

3 LAB EXPERIMENT

OBJECT:

An Introduction to National Instrument Multisim 14.0 software.

INTRODUCTION

1. MULTISIM:

Multisim integrates industry-standard SPICE simulation with an interactive schematic environment to instantly visualize and analyze electronic circuit behavior. Its intuitive interface helps educators reinforce circuit theory and improve retention of theory throughout engineering curriculum. By adding powerful circuit simulation and analyses to the design flow, Multisim helps researchers and designers reduce printed circuit board (PCB) prototype iterations and save development costs.

2. MULTISIM OPTION:



1. MULTISIM FOR EDUCATION:

Multisim for Education is circuits teaching application software for analog, digital, and power electronics courses and laboratories.



2. MULTISIM FOR DESIGNER:

Multisim provides engineers the SPICE (Simulation Program with Integrated Circuit Emphasis) simulation, analysis, and PCB design tools to quickly iterate through designs and improve prototype performance.

3. WHAT CAN YOU DO WITH MULTISIM FOR EDUCATION?

Multisim empowers students to understand the circuits in a way that maximizes student learning and real-world preparedness. Explore the subjects below to see how Multisim can improve your program.

1. ANALYZE CIRCUIT BEHAVIOUR:



Simulated benchtop instruments and advanced analyses in Multisim lend a thorough understanding of circuit behavior, which reinforces textbook theory.

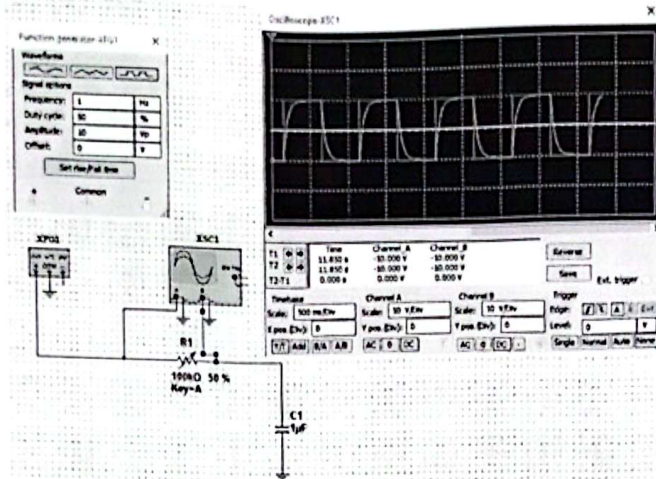


Figure 1: Analyze Circuit response of RC circuit.



2. TEACH ELECTRONICS:

As a learning tool, Multisim connects abstract theory to concrete signals through intuitive design, interactive simulation, and seamless hardware integration.

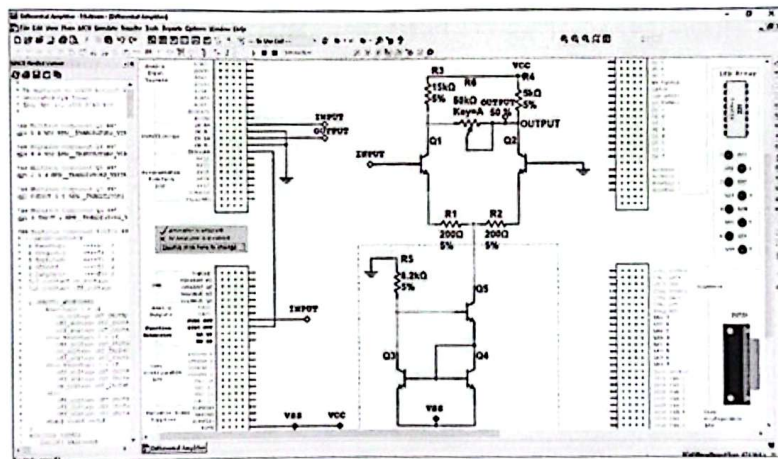


Figure 2: Circuit of Differential Amplifier.



3. REINFORCE THEORY BY COMPARING REAL AND SIMULATED SIGNALS:

Multisim embraces the need to take a hands-on approach to engineering education.

