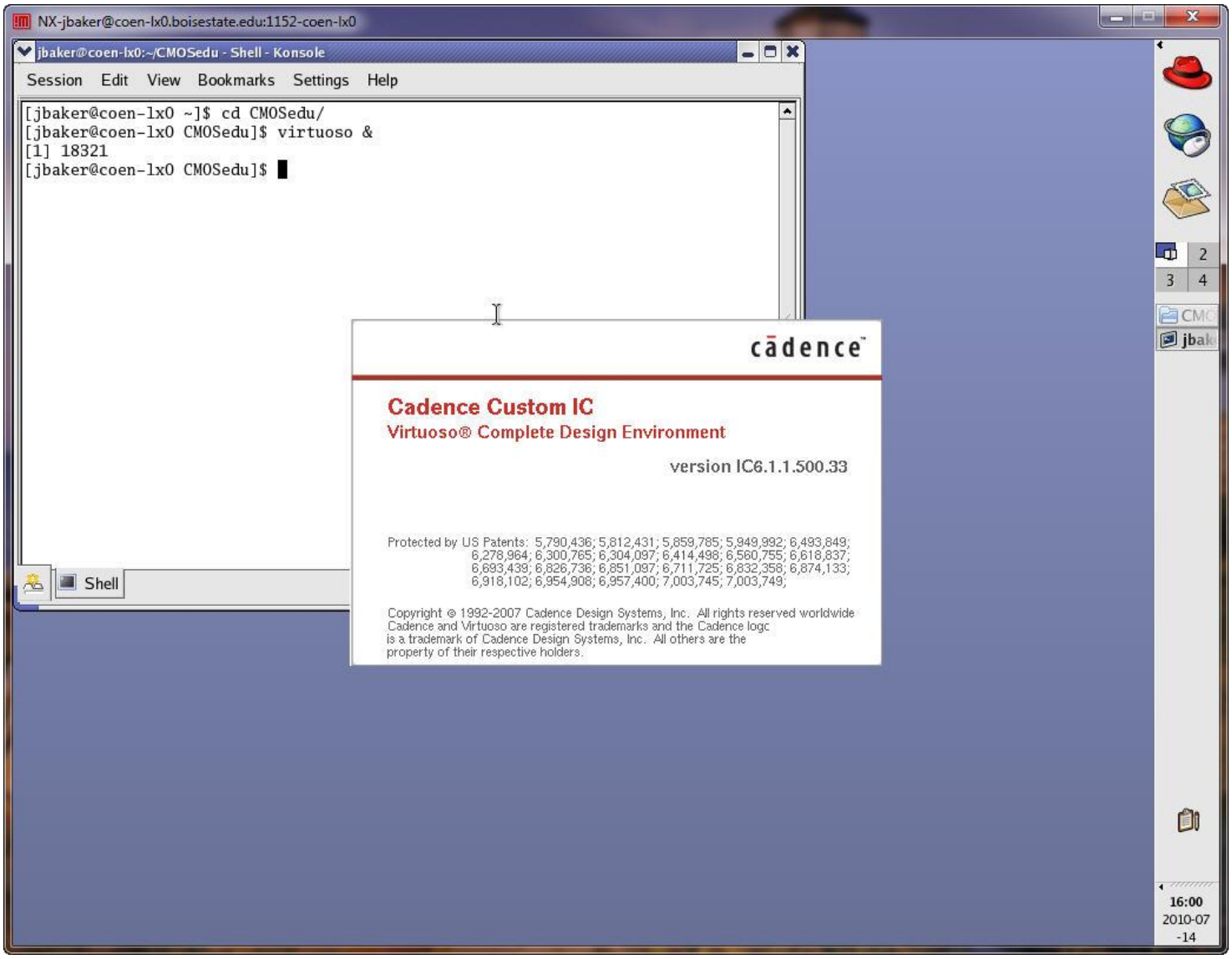
Cadence Design System Tutorials from CMOSedu.com (Return)

Tutorial 2 – Layout and simulating the IV curves of PMOS and NMOS devices

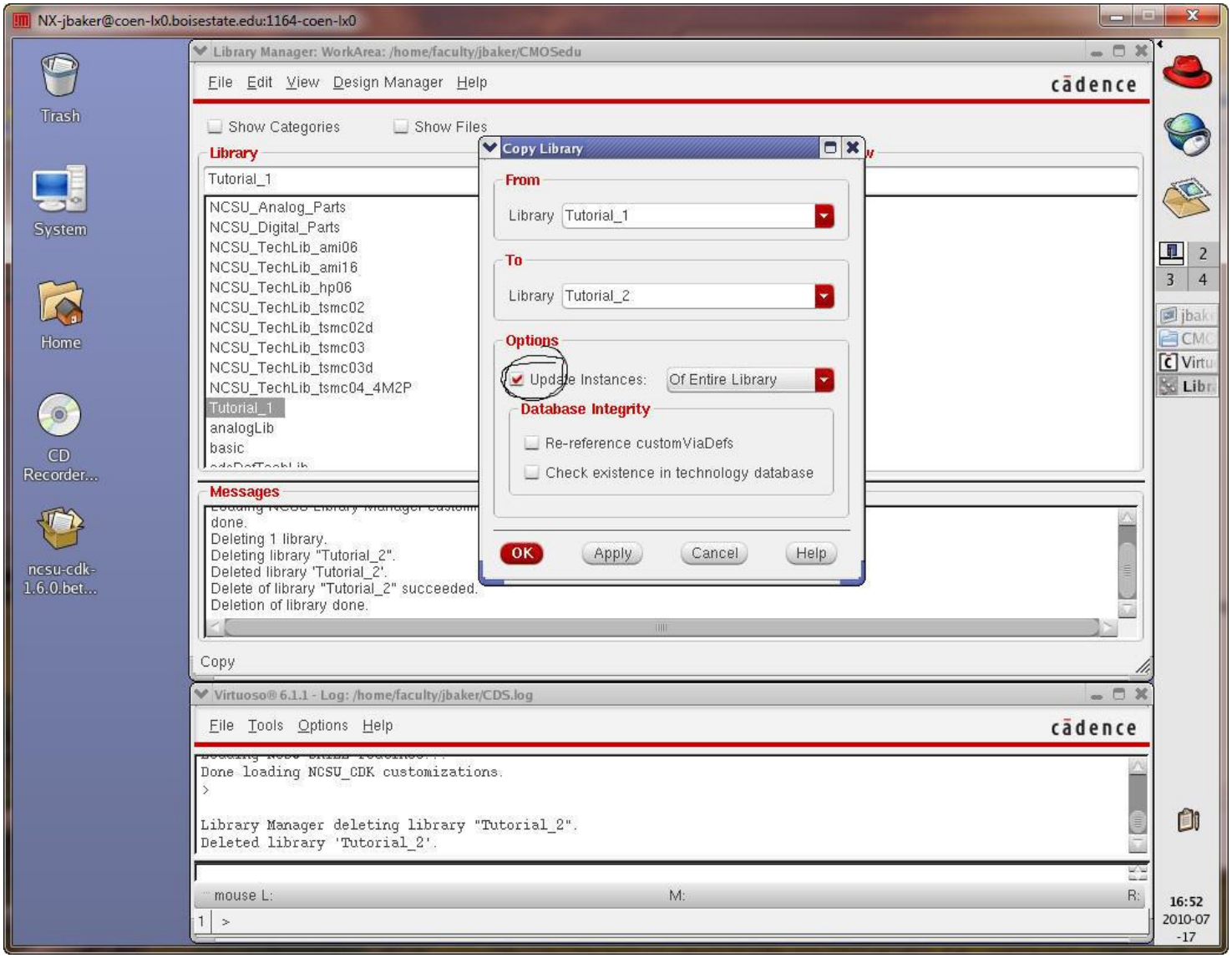
In this tutorial we'll lay out and simulate the operation of NMOS and PMOS transistors using the C5 process from Tutorial 1.
Note that we'll simulate both the schematic and layout (extracted) views.

To begin Tutorial 2 first start Cadence's Virtuoso in the CMOSedu directory as seen below.

