

Oregon State University

Analyzing Complex RLC Circuits:
Using Math, Simulation, and Tools to Solve Circuit
Problems

Noah Bean
WR 227
08/06/2023

Topic: Analyzing Complex RLC Circuits

Intended Audience: Undergraduate Electrical Engineering Students

Introduction:

Analyzing a circuit for various properties is an activity that all undergraduate Electrical Engineering students are expected to know how to do. A commonly used circuit type is an AC RLC circuit, which has applications for filtering signals for specific frequencies. These circuits use a voltage source that creates alternating current (AC) and contain any number of resistors (R), inductors (L), and capacitors (C) that are either in a series or parallel configuration with each other. These instructions explain how to analyze an RLC circuit for current and voltage by using mathematical theory, circuit simulation software, and testing a physical circuit.

These instructions are divided into 10 steps:

Analyzing the circuit with mathematical theory:

1. Draw and Label the circuit Schematic
2. Convert the circuit components into the corresponding impedance values
3. Write down a system of equations for solving the circuit
4. Use a calculator to solve the system of equations

Simulating the circuit using LTSpice:

5. Install LTSpice on a computer
6. Build and Label the circuit schematic
7. Simulate the circuit

Testing the physical circuit:

8. Purchase the necessary components
9. Build the circuit on a breadboard
10. Use a multimeter to test the circuit

Note: Calculating, simulating, and testing a circuit for voltage and current should result in the same answers, but undergraduate students must be able to use multiple techniques in order to be thorough in their analysis.

Caution: Building circuits can be a dangerous activity that could lead to serious injury or death. Always use proper safety equipment and good judgment when building a circuit.

Required equipment:

- Pen/Pencil
- Paper
- calculator
- resistor
- inductor
- capacitor

- multimeter
- AC power source
- computer with internet access
- LTSpice software
- Safety Equipment including safety glasses, gloves, and a fire extinguisher rate for electrical fire

Definitions:

Kirchhoff's Voltage Law: $\sum_{loop} \Delta V = 0$

Kirchoff's Current Law: $\sum I_{in} - \sum I_{out} = 0$

Ohm's Law: $V = IR$

Voltage: Electrical Potential Difference

Current: Change in position of a group of charged particles with respect to time

Impedance: A complex number that represents reactance and resistance in an electrical component

Inductor: An electrical component that stores energy in an electrical field

Resistor: An electrical component that dissipates energy

Capacitor: An electrical component that stores energy in a magnetic field

Complex Algebra: $j = \sqrt{-1}$, $(a + jb) * (c + jd) = (a * c) + j(b + d)$

LTSpice: A free circuit simulation software created by Analog Devices

Breadboard: A circuit prototyping tool that allows electrical components to be connected without using solder

AC power supply: A voltage source that creates alternating current

Multimeter: A tool that measures current and voltage in a circuit

[3], [4]

Analyzing the Circuit With Mathematical Theory:

In this section, mathematical theory will be used to predict how an RLC circuit should behave.

Using mathematical theory is often the least accurate method for determining the correct results,

but actually calculating the desired values provides design intuition for how the electrical components will relate to each other. This method requires the usage of a pen or pencil, and paper.

1. Draw and label the circuit schematic. Start this process by including a voltage source on the left side of the paper by drawing a circle with a sine wave in the center. Then draw a closed loop for the circuit by making a line that connects the top side of the voltage source to bottom of the voltage source in a clockwise arc. Make sure to leave enough room for the other circuit components in this closed loop. Then, add a resistor by drawing a “zig zag” on the loop, add an inductor by drawing a “spring” on the loop, and add a capacitor by drawing two parallel lines that are both perpendicular to the loop. Finally, add the corresponding values for each component by labeling the inductance of the inductor in units of Henries, the capacitance of the capacitor in units of Farads, the resistance of the resistor in units of Ohms, the voltage of the AC voltage source in units of volts, and the frequency of the voltage source in units of Hertz [3], [4]. Please see Figure 1 for more information.

