Multisim Basics Tutorial
Building and Simulating Passive Circuits

ECEN 222, Spring 2026

University of Nebraska-Lincoln
Maxx Seminario

Introduction

This tutorial will guide you through the basics of using Multisim to build, simulate, and analyze simple circuits with passive components (resistors, capacitors, inductors and sources). By the end of this tutorial, you will be able to:

- Place components on the workspace
- Wire components together
- Configure component values
- Add voltage and current probes
- Run DC simulations
- Measure voltages and currents in your circuit

1 Tutorial 1: Simple Voltage Divider

We'll start with the most basic circuit: a voltage divider consisting of a DC voltage source and two resistors in series.

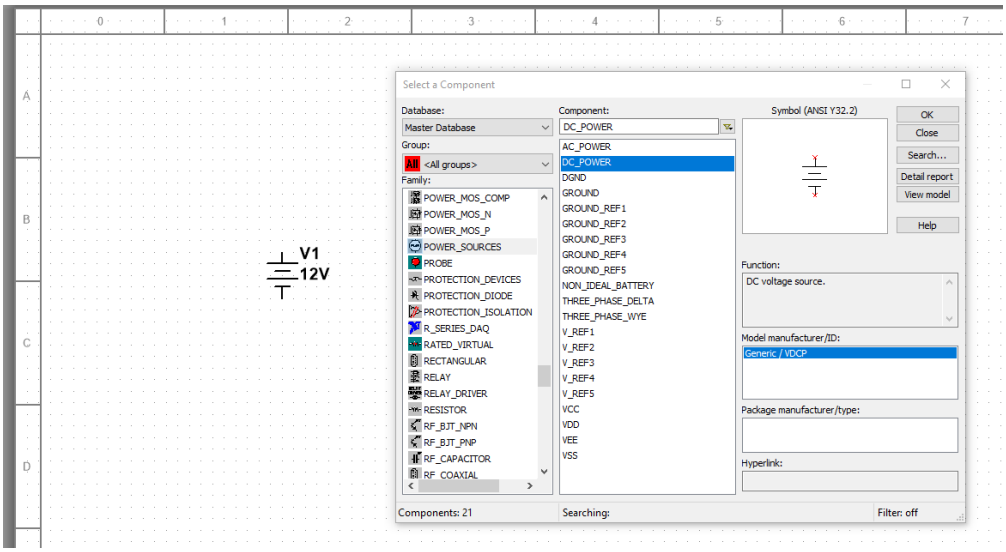
1.1 Step 1: Create a New Project

1. Open Multisim
2. Click **File** → **New** → **Blank**
3. Save your project: **File** → **Save As...** Name it `voltage_divider`

1.2 Step 2: Place the Voltage Source

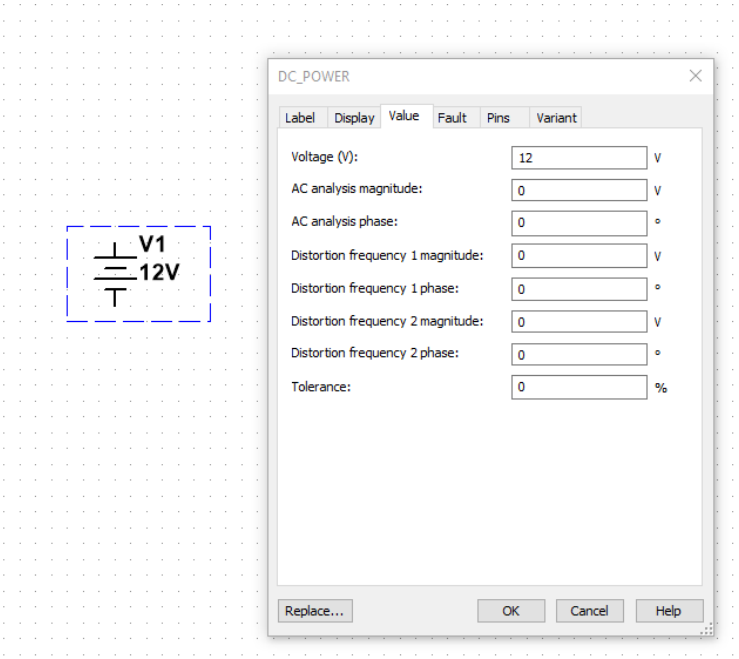
1. On the right side of the screen, locate the **Component Toolbar**
2. Click on **Place Source** (it looks like a battery symbol) or press `Ctrl+W`
3. In the dialog box:
   - Family: **POWER_SOURCES**
   - Component: **DC_POWER**
4. Click **OK**

