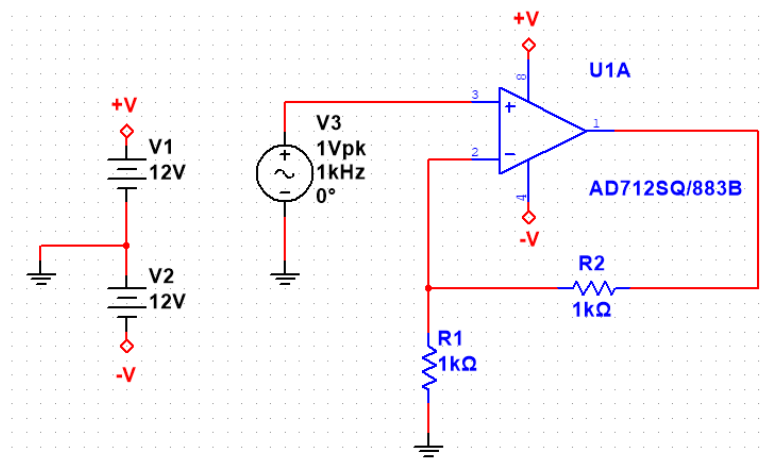


Figure 12. Non-inverting amplifier.

Simulation

1. From the **Instruments** toolbar click on **Tektronix oscilloscope** icon and place it above the circuit.

2. Connect **CH 1** of the oscilloscope to the non-inverting input of the OpAmp (pin 3) and **CH 2** to the output (pin 1).
3. Start the simulation.
4. Double click the oscilloscope to open the **Front panel** window.
5. On front panel click on **Power** button to turn the oscilloscope on.
6. If **CH 2** trace is not shown, click on **CH 2 MENU** button to turn it on.
7. Adjust the timebase to display a few cycles of the waveforms.
8. Measure the amplitude of input and output signals. You can use the **MEASURE** button on the front panel for more precise measurement (see the information in blue text box above Figure 6).

The gain of a non-inverting OpAmp circuit is equal to $A_V = 1 + R_2/R_1$.
Verify the gain of the amplifier from the measured values.

9. Stop the simulation.
10. Calculate and change the value of **R2** to achieve the gain of 10.
11. Run the simulation and verify the gain by measuring the amplitude of input and output signals.
12. Stop the simulation.