Note: make sure to designate a part of the circuit where voltage = 0 and call this “ground”

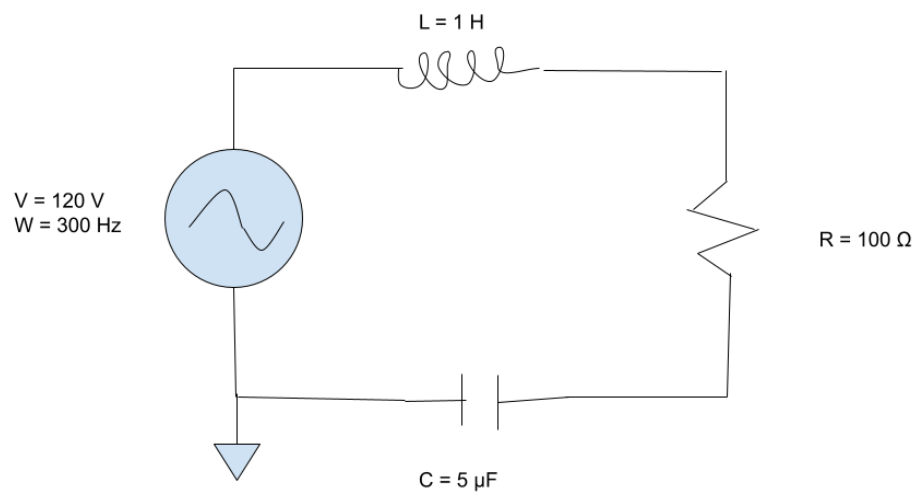


Figure 1: An RLC circuit with an inductor at the top, a capacitor at the bottom, a resistor on the right side, and a voltage source on the left side.

Alt text: Image of an electrical circuit drawing that includes a label for an inductor at the top with units of 1 Henry, a capacitor at the bottom labeled with 5 microFarads, resistor labeled with 100 Ohms on the right side, and an alternating current voltage source on the left side labeled with 120 volts and 300 Hertz. A ground node has also been designed at the bottom left side.

2. Convert the circuit components into their corresponding impedance values. Resistors are passive circuit components that do not change the phase of the circuit, so they only have resistance (R) values. However, capacitors and inductors are active circuit components so they both have a reactance (X) and change the phase of the circuit. In order to compare capacitors, inductors, and resistors, a new value must be used that combines reactance and resistance and this value is called impedance. Impedance can be described using the mathematical relationship:

$$Z = R + X$$

Capacitor, Inductor, and resistor impedance can be found by using the mathematical relationships:

$$X_C = \frac{1}{j\omega C}$$

$$X_L = j\omega L$$

$$X_R = R$$

Where X_C is the impedance of a capacitor, X_L is the impedance of an inductor, X_R is the impedance of a Resistor, j is the imaginary number, C is the capacitance, L is the inductance, R is the resistance, and ω is the angular frequency of the AC voltage source [3], [4]. Please see Figure 2 for more information.

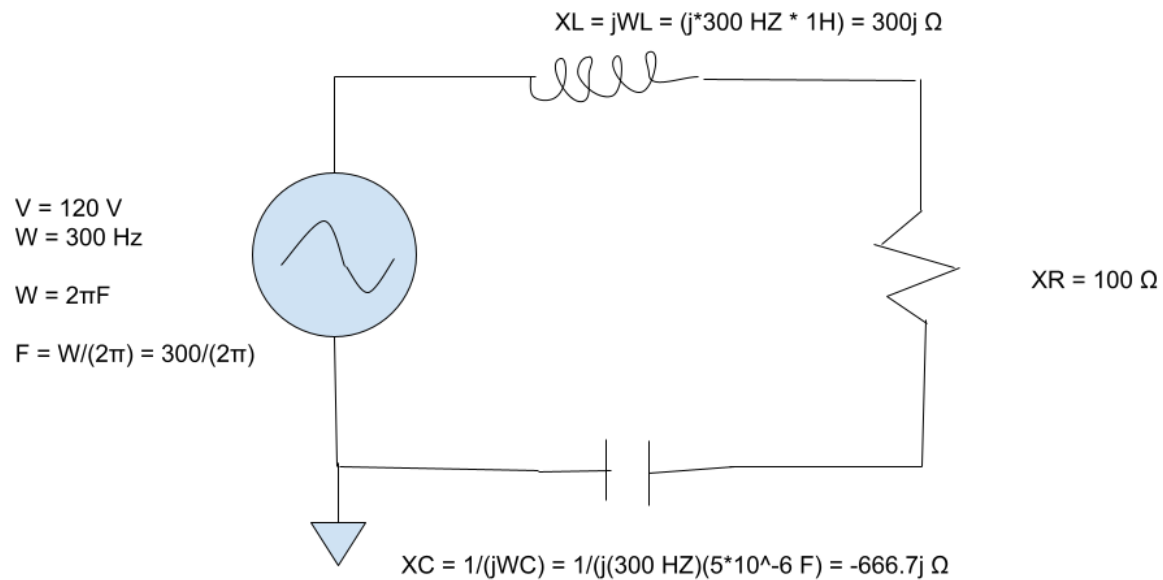


Figure 2: An RLC circuit with labeled impedance values for a resistor, capacitor, and inductor.