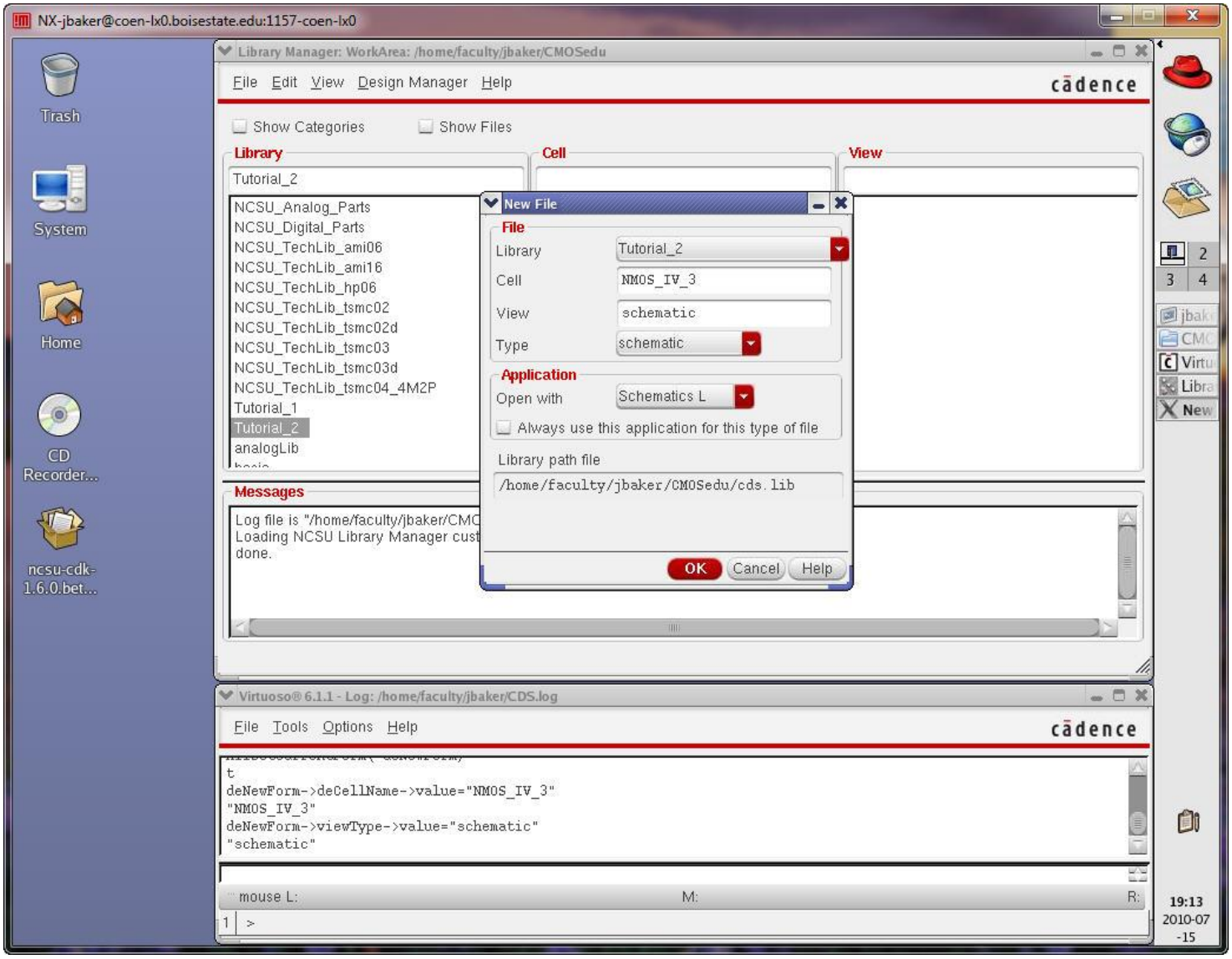


Next copy Tutorial_1 into a new library called Tutorial_2 by right clicking on Tutorial_1 (below).
Note that if you don't select Update Instances many of the cells in Tutorial_2 will reference Tutorial_1 (we want the libraries to be self-contained).

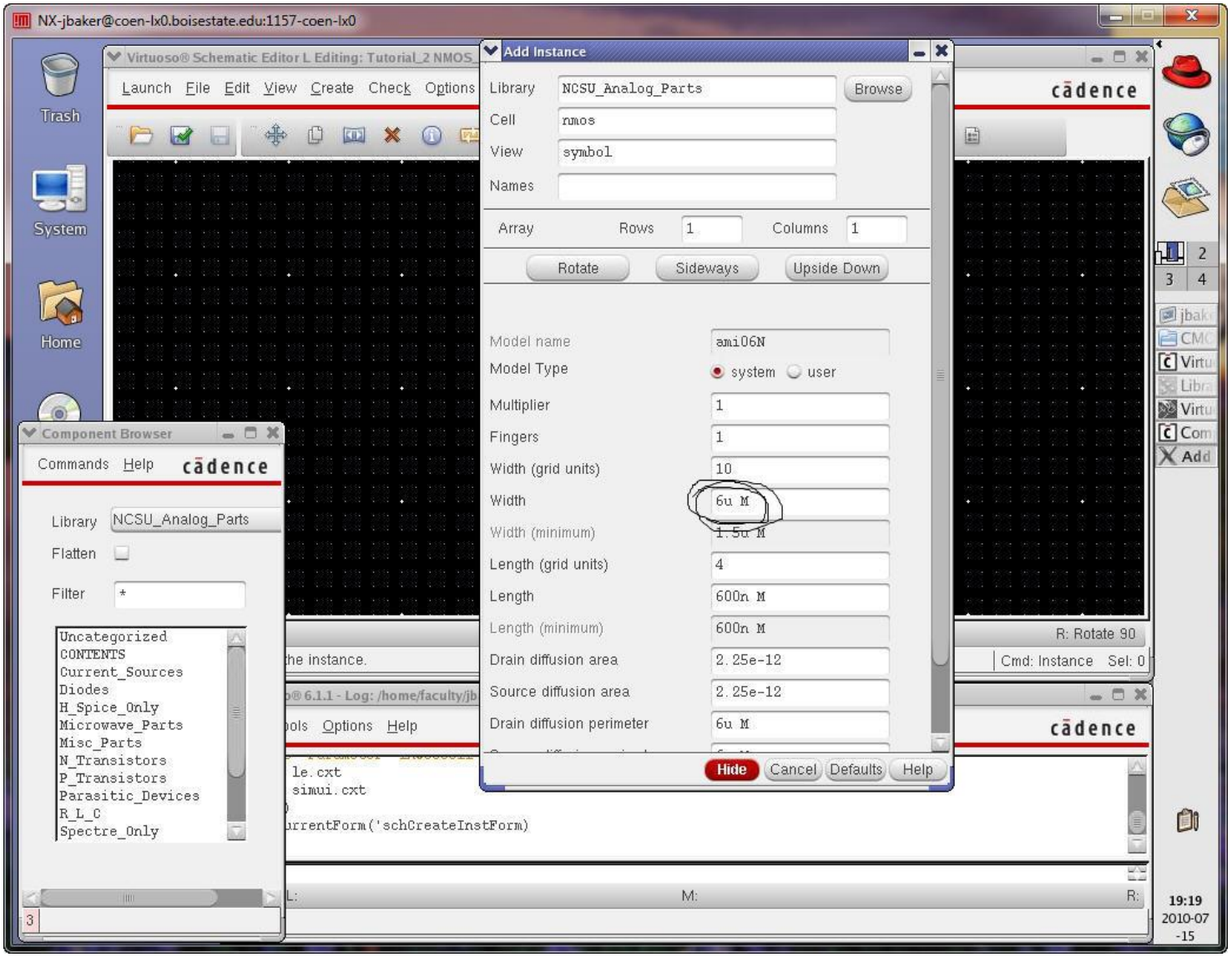
After pressing OK you will be asked where the Tutorial_2 directory should be placed. The default location should be the same as Tutorial_1, that is,
CMOSedu, so hit OK a second time to create the library.

