Chapter 3 Lab Writeup

Zach Thompson et all

February 9, 2017

Overview

The purpose of this lab is to evaluate non-ideal components and to simulate them using LTSpice. This is necessary because components in real life do not have ideal properties and are not perfect. This impacts the circuit and is necessary to understand how to simulate. In this lab we evaluate how the given value of a resistor varies from it's measured value as well as how other components vary in measurement from their given values.

Process

In order to achieve the purpose of this lab several tools were used. The tools used include an oscilloscope, a function generator, a voltage source, a digital multi-meter, and LTSpice. LTSpice was used to simulate the circuits and the rest of the tools were used to measure a physical circuit.

Several circuits were analyzed. First a resistor network voltage divider with a DC source was analyzed, then again with an AC source. After that an LR circuit was analyzed.

Results

Here are the results for the various experiments that were conducted in this lab.

Non-Ideal Resistors

For the given circuit an output voltage was found for the voltage divider. The output voltage was found to be 2.5V. Then the given circuit was constructed with non-ideal components. For 100Ω resistors the resistance tolerance was found to be within 5Ω of the given resistance. With the circuit constructed from non-ideal components the output voltage was found to be 2.498V which was very close to the ideal voltage.

Non-Ideal Voltage Source

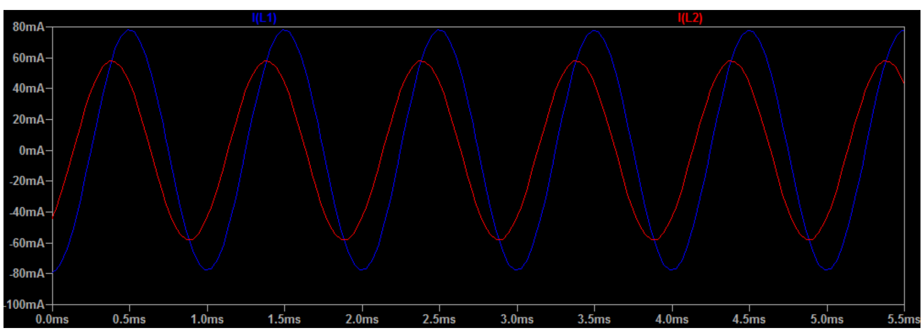
For this part of the lab an AC circuit with a resistive load was analyzed. The circuit was setup with two different loads and measured. For both circuits the amplitude of the voltage source was measured to be 9V. The voltage was then measured across a $1k\Omega$ resistor and was found to be 8.4V. The resistor was then replaced with a 220Ω resistor and the voltage was found to be 7.2V.

Using the recorded values and the given equations several properties of the voltage source could be found. R_{out} was found to be 77.39Ω. With R_{out} we were then able to calculate X_{out} to be $j113.181$. Now the total impedance of the function generator could be found as $77.39 + j113.181\Omega$ in rectangular form and $137.11\angle 55.64°$ in polar form.

Other Non-Ideal Components

An inductor was inspected next. The resistance of a 10mH inductor was measured with a digital multimeter and the equivalent series resistance was found to be 22.17Ω.

Modeling Non-Ideality in LTSpice



Conclusion

In this lab we explored the difference between modeled and real circuits. Tolerance is the aspect of this difference applied to the component level. Tolerances scale by percentages of total value, so variance in output is scaled by the value of the component. While modeled circuits are useful tools in predicting real outcomes, we must take care to not take these models as absolutes, but rather as an approximate guide to circuit design and development.

Study Questions

2. Calculate Peak instantaneous and RMS power dissapated from a 220Ω() resistor supplied by a 9V 1KhZ signal:
$v(t) = 4.5cos(2000pit)V = 4.50\angle 0° V$
$i(t) = V(t)/Z = (4.5 < 0)/220 = 0.02\angle 0° A$
$P(inst.) = (4.5)(0.02) = 92mVA$
$P(real) = (1/2)(4.5)(0.02) = 46mW$

3. A few examples of when it would be important to account for the non-ideal characteristics in your circuit design:
-When you expect a current of voltage value to be zero i.e. across a bridge of two similar paths
-When using sensitive equipment it is important to not overload components
-When your circuit is expected to perform over a variety of supplied voltage of current, it is important to understand how tolerances might scale