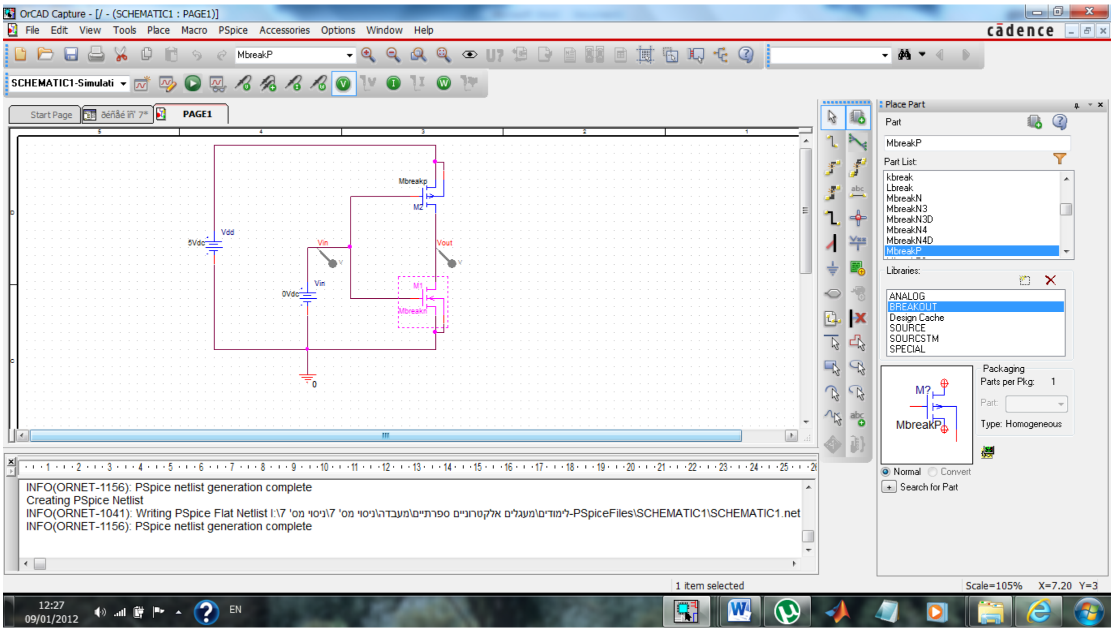
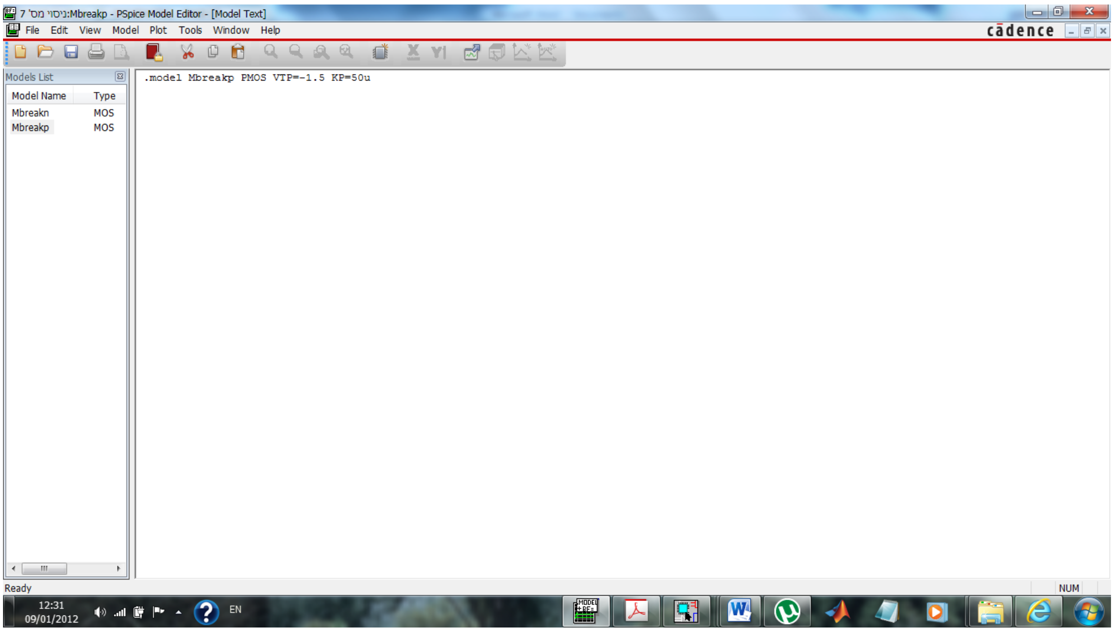
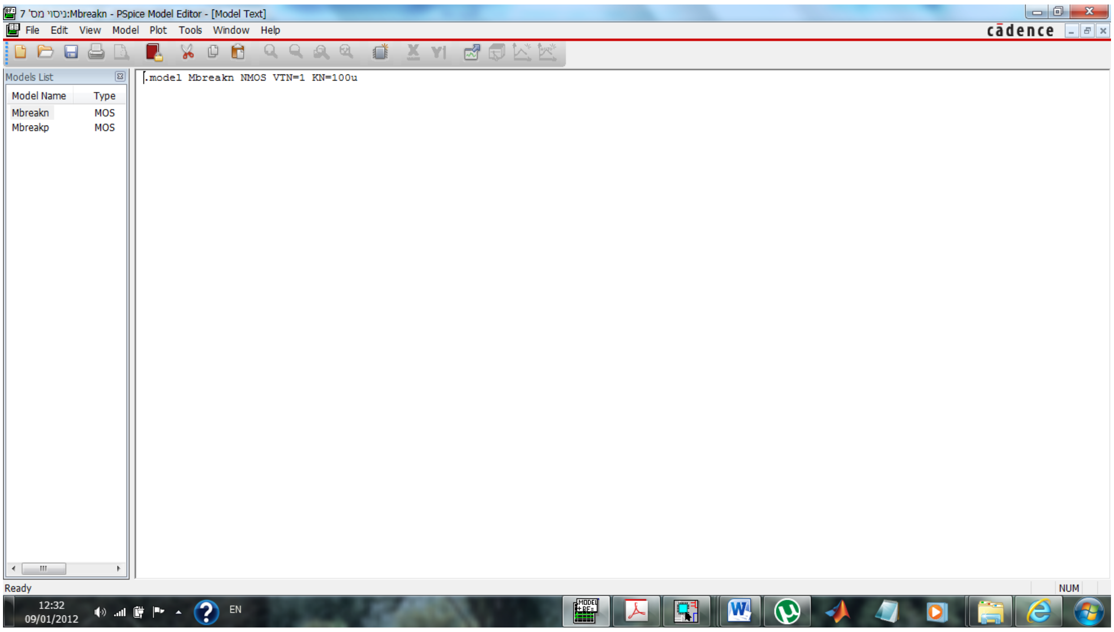
Hi.
I'm trying to simulate the following (static) CMOS inverter circuit with PSPICE, but can't figure out why I'm receiving an error message, which is also quite unclear:



I also defined the PMOS and NMOS transistors in the following way:





The circuit's parameters are given below:

$$V_{DD} = 5\,\mathrm{V};\quad V_{TN} = 1\,\mathrm{V};\quad V_{TP} = -1.5\,\mathrm{V};\quad K_N = 100\,\frac{\mu\mathrm{A}}{\mathrm{V}^2};\quad K_P = 50\,\frac{\mu\mathrm{A}}{\mathrm{V}^2}$$

And the simulation used is a DC sweep (primary sweep only) simulation, as V_{in}'s start value is $0\,\mathrm{V}$, its end value is $5\,\mathrm{V}$ and the incerement chosen is $0.01\,\mathrm{V}$.

The error message I get apperars in the next page:

**** 01/09/12 12:47:31 ****** PSpice 16.5.0 (April 2011) ****** ID# 0 ********

** Profile: "SCHEMATIC1-Simulation 1" [I:\לימודים\מעגלים אלקטרוניים ספרתיים\מעבדה\ניסוי מס' 7\Lab 7-PSpiceFiles\SCHEMATIC1\Simulat

**** CIRCUIT DESCRIPTION

****

** Creating circuit file "Simulation 1.cir"
** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT SIMULATIONS

*Libraries:
* Profile Libraries :
* Local Libraries :
.LIB "../../../lab 7-pspicefiles/lab 7.lib"
* From [PSPICE NETLIST] section of C:\Cadence\SPB_16.5\tools\PSpice\PSpice.ini file:
.lib "nom.lib"

*Analysis directives:
.DC LIN V_Vin 0 5 0.01
.PROBE V(alias(*)) I(alias(*)) W(alias(*)) D(alias(*)) NOISE(alias(*))
.INC "..\SCHEMATIC1.net"

**** INCLUDING SCHEMATIC1.net ****
* source LAB 7
M_M2 VOUT VIN N15255 N15255 Mbreakp
V_Vin VIN 0 0Vdc
M_M1 VOUT VIN 0 0 Mbreakn
V_Vdd N15255 0 5Vdc

**** RESUMING "Simulation 1.cir" ****
.END

INFO(ORPSIM-15423): Unable to find index file "lab 7.ind" for library file "lab 7.lib".

INFO(ORPSIM-15422): Making new index file "lab 7.ind" for library file "lab 7.lib".

Index has 2 entries from 1 file(s).

**** FROM LIBRARY "../../../lab 7-pspicefiles/lab 7.lib" ****
.model Mbreakp PMOS VTP=-1.5 KP=50u
--------------------$
ERROR -- 'VTP' is not a model parameter name

.model Mbreakn NMOS VTN=1 KN=100u
--------------------$
ERROR -- 'VTN' is not a model parameter name

What am I doing wrong, and how to solve that issue?

Thanks! ☺