1) Using the figure below, estimate the transistor model (Level 1) parameters: $V_{TO}$, $\gamma$, $k'$, and $\lambda$. Assume $W/L=10$, $\phi_f=0.3V$. Explain your method.

2) An NMOS transistor has the following parameters:

$W=1.3 \ \mu m$, $L=0.13 \ \mu m$, $V_{TO}=0.4 \ V$, $\gamma=0.5 \ V^{1/2}$, $k'=100 \ \mu A/V^2$, and $\lambda=0.01 \ V^{-1}$

Sketch $g_m$-$V_{ds}$ characteristics for $V_{ds}$ from 0 to 1V and $V_{bs}=0 \ V$ and $-0.5 \ V$. Assume $V_{gs}=1 \ V$. 
3) The figure below is the ln(Id)-Vgs characteristic with and without subthreshold modeling. Assuming the subthreshold current can be approximated with,

\[ I_d = I_o e^{\frac{V_{gs} - V_{th}}{V_T}}, \text{ where } V_T = 26mV\]

Estimate the value, n, in this model.

![Graph showing ln(Id) vs. Vgs with and without subthreshold modeling.]

4) Learn to run HSPICE:
   a) Setup the HSPICE environment with the following instructions:
      http://bwrc.eecs.berkeley.edu/classes/ee140/howtos.htm
   b) Copy the SPICE file (demo3.sp) to your working directory from
      http://bwrc.eecs.berkeley.edu/classes/ee140/notes.htm
   c) Execute HSPICE and Awaves:
      c199.> hspice demo3.sp >! demo3.lis
      >info:      ***** hspice job concluded
      c199.> awaves &
   d) View the results from awaves:
      Load the design from Design->Open, and Choose the waveforms you
      would like to see from Results Browser.
   e) Print out the Id-Vgs, and gm-Vgs curves.