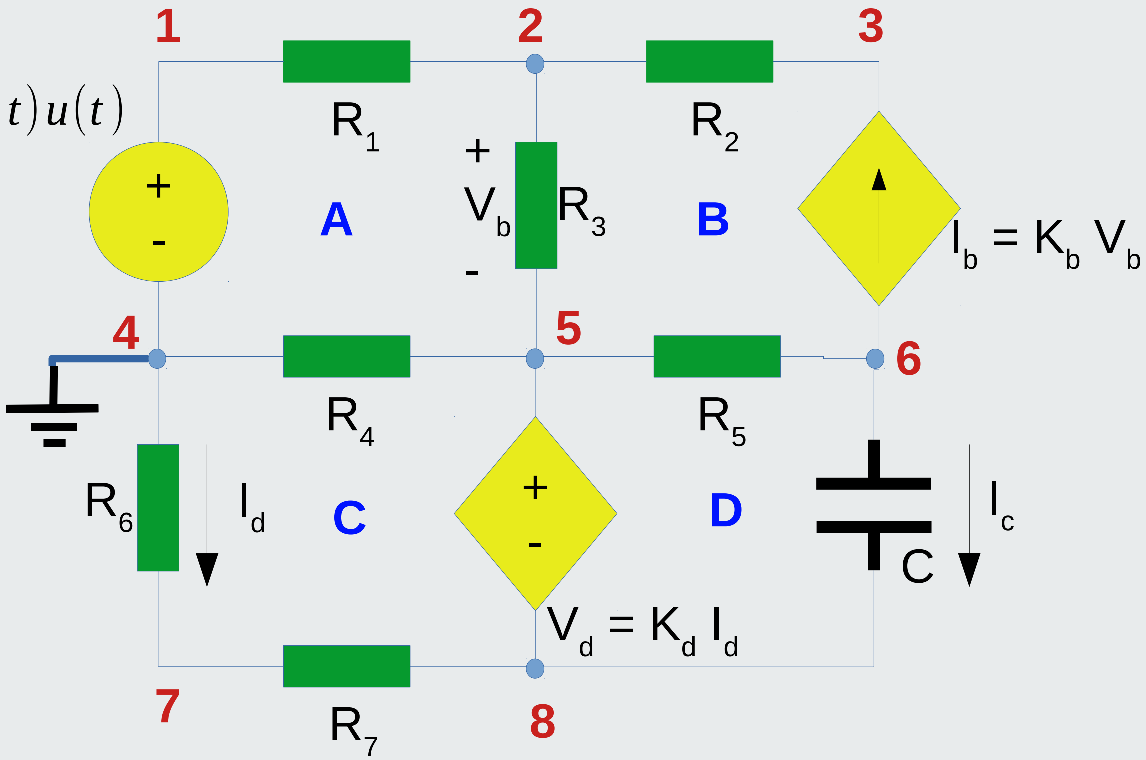
TÉCNICO
LISBOA

Circuit Theory and Electronics Fundamentals

Lab 2: RC Circuit Analysis

$$v_s(t) = V_s u(-t) + \sin(2\pi f t) u(t)$$

$$u(t) = \begin{cases} 0, t < 0 \\ 1, t \geq 0 \end{cases}$$



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 Ngpice'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=1kHz. 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=1KHz, 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 <u>side by side</u> looking for exactness or discrepancy, and explaining both. Read the repository's README file.