February 22, 2018

# ECEN 474

## Homework #2 Notes

Due: 3-6-2018, 5:00PM

**Homeworks will not be received after due.**

Instructor: Sam Palermo

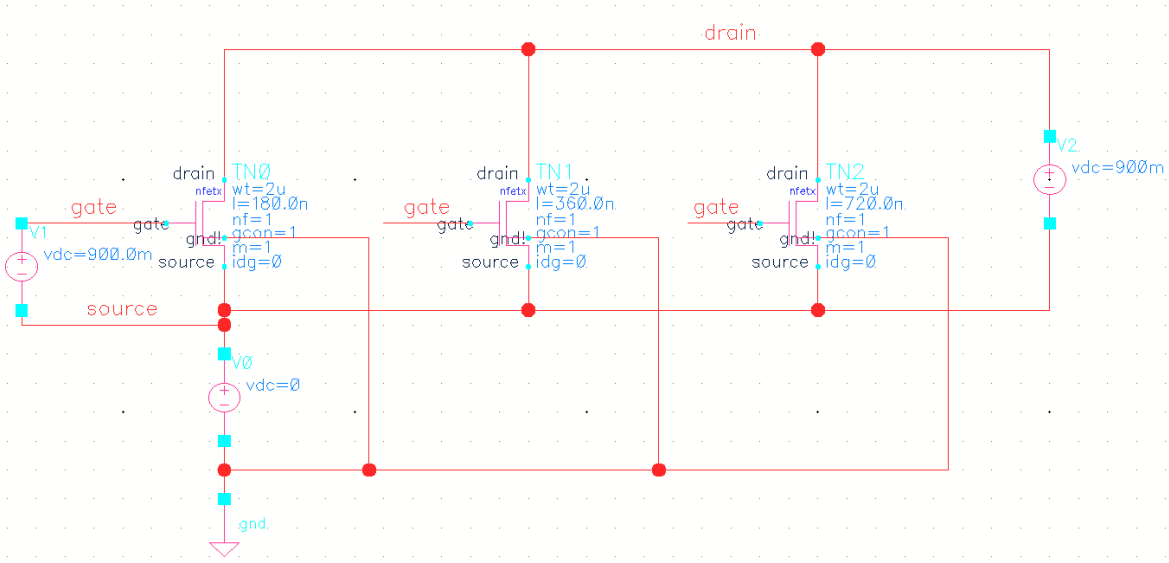
1. **(50 points)** Technology Characterization for Design

In this problem, we will extract some of the key MOSFET parameters used in circuit design. For all the following, extract the required data and plots for both the NMOS AND the PMOS device. For the PMOS device swap the polarity of the given bias conditions, i.e use $V_{SD}$ for $V_{DS}$ and $V_{SG}$ for $V_{GS}$.

a) Threshold Voltage Dependency on Width & Length.

- Plot $V_T$ vs Length for a range of $0.18\mu m \leq L \leq 1\mu m$. Use $W=2\mu m$ and $V_{DS}=V_{GS}=0.9V$. (one plot w/ one curve for both NMOS and PMOS versions)
- Plot $V_T$ vs Width for a range of $0.5\mu m \leq W \leq 6\mu m$. Plot for $L=0.18\mu m$, $L=0.36\mu m$, and $L=0.72\mu m$. Use $V_{DS}=V_{GS}=0.9V$. (one plot w/ 3 curves for both NMOS and PMOS versions)