Alt text: Image of an electrical circuit with the corresponding impedance values for a resistor on the left side, a capacitor on the bottom, and an inductor on the top as well as an AC voltage source.

- Write down a system of equations for solving the circuit. The relationship between the circuit components can be described by using Kirchhoff's voltage law and Kirchhoff's current law. These equations are given by:

$$\text{Kirchhoff's Voltage Law: } \sum_{\text{loop}} \Delta V = 0$$

$$\text{Kirchhoff's Current Law: } \sum I_{\text{in}} - \sum I_{\text{out}} = 0$$

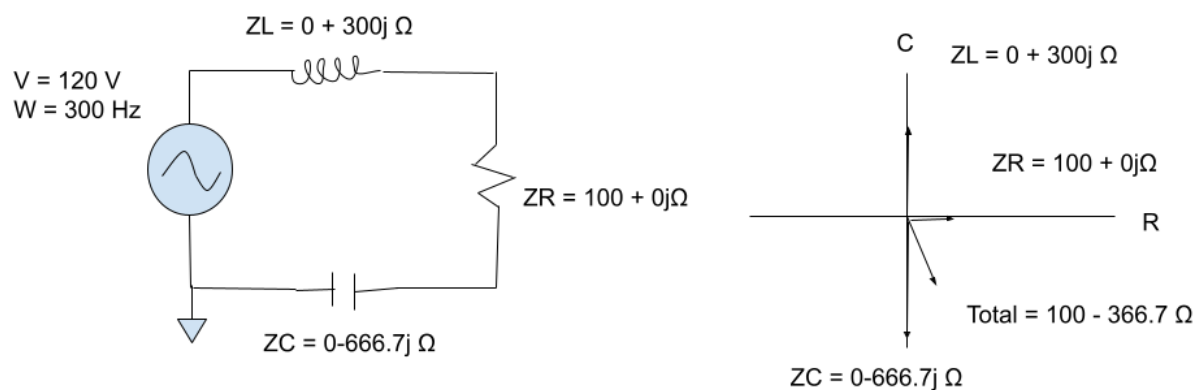
Kirchhoff's Voltage Law states that the total voltage is conserved in a loop, while Kirchhoff's current law says that current is conserved in a node. In a circuit with a single loop, the voltage drop across the sum of all components in the loop will be equal to the voltage provided by the source. In a circuit with a single loop, the current flowing through each component must be the same. The total impedance of a circuit with all of the components in series can be obtained by adding the impedance of each separate

component. Once the total impedance is obtained, the magnitude of the impedance and the phase angle of the impedance can be obtained by using the trigonometric formulas:

$$|Z_{total}| = \sqrt{R^2 + X^2}$$

$$\theta = \tan^{-1}\left(\frac{X}{R}\right)$$

Note: Complex numbers can be treated like vectors by applying the usual vector rules [3], [4]. Please see Figure 3 for more information.



$$Z_{total} = Z_L + Z_C + Z_R = (0 + 300j \Omega) + (0 - 666.7j \Omega) + (100 + 0j \Omega) = 100 - 366.7 \Omega$$

$$Z_{\text{magnitude}} = \sqrt{R^2 + X^2} = \sqrt{(100)^2 + (-366.7)^2} = 380.091 \Omega$$

$$Z_{\text{angle}} = \theta = \tan^{-1}(X/R) = \tan^{-1}(-366.7/100) = -74.75 \text{ degrees}$$

Figure 3: Mathematical equations for finding the total impedance for an RLC circuit with a single loop.

Alt text: Figure of a mathematical method for finding the total impedance vector of an RLC circuit in a loop by adding the impedance vectors of each individual component and then finding the magnitude and angle of this impedance.

4. Use a calculator to solve the system of equations. Once the total impedance has been calculated, Ohm's law can be used to find the total current in the loop and the voltage drop across each capacitor. Ohm's Law can be described by:

$$V_{RMS} = I_{RMS} * Z_{total}$$

Ohm's law states that voltage amplitude is equal to the product of the current amplitude and the total impedance. The voltage drop across each component can be calculated by again using Ohm's law to find the voltage, given the impedance for each component and the total current [3], [4]. Please see Figure 4 for more information.

