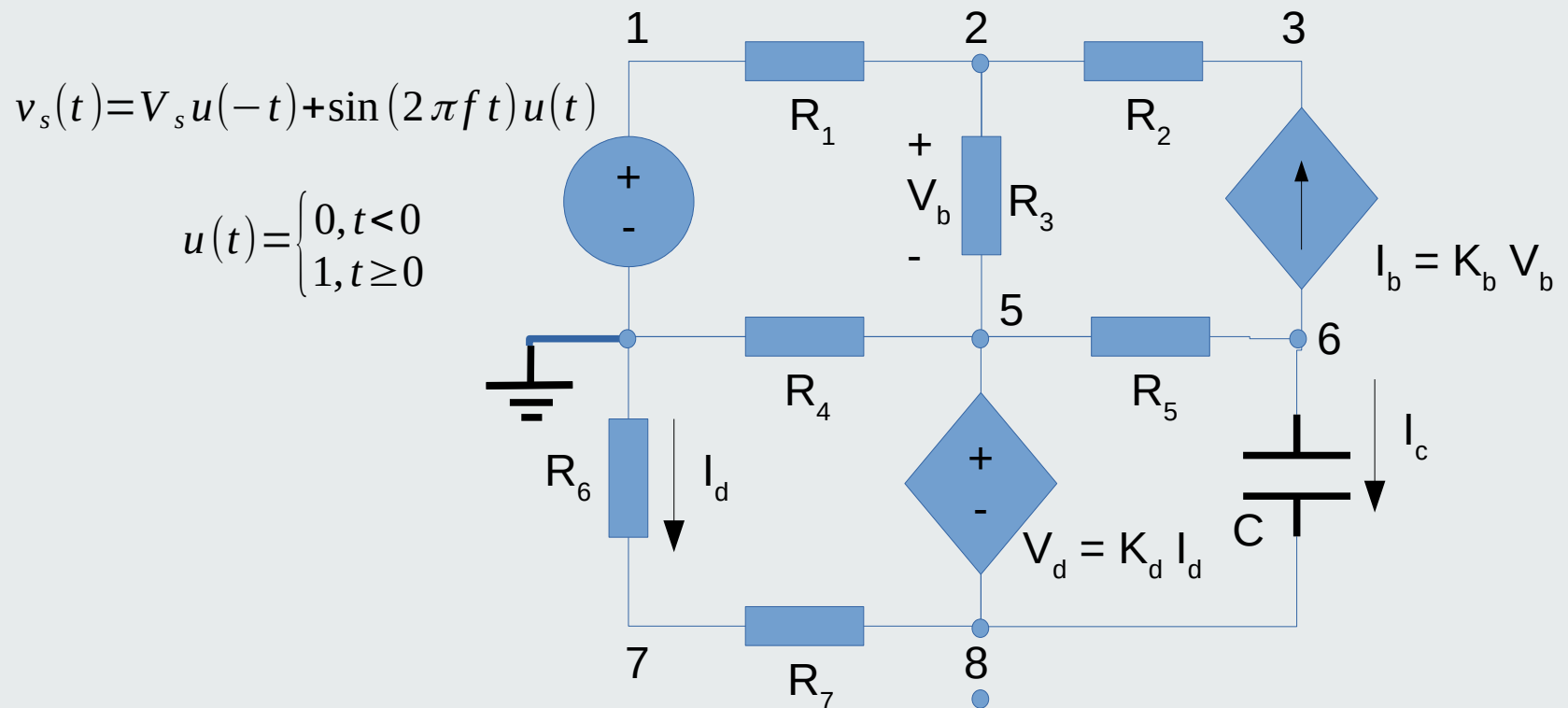


Circuit Theory and Electronics Fundamentals

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 for $f = 1\text{KHz}$. Suggestion: use a phasor voltage source $V_s = 1$; replace C with its impedance Z_C ; run nodal analysis to determine the phasor voltages in all nodes. Print a table with the complex amplitudes in the nodes.
- 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.py* 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.