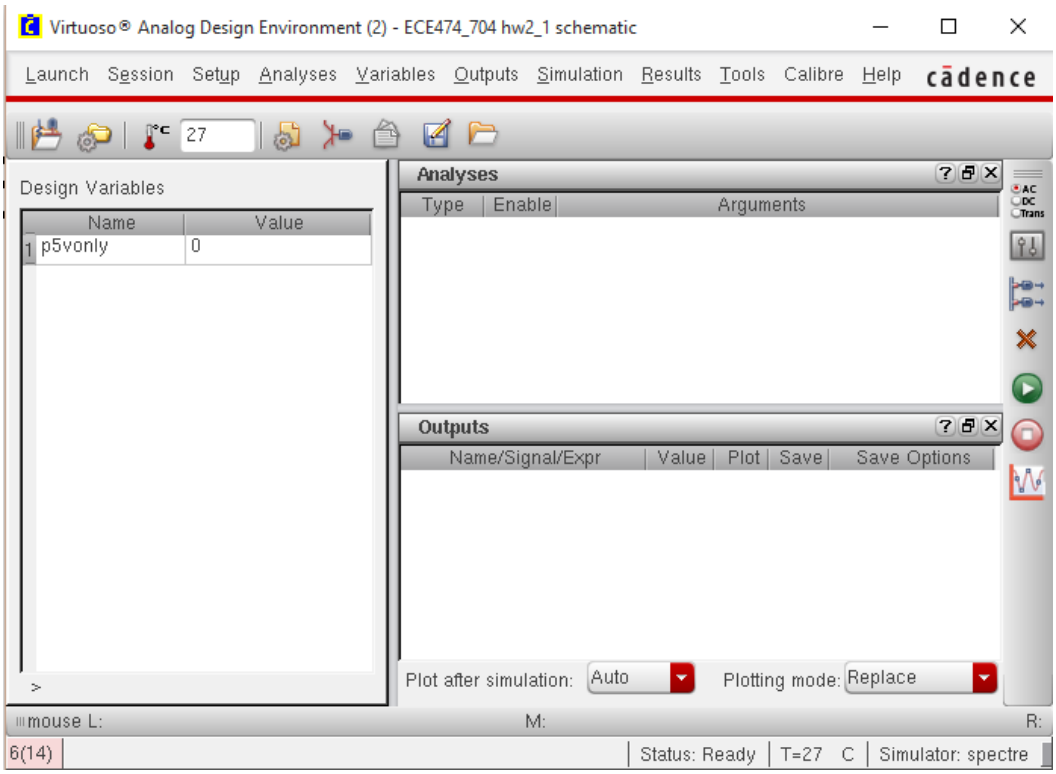
For parts (a)-(d), I used the following schematic in Cadence. It has 3 transistors which all have a default W=2µm, but transistor N0 has L=0.18µm, N1 has L=0.36µm, and N2 has L=0.72µm.