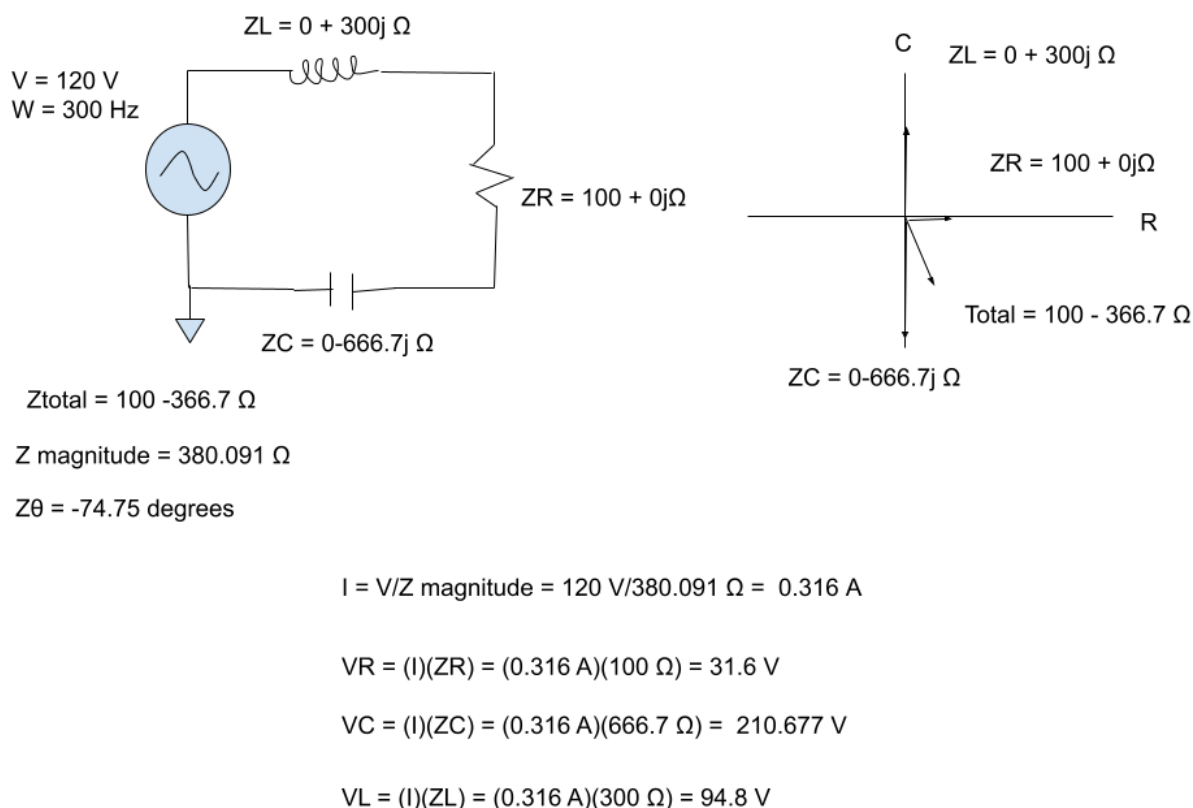


Figure 4: Calculation to find the total current in an RLC circuit with a single loop and the voltage drop across each component.

Alt text: Image of a calculation to find the total current in a circuit and the voltage drop across each component by using Ohm's Law.

Simulating The Circuit Using LTSpice:

In this section, a computer that can connect to the internet will be used to run LTSpice software in order to predict the behavior of an RLC circuit. Using software is generally the quickest way to solve the circuit, and the results are more accurate than hand calculations. However, the calculations are not as accurate as physically testing the circuit and the behavioral intuition of solving the equations is dampened.

- Install LTSpice on the computer. First, go to the Analog Devices website where the LTSpice software can be found (<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>). Then, download the correct version of the software for your computer's operating system. After opening the installation software, a series of prompts will appear. Select "next" on the setup wizard opening page. Then agree to the user agreement. Next, select the file location for the software to be installed. After that, wait for the software to install. Finally, Select the option to launch LTSpice and the setup wizard will close. The LTSpice software should open up with a blank page [1]. Please see Figure 5 for more information.