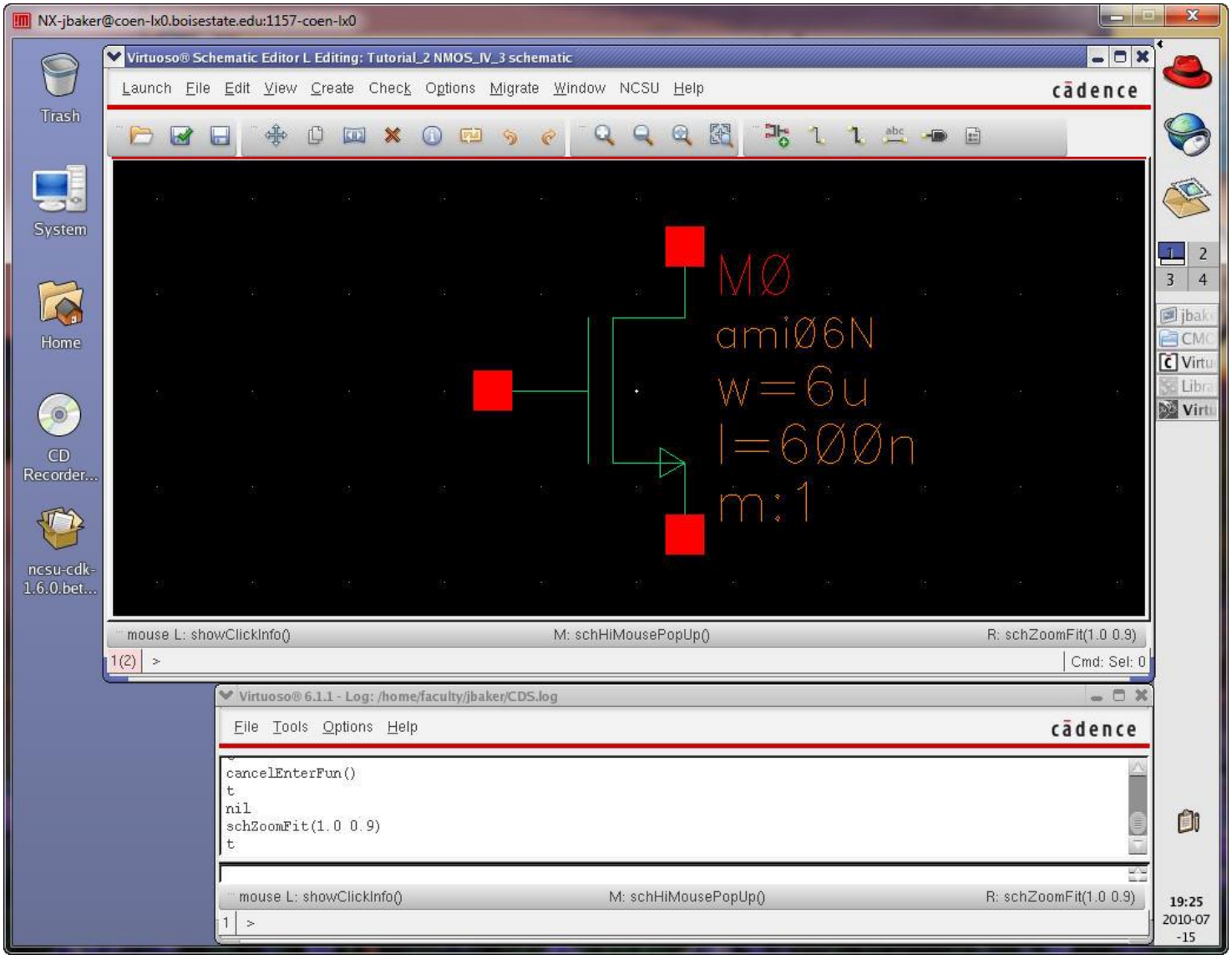


Next, with the Tutorial_2 Library highlighted, select in the Library Manager File -> New -> Cell View.
Create a schematic cell called NMOS_IV_3 as seen below. The 3 indicates we'll use a 3 terminal transistor.
Using a 3-terminal NMOS the body (p-well or p-substrate) is connected to gnd!

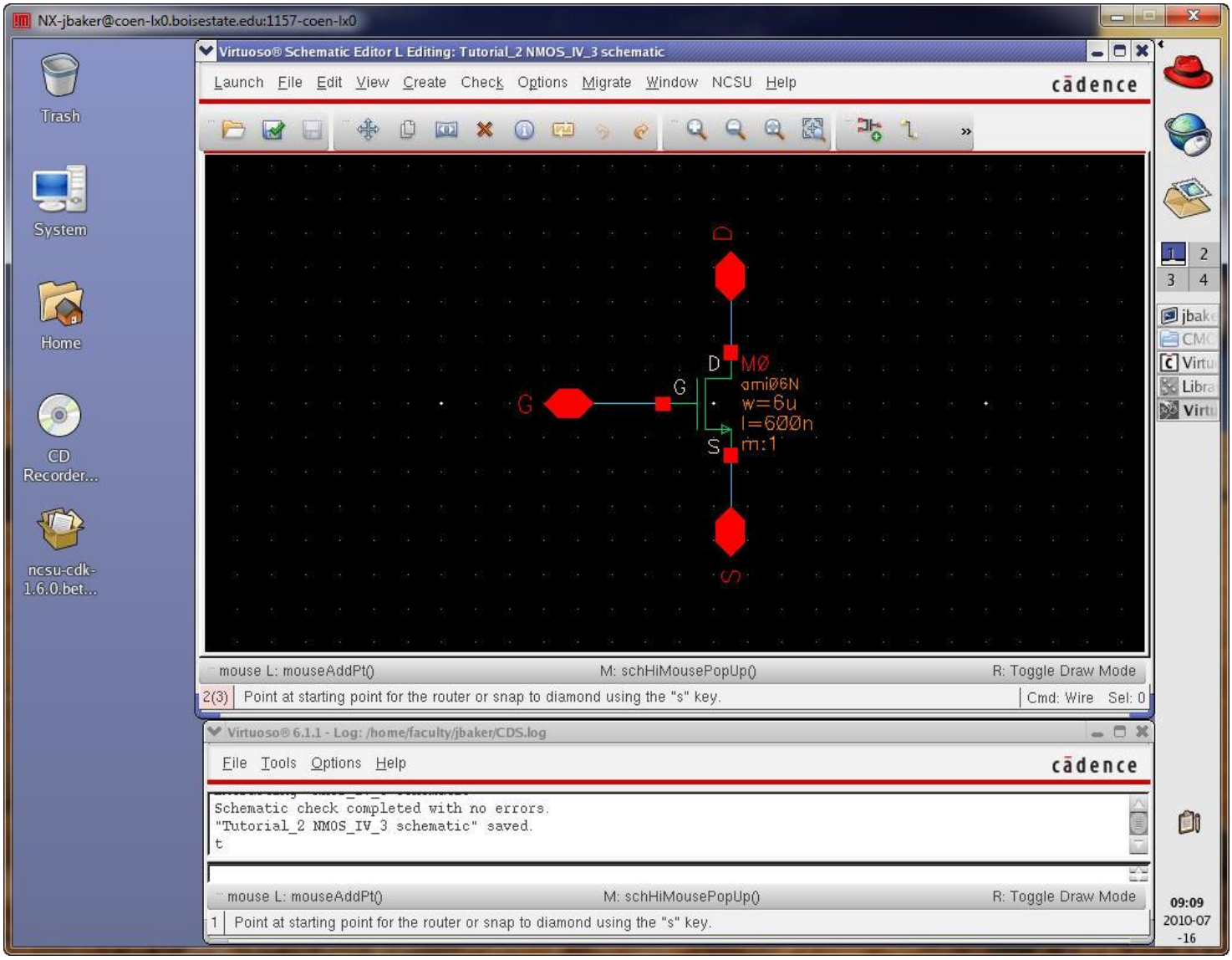


Press i (or use the menu Create -> Instance) and select, in the Component Browser Window, NCSU_Analog_Parts, N_Transistors, nmos.
Remember that often Windows are opened that are behind other windows so you have to use the Task bar to bring them to the front.
Set the transistors Width to 6u and leave its length at 600 nm.
Place the transistor and fit the screen (press **f**).