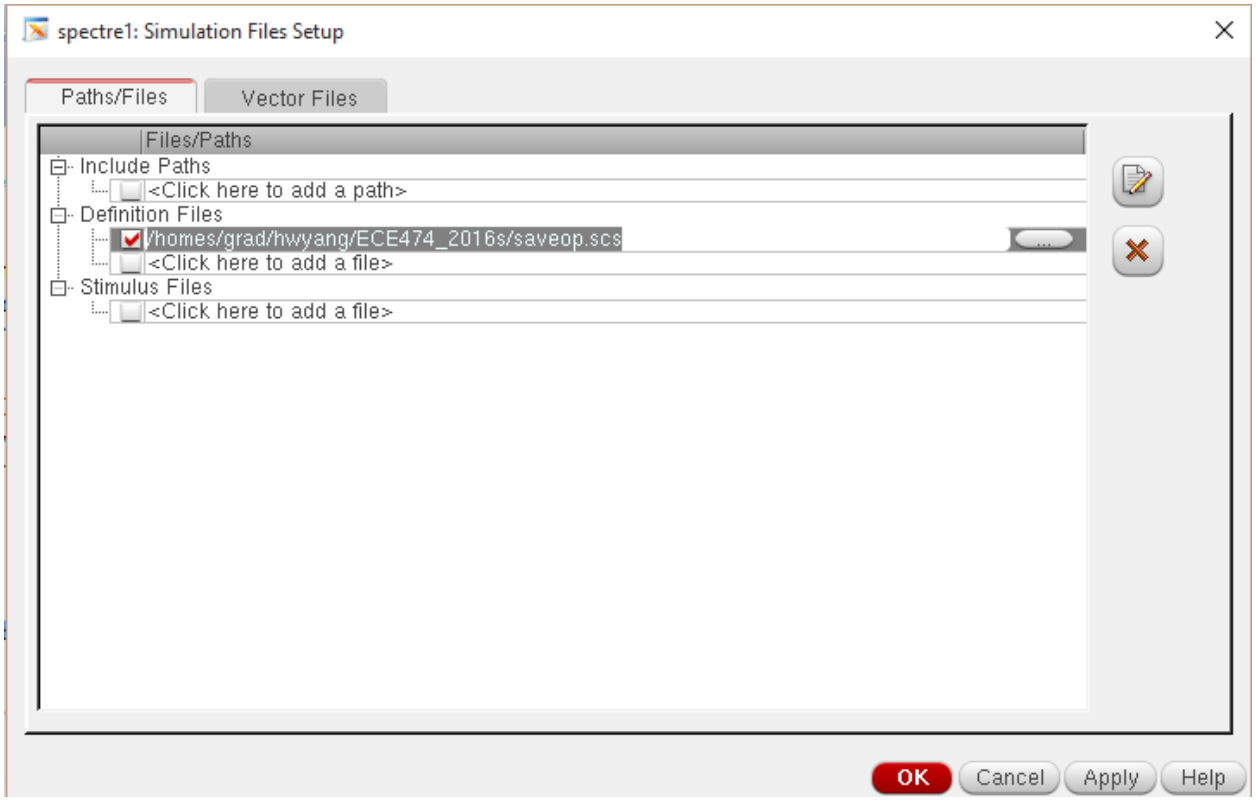

Unfortunately, Cadence Analog Design Environment (ADE) does not by default save the operating point information of the transistors and, to my knowledge, you cannot directly tell ADE to do this through any of the menus. In order to do this you must include a "Definition File", which I call "saveop.scs". This file has the following three lines to save the three transistors' operating point information:

save TN0:all
save TN1:all
save TN2:all

To include the file in the simulation, in the ADE window:

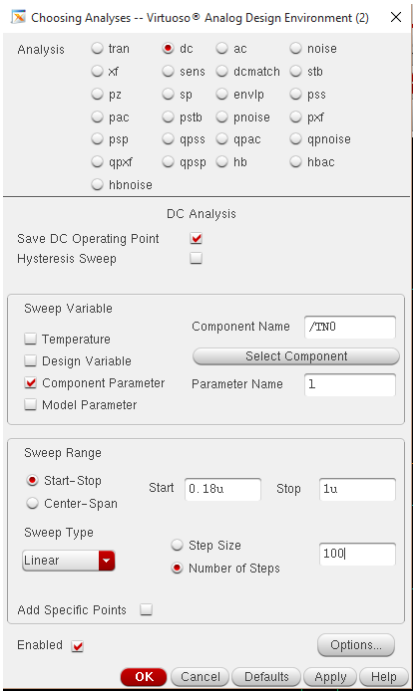



Select Setup -> Simulation Files. You should get a window which looks like this

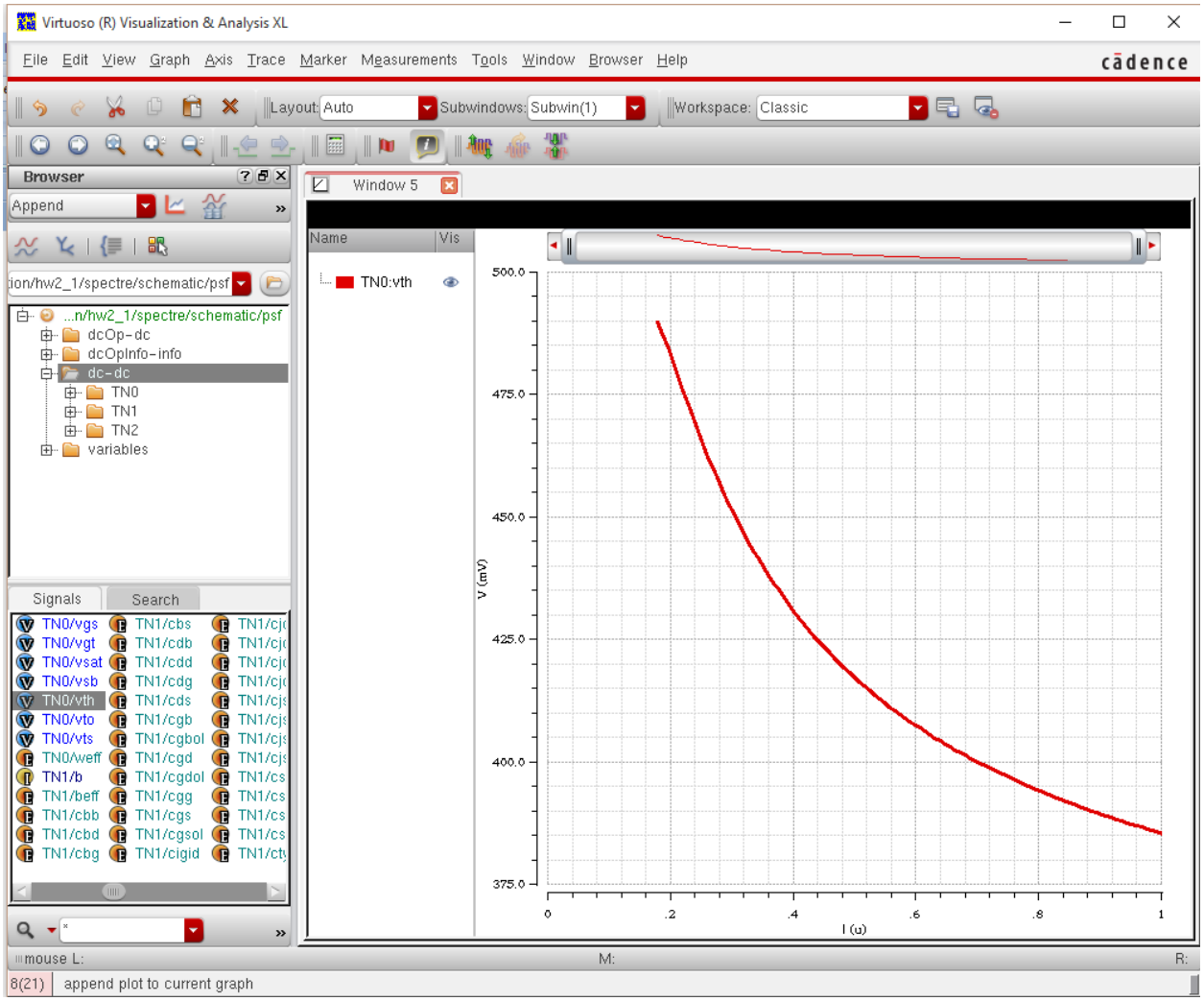



Click on “Click here to add a file” in the Definition Files part and browse to the path of your “saveop.scs” file. Click “OK” and it should save the transistor operating point info during the simulations.

All the simulation data for the first problem should be extracted with DC simulations. To simulate the first part of part (a), you just need to look at one of the transistors because I ask you to sweep L with only one W=6µm. Here are the settings I use:

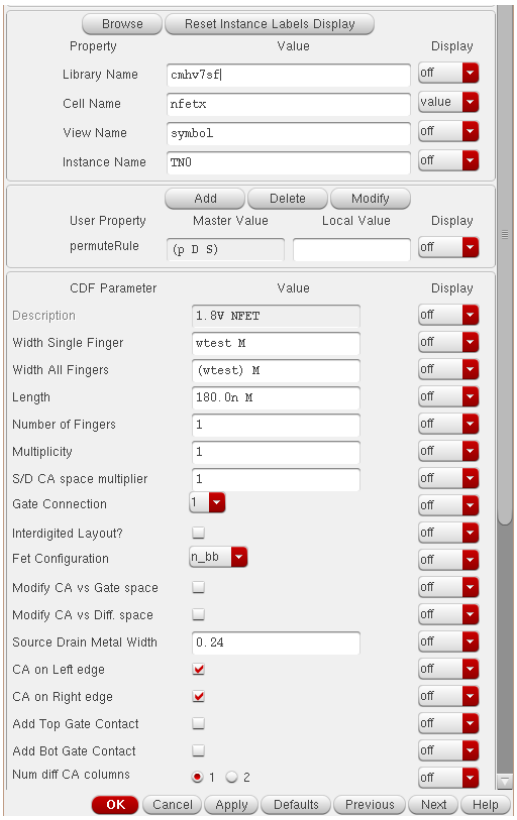



Here I am sweeping the Component Transistor "/TN0" and Parameter "l" from 0.18µm to 1µm with 100 points. After running the simulation, in the ADE select Tools -> Results Browser and the following window should pop up.