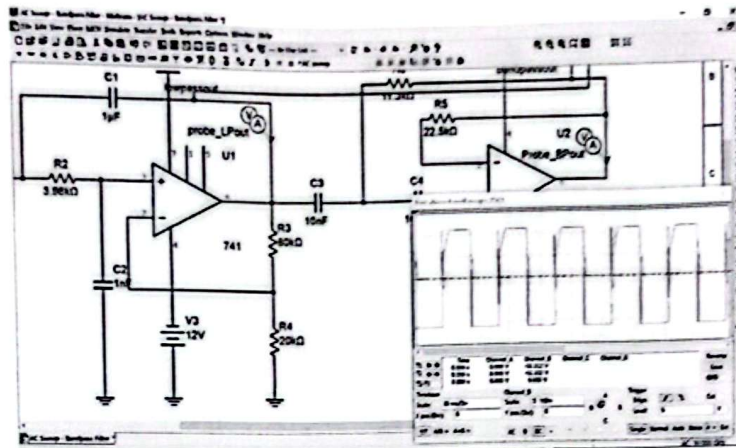


Figure 3: Band pass filter circuit simulation.



4. TEACH DIGITAL LOGIC AND DEPLOY IT TO HARDWARE:

Multisim goes beyond standard SPICE simulation to include an extensive digital component library that you can simulate and deploy to any Digilent FPGA device.

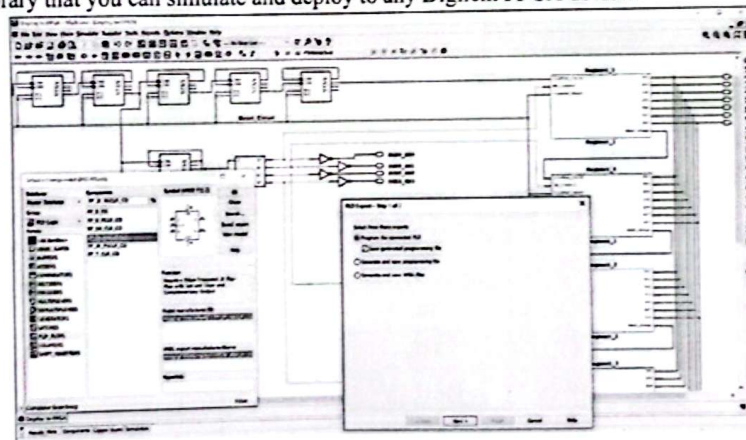



Figure 4: FPGA circuit design.

4. HOW TO START WITH NI MULTISIM 14.0:

1. Open The National Instrument Multisim 14.0 by clicking  over this icon.
2. This Window will appear.

3. Click to Create the new file.

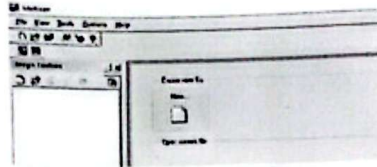


Figure 5: Creating new file.

4. Go for the blank icon and click on the create button.

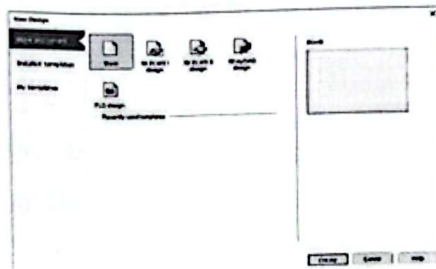


Figure 6: Selecting type of design.

5. Now you can design your desired electronic circuit on this window.

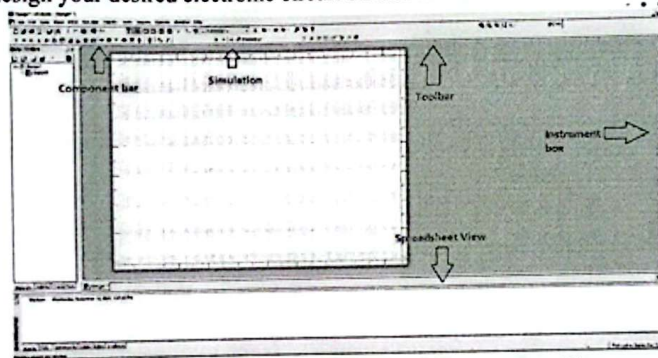
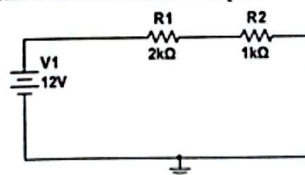


Figure 7: Main window of NI Multisim with labelling.

5. TASK 1:

Design the given circuit on Multisim and observed the output:



5.1. DRAWING SCHEMATIC IN MULTISIM 14.0:

1. Selecting Components: By clicking this circle area component window will appear select component which is necessary to design circuit.

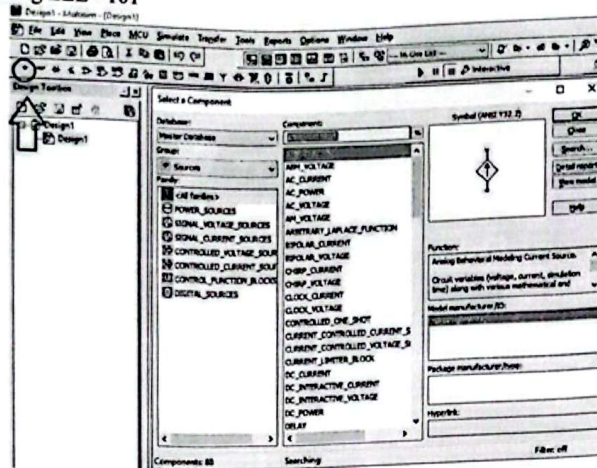


Figure 8: Component Selecting Window.

- Place all the component on the blank sheet which is required to design your circuit.

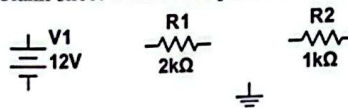


Figure 9: Selected component on the main window.

- Wiring the Schematic.

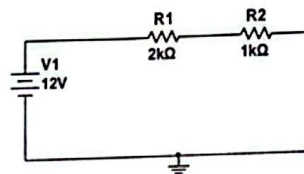


Figure 10: Complete circuit after connection.

- Select the instrument from instrument box. Place the instrument through which you can measure the output. i.e. Multimeter.

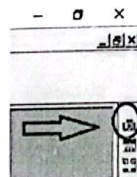


Figure 11: Selecting Multimeter from Instrument box.

- Measure the value by connecting your instrument to the circuit in an appropriate way.
- Click the play button in the simulation box.
- Now you are able to see your output on the display window which will appear after double clicking the instrument.

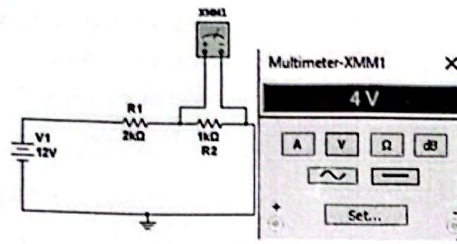
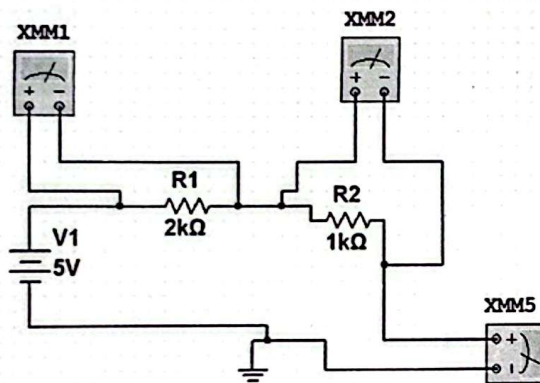


Figure 12: simulating the circuit and visualizing the output.