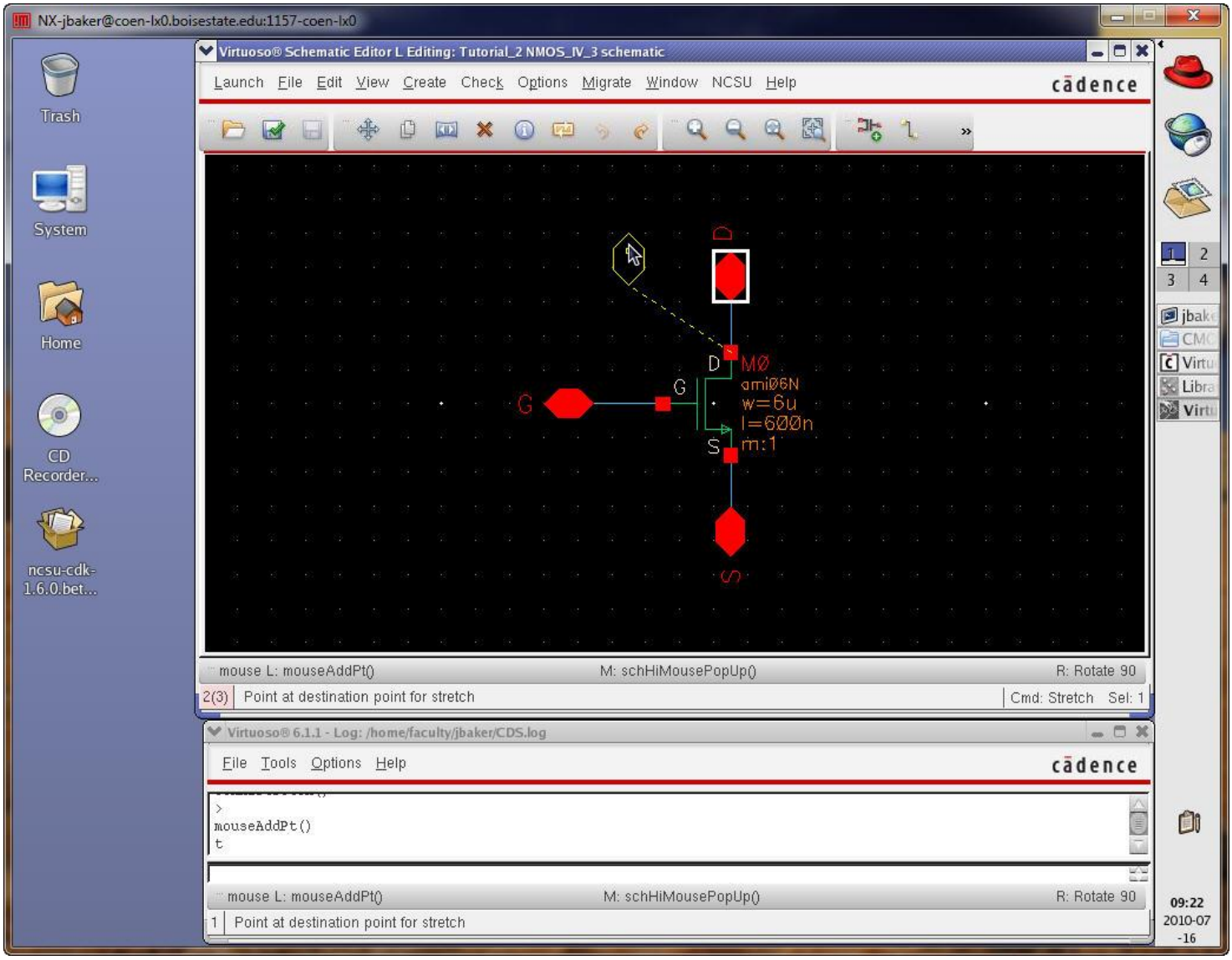

NX-jbaker@coen-lx0.boisestate.edu:1157-coen-lx0
Trash
System
Home
Virtuoso® Schematic Editor L Editing: Tutorial_2 NMOS_
Launch File Edit View Create Check Options
cadence
Add Instance
Library NCSU_Analog_Parts
Browse
Cell nmos
View symbol
Names
Array Rows 1 Columns 1
Rotate Sideways Upside Down
Model name ami06N
Model Type system user
Multiplier 1
Fingers 1
Width (grid units) 10
Width 6u M
Width (minimum) 1.5u M
Length (grid units) 4
Length 600n M
Length (minimum) 600n M
Drain diffusion area 2.25e-12
Source diffusion area 2.25e-12
Drain diffusion perimeter 6u M
Hide Cancel Defaults Help
Component Browser
Commands Help
cadence
Library NCSU_Analog_Parts
Flatten
Filter *
Uncategorized
CONTENTS
Current_Sources
Diodes
H_Spice_Only
Microwave_Parts
Misc_Parts
N_Transistors
P_Transistors
Parasitic_Devices
R_L_C
Spectre_Only
R: Rotate 90
Cmd: Instance Sel: 0
19:19
2010-07
-15



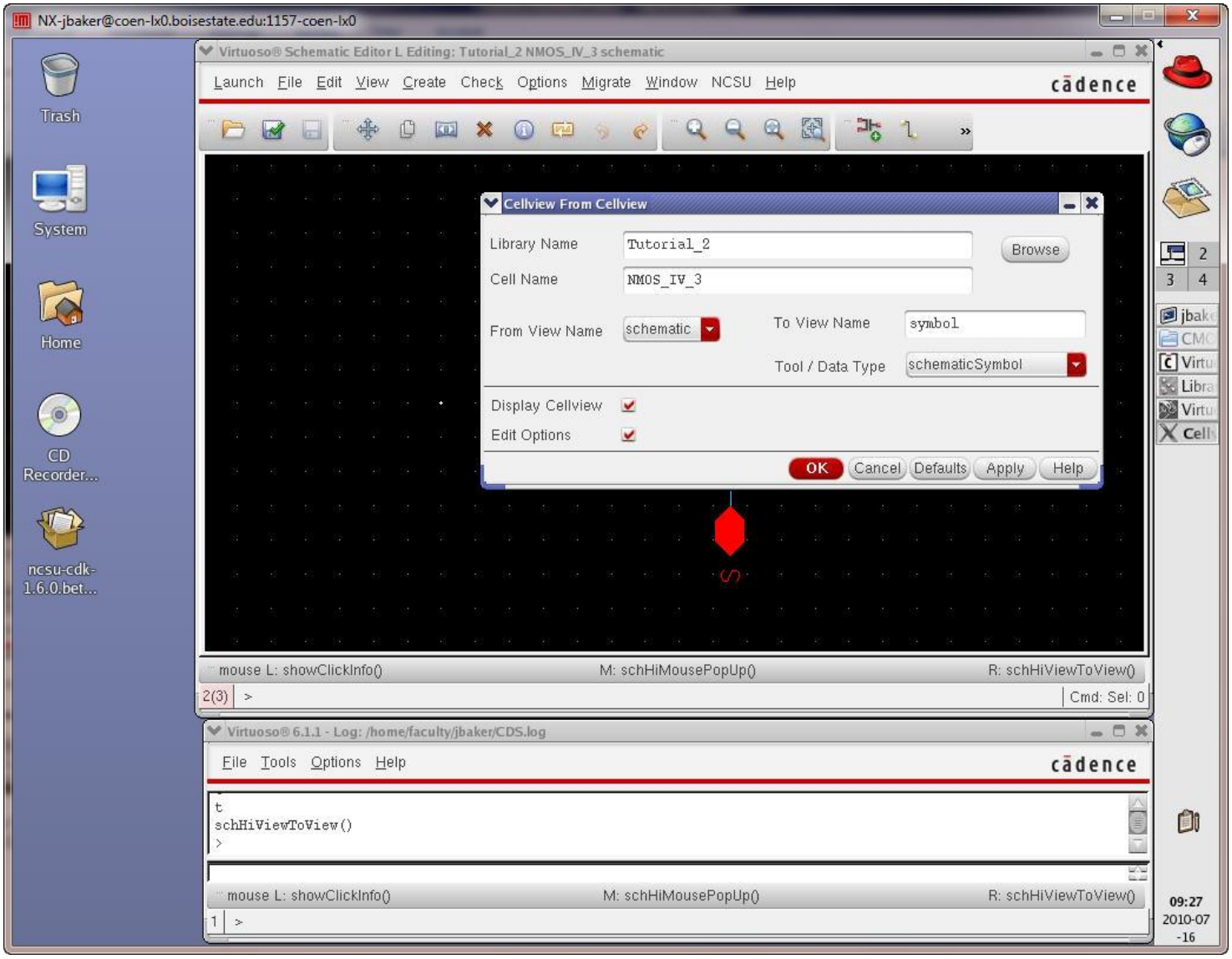
Next add Pins with the direction inputOutput and wires as seen below.
Remember that right+clicking the mouse when instantiating the Pin rotates it.
Z (zoom out by 2) and **f** (fit) can be used while instantiating the pin.
w can be used to wire the Pin to the MOSFET.
Check and Save the schematic when finished.