5. Move your mouse to the workspace and click to place the voltage source
6. Press `ESC` to stop placing components



1.3 Step 3: Configure the Voltage Source

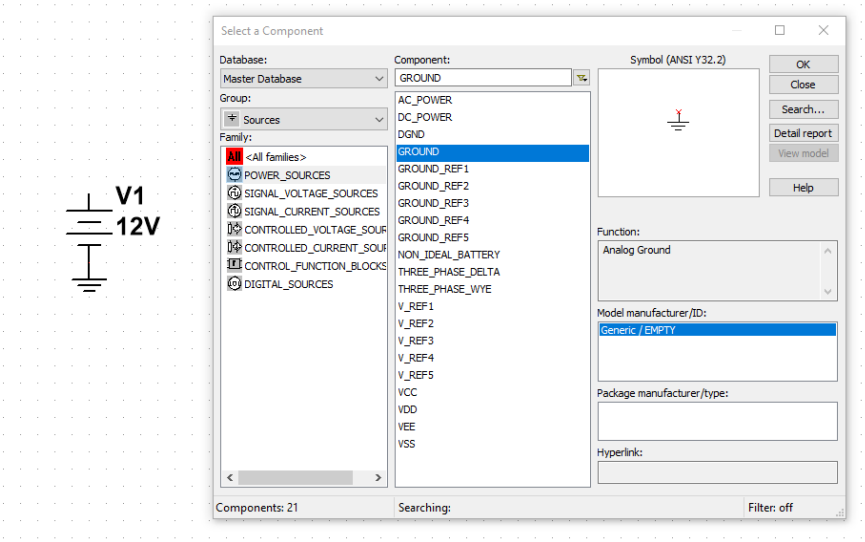
1. Double-click on the voltage source you just placed
2. In the properties window, set the **Voltage (V)** to `12V`
3. Click **OK**



1.4 Step 4: Place Ground

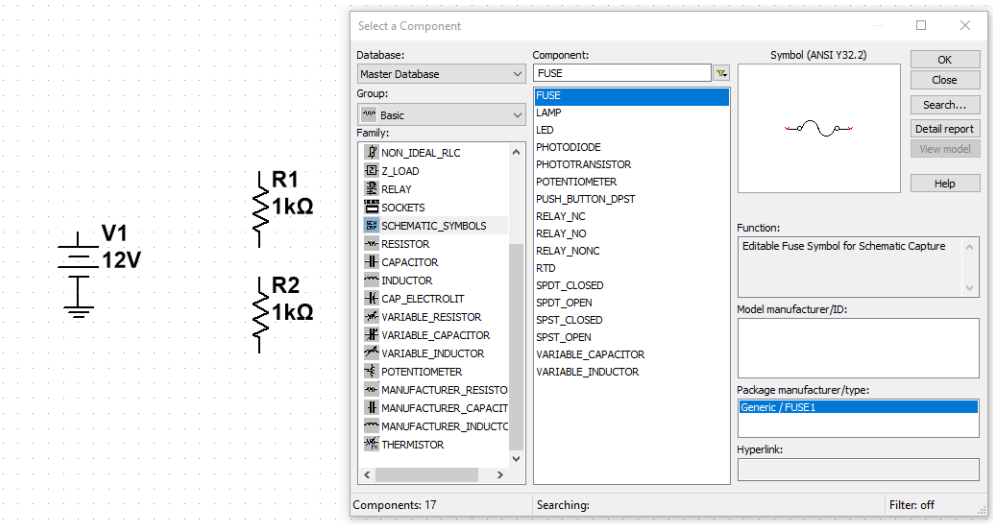
Every circuit needs a ground reference!

1. Click **Place Ground** button on the toolbar (or press `Ctrl+G`)
2. Click below the negative terminal of the voltage source to place the ground
3. Press `ESC`



1.5 Step 5: Place Resistors

1. Click **Place Component** (or press `Ctrl+W`)
2. In the dialog box:
   - Group: **Basic**
   - Family: **RESISTOR**
   - Component: Select any resistor (e.g., `1kΩ`)
3. Click **OK**
4. Place the first resistor to the right of the voltage source
5. Without pressing ESC, place a second resistor below the first one
6. Press `ESC` when done



1.6 Step 6: Configure Resistor Values

We'll create a voltage divider where $V_{out} = 4V$ when $V_{in} = 12V$.