8. Now measure the voltage across each resistor.

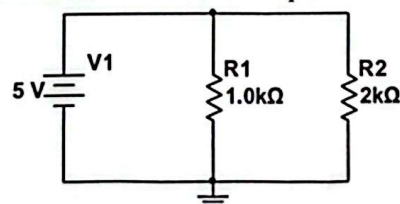
5.2. RESULTS AND DISCUSSION:

$V_{in} (V)$	$R_1 (\Omega)$	$R_2 (\Omega)$	$R_T (\Omega)$	$V_{R1}(V)$	$V_{R2}(V)$	I
5	2K	1K	3K	3.333V	1.667V	1.667mA



6. TASK 2:

Design the given circuit on Multisim and observed the output:



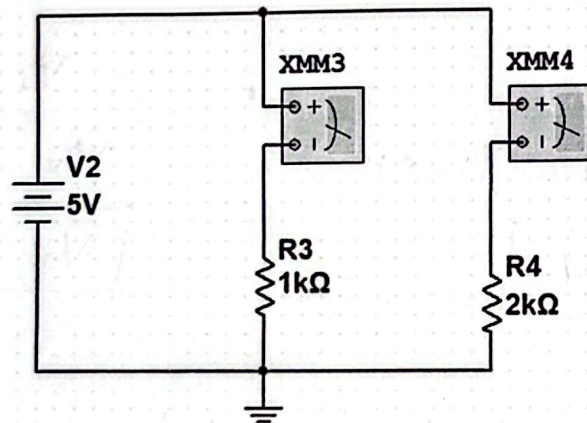
1.1. PROCEDURE:

1. Selecting Components: By clicking this circle area component window will appear select component which is necessary to design circuit.
2. Place all the component on the blank sheet which is required to design your circuit.
3. Wire the Schematic as per the given circuit diagram.
4. Select the instrument from instrument box. Place the instrument through which you can measure the output. i.e. Multimeter.

5. Measure the value by connecting your instrument to the circuit in an appropriate way.
6. Click the play button in the simulation box.
7. Now you are able to see your output on the display window which will appear after double clicking the instrument.

1.2. RESULTS AND DISCUSSION:

V_{in} (V)	R_1 (Ω)	R_2 (Ω)	R_T (Ω)	I_{R1} (V)	I_{R2} (V)	V
5	1K	2K	0.666K	5mA	2.5mA	5V



EXERCISE:

Question 1:

Explain why the voltage is not same across each resistor?

In Task 1, voltage is not same across each resistor because the circuit is connected in series configuration. In series configuration, current remains the same along the single path but the voltage drop across each electrical component.

Question 2:

Calculate the values of voltage and current across each resistor to verify that the outputs obtained from Multisim are correct?

QUESTION 02 (Calculations)

→ TASK 01 (Series Circuit)

$$R_T = R_1 + R_2 = 2\text{ k}\Omega + 1\text{ k}\Omega = 3\text{ k}\Omega$$

$$I_T = \frac{V_T}{R_T} = \frac{5}{3 \times 10^3} = 1.667\text{ mA}$$

$$\boxed{I_1 = I_2 = I_T = 1.667\text{ mA}}$$

$$\Rightarrow V_1 = I_1 R_1$$

$$\Rightarrow V_1 = (1.667 \times 10^{-3}) (2 \times 10^3)$$

$$\boxed{V_1 = 3.334\text{ V}}$$

$$V_2 = I_2 R_2$$

$$V_2 = (1.667 \times 10^{-3}) (1 \times 10^3)$$

$$\boxed{V_2 = 1.667\text{ V}}$$

→ TASK 02 (Parallel Circuit)

$$R_T = \frac{R_1 \times R_2}{R_1 + R_2} = \frac{(1)(2)}{(1+2)} = \frac{2}{3} = 0.667\text{ k}\Omega$$

$$\Rightarrow \because V_T = I_T R_T \Rightarrow I_T = \frac{V_T}{R_T} = \frac{5}{0.667 \times 10^3} = 7.49\text{ mA}$$

$$\boxed{I_T = 7.49\text{ mA}}$$

Using current divider rule:

$$\Rightarrow I_1 = I_T \left(\frac{R_2}{R_1 + R_2} \right) = 4.999\text{ mA} \approx 5\text{ mA}$$

$$\Rightarrow I_2 = I_T \left(\frac{R_1}{R_1 + R_2} \right) = (7.49 \times 10^{-3}) \left(\frac{1 \times 10^3}{1 \times 10^3 + 2 \times 10^3} \right)$$

$$\Rightarrow I_2 = 2.49\text{ mA} \approx 2.5\text{ mA}$$

$$\boxed{I_1 = 5\text{ mA}}$$

$$\boxed{I_2 = 2.5\text{ mA}}$$