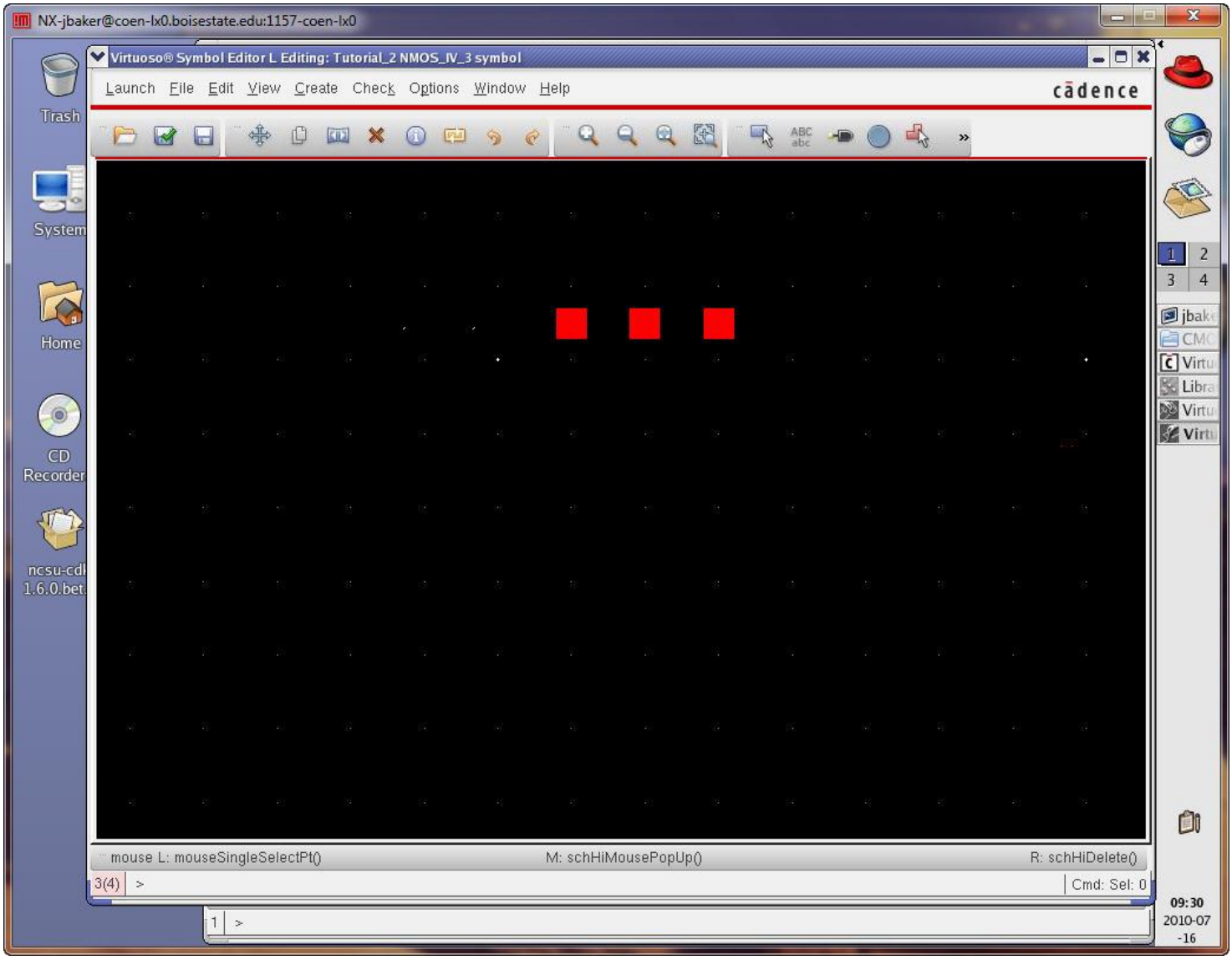
Before making a symbol for this schematic let's talk about stretching (Bindkey **s**).
We know that to move an object we select it then press **m**.
Stretching is similar except that the object stays attached to the wire or whatever you haven't selected.
Below is an example.



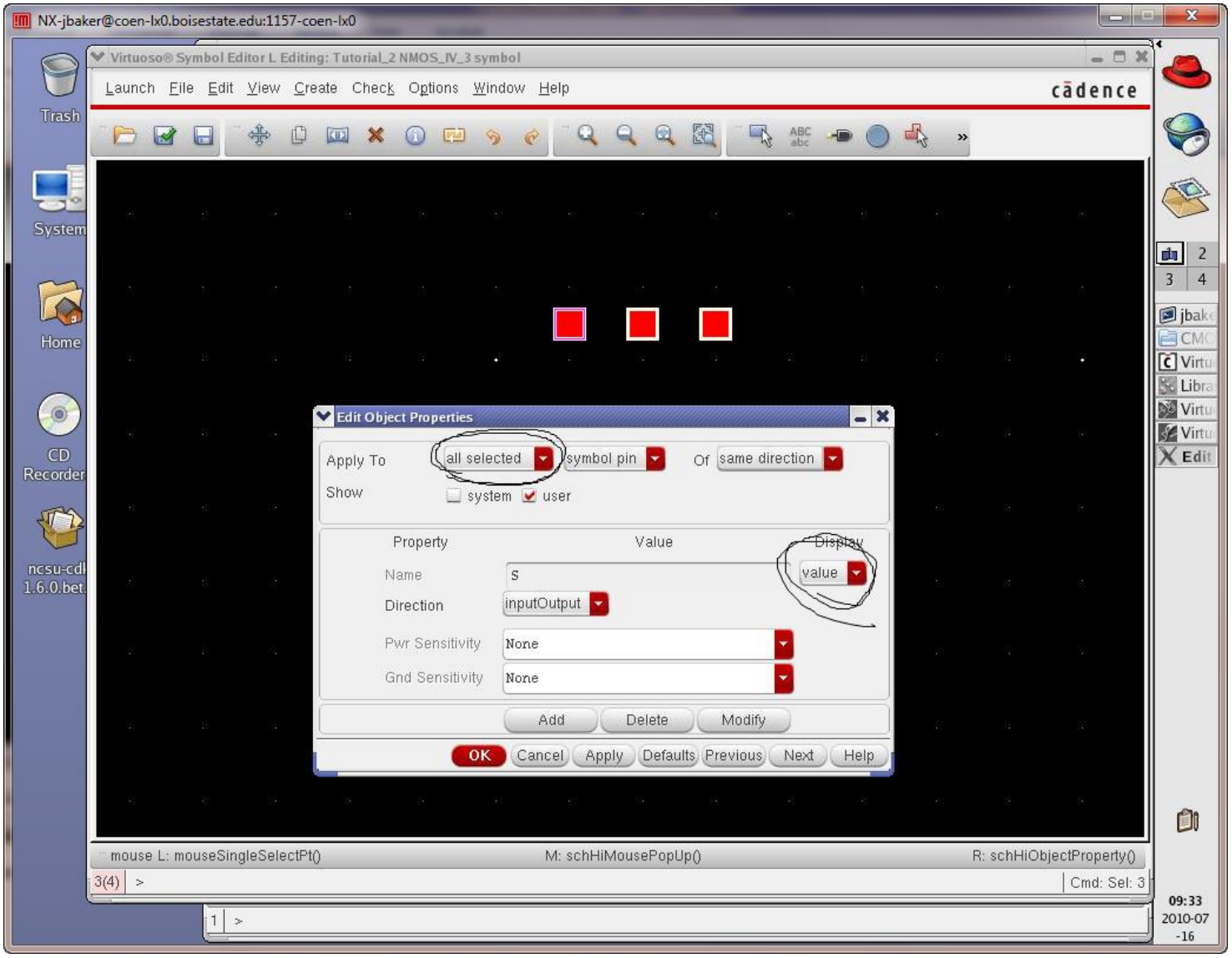
Next create a symbol view for this schematic.
Create -> Cell View -> From Cell View



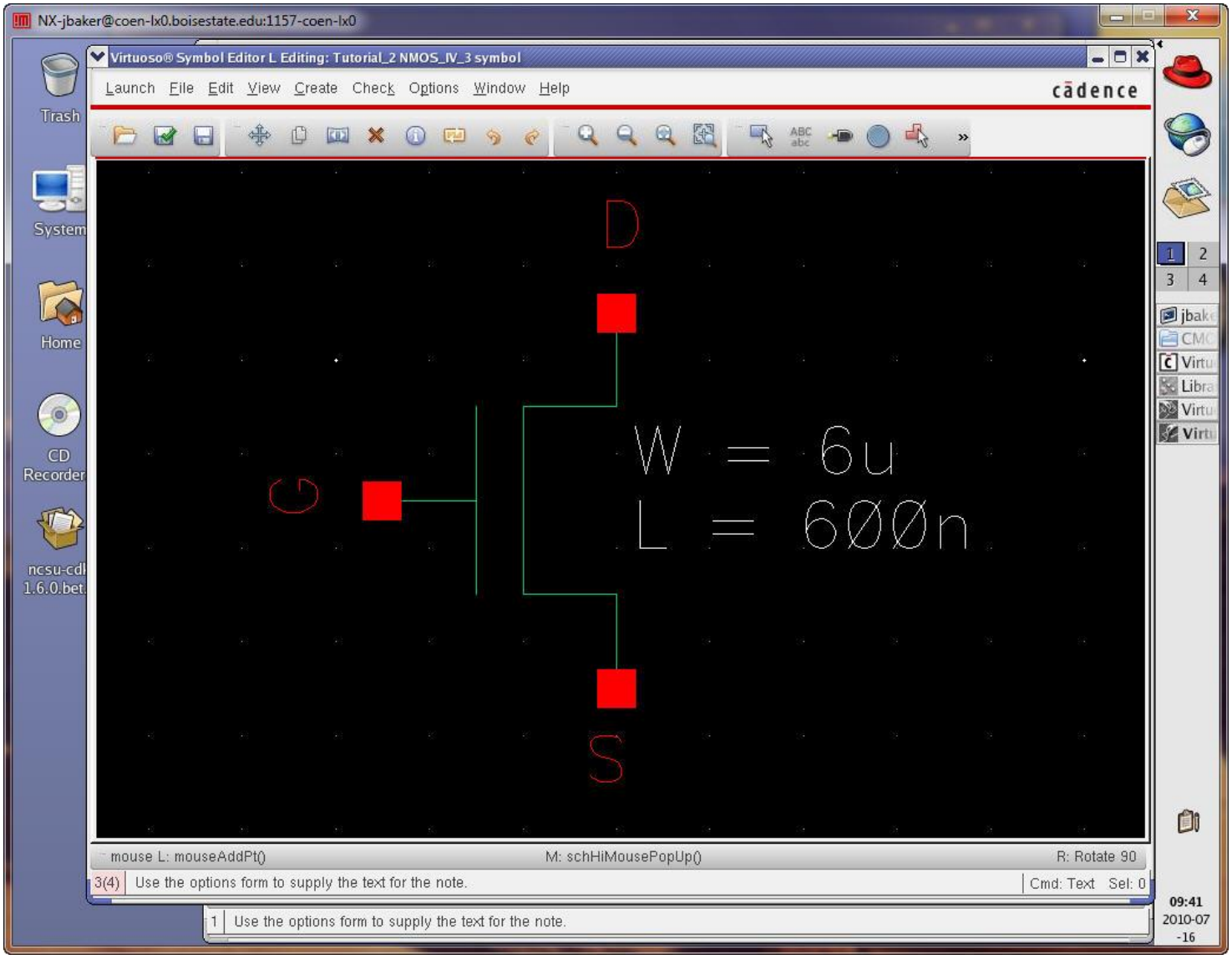
Virtuoso will ask where the pins should be located (it doesn't matter). Hit OK.
Delete everything in the cell except the pins.