1. Double-click on the top resistor (R1)
2. Set the **Resistance** to `2kΩ` (or `2000`)
3. Click **OK**
4. Double-click on the bottom resistor (R2)
5. Set the **Resistance** to `1kΩ` (or `1000`)
6. Click **OK**

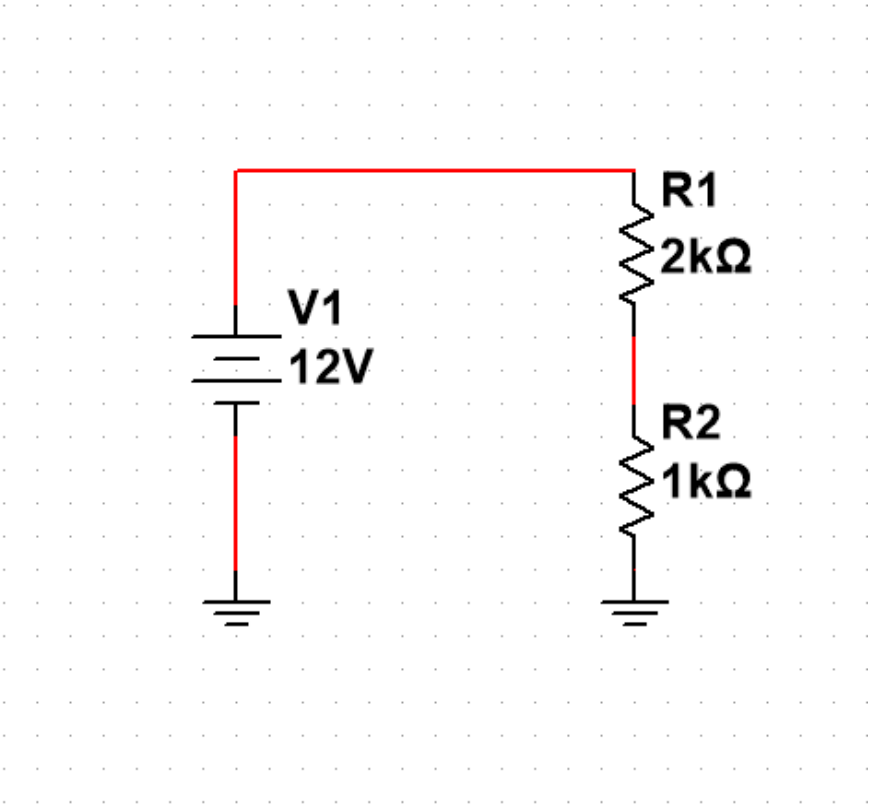
Circuit Theory Reminder

For a voltage divider: $V_{out} = V_{in} \times \frac{R_2}{R_1+R_2} = 12V \times \frac{1k}{2k+1k} = 4V$

1.7 Step 7: Wire the Components

Now we'll connect everything together.

1. Click on the **positive terminal** of the voltage source (a small circle/dot)
2. Move the cursor to the **left terminal** of R1 and click
3. A wire will be created automatically. Wires automatically route in straight lines. You can click multiple times to create corners if needed.
4. Connect the **right terminal** of R1 to the **left terminal** of R2
5. Connect the **right terminal** of R2 to the ground
6. Connect the ground to the **negative terminal** of the voltage source