Double click on “dc-dc” and you should be able to see the DC simulation outputs, including all the operating point information. Double-click on TN0/vth and it should plot it for you as shown above. (Note, plot background is black by default)

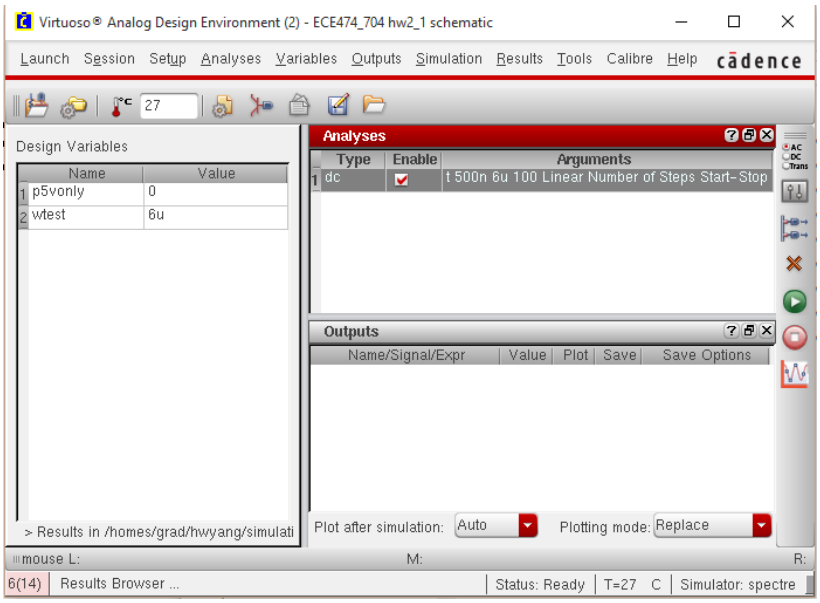
To simulate the second part of part (a), you need to look at all three transistors because I ask you to sweep W for the 3 Ls. In order to do this in one simulation, I create a “Design Variable” called “wtest” which is the common width of the transistors. For all 3 transistors in the schematic, I “Edit Object Properties” (select and press “q”) and change the “Width” to “wtest”, as shown below.



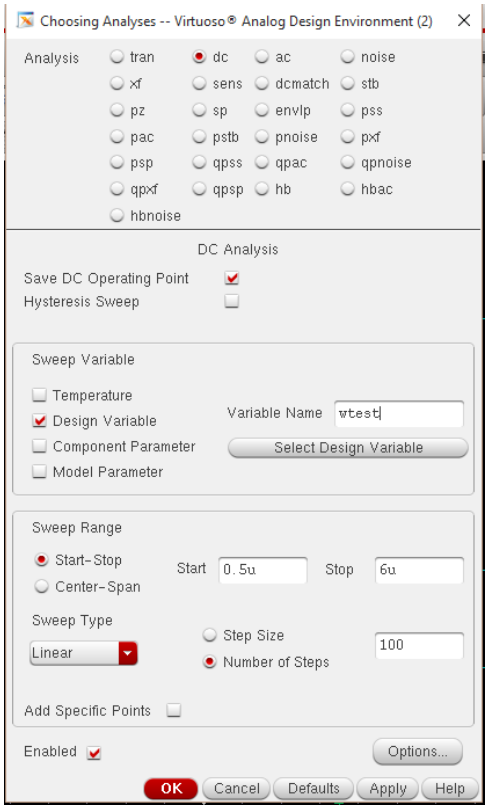

In the ADE, you will need to set the default variable value by selecting Variables ->Edit

