Lab 1: Welcome to ltSpice & RF Lab Cert

In this lab you are going to install your simulator for this course, familiarize yourself with different kinds of simulation, and go through training to access the RF lab.

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.
4. Describe the three common ways to break RF equipment
5. Use a torque wrench to connect a SMA cable to a SMA receiver

We haven't talked about everything you need to know for practical questions below in class. That's normal, and I expect you to search the internet as part of this assignment. Doing such a search is useful practice for teaching yourself tricky engineering stuff in the future.

Practical Questions

1. Find an example of a commercial (ie: costs money) RF circuit simulator
2. Find an example of a commercial field solver
3. Look up the cost of a SMA cable (both cheap and fancy).
4. Find a picture of a failing SMA cable. Annotate how you can tell it's failing.
5. What is a "mating plane" when discussing electrical cables?
6. Find a picture of the inside of a SMA cable. Why is the shield of the SMA cable there?
7. What's the difference between a circuit simulator and a field solver?
8. What assumptions does ltSpice make during a DC simulation? Do capacitors matter?
9. What assumptions does ltSpice make during an AC simulation?
10. What assumptions does ltSpice make during a transient simulation?

Theory Questions

1. From class, what are the three major ways to damage RF equipment?
2. What is the power dissipated in a 50 Ohm resistor (which is what most of our instruments look like) if it is driven by each of the following voltages? Express answers in both mW and dBm.
    a. a 1Vpp sinusoid with 0VDC offset
    b. a 1Vpp sinusoid with 1VDC offset
    c. a 1Vpp square wave with 0VDC offset
3. Imagine 0dBm power is being dissipated in a load. First, express the load power in mW. Second, express the power level in dBm if the load power were multiplied by the following factors: 2x, 4x, 8x, 0.5x, 10x, 100x, 200x, 0.1x . Logarithmic math is your friend here, and this problem may go faster if you look for additive patterns in the results.

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 divider output node and ground. Add a voltage source between the divider input and ground and give it a DC value of 1V, an AC amplitude of 1V and an appropriate transient behavior based on question 3.
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 resistor between out and ground steps 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. Assume port 1 is between the in node and ground and port 2 is between the out node and ground. Add loads as appropriate. No need to know what the S-parameter results means yet, this is just practice digging up information and debugging.
5. Follow the instructions here to get RF lab certified: https://pages.hmc.edu/mspencer/rf_lab/

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.