Matthew Spencer – Fall 2020

Lab 1: Welcome to ltSpice

In this lab you are going to install your simulator for this course and familiarize yourself with different kinds of simulation.

After this lab, you will be able to:

1. Run ltSpice on your computer
2. Carry out common simulations (DC, AC, transient) in a circuit simulator.
3. Describe the assumptions made in common types of circuit simulation.

Practical Questions

1. Find an example of a commercial (ie: costs money) RF circuit simulator
2. Find an example of a commercial field solver

Theory Questions

1. What's the difference between a circuit simulator and a field solver?
2. What assumptions does the simulator make during a DC simulation? Do capacitors matter?
3. What assumptions does the simulator make during an AC simulation?
4. What assumptions does the simulator make during a transient simulation?

We haven't talked about these in class, so dig on the internet and be ask questions in the Discord.

Lab Instructions

1. Install ltSpice from this link. https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html#
2. Use ltSpice to make a resistor divider between a 10kohm and a 20kohm resistor such that the output is 2/3 of the input. Add a 1µF capacitance between the output node and ground.
3. Carry out the following simulations and compare the results to hand calculations. That comparison can consist of identifying corner frequencies, describing the shape of curves, or many other possibilities.
    a. A DC operating point simulation.
    b. A DC sweep where the lower resistor is swept from 5kOhm to 30kOhm.
    c. An AC simulation that captures 2 decades around the corner frequency.
    d. A transient simulation of a step response.
4. Find a website with a tutorial on how to do S parameter simulations. Follow the instructions and include the result. No need to know what the S-parameter results means yet, this is just practice digging up information and debugging.

Required data in lab notebook: Schematic of divider, each simulation result in section 3, calculations showing simulations match experiments in section 3, schematic and results from section 4.