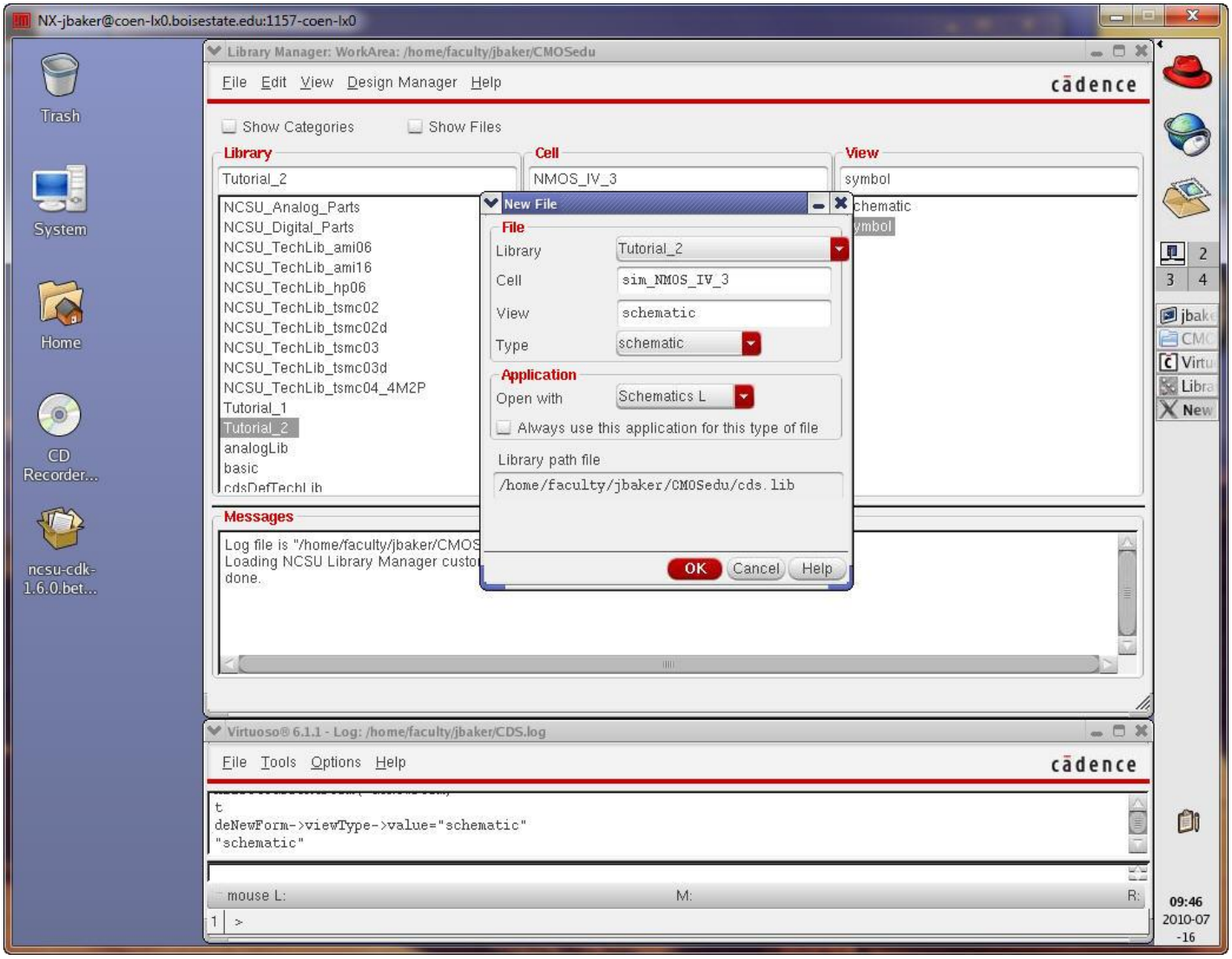
Notice we can't see the pin names.
Select all of the pins and then edit their properties (**q**).
Select "all selected" and Display "value" as seen below.



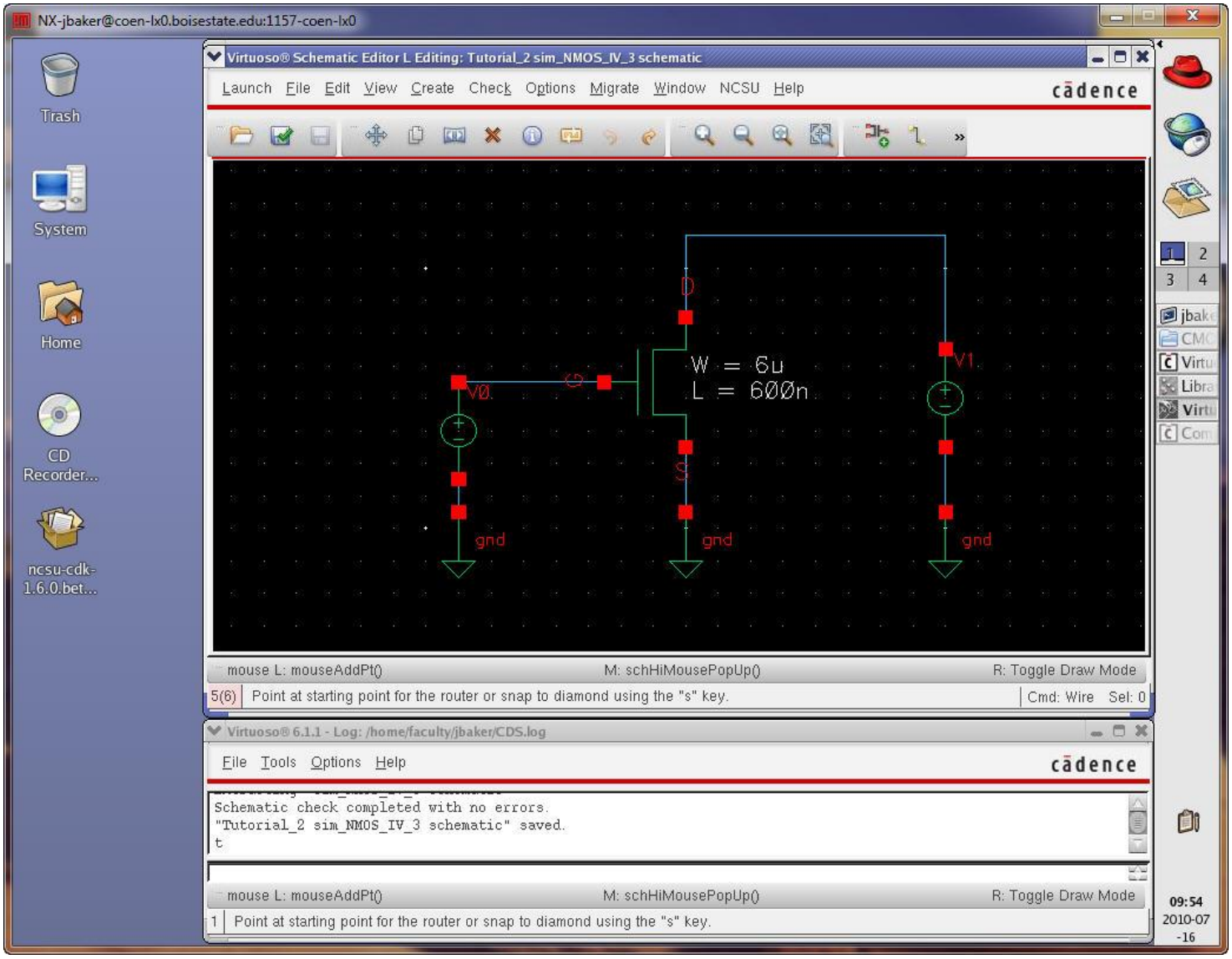
Draw a MOSFET symbol and move the pins (rotate as necessary) to get (something similar to) the following.
Remember to draw a line use Create -> Shape -> Line.
Adding the width and length text using Create -> Note -> Text
When finished "Check and Save"