Name = wtest
Value (Expr) = 6u Note, this is the default value

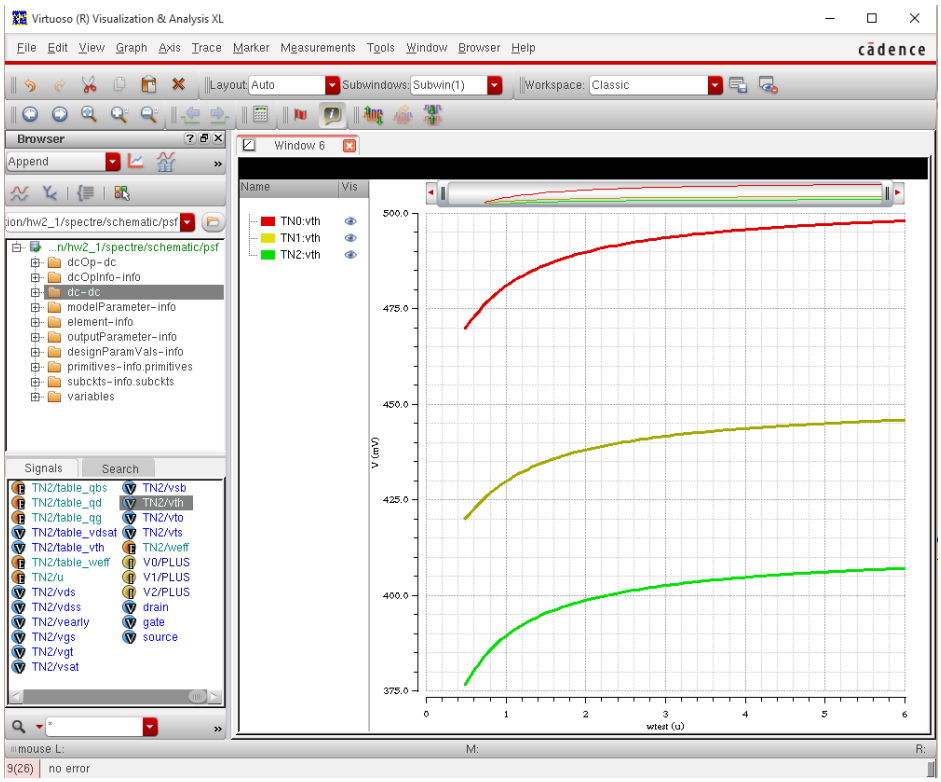
Now your ADE window should look as follows, with the Design Variables info on the left.




Here are the settings I use for the DC simulation:



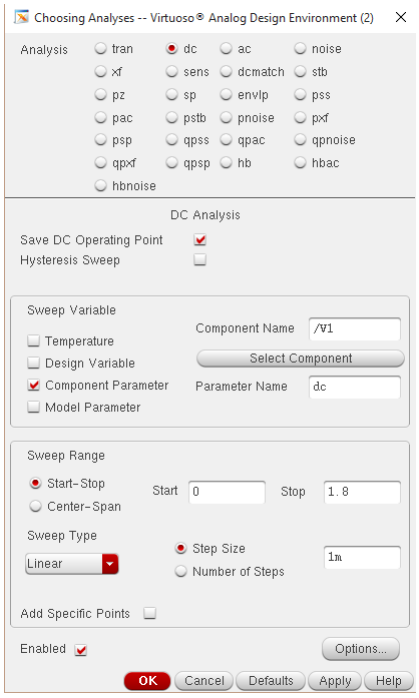

Here I am sweeping the Design Variable "wtest" from 0.5µm to 6µm with 100 points. After running the simulation, refresh the results browser and you should be able to plot the three transistors' $V_T$ on one plot as shown below.



b) For a unit device finger of W=2µm and $V_{DS}$=0.9V, extract a data table with the following device parameters from a DC sweep of $V_{GS}$ from 0 to 1.8V with a step size of 1mV. Extract three sets of data for L=0.18µm, L=0.36µm, and L=0.72µm.

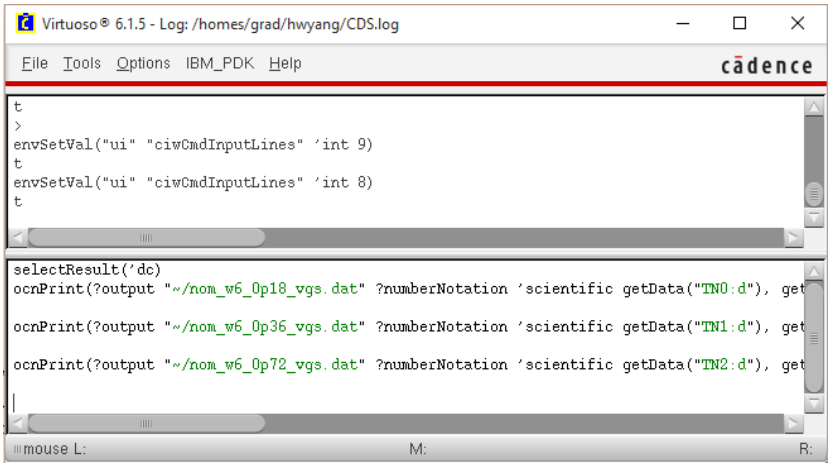
- $V_{GS}$, $I_D$, $g_m$, $g_{ds}$, $V_T$, $C_{gg}$, $C_{gs}$, $C_{gd}$, $C_{dd}$, $C_{ss}$, $C_{jd}$, $C_{js}$

To extract all this data, you simply need to run one DC simulation, sweeping $V_{GS}$ from 0 to 1.8V. Here are the settings that I use:




