

Chapter 3 Lab Writeup

Zach Thompson et al

February 8, 2017

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 its measured value as well as how other components vary in measurement from their given values.

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.