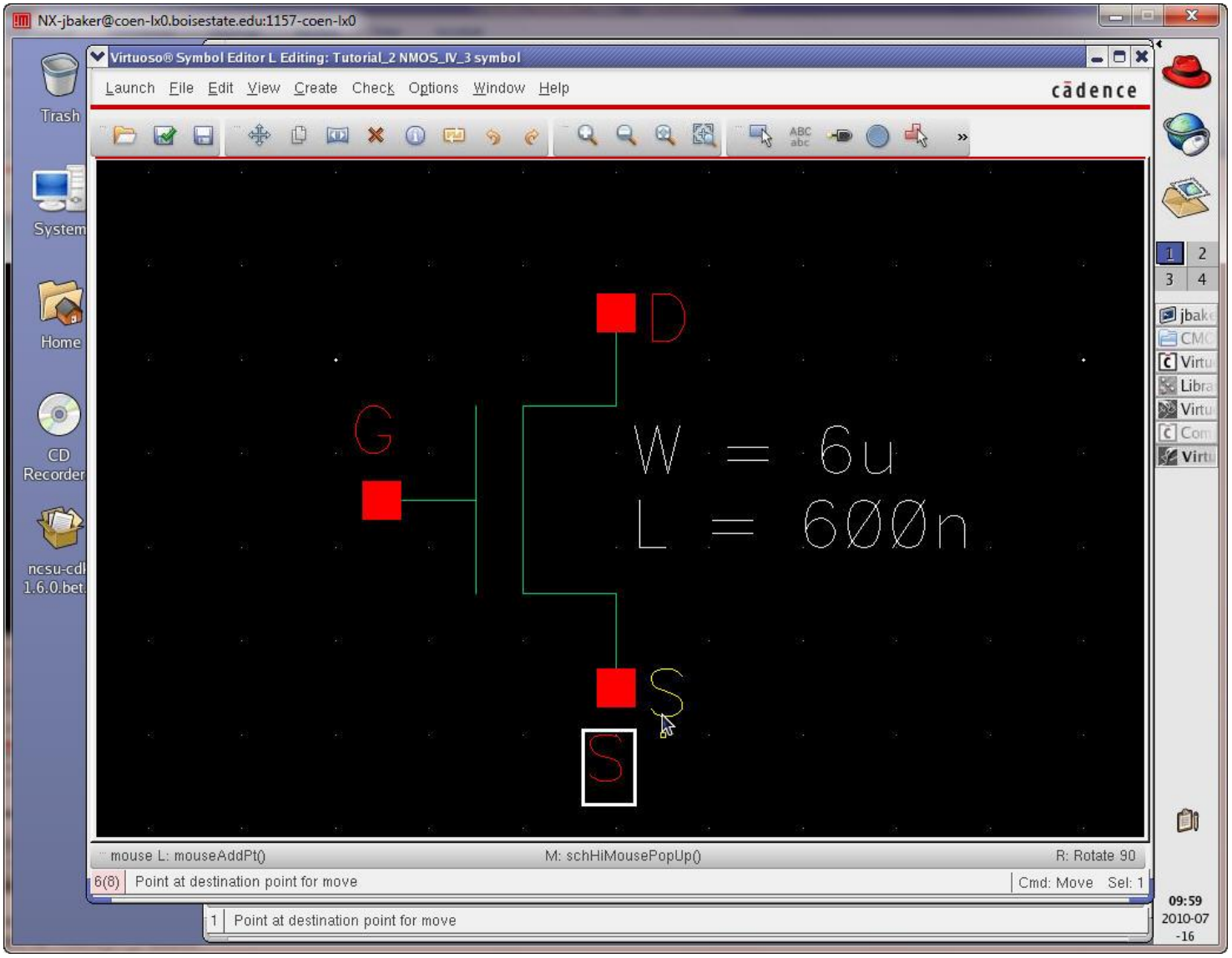
Next, create, File -> New -> Cell View, a cell (schematic) called sim_NMOS_IV_3.

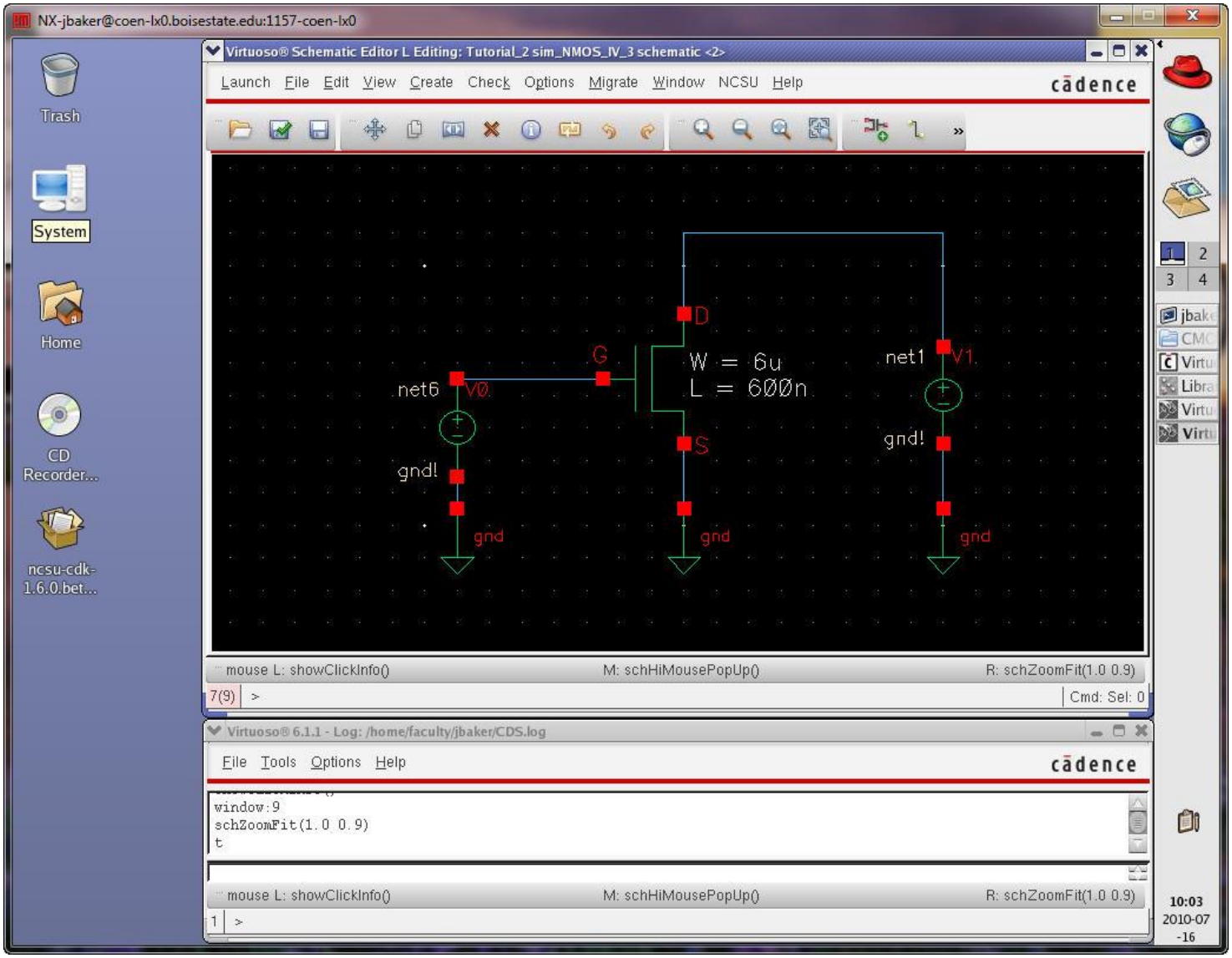


Instantiate the NMOS_IV_Cell View just created and the DC voltage sources seen below (and wire things together).

