CENG 215 Circuits and Electronics

LAB #7 Feuille

Place: PC Lab

Aim

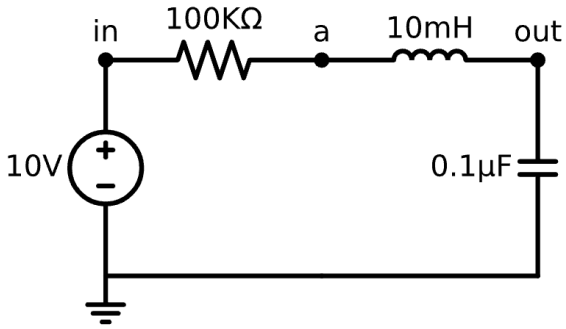
To build and analyze first and secont order networks in PySpice and to compare the analysis results with the theoretical analysis results.

Materials/Devices:

PySpice

Work to be done:

The following circuit is a second order R-L-C circuit. Assume that the capacitor and inductor are initially discharged and 10V voltage source is applied at time zero.



1. Simulate the circuit and plot the capacitor voltage $V_{out}(t)$ and the current.

   (Use the following voltage source. You can change the parameters passed if necessary)

```
source = circuit.PulseVoltageSource('input', 'in', circuit.gnd,
                            initial_value=0@u_V, pulsed_value=10@u_V,
                            pulse_width=100@u_ms, period=200@u_ms)
```

2. Change the resistor value to observe the following characteristics of the circuit. Plot the output voltage for each case:
   a. Over-damped
   b. Critically-damped
   c. Under-damped

Final Remarks

-