Here I am sweeping the Component Voltage Source "/V1" and Parameter "dc" from 0 to 1.8V with a step of 1mV. **After running the simulation, I would suggest to NOT USE THE CALCULATOR TO PLOT THE REMAINING PLOTS, as the cadence calculator is somewhat inconvenient to use. Instead, I would save the data to a text file and plot the remaining plots using MATLAB.**

In order to extract the data, type the following four lines in the main cadence CIW command window:



selectResult('dc)
ocnPrint(?output "~/nom_w6_0p18_vgs.dat" ?numberNotation 'scientific getData("TN0:d"), getData("TN0:gm"), getData("TN0:gds"), getData("TN0:vth"), getData("TN0:cgg"), getData("TN0:cgs"), getData("TN0:cgd"), getData("TN0:cdd"), getData("TN0:css"), getData("TN0:cjd"), getData("TN0:cjs"))

ocnPrint(?output "~/nom_w6_0p36_vgs.dat" ?numberNotation 'scientific getData("TN1:d"), getData("TN1:gm"), getData("TN1:gds"), getData("TN1:vth"), getData("TN1:cgg"), getData("TN1:cgs"), getData("TN1:cgd"), getData("TN1:cdd"), getData("TN1:css"), getData("TN1:cjd"), getData("TN1:cjs"))

ocnPrint(?output "~/nom_w6_0p72_vgs.dat" ?numberNotation 'scientific getData("TN2:d"), getData("TN2:gm"), getData("TN2:gds"), getData("TN2:vth"), getData("TN2:cgg"), getData("TN2:cgs"), getData("TN2:cgd"), getData("TN2:cdd"), getData("TN2:css"), getData("TN2:cjd"), getData("TN2:cjs"))

Hit return and this should write the data to 3 text files in your home directory (the ?output variable is the file name). The data is written in column format in the order as given above with an additional first variable being the sweep parameter "$V_{GS}$". Note in order to load the data into MATLAB for plotting, you will need to delete the header in the top line of the three data files. Also, potentially typing the above lines multiple times can be tedious. So I generally create a text file with the above lines and "copy-and-paste" the lines into the main cadence window using the mouse middle button.

In MATLAB, you can use the following command to import the data from the *.dat file.

**[wtest,id,gm,gds,vth,cgg,cgs,cgd,cdd,css,cjd,cjs]**
**=textread('/homes/grad/USER_id/YOUR_File_Name.dat','%f %f %f %f %f %f %f %f %f %f %f %f','delimiter',',','headerlines',2);**

c) Essential Design Plots. Using the table data from part (b), plot the following
   - $g_m/I_D$ vs $V_{ov}=V_{GS}-V_T$
   - $g_m/g_{ds}$ vs $g_m/I_D$
   - $f_T$ vs $g_m/I_D$
   - $I_D/W$ vs $g_m/I_D$

   For the 4 plots for both the NMOS and PMOS versions, each one should have 3 curves for the 3 channel lengths.

d) Other Useful Design Plots. Using the table data from part (b), plot the following
   - $g_{ds}/W$ vs $g_m/I_D$
   - $C_{gg}/W$ vs $g_m/I_D$
   - $C_{dtotal}/W$ vs $g_m/I_D$, where $C_{dtotal}=C_{dd}+C_{jd}$
   - $C_{stotal}/W$ vs $g_m/I_D$, where $C_{stotal}=C_{ss}+C_{js}$

   For the 4 plots for both the NMOS and PMOS versions, each one should have 3 curves for the 3 channel lengths.

Grading is 2.5points/plot for the 20 total plots.

2. **(50 points)** Table-Based ($g_m/I_D$) Design of Common-Source Amplifier

Using the table-based design procedure outlined in Lecture 7, design the common-source amplifier below to satisfy the following specifications.

- 0.18μm technology
- $|A_v| \geq 5V/V$
- $f_u \geq 200MHz$
- $C_L = 5pF$
- Vdd = 1.8V

Design procedure counts for 30 points.

Turn in the following to validate the design performance:

- Schematic with DC operating points and the bias current labeled (5 points)
- Print-out with small-signal device parameters. Highlight the critical small-signal parameters, such as $g_m$, $g_{ds}$, etc. (5 points)
- AC frequency response with the DC gain and unity-gain frequency labeled. (10 points)