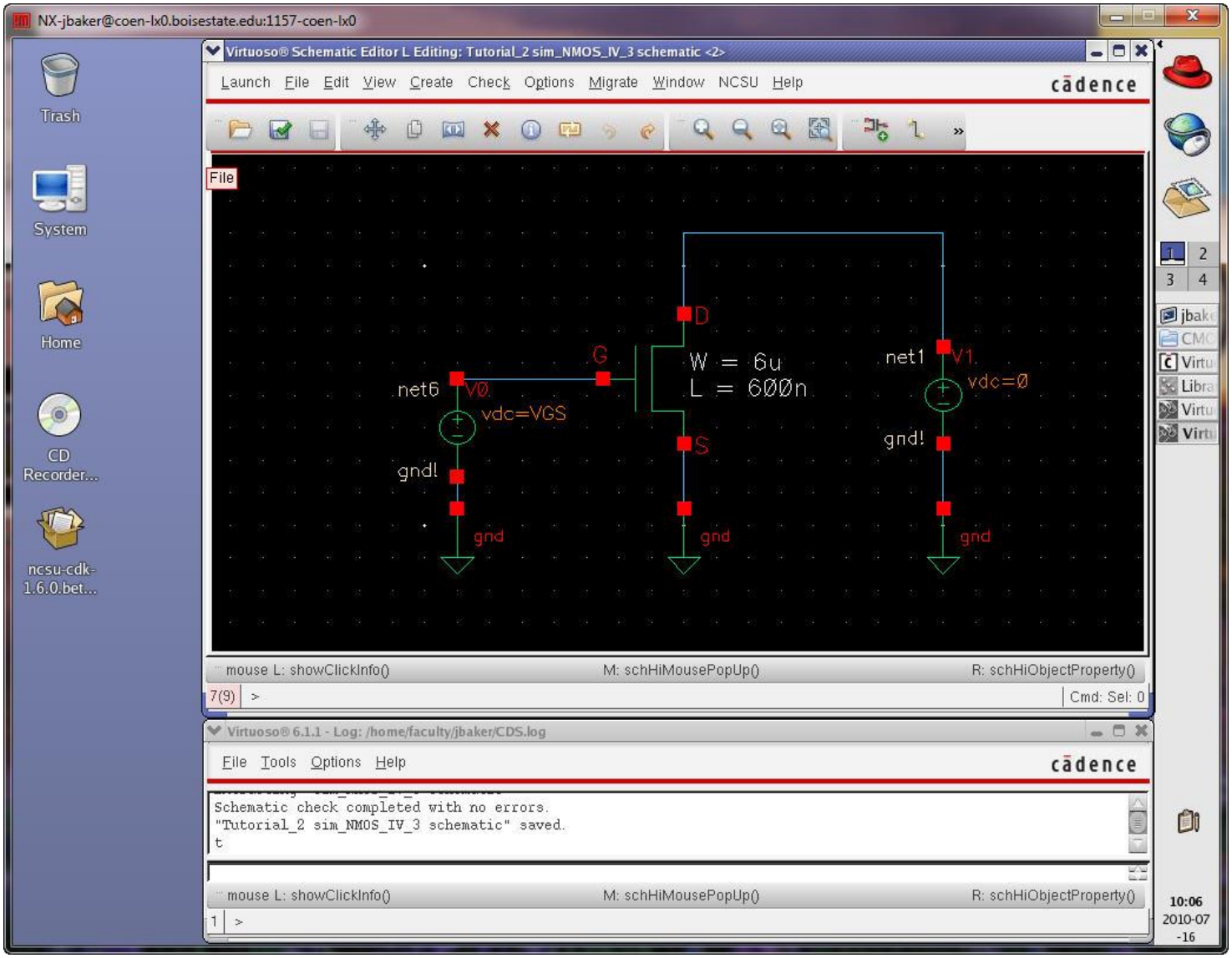


Notice how the wires are drawn through the pin names? This is sloppy so let's fix it.
Use the Library Manager to open the NMOS_IV_3 symbol.
Move the labels as seen below. You'll need to rotate the G once and the S twice (right+click the mouse while moving).
"Check and Save" the Symbol, close it, then return to the sim_NMOS_IV_3 schematic.

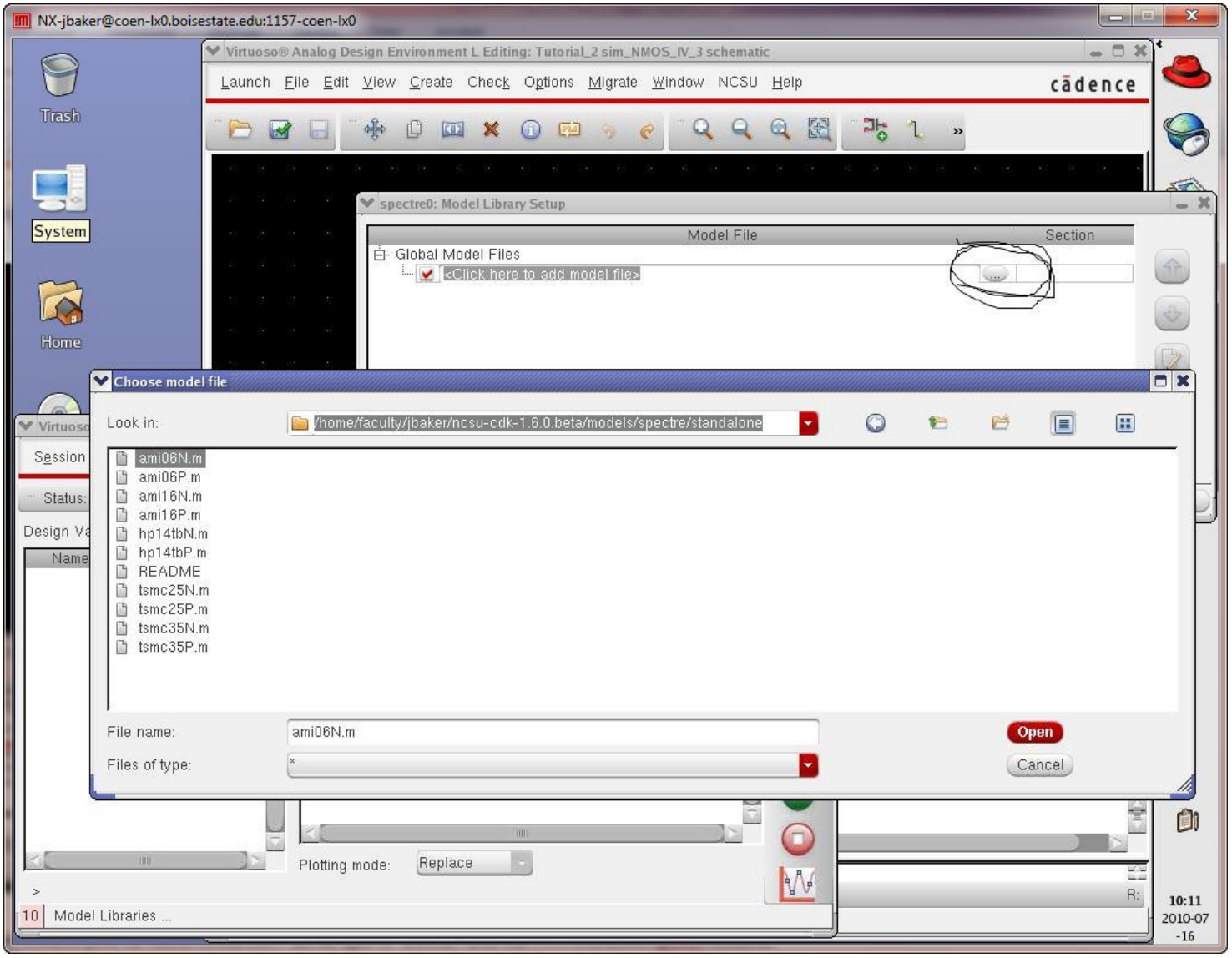

NX-jbaker@coen-lx0.boisestate.edu:1157-coen-lx0
Virtuoso® Symbol Editor L Editing: Tutorial_2 NMOS_IV_3 symbol
Launch File Edit View Create Check Options Window Help
cadence
Trash
System
Home
CD Recorder
D
G
W = 6u
L = 600n
S
S
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Rotate 90
6(8) Point at destination point for move
Cmd: Move Sel: 1
1 Point at destination point for move
09:59
2010-07-16



