

ABHISHEK SHARMA

COMPUTER SCIENCE AND ENGINEERING DEPARTMENT

SECTION : "I"

ROLL NO.: 01

YEAR : THIRD

ENROLLMENT NO.: 12019009001127

ANALOG ELECTRONICS CIRCUIT LABORATORY

DAY 1

ASSIGNMENT NO.: 01

DATE : 13.07.2021

UNIVERSITY OF ENGINEERING & MANAGEMENT, KOLKATA

◆ Multisim 14.2 Software Description and Installation.

NI Multisim (formerly **MultiSIM**) is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. Multisim is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation. Multisim was originally created by a company named Electronics Workbench Group, which is now a division of National Instruments. Multisim includes microcontroller simulation (formerly known as MultiMCU), as well as integrated import and export features to the printed circuit board layout software in the suite, NI Ultiboard.

Let's move into the installation process :

To download the software here is the link :

After downloading the .rar file from the given link, install the software as shown in the video here.

After successful installation of the software, let's deploy a basic electronics circuit using the software.

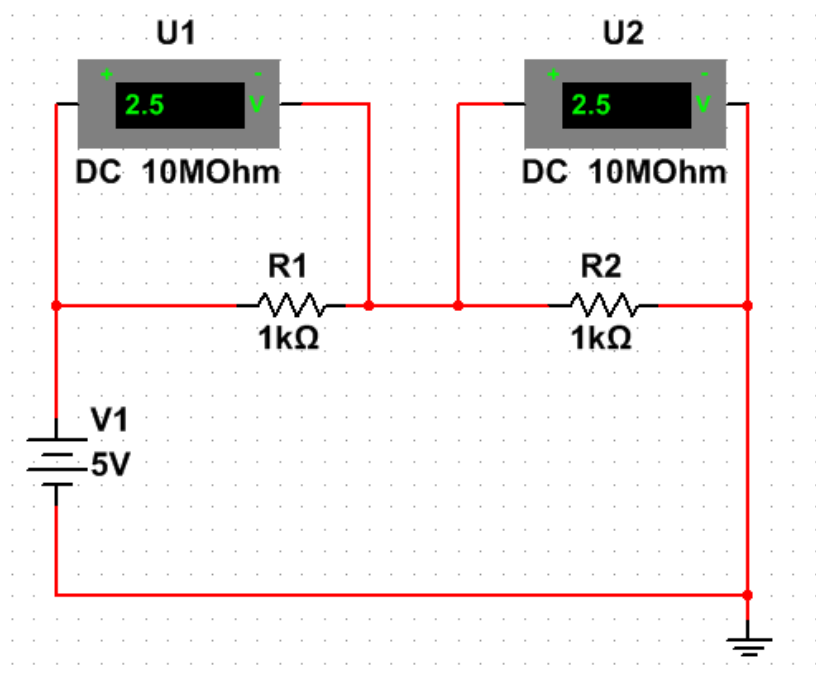
Circuit 1 :

Here we have deployed a simple circuit connecting two resistors of $1\text{k}\Omega$ and providing the DC voltage of 5V. Now finding out the voltages and currents across the resistors using the simulation.

Steps to create the circuit :

1. From the Components find out the Power source DC, resistors and the ground.
2. Place them at right positions.
3. Connect them using the wires and from the right of the toolbox find out the voltmeter or, you can find the voltmeter from the Components -> Indicator section too.
4. Save the circuit.
5. Simulate the circuit and take the outputs.

Simulating and taking the values of the voltage using the voltmeter -



DC Power Source 5V
Resistors $1\text{k}\Omega$

Voltage across the each resistors
2.5V (Ans.)

Circuit 2 :