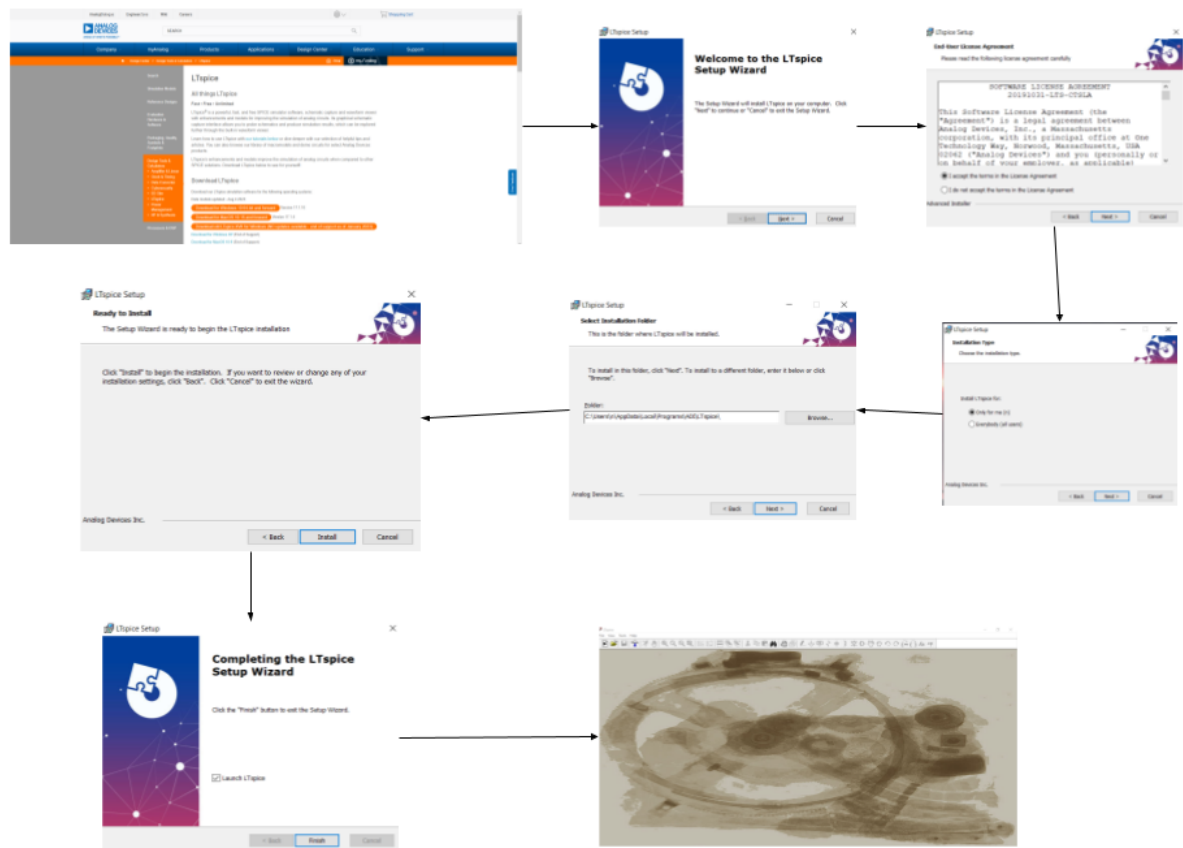


Figure 5: The LTSpice software installation process.

Alt text: Image of the LTSpice software installation starting with downloading the software from the download side and ending with opening the software to the home page.

- Build and Label the circuit Schematic. In the top right corner, select "File", then select "New Schematic". This will open up a new schematic page that has a gray background. Copy the schematic onto this page by finding the correct components at the top of the screen or using the correct shortcut. The resistor can be obtained by selected the 'zig zag' button or using the shortcut 'r', the inductor can be obtained by selected the 'curly'

button or using the shortcut 'L', and the capacitor can be obtained by selected the 'parallel line' button or using the shortcut 'C'. The voltage source can be obtained by clicking on the 'component' button (shortcut 'F2') and typing 'voltage' into the search bar. Once all of the components have been placed, they can be wired together using the 'pencil' button or the shortcut 'F3'. Next, each component should be labeled with the corresponding value. Right clicking on the component will bring up a menu to customize the values. To customize the voltage source right click on the component, go to the 'advanced' section, change the function to a "SINE" wave, and enter the correct voltage amplitude in 'Amplitude[V]' and the correct frequency in 'Freq[Hz]' [1]. Please see Figure 6 for more information.

Note: Do not forget to include Ground (shortcut 'g')

Note: The components can be rotated with shortcut 'ctrl-r'

Note: Units with a 'micro' prefix can be named by adding a 'u' and units with a 'nano' prefix can be named by adding a 'n'

Troubleshooting: Do not forget to convert radial frequency into frequency using the relationship $f = \omega/2\pi$

