PROJECT 1 – PART 1
ECE-723 Spring 2011

Description

In the first part of Project 1 you will simulate the Voltage-Controlled Oscillator (VCO) from Figure 10.21 using LTSpice. LTSpice is already installed in the computer cluster room. You may want however to download and install LTSpice on your own computer. Here is the link:
http://www.linear.com/designtools/software/index.jsp

To learn how to use LTSpice read the provided starter and user guides. In addition, you may also read online tutorials (links are provided on the website of this class).

When implementing your circuit, you will search and find from the Linear Technology models library the Op Amp, the Comparator, and the transistor that best match the components from the schematic diagram of Fig. 10.21. Your simulation should be for the range 1V…10V of v_I (with increments of 1V).
The results of your simulation will be later compared to your experimental measurements on the hardware prototype of the same circuit (to be done in part 2 of this project).

Delivarables

10 Points: Print out of your own schematic diagram (**not** a .txt Spice netlist and **not** a scanned version from textbook).
20 Points: Print outs of the triangular and square waveforms for v_I = 1V, 5V, and 10V.
20 Points: Written report of about one page or more with a discussion of: (i) LTSpice as a simulator and (ii) your simulation results. This discussion should contain a table with values of the output frequency for integer values of v_I within the range of interest.

NOTE: *The due date will be discussed in class.*