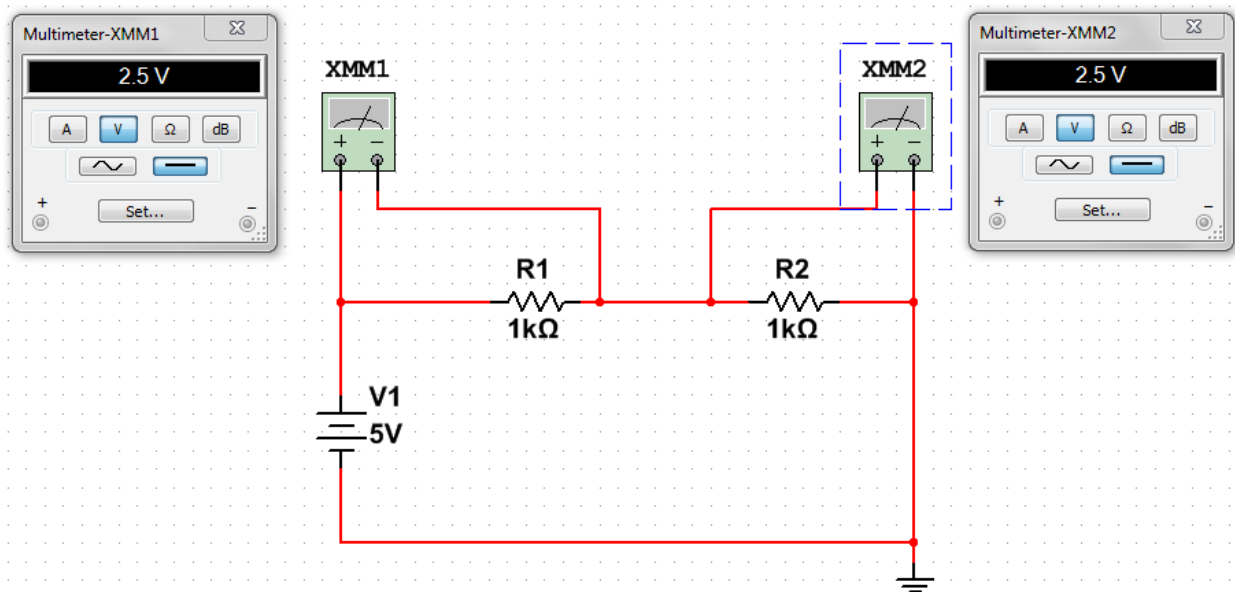
Here we have deployed a simple circuit connecting two resistors of $1k\Omega$ and providing the DC voltage of 5V. Now finding out the voltages and currents across the resistors using the simulation.

(Using Ammeter here)

Steps to create the circuit :

1. From the Components find out the Power source DC, resistors and the ground.
2. Place them at right positions.
3. Connect them using the wires and from the right of the toolbox find out the ammeter or, you can find the ammeter from the Components -> Indicator section too.
4. Save the circuit.
5. Simulate the circuit and take the outputs.

Simulating and taking the values of the voltage using the ammeter -

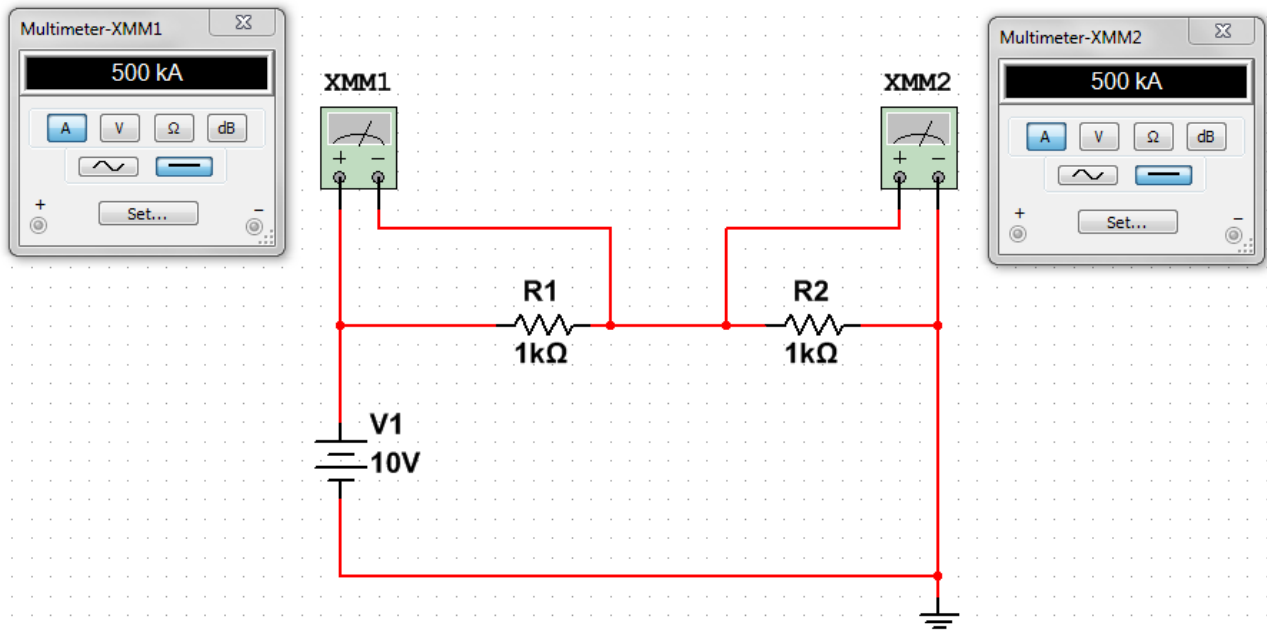


DC Power Source 5V

Resistors $1k\Omega$

Voltage across the each resistors 2.5V (Ans.)

