Brac University

Semester: Fall 2022
Course No: CSE250
Course Title: CIRCUITS AND ELECTRONICS
Section:

Simulation Project
Full marks: 20
Deadline: December 15, 2022

All the questions are mandatory. ***Find the submission guidelines on page 3.***

Question 1 of 2 *[10 marks]*

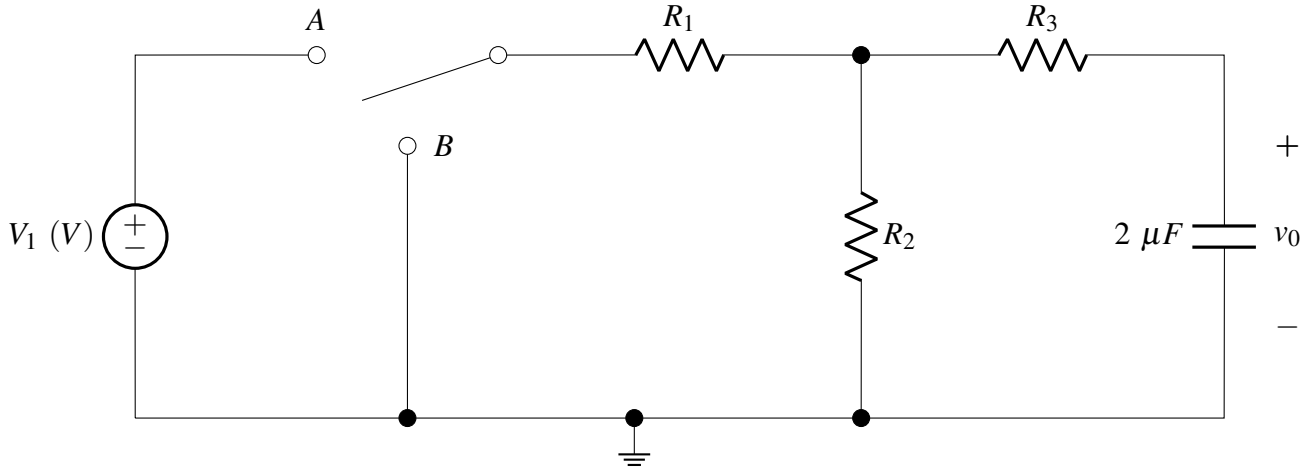
The switch in the circuit below alternates between positions A and B continuously and periodically. The voltage response of the capacitor over time upon switching is also shown below. Assume that the time it takes for the capacitor to be fully charged and discharged in milliseconds is equal to the sum of the eight digits of your student ID. For example, the charging/discharging time for a student ID of 14301234 is (1+4+3+0+1+2+3+4) = 18 ms.

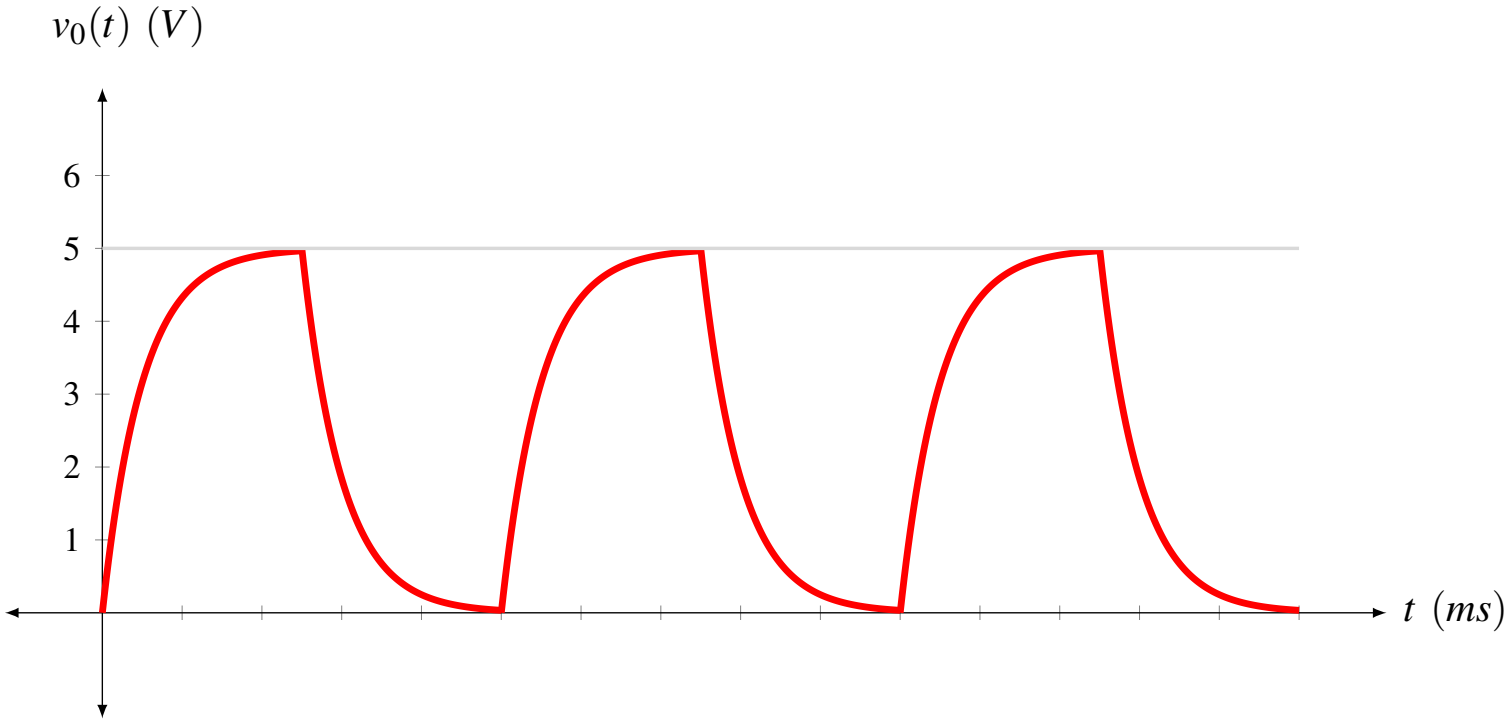
(i) Design the circuit (values of R_1, R_2, and R_3) so that the capacitor charges or discharges following the time constant you derived from your student ID.

