

Problem Set 1 Due February 2, 2015

In this homework assignment you will get familiar with Agilent ADS, the main simulation tool that we will use in this class. The assignments themselves are on page 12 and 13.

1. Connecting using the instructional accounts

ADS is installed on the hpse and t7400 servers:

hpse-9.eecs.berkeley.edu, hpse-10.eecs.berkeley.edu, ... hpse-16.eecs.berkeley.edu

t7400-1.eecs.berkeley.edu, t7400-2.eecs.berkeley.edu, ..., t7400-12.eecs.berkeley.edu

For running the graphics remotely (from Windows or Mac) the following tools are recommended:

1. NoMachine: <http://www.nomachine.com/download>

Use SSH for the protocol.

In the Authentication step choose "Use the NoMachine login", and choose a key for the appropriate server. The keys can be downloaded from <http://inst.eecs.berkeley.edu/pub/nxkeys/>. hpse.client.id_dsa.key should be used for the hpse servers, and t7400.client.id_dsa.key for the t7400 servers.

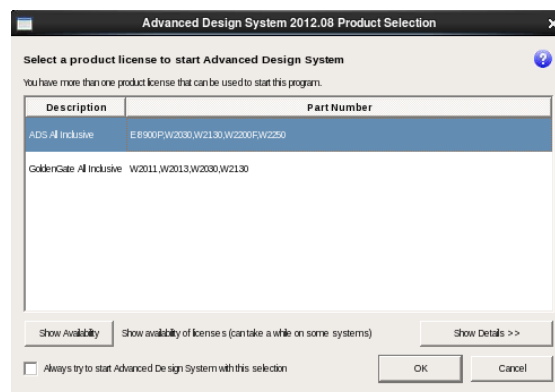
Choose "Don't use a proxy".

2. X2Go: <http://wiki.x2go.org/doku.php/download:start>

You don't need to have a key for this client.

After connecting, open a terminal and type "ads_local" to start the program.

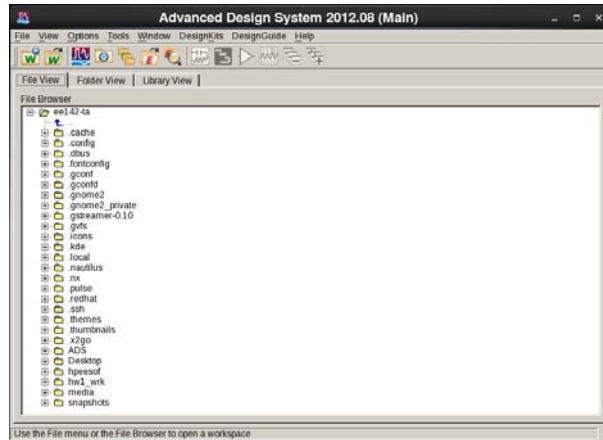
At the first time you run the program, the following window will appear:



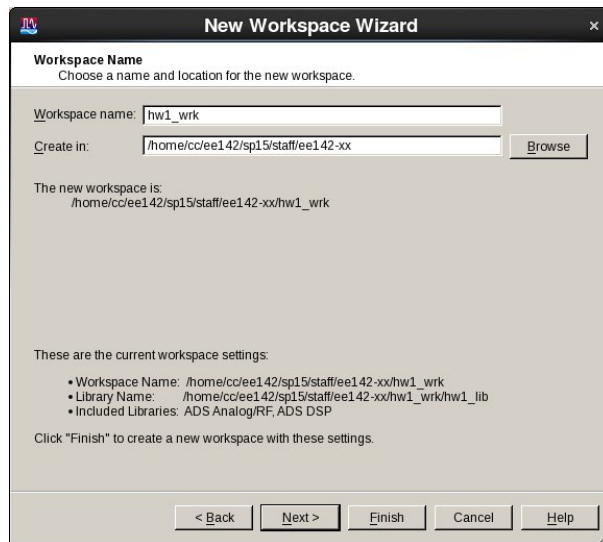
Choose the first option (ADS All inclusive) and check the "Always try to start Advanced Design System with this selection" to remove this window for the next times.

2. ADS overview

When opening ADS, you'll see the main window:

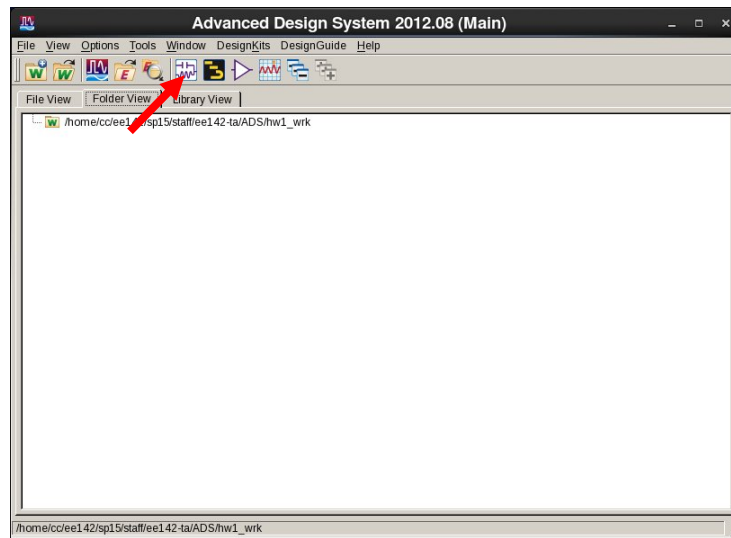


The designs in ADS are combined under "workspaces". Each workspace can contain multiple design schematics. To open a new workspace: File -> New -> Workspace. The workspace name should end with `_wrk` and cannot contain spaces:



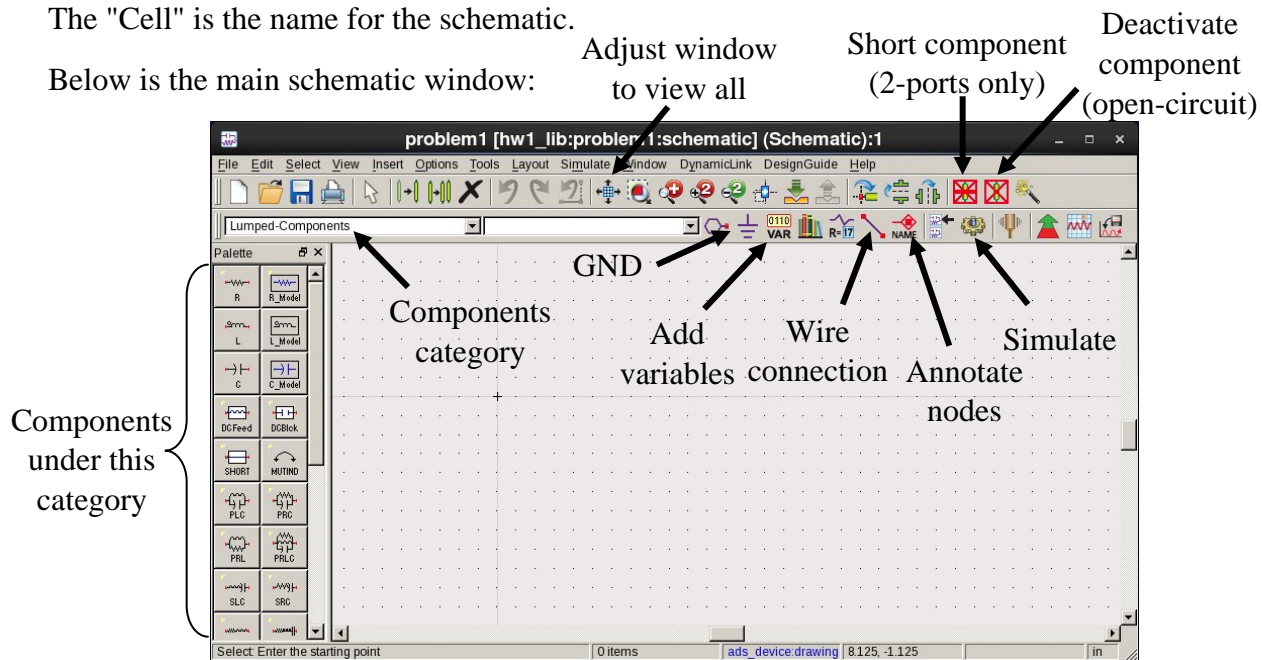
Click Finish in this window to create the workspace.

To add a schematic to the workspace, click on "New schematic window":



The "Cell" is the name for the schematic.

Below is the main schematic window:



The most useful buttons are annotated above.

To add a component, choose the appropriate category and click on the component on the left side of the screen.

In ADS, simulation setups and variables are also added to the schematic as components.

Useful components:

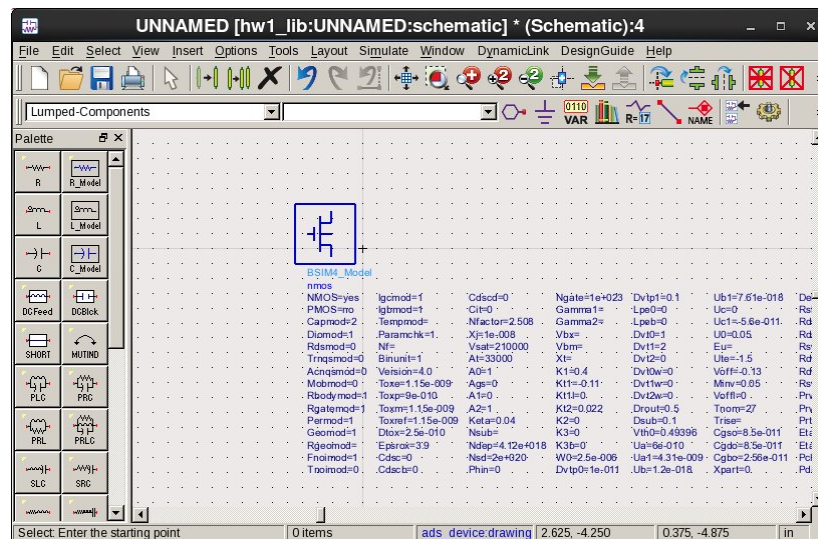
Category	Components
Lumped-Components	R,L,C DCfeed - infinite inductor DCBlock - infinite capacitor TF, TF3 - transformers
Sources-Controlled	VCVS, VCCS, ...
Sources-Freq Domain	V_DC, I_DC - DC sources V_AC, I_AC - AC sources (for AC simulation)
Sources-Time Domain	Sine, Pulse - for transient simulations
Simulation-DC	DC - DC simulation
Simulation-AC	AC - AC simulation
Simulation-Transient	Trans - Transient simulation
Any simulation category	PrmSwp - Parametric sweep Sweep Plan - setup parameters for parametric sweep
Probe Components	I_probe - current probe
Devices - BJT	BJTNPN, BJTPNP - bipolar transistors BJT - transistor model
Devices - MOS	NMOS, PMOS - MOS transistors for level1-level3 spice models Level 1, Level 2, Level 3 - spice models Bsim 4 (N,P) - MOS transistors for BSIM4 models Bsim4 (M) - BSIM4 model

3. Importing a PTM model for 32nm devices

In this class we will use the Predictive Transistor Model (PTM) BSIM4 models (<http://ptm.asu.edu/>). The models are under <http://ptm.asu.edu/latest.html>. We will use the 32nm PTM HP model from September 30, 2008 (see link below). To import the 32nm models into ADS:

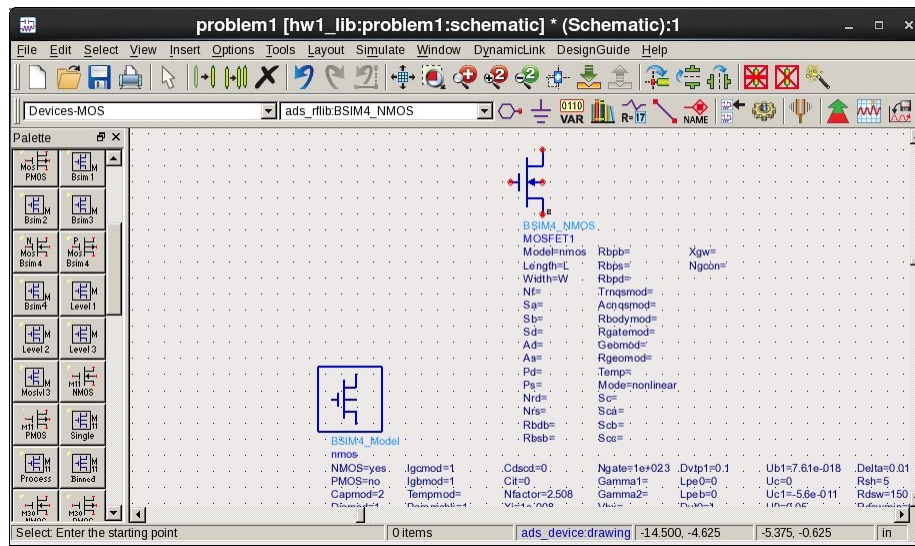
- In a schematic window File > Import
- File type: "Netlist File" (default)
- Import file name - download the model file from http://ptm.asu.edu/modelcard/HP/32nm_HP.pm
- Click on "Options"
- Input Netlist Dialect: HSPICE
- Translated Output Format: ADS Schematic
- Click OK, and OK in the Import window to import the file.

A new schematic window will be created with the nmos and pmos model components:



Copy the model components to your schematic. When placing a new BSIM4 device, the default model for NMOS will be BSIM4M1. Change the name to nmos (the imported model).

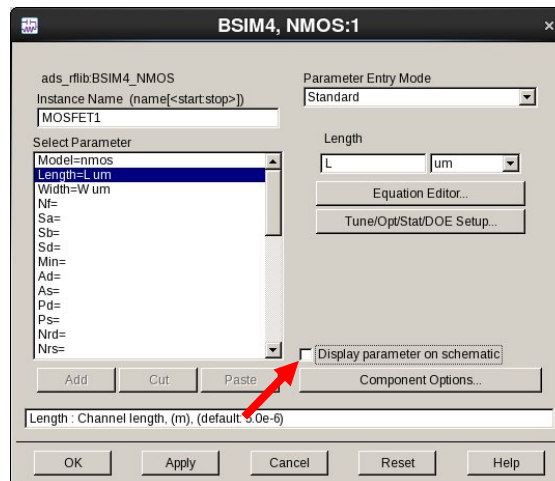
Now the schematic should look as follows:



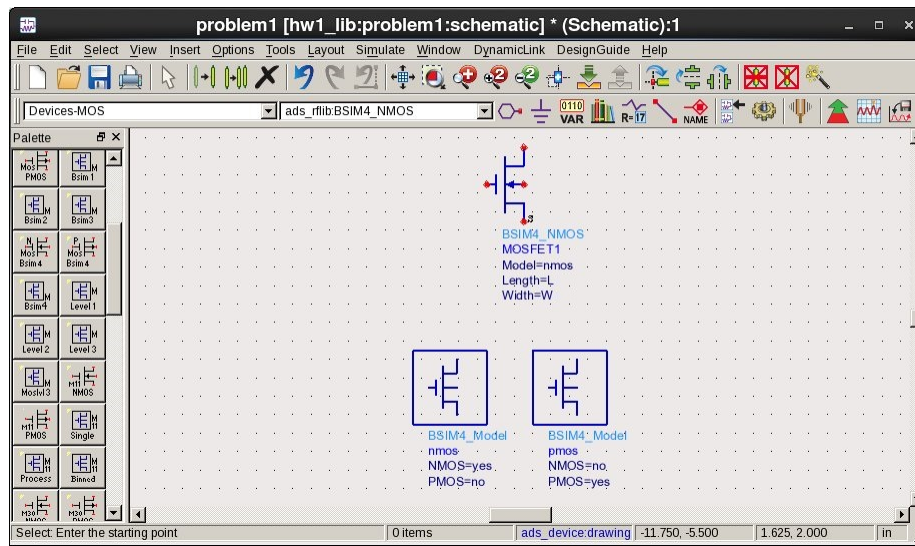
To remove the unnecessary parameters from the display:

- Double click on the component
- Select "Component Options"
- Parameter visibility -> Clear all

Now you can choose which parameters to display on the screen:



So when showing the important parameters only, the schematic is:

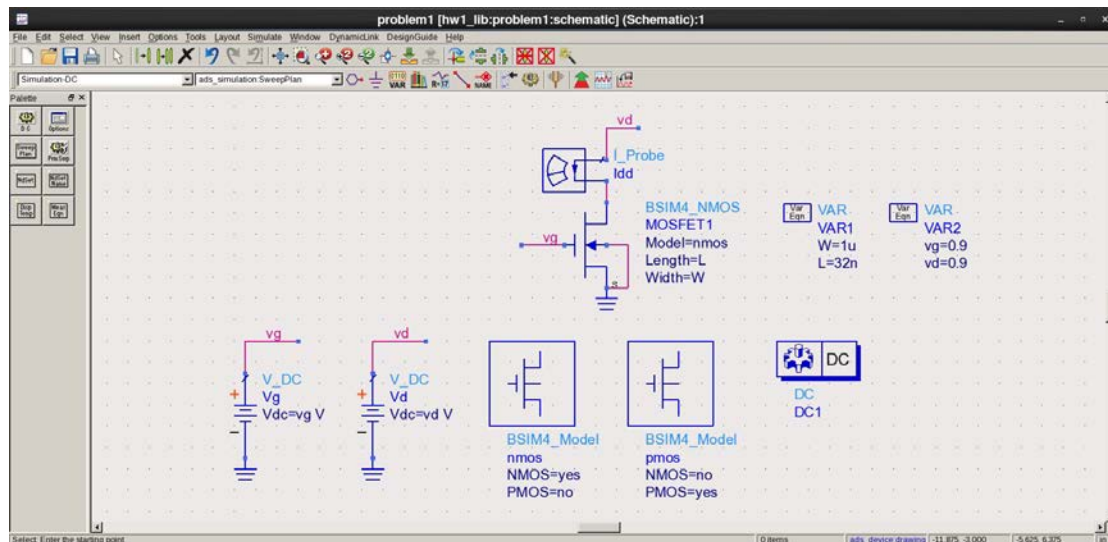


To move the component parameters on the screen:

F5 -> Click on the parameters -> Move them on the screen

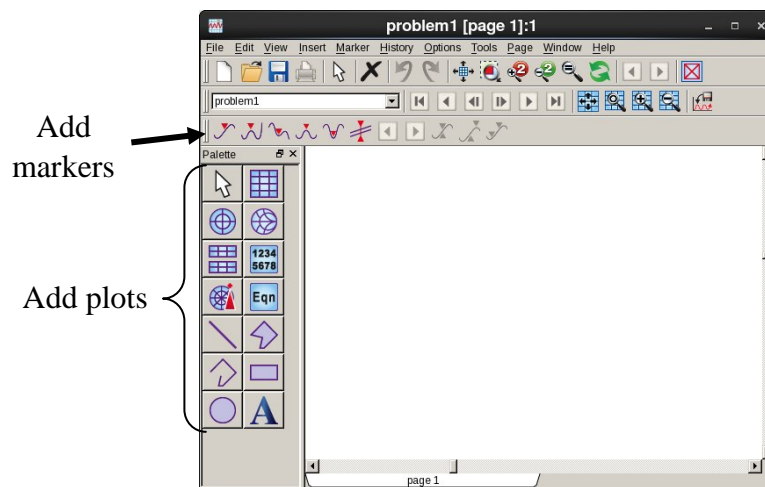
4. DC simulation

Let's start with the simplest DC simulation (transistor size is $W/L=1\mu/32\text{nm}$):



Note that the simulation setup (DC1) and the variables (VAR1, VAR2) are placed as components to the schematic.

When clicking "simulate" a simulation results window will open:



On the left you can see different types of plots that you can choose from. The most useful are:



Rectangular plot



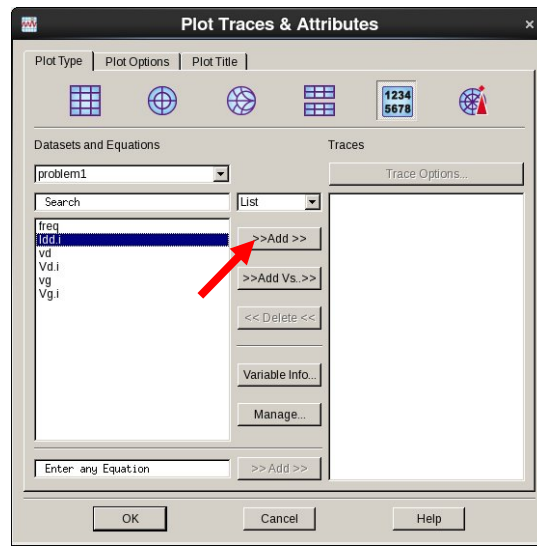
List (a table)



Smith chart (we'll study it later on in this class)

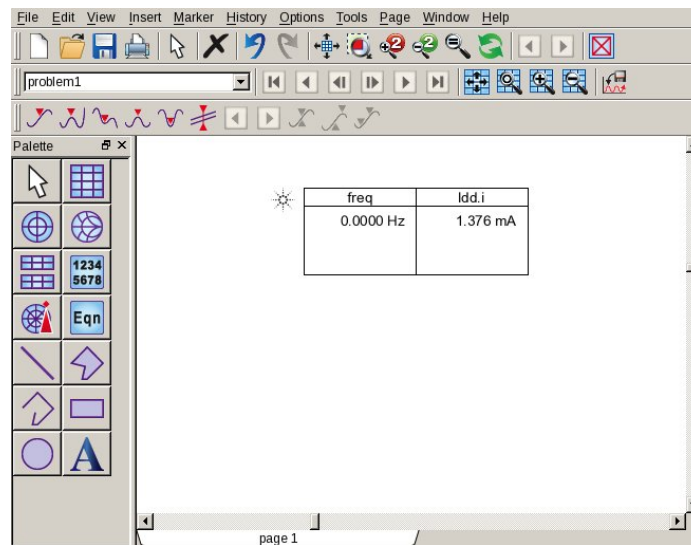
Defining output equations

Now we only have a single result. We'll choose List:



On the left you can see the available results. Choose Idd.i (the drain current) and click Add.

You will see the following table:



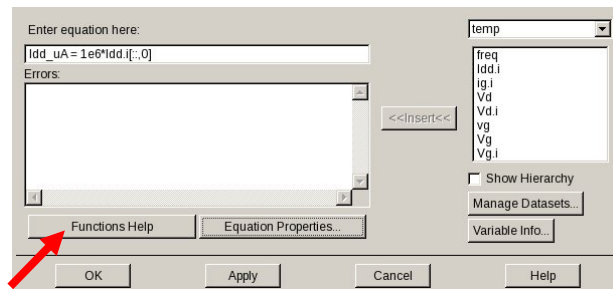
DC simulation - parametric sweep

Now we would like to sweep the gate voltage. To perform a parametric sweep, place the PrmSwp component. Choose *vg* as the parameter to sweep, and specify the sweep range. Under the Simulations tab, you should specify which simulations should be performed for this sweep. Type DC1 (the only simulation so far).

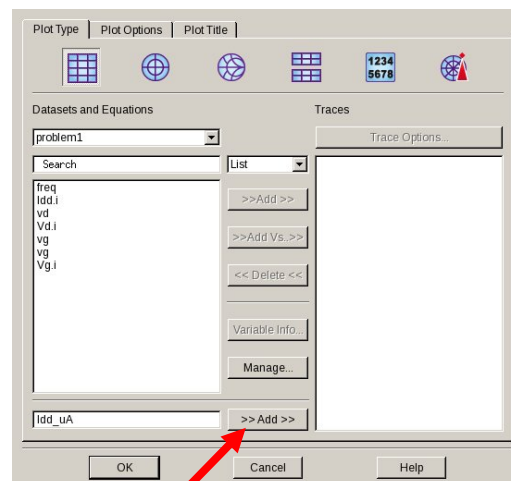
Now the current is a function of the frequency and the gate voltage. The output expressions in ADS are multidimensional matrixes. Each sweep introduces a dimension, and the frequency is an inherent dimension in DC and frequency-domain simulations. The independent (swept) variable is associated with the data (for example, the DC drain current). This independent is propagated through expressions, so that the results of equations are automatically plotted or listed against the relevant swept variable.

To display the current as a function of the gate voltage only, we'll define an output equation. Click on Equation on the left, place it into the window, and type `Idd_uA = 1e6*Idd.i[:,0]`. This is similar to a Matlab syntax `A[:,1]`. Here the first index is the swept gate voltage, and the second index is the frequency. We have chosen the first frequency (which is the only one in DC simulation).

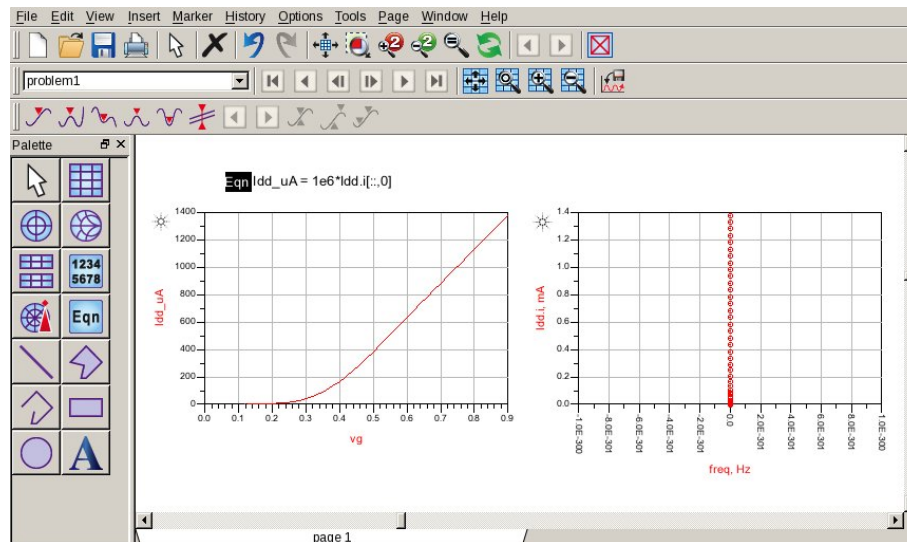
For the expressions in the equations you can use standard mathematical operations, and built-in functions. To get help on the functions, you can use the ADS help under the equation window:



To display the new `Idd_uA` result, click on Rectangular plot on the left, and type the name in the bottom, and click Add:

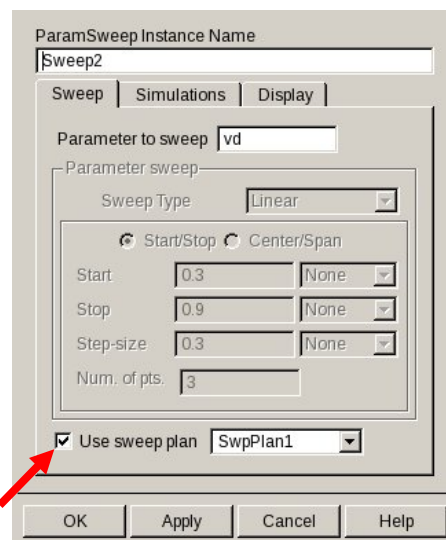


The new equation result compared to the direct $I_{dd,i}$:

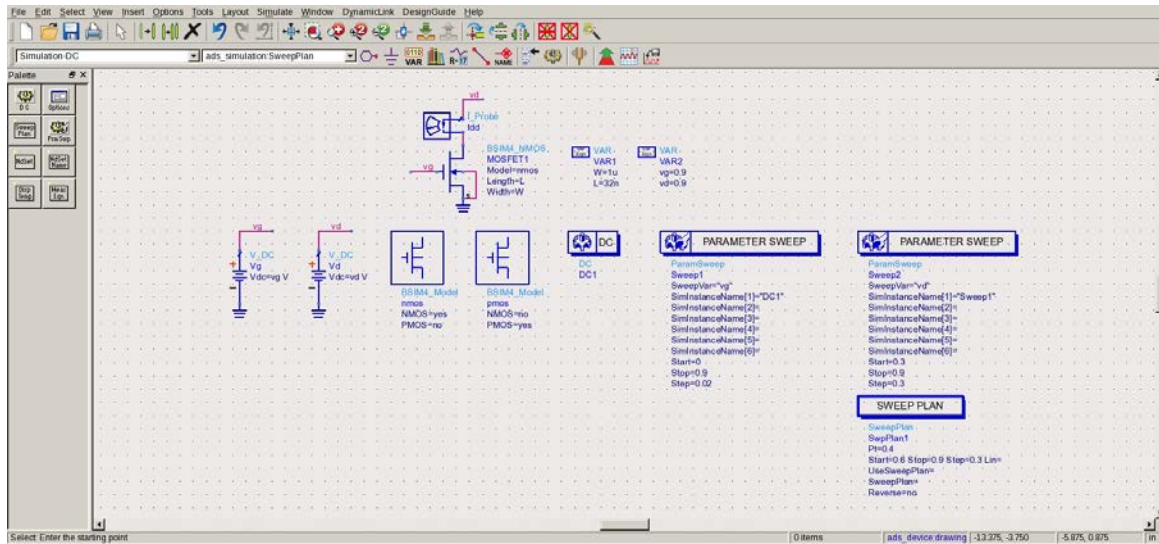


Now we would like to sweep more than one parameter. For example, we will sweep the gate voltage for several different drain voltages. We will place an additional PrmSwp component for the drain voltage. Under the Simulations tab, we will write Sweep1, which is the gate voltage parametric sweep.

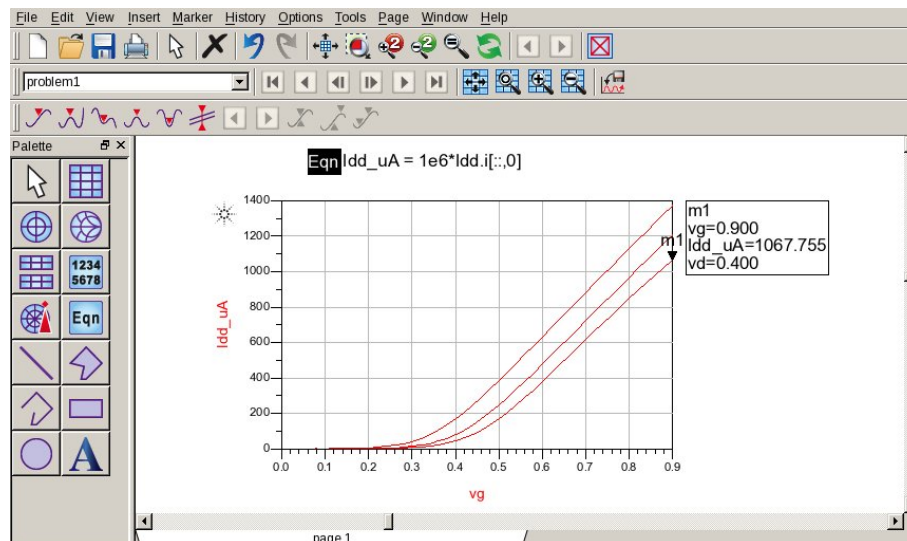
In order to set a more complicated sweep type than a simple linear sweep, we can use the "Sweep Plan" component. In a sweep plan you can add several sweep types (single point, linear, and log). To use a sweep plan in a parametric sweep, select the "Use sweep plan" option and choose the right plan.



The new schematic will look as follows:



And the simulation result:

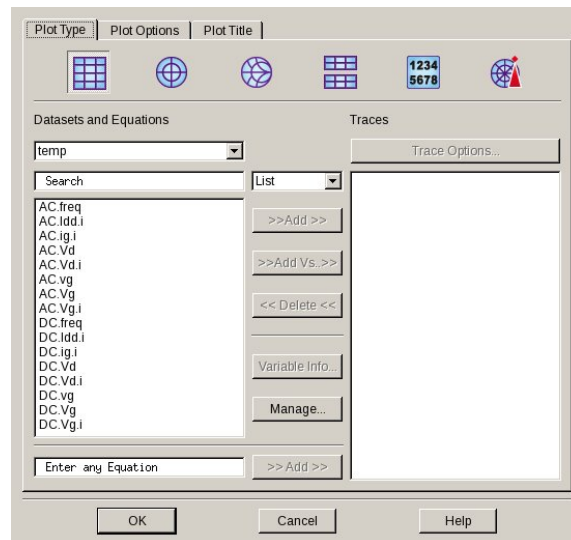


DC simulation - assignment

- For the same transistor size $W/L=1\mu\text{m}/32\text{nm}$ and $V_{dd}=0.9\text{V}$, generate plots of I_{ds} vs V_{gs} (for 3 different V_{ds} voltages), and I_{ds} vs V_{ds} (for 3 different V_{gs} voltages).
When sweeping V_{ds} , use three values of V_{gs} , one below "threshold", a little above "threshold", and $V_{gs} = V_{dd}$. For V_{gs} sweeps, pick a V_{ds} point below saturation, around the knee of saturation, and $V_{ds} = V_{dd}$.
- Repeat part 1 with long-channel device. Compare the performance of short-channel and long-channel devices.

5. AC simulation

After performing the AC simulation, the syntax of the output expressions will change. When adding a new plot, you'll see that now you can choose between AC and DC simulation results:



When adding more simulations / parametric sweeps, the syntax may also change. You can always see the correct syntax by adding a new plot or double-clicking an old one.

AC simulation - assignment

3. For the same parameters of question1, plot the g_m versus V_{gs} and V_{ds} . Explain qualitatively the behavior of these plots.
4. For the same parameters of question1, plot r_o and A_v (low-frequency voltage gain) versus V_{ds} . Explain qualitatively the behavior of these plots. Compare the voltage gain result to long-channel devices.
5. **(242A)** Plot the f_T of the device versus V_{gs} . Explain qualitatively the behavior of these plots. Compare the result to long-channel devices. You may want to use the DCfeed component (infinite inductor) and the cross() function (finds a zero-crossing of a function).