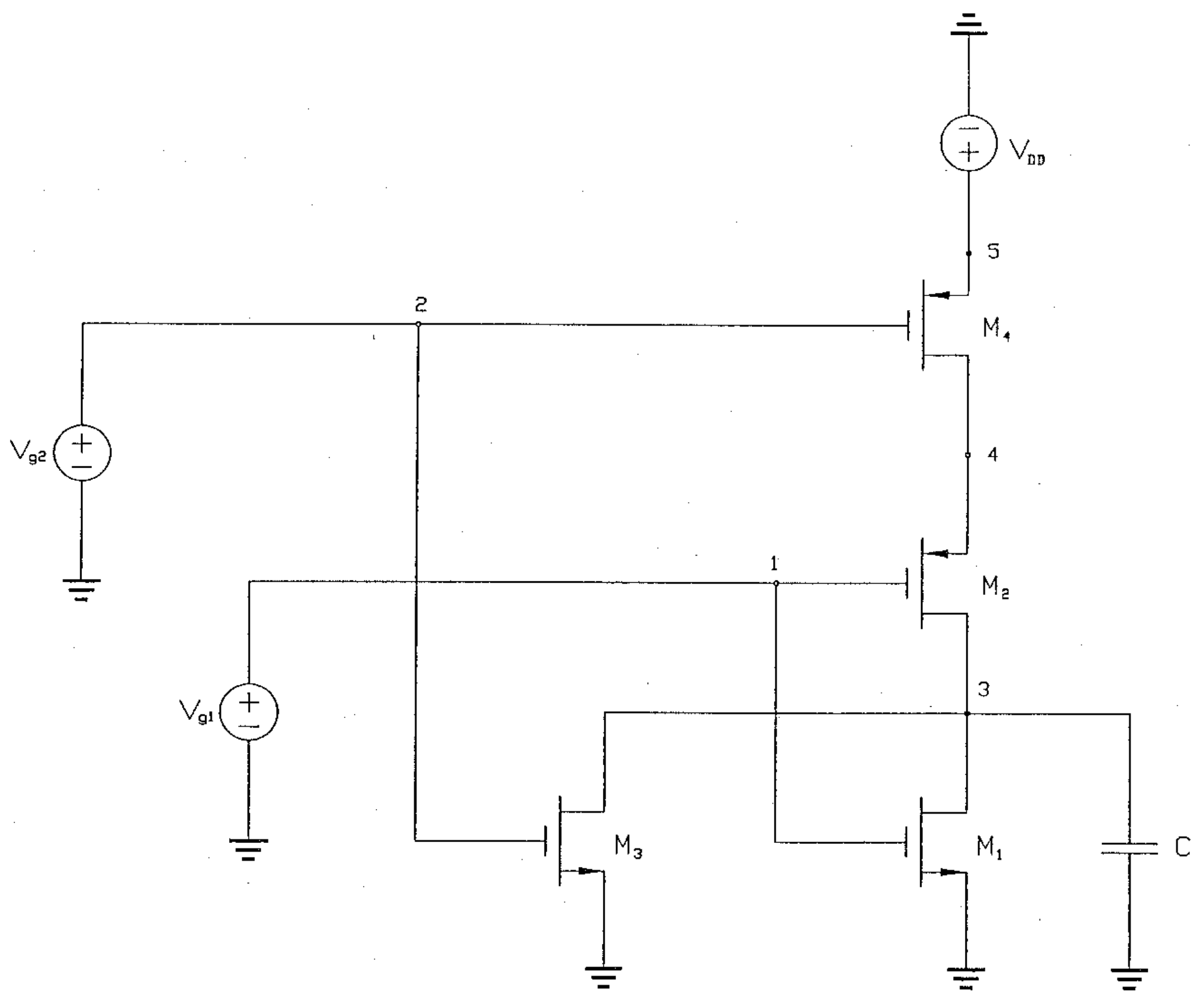
ELEN 118
A. I. Zecevic

PROJECT # 5: Transient Analysis of Digital Circuits

The circuit below represents a CMOS nor gate, with a DC supply of $V_{DD} = 5V$ and a loading capacitor $C = 0.25pF$.



The device parameters are: $K = 2 \times 10^{-5}\ A/V^2$, $V_{Tn} = 1V$ and $V_{Tp} = -1V$, and all pulse sources are assumed to have: $V_1 = 0$, $V_2 = 5V$ and $T_r = T_f = 1ns$.

PROBLEM 1.

Use the PMOS and NMOS transistor models to formulate the circuit equations for transient analysis. Indicate G, E, $p(x)$ and w explicitly.

PROBLEM 2.

a) Write an m - file that will perform a *partial transient analysis*, the results of which can be used to obtain a good initial approximation for the DC solution. The input arguments for this function should be matrices E and G, time t_{end}, step size h and *three* 1×7 vectors corresponding to the different pulse sources (note that in this stage of the analysis the DC power supply is treated as a *step function*). The output should be the *waveform vectors* for all voltages, the set of *time points*, and the *number of iterations* that were needed in each time point. In other words, this function should look something like: **[*tout, out, iter*] = *nor*(*tend, h, E, G, a, b, c*)**.

b) Use this function to perform a partial transient analysis with t_{end} = 10ns and $h = 2 \times 10^{-10}$. Record *all* the voltage (and current) values at $t = t_{end}$, and use this as the initial approximation for the DC solution.

PROBLEM 3.

a) Write an m - file that will compute a DC solution for this circuit. The input arguments for this function should be *matrix G*, the *initial guess* x_0 (obtained in Problem 2), and the *convergence criterion e*. The output should contain the *solution vector* and the *number of iterations* needed for convergence. The corresponding format should be: **[*x, iter*] = *newt*(*G, e,* x_0)**.

b) Use this function to compute the *exact* DC solution (choosing $e = 10^{-9}$ as the convergence criterion for Newton's method).

PROBLEM 4.

a) Write an m - file that will perform a *complete transient analysis* of this circuit. The input arguments for this function should be matrices E and G, time t_{end}, step size h, DC solution x_0 and *two* 1×7 vectors that correspond to the input pulse sources (recall that in this stage the DC power supply voltage is treated as a *constant*). The output should be the *waveform vectors* for all voltages, the set of *time points*, and the *number of iterations* that were needed in each time point. Your function should have the form: **[*tout, out, iter*] = *nordc*(*tend, h, E, G, a, b,* x_0)**.

b) Use this function to obtain waveforms for voltages $V_3(t)$ and $V_4(t)$ in case input 1 is a LOGIC ONE and input 2 is a LOGIC ZERO (assume that $t_{end} = 1.5 \times 10^{-7}$ sec, $h = 2 \times 10^{-10}$, and that the input pulse width is 100ns). Plot *both* waveforms in Matlab.

c) Perform a transient analysis of this circuit in SPICE, assuming the same input conditions as in part b). Print out your input file, and obtain plots for voltages $V_3(t)$ and $V_4(t)$.

d) Compare the results of your program to those obtained using SPICE. Do you see any significant differences?