(ii) Determine the required supply voltage so that the amplitude of the output voltage matches with that shown in the plot.

(iii) Regenerate the following plot by simulating the designed circuit in LTspice.

[Hint: Remember how we modeled such switching mechanism in our lab experiment?]





Question 2 of 2 *[10 marks]*

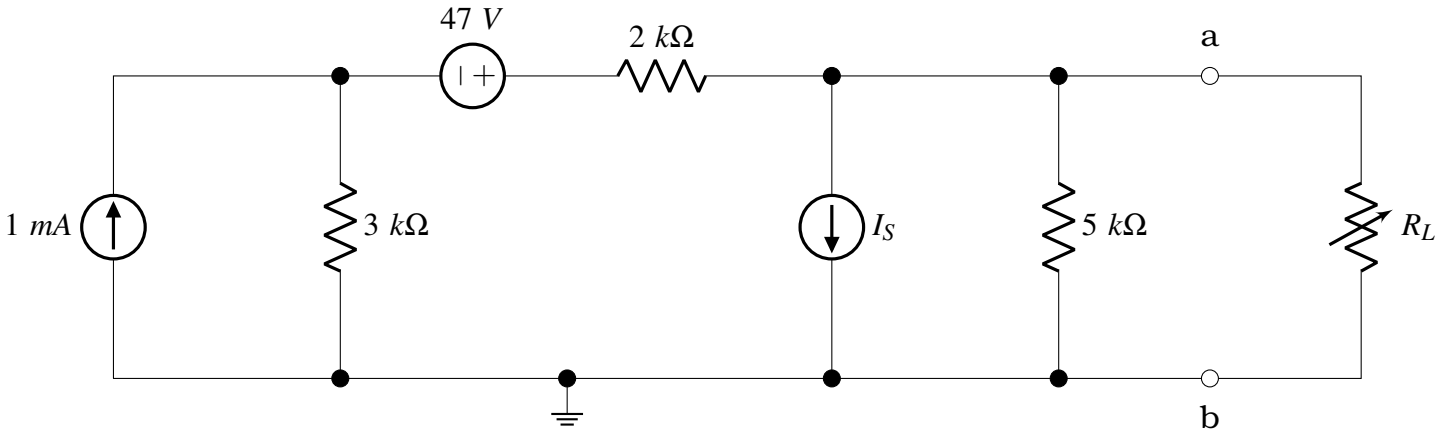
Consider the following circuit. The Thevenin equivalent of the circuit with respect to the load terminals $a-b$ is also illustrated, where V_{Th} and R_{Th} represent the Thevenin voltage and resistance, respectively. The power of the load R_L as the load fluctuates is plotted against R_L. Take the maximum power, in milliwatts unit, equal to the sum of the eight digits of your student ID. For example, for a student ID of 14301234, the maximum power of the load is (1+4+3+0+1+2+3+4) = 18 mW.

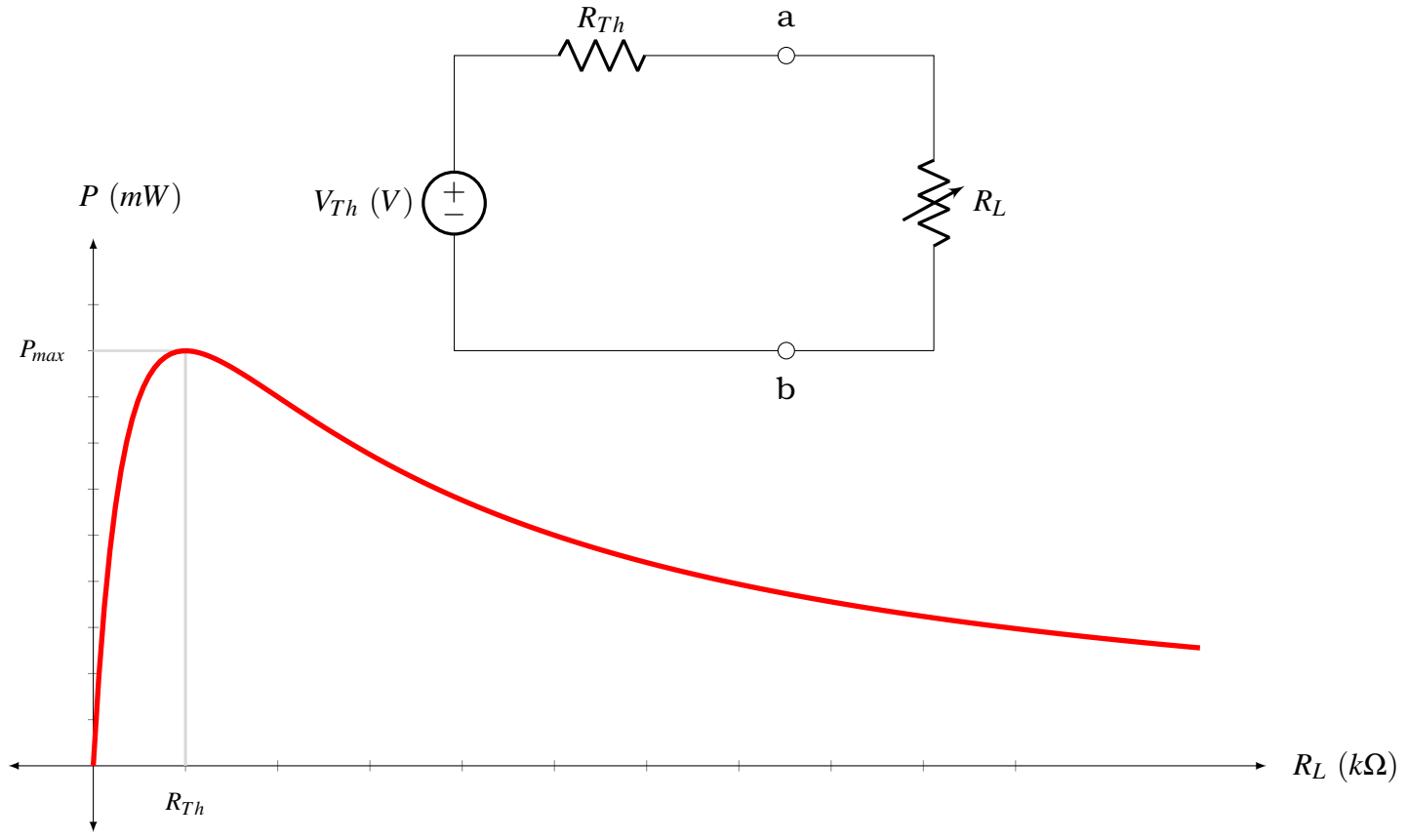
(i) Determine the value of R_L that will draw the maximum power from the rest of the circuit.

(ii) Regenerate the power vs R_L curve by simulating the reduced Thevenin circuit in LTspice.

(iii) Determine the value of the current source I_S from the original circuit.

(iv) Now simulate the original circuit in LTspice and generate the power vs R_L curve and match with that one derived in step (ii). Comment on your results.





Submission Guidlines

You will be provided a google form where you must submit a report and all the LTspice simulation files *(.asc files and .plt files)*.

Report should include the following things:

(i) Cover page.

(ii) Methodology *[In this part you must describe your technique or strategy for solving the problems. You may use flowcharts to decorate your report].*

(iii) Necessary calculations. *[You must specify and use your student ID in the calculation as instructed in each of the questions].*

(iv) Optimized circuit diagrams (*[Drawn]*).

(v) Simulated circuit diagrams (*[full-screen screenshots from LTSpice*). *Simulation commands must be seen in the screenshots along with the circuit diagrams].*

(vi) Simulated characteristics curves.