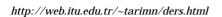# HOMEWORK 1

Due: March 22, 2013, at 14:00

- **Solve each question on the space below each question. If necessary add an extra page.**
- **Submissions will be made to the instructor on paper. Electronic submissions (by e-mail, etc.) will not be accepted.**
- **Late submissions will not be accepted. If you have no lectures on thursday, submit your homework earlier.**
- **The homework must be your own work.**

Any homework which does not comply to the rules above, will receive an automatic grade of zero.

The following are given for a 0.35 µm CMOS process:

V_{DD}=3.3 V, V_{tn}=0.7 V, V_{tp}= −0.7 V, V_{tL}= −1 V, $\mu_n C_{ox}$=160 µA/V², $\mu_p C_{ox}$=80 µA/V², ε_{ox}=3.45*10⁻¹¹ F/m, t_{ox}=7.75 nm, γ=0.45 V^{1/2}, $2|\Phi_F|$=0.6 V.

1. V_{GG}=6 V and k_D / k_L=10 is given for a non-saturated enhancement-mode NMOS loaded inverter. Calculate V_{OH}, V_{OL} and V_{TH}.

2. Design a CMOS inverter so that V_{TH}=1.7 V. Calculate V_{IH} and V_{IL}.

1. SPICE simulations

For each question:
Obtain the voltage transfer characteristic (VTC). Show the values of V_{TH}, V_{IL} and V_{IH} on your plots.

Use the MBREAK transistor from the library with the model parameters given in the table below. Use LEVEL=2.

Attach the SPICE output file (only for DC) and screen-view of node voltages and transistor currents of your circuit to your homework.

Attach all necessary simulation result plots to your homework.

Compare your calculations and the theory to your SPICE simulation results. Comment on all your results.

Table 1   Active device parameters and their SPICE equivalents

	Device parameter	SPICE model parameter
MOSFET	μC_{ox}	+ KP
	V_t	+ VTO
	t_{ox}	+ TOX
	γ	+ GAMMA
	$2\|\Phi_F\|$	+ PHI