Multisim Transient Analysis Tutorial
Time-Domain Analysis of Passive Circuits

ECEN 222, Spring 2026

University of Nebraska-Lincoln
Maxx Seminario

Introduction

This tutorial introduces AC (alternating current) analysis in Multisim, focusing on analyzing passive circuits in the time domain. You'll learn to run a transient simulation in Multisim.

Prerequisites:

- Completion of Multisim DC Basics tutorial
- Understanding of phasor notation and complex numbers
- Familiarity with impedance concepts: $Z_R = R$, $Z_L = j\omega L$, $Z_C = 1/(j\omega C)$

Learning Objectives:

- Build AC circuits with sinusoidal sources
- Use the AC function generator to create AC signals
- Measure AC voltages and currents with the oscilloscope

1 Tutorial 1: Simple RC Circuit - Transient Analysis

We'll start with a series RC circuit driven by an AC voltage source to observe the transient response.

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 `AC_RC_circuit`

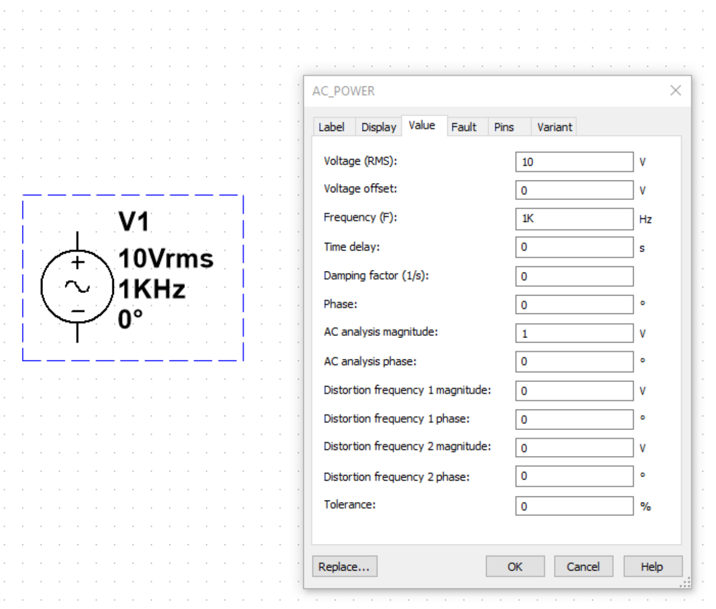
1.2 Step 2: Build the Circuit

Components needed:

- One AC voltage source (10V RMS at 1kHz)
- One resistor (1kΩ)
- One capacitor (100nF = 0.1μF)
- Ground

1.2.1 Place the AC Voltage Source

1. Click **Place Source** or press `Ctrl+W`
2. Family: **POWER_SOURCES**
3. Component: **AC_POWER**
4. Click **OK** and place it on the workspace
5. Double-click the AC source
6. Set:
   - **Voltage (RMS)**: `10` (10V RMS)
   - **Frequency (F)**: `1000` or `1k` (1 kHz)
   - **Phase**: `0` degrees
7. Click **OK**



AC Source Parameters

- **RMS voltage**: The root-mean-square voltage. For a sinusoid, $V_{RMS} = V_{peak}/\sqrt{2} = 0.707V_{peak}$
- **Frequency**: In Hz (cycles per second). Angular frequency $\omega = 2\pi f$
- **Phase**: Initial phase shift in degrees

For a 10V RMS source at 1kHz: $v(t) = 10\sqrt{2}\cos(2\pi \cdot 1000 \cdot t)$ V (peak = 14.14V)

