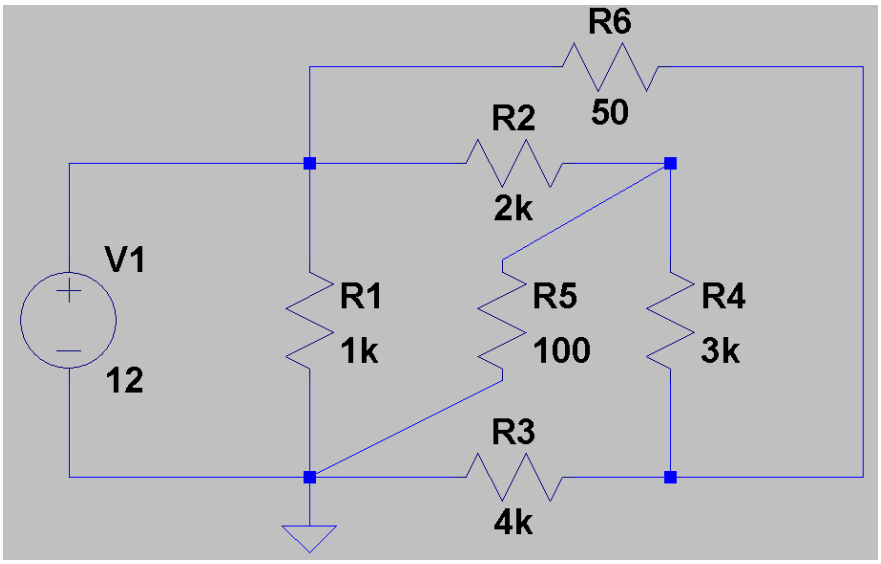
XMUT203 Analogue Circuits and Systems

Laboratory 1: LTSpice (Introduction)

Due date: Friday, 29th September 2023 (to Co-teacher)

Exercise 1: DC Operating Points Simulation

In this first exercise, we will find DC operating points for a very simple resistive bridge circuit given below using LTSpice.



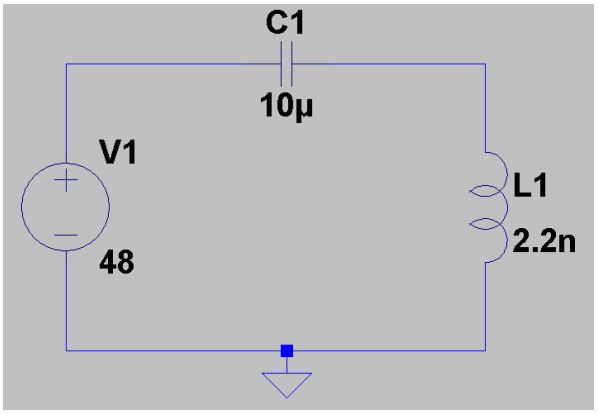
a. Construct the circuit given above as a schematic in LTSpice. [10 marks]

b. Calculate and perform *DC Operating Point* simulation of the circuit. [20 marks]

- All voltages in the circuit.
- All currents in the circuit.
- All powers generated by the voltage source and dissipated in the components.

Exercise 2: Transient Analysis Simulation

In this second exercise, we will carry out transient analysis for a very simple series RC circuit given below using LTSpice.



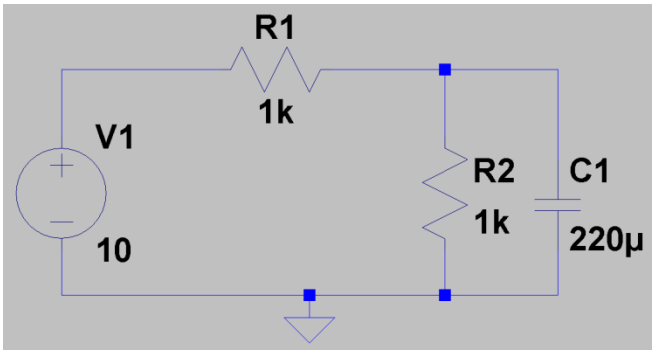
a. Construct the above given circuit as a schematic in LTSpice. [10 marks]

b. Perform *Transient Analysis* simulation of the circuit (Stop Time = 1 s; Maximum Step = 0.0001). Create plots of the following: [20 marks]

- All voltages across the devices in a graph.
- All currents through the devices in a graph.
- All powers generated by voltage source and dissipated in the components in a graph.

Exercise 3: DC Analysis Simulation

In this third exercise, we will perform DC analysis simulation for a very simple series-parallel RC circuit given below using LTSpice.



a. Construct the above given circuit as a schematic in LTSpice. [10 marks]

b. Calculate and perform *DC Operating Point* simulation of the circuit. [15 marks]

- All voltages in the circuit.
- All currents in the circuit.
- All powers generated by the voltage source and dissipated in the components.

c. Perform *Transient Analysis* simulation of the circuit (Stop Time = 1 s; Maximum Step = 0.0001). Create plots of the following: [15 marks]

- All voltages across the devices in a graph.
- All currents through the devices in a graph.
- All powers generated by voltage source and dissipated in the components in a graph.

NOTE: Attach snapshots of all LTspice schematics and simulation results in the appendix of your report.

Marking Schedule

Student Name : _____

Student ID : _____

No	Description	Mark	Your Mark	Remark
	Exercise 1 – DC Operating Point			
1a.	Schematic circuit.	10		
1b.	DC operating point simulation.	20		
	Exercise 2 – Transient Analysis			
2a.	Schematic circuit.	10		
2b.	Transient analysis simulation.	20		
	Exercise 3 – DC Analysis			
3a.	Schematic circuit.	10		
3b.	DC operating point simulation.	15		
3c.	Transient analysis simulation.	15		
	Total	100		

Comment: