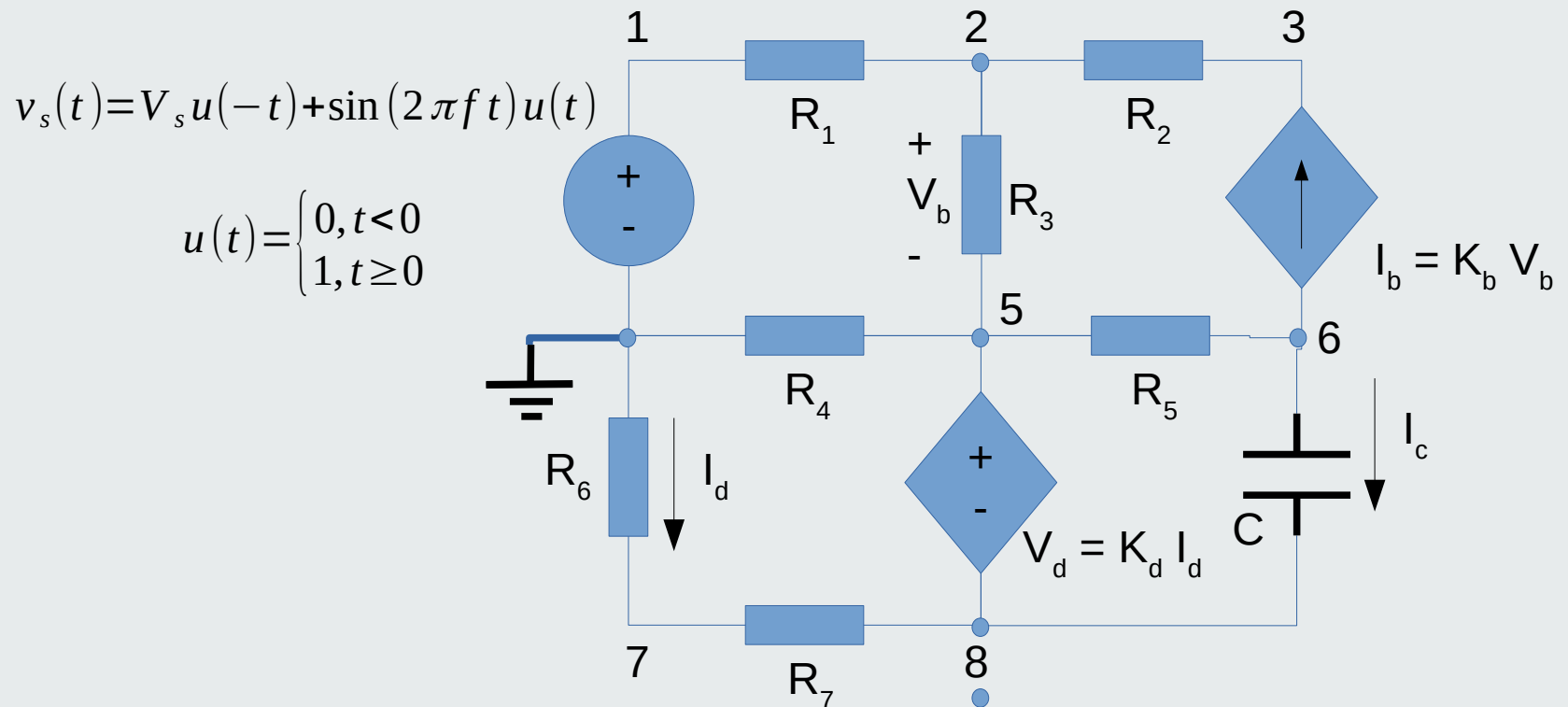


# **Circuit Theory and Electronics Fundamentals**

Lab 2: RC Circuit Analysis

# The RC circuit



# Simulation Analysis

- 1) Simulate the operating point for  $t < 0$ , to obtain the voltages in all nodes and the currents in all branches. Print a table with the results.
- 2) Simulate the operating point for  $v_s(0)=0$ , replacing the capacitor with with a voltage source  $V_x = V(6)-V(8)$ , where  $V(6)$  and  $V(8)$  are the voltages in nodes 6 and 8 as obtained in 1). Explain why you need this step. Print a table with the results.
- 3) Simulate the natural response of the circuit, using the boundary conditions  $V(6)$  and  $V(8)$  as obtained in 2) (use Ngspice directive `.ic V(6)=... v(9)=...`). Use Ngspice's transient analysis mode to get  $v_6(t)$  in the interval  $[0, 20]$  ms. Plot the result.
- 4) Simulate the natural and forced response on node 6 by repeating step 3) with  $v_s(t)$  as given in Fig. 1 and  $f=1\text{kHz}$ . Plot both the stimulus and the response.
- 5) Simulate the frequency response in node 6 (frequency logscale, magnitude in dB, phase in degrees) for the frequency range 0.1 Hz to 1 MHz. Plot both  $v_s(f)$  and  $v_6(f)$  in the same figure and explain how and why they differ.

# Theoretical Analysis

- 1) Use the nodal method to determine the voltages in all nodes and currents in all branches for  $t < 0$ . Print a table with the results.
- 2) Determine the equivalent resistance  $R_{eq}$  as seen from the capacitor terminals.  
Suggestion: make  $V_s = 0$  and replace the capacitor with a voltage source  $V_x = V(6) - V(8)$ , where  $V(6)$  and  $V(8)$  are the voltages in nodes 6 and 8 as obtained in 1); run nodal analysis to determine the current  $I_x$  supplied by  $V_x$ ; compute the equivalent resistor as  $R_{eq} = V_x / I_x$ , and the time constant. Explain why you need this procedure, and print a table with the computed results.
- 3) With the results obtained in 2), determine the natural solution  $v_{6n}(t)$ , in the interval  $[0, 20]$  ms, using the capacitor voltage  $V_x$  for  $t < 0$  as the initial condition. Plot the result, identifying all axes, signals displayed and units used.
- 4) Determine the forced solution  $v_{6f}(t)$  in the same interval. Suggestion: use a phasor voltage source  $V_s$ ; replace  $C$  with its impedance  $Z_C$ ; run nodal analysis to determine the phasor voltages in all nodes. Print a table with the results.
- 5) Determine the final total solution  $v_6(t)$ , converting the phasors to real time functions for  $f = 1\text{kHz}$ , and superimposing the natural and forced solutions. Plot both  $v_s(t)$  and  $v_6(t)$  in the same figure in the interval **[-5, 20]** ms.
- 6) Determine the frequency responses  $v_c(f) = v_6(f) - v_8(f)$ , and  $v_6(f)$  (frequency logscale magnitude in dB, phase in degrees) for frequency range 0.1 Hz to 1 MHz. Plot  $v_s(f)$ ,  $v_c(f)$  and  $v_6(f)$  in the same figure and explain how and why they differ.

# Lab report

- 1) The Python script, *t2\_datagen.m* output should be redirected to a file *data.txt* using bash's ">" facility.
- 2) The octave script should read and use the data file *data.txt*, and output a data file for Ngspice (directive `.include`). Use file I/O functions such as *fprintf* and *fscanf*.
- 3) Ideally, generating a new data set with a different student number and re-running the top Makefile should produce consistent results.
- 4) Produce all tables and plots required in the simulation and analysis sections
- 5) Compare Octave and Ngspice results side by side looking for exactness or discrepancy, and explaining both. Read the repository's README file.