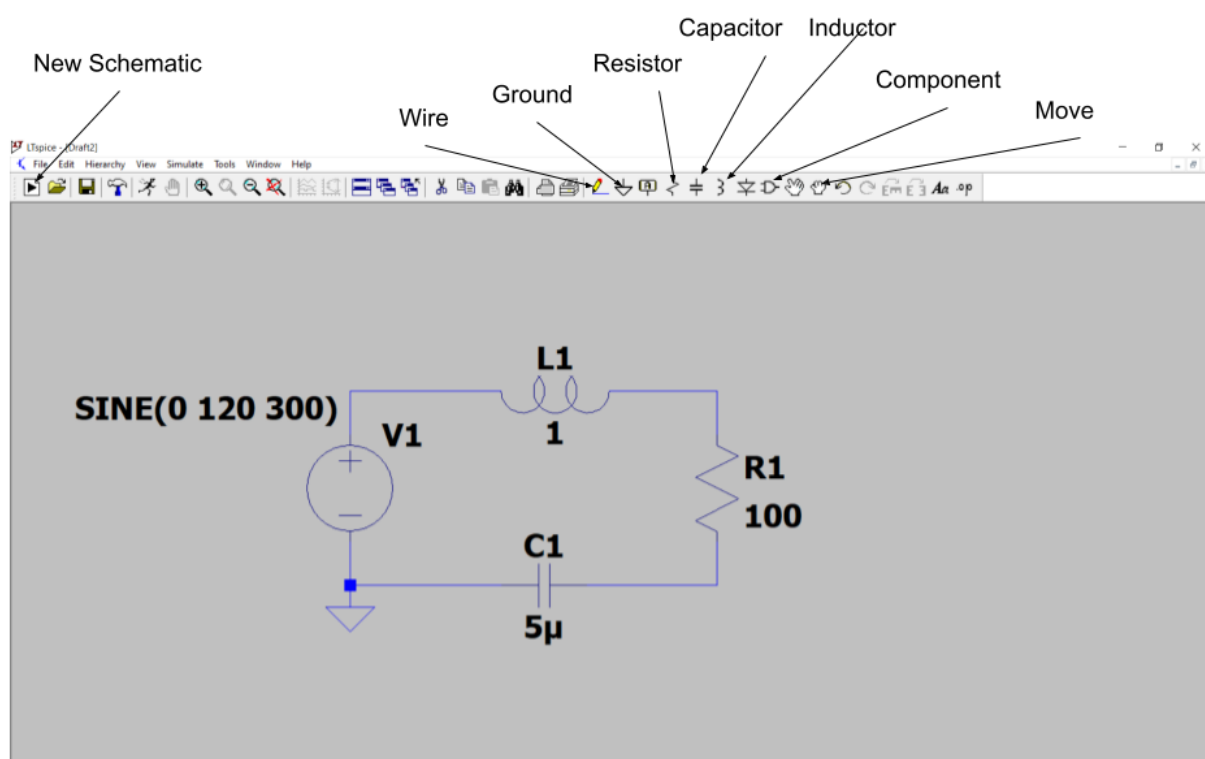


Figure 6: A schematic for an RLC circuit built using the LTSpice software

Alt text: Image of a schematic for an RLC circuit built using the LTSpice software with the necessary buttons highlighted.

7. Simulate the circuit. Once the circuit schematic is built, click on the 'simulate' button at the top right of the screen and this will open up a new window to edit the simulation commands. Click on the 'Transient' tab and enter a 'stop time', 'time to start saving data', and a 'maximum timestep'. Once the simulation commands have been set, click on the 'run' button under the simulate tab at the top right of the screen. The simulation results will appear at the top of the screen above the circuit schematic [1]. Please see Figure 7 for more information.

Note: Each entry in the simulation command window must be greater than 0.

Troubleshooting: Make sure to zoom in on the simulation output to see the best image results.

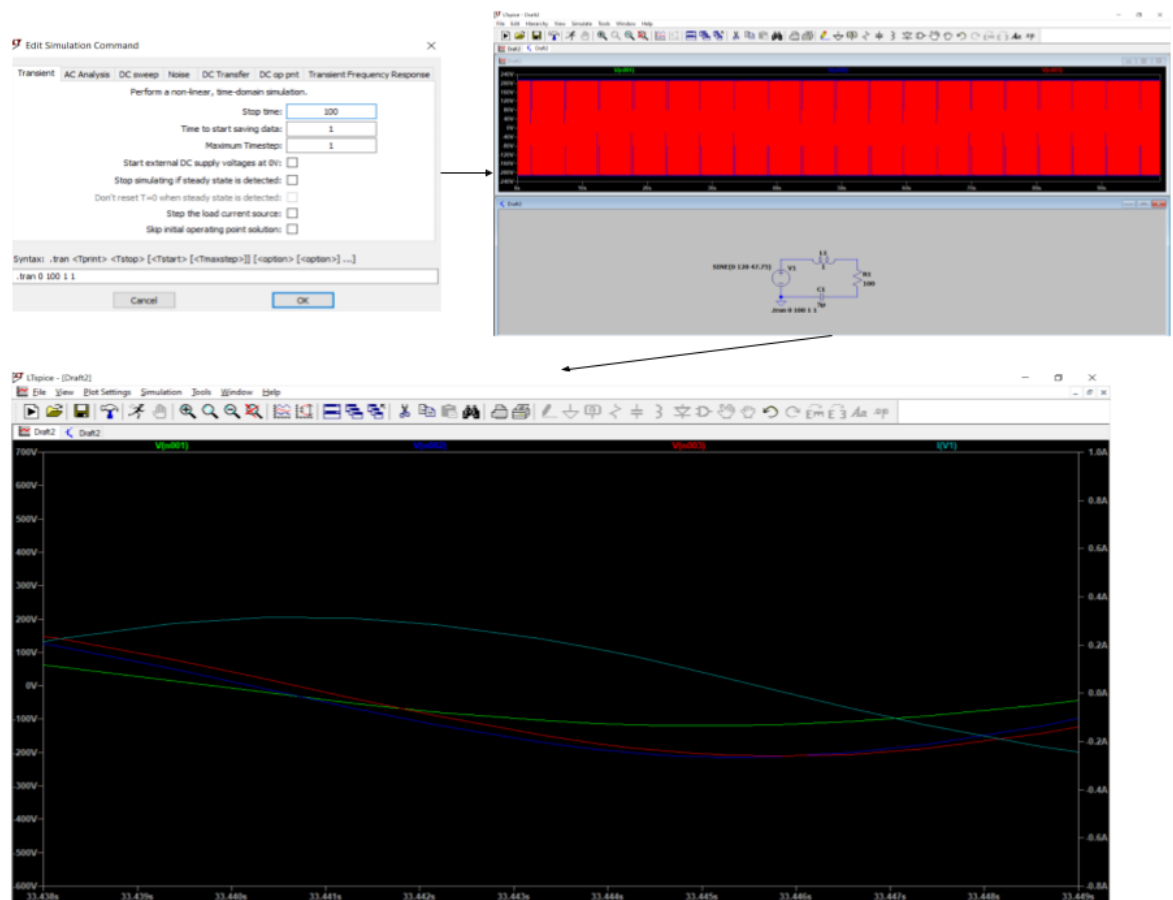


Figure 7: The process for simulating an RLC circuit using LTSpice software.

Alt text: Image of the process for simulating an RLC circuit using the LTSpice software by setting the stop time to 100, the time to start saving data at 1, and the maximum timestep at 1, with the final simulation results zoomed in.

Testing the physical circuit:

In this section, hand tools and electrical components will be used to physically test the circuit. Testing each component is certainly the most accurate method for finding the desired values, but it is also the most expensive and time consuming technique. Testing a physical circuit is also the least intuitive method because the circuit acts like a 'black box' with all of the variable relationships hidden from the student.

8. Purchase the necessary components. First, go to a website where electronic components are sold. A commonly used website for purchasing electronic components is DigiKey (<https://www.digikey.com/>). Add the necessary components to the shopping cart and pay for the order. The order should arrive with shipping time between a week and a month, depending on the rarity of the component.

Troubleshooting: If a specific component size is not available to purchase, the correct size can be made by adding different components in either series or parallel to get the desired result.

9. Build the circuit on a breadboard. Once the components have arrived, construct the circuit prototype by placing each component on a breadboard with the correct connections. Please see Figure 8 for more information.

Warning: Constructing a circuit can be dangerous and life threatening. In order to maximize safety, wear safety glasses and gloves when handling the circuit components. Also, make sure to have a fire extinguisher nearby that is rated for electrical fires.

Note: A breadboard has two vertical parallel rails on the left and the right sides that should be denoted as ground and voltage, while the horizontal parallel lines in the center should be used to make connections between wires and components [2].

10. Use a multimeter to test the circuit. Once the breadboard prototype has been made, connect the AC power supply to the voltage strip and the ground strip. Then, connect the Oscilloscope multimeter to each component by placing the red lead on one side of the component and the black lead on the other side of the component. Finally, record the voltage and current for each part. Please see Figure 8 for more information.

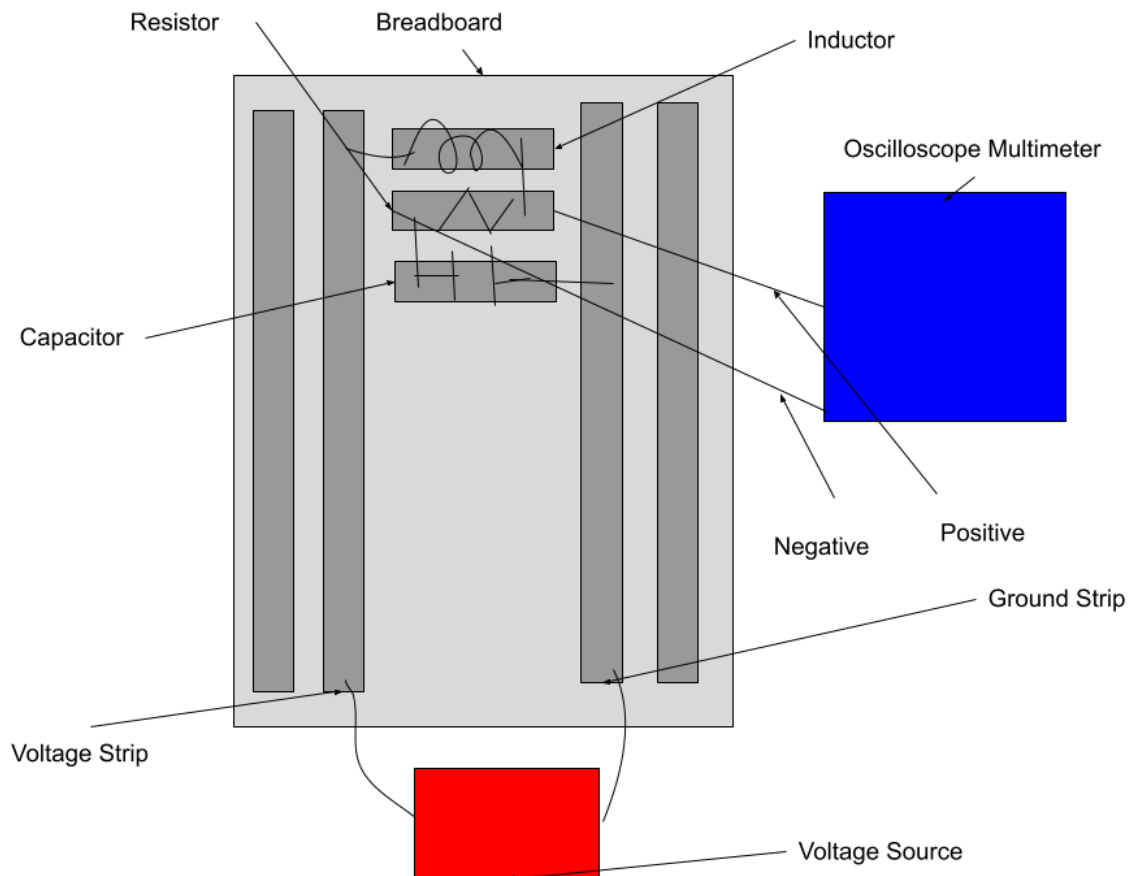


Figure 8: A breadboard prototype of an RLC circuit with a voltage source and oscilloscope connected.

Alt text: An image of a diagram for how to configure an RLC circuit on a breadboard and how to connect a voltage source to the breadboard and an oscilloscope to a circuit component.

Conclusion: Analyzing circuits is a very important skill for undergraduate Electrical Engineering students. There are multiple different ways to find the current through a component or voltage drop across a component, including using mathematical theory, simulation software, and using hand tools. Each technique has its own strengths and weaknesses, so choosing the correct method for the given constraints in the problem will make life much easier for an engineering student.

References:

1. Bramble, S. (n.d.). *LTspice tutorial: The complete course*. LTspice Tutorial | The Complete Course. http://www.simonbramble.co.uk/lt_spice/ltspice_lt_spice.htm [Accessed August 06th 2023]

2. Shams, & Instructables. (2017, October 31). *How to use a Breadboard*. Instructables. <https://www.instructables.com/How-to-use-a-breadboard/> [Accessed August 06th 2023]
3. Ulaby, F. T., Maharbiz, M. M., & Furse, C. (2018). *Circuit analysis and Design*. Michigan Publishing. [Accessed August 06th 2023]
4. YouTube. (2013). *Physics 49 RCL Circuits (1 of 2) Reactance and Impedance explained example*. *YouTube*. Retrieved August 6, 2023, from https://www.youtube.com/watch?v=IVS-vjHdC1c&t=5s&ab_channel=MichelvanBiezen [Accessed August 06th 2023]