(A) Now let's check the current at each node of the resistors using the same ammeter. But this time change the 'V' to 'A' indicator in the ammeter.



DC Power Source 10V

Resistors 1kΩ

Current across the each resistors 500kA (Ans.)

Circuit 3 :

Visualizing the oscilloscope curves using the software.

Requirements :

1. AC Power Source
[Config. : 2.12 vrms value and 50 Hz, this will provide the voltage of 2.99V]
2. Diode [1N4002G]
3. Resistor [1 pc.] - 1 k Ω
4. Oscilloscope

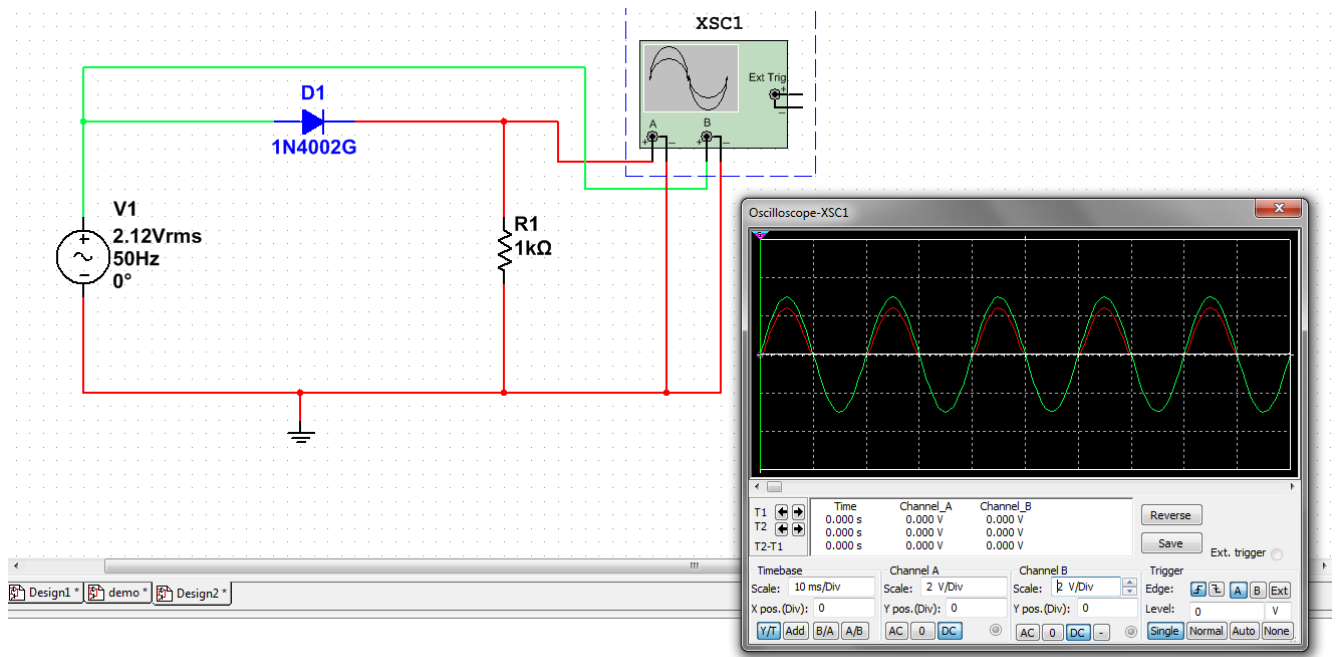
Steps to be followed :

1. From the Components find out the Power source AC, resistors and the ground and configure the data in the resources, as given in the requirements.
2. Place them at right positions.
3. Connect them using the wires and from the right of the toolbox find out the Oscilloscope. and make the connections with the circuit.
4. Save the circuit.
5. Simulate the circuit and take the outputs.

Additional Tip :

To make the waves/curves more attractive change the colour of the curves by clicking on the particular wire connected to the oscilloscope.

Now, let's deploy the final circuit with the visualization of the curves using the oscilloscope.



---- O ----