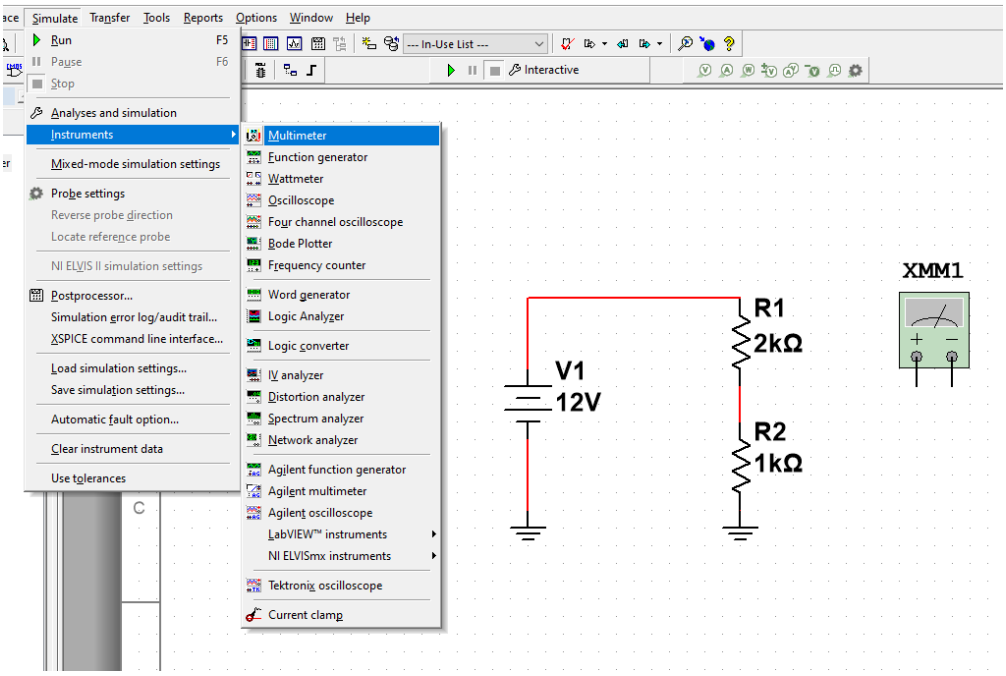


Your circuit should now look like a complete voltage divider with a 12V source, R1 (2kΩ), and R2 (1kΩ) in series.

1.8 Step 8: Add a Multimeter to Measure Voltage

Let's measure the voltage across R2 (this is V_{out}).

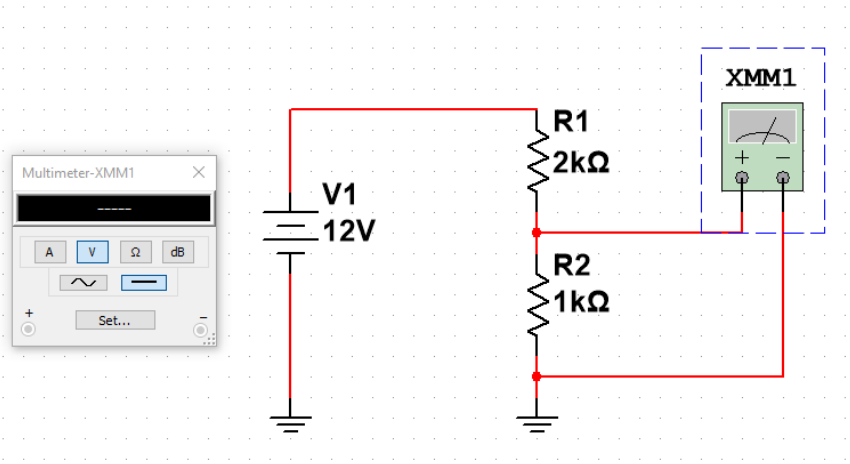
1. Click **Place Indicator** on the toolbar (or go to **Simulate** → **Instruments**)
2. Select **MULTIMETER**
3. Click **OK**
4. Place the multimeter on your workspace
5. Press `ESC`



1.9 Step 9: Connect the Multimeter

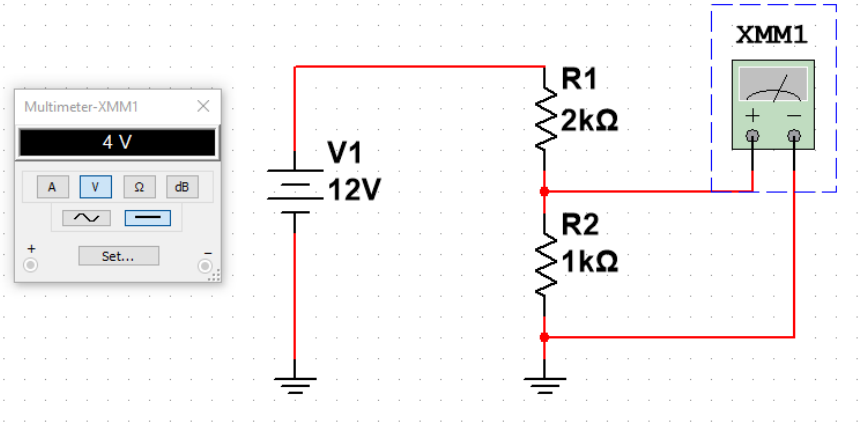
Double clicking the multimeter reveals options of A (ammeter), V (voltmeter), and Ω (ohmmeter).

1. Select the **V mode** of the multimeter to the node between R1 and R2 (where we want to measure V_{out})
2. Wire the **COM (-) terminal** to ground
3. Double-click the multimeter
4. Ensure it's set to measure **Voltage (V)** in DC mode (flat line)
5. Click **OK**



1.10 Step 10: Run the Simulation

1. Click the **Run** button (green triangle) at the top of the screen or press `F5`
2. The multimeter should display approximately `4.000 V`
3. Click the **Stop** button (red square) to stop the simulation



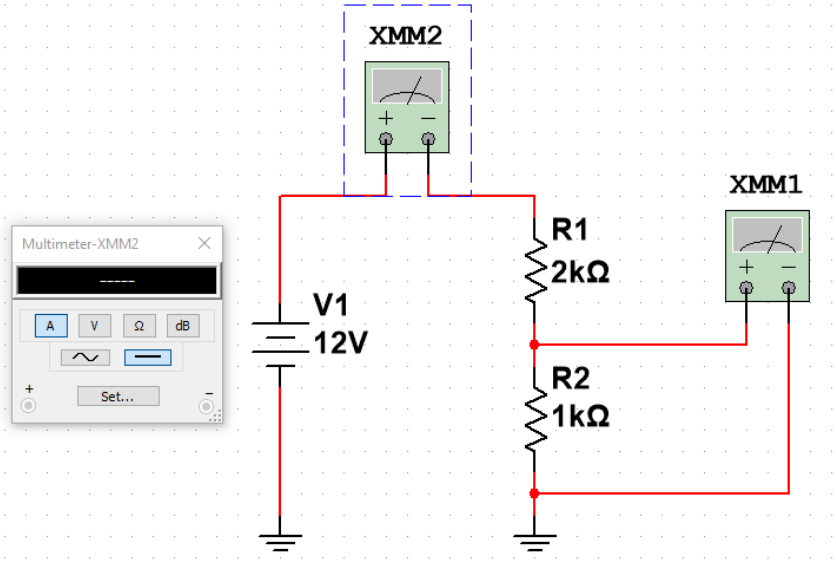
Success!

If you see 4V, congratulations! Your voltage divider is working correctly.

1.11 Step 11: Measure Current

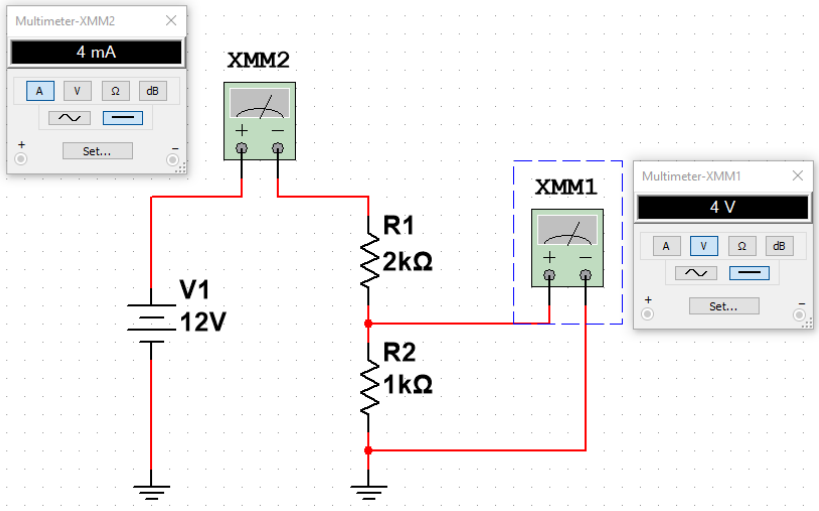
Now let's measure the current flowing through the circuit.

1. Stop the simulation if it's running
2. Place a second multimeter on your workspace
3. To measure current, we need to insert the ammeter **in series** with the circuit
4. Delete the wire between the voltage source and R1 (click on it and press `Delete`)
5. Wire the **positive terminal** of the voltage source to the **A terminal** of the multimeter
6. Wire the **COM terminal** of the multimeter to R1
7. Double-click the multimeter and set it to measure **Current (A)** in DC mode
8. Click **OK**



1.12 Step 12: Verify the Current Measurement

1. Run the simulation (press `F5`)
2. The ammeter should read approximately `4.000 mA`
3. This makes sense: $I = \frac{V}{R_{total}} = \frac{12V}{2k\Omega + 1k\Omega} = \frac{12V}{3k\Omega} = 4mA$



Success!

If you see 4mA, you've successfully measured the current in your circuit!

2 Conclusion

You now have the basic skills for DC circuit simulation in Multisim:

- Build circuits with passive components
- Configure component values
- Measure DC voltages and currents using multimeters
- Run DC simulations and verify results

These skills will be essential for completing your assignments and lab work. Always verify your Multisim results with hand calculations to ensure you understand the underlying circuit theory.