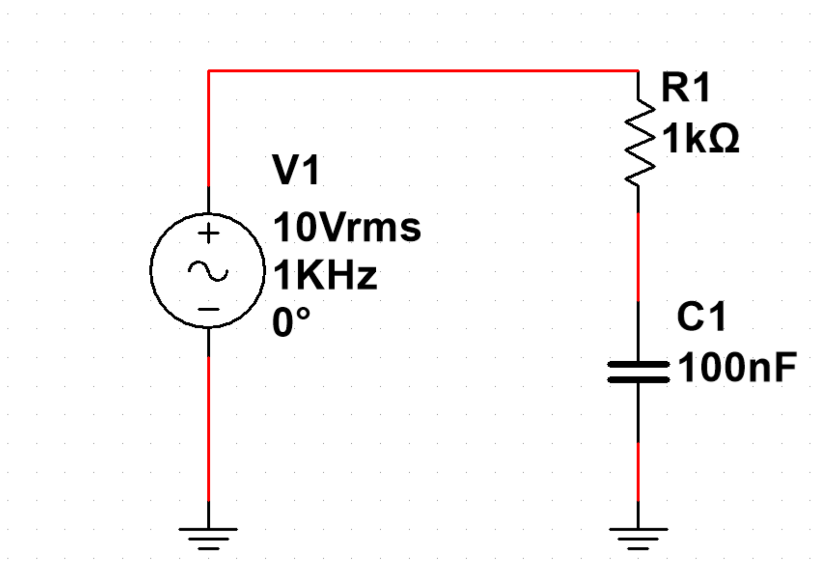
1.2.2 Place Resistor and Capacitor

1. Place a resistor (1kΩ) to the right of the AC source
2. Place a capacitor below the resistor
3. Double-click the capacitor and set it to `100nF` or `100e-9`
4. Place ground below the capacitor

1.3 Step 3: Wire the Circuit

Create a series RC circuit:

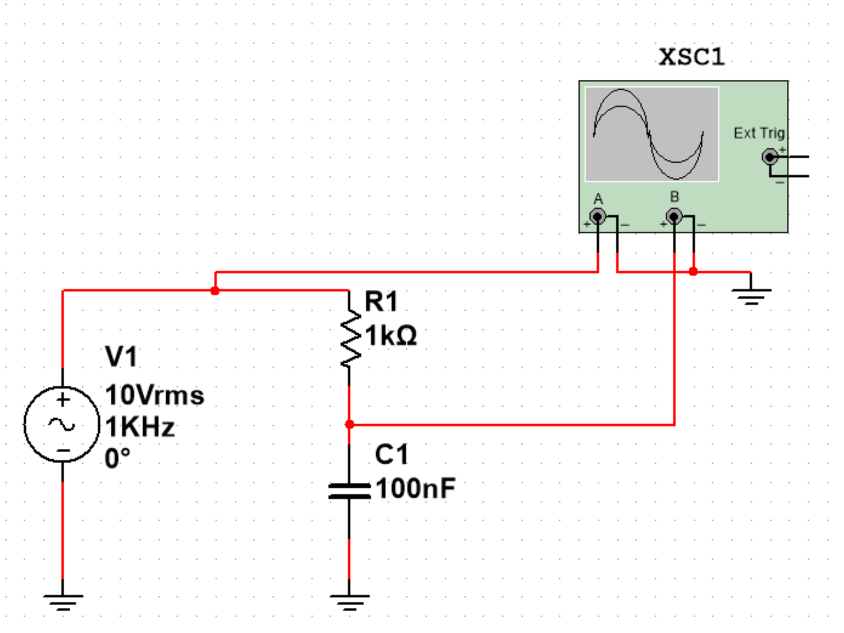
1. Connect the positive terminal of the AC source to the resistor
2. Connect the resistor to the top terminal of the capacitor
3. Connect the bottom terminal of the capacitor to ground
4. Connect ground back to the negative terminal of the AC source



1.4 Step 4: Add Oscilloscope for AC Measurement

1. Go to **Simulate** → **Instruments** → **Oscilloscope**
2. Select the **4-Channel Oscilloscope**
3. Place it on your workspace
4. Connect **Channel A** (positive) to the positive terminal of the AC source (input voltage)
5. Connect **Channel A ground** to circuit ground
6. Connect **Channel B** (positive) to the node between R and C (capacitor voltage)

7. Connect **Channel B ground** to circuit ground

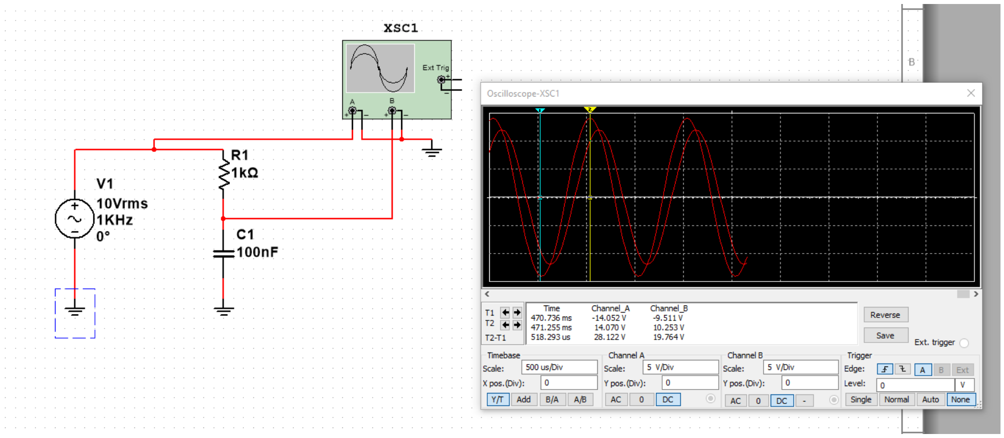


1.5 Step 5: Configure the Oscilloscope

1. Double-click the oscilloscope
2. Set **Time base**: 500μs/div or 0.5ms/div (to see about 2 cycles at 1kHz)
3. Set **Channel A scale**: 5V/div
4. Set **Channel B scale**: 5V/div
5. Set **Trigger**:
    - Source: Channel A (input signal)
    - Edge: Rising
    - Level: 0V
6. Enable both Channel A and Channel B

1.6 Step 6: Run the Simulation

1. Click **Run** (F5)
2. Observe both waveforms on the oscilloscope
3. Channel A (input): Shows peak voltage of $10\sqrt{2} \approx 14.14$V (remember: oscilloscope shows peak, source is set to 10V RMS)
4. Channel B (capacitor): Smaller amplitude sinusoid, phase-shifted
5. Notice the capacitor voltage **lags** the input voltage



1.7 Step 7: Measure Magnitude and Phase

Use the oscilloscope cursors to measure:

1. **Magnitude of V_C**: Measure the peak-to-peak voltage of Channel B and divide by 2
2. **Phase shift**: Measure the time delay between zero-crossings of the two signals
   - Place cursor T1 at a rising zero-crossing of Channel A (input)
   - Place cursor T2 at the next rising zero-crossing of Channel B (capacitor)
   - Time difference: Δt
   - Phase shift: $\phi = \frac{\Delta t}{T} \times 360$ where $T = 1/f = 1$ ms

1.8 Step 8: Verify with Theory

Calculate the theoretical values:

At $f = 1$ kHz, $\omega = 2\pi f = 2\pi(1000) = 6283$ rad/s

Capacitive reactance:

$$X_C = \frac{1}{\omega C} = \frac{1}{6283 \times 100 \times 10^{-9}} = \frac{1}{6.283 \times 10^{-4}} \approx 1592\,\Omega$$

Total impedance:

$$\mathbf{Z}_{tot} = R + \frac{1}{j\omega C} = 1000 - j1592 = 1880\angle - 57.9\,\Omega$$

Capacitor voltage (using voltage divider):

$$\mathbf{V}_C = \mathbf{V}_s \times \frac{\mathbf{Z}_C}{\mathbf{Z}_{tot}} = 10\angle 0 \times \frac{1592\angle - 90}{1880\angle - 57.9} = 8.47\angle - 32.1 \text{ V (RMS)}$$

Your oscilloscope should show:

- $|V_C| \approx 8.47\sqrt{2} \approx 12.0$ V peak (oscilloscope shows peak values)
- Or equivalently, 8.47V RMS if you convert the oscilloscope reading
- Phase lag ≈ -32 (capacitor voltage lags input)

Success

If your measurements match theory, you've successfully performed transient circuit analysis in Multisim!

2 Conclusion

You now have the skills to perform AC transient analysis in Multisim:

- Configure AC voltage sources with RMS amplitude, frequency, and phase
- Build series RC, RL, and RLC circuits
- Use oscilloscope to measure AC voltages and observe waveforms
- Measure magnitude and phase relationships between signals
- Verify theoretical impedance calculations with measurements

These skills are essential for understanding frequency-domain circuit behavior.