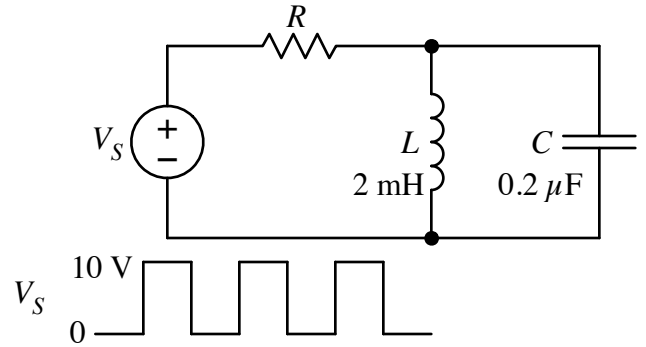


The RLC circuit at right is driven by a square-wave source. The source waveform is low at 0 V for 1 ms and then high at 10 V for 1 ms. ($T = 2$ ms, $f = 500$ Hz.)

Use a transient simulation in SPICE to plot: a) the inductor current, b) the capacitor current, c) the resistor current, and d) the voltage across the inductor-capacitor parallel combination.



Run the simulations and make plots for two different values of R : 25 Ω and 200 Ω .

Run the simulation for at least 3 periods of the source waveform. You may need to read up on how to run transient simulation using the VPULSE source. In submitting your work, you should turn in two sheets of paper. Each sheet should have the SPICE schematic and the four plots corresponding to one value of resistance. With a small amount of effort, you should be able to copy and paste the pieces into a single word processor page, resizing and arranging so that the plots are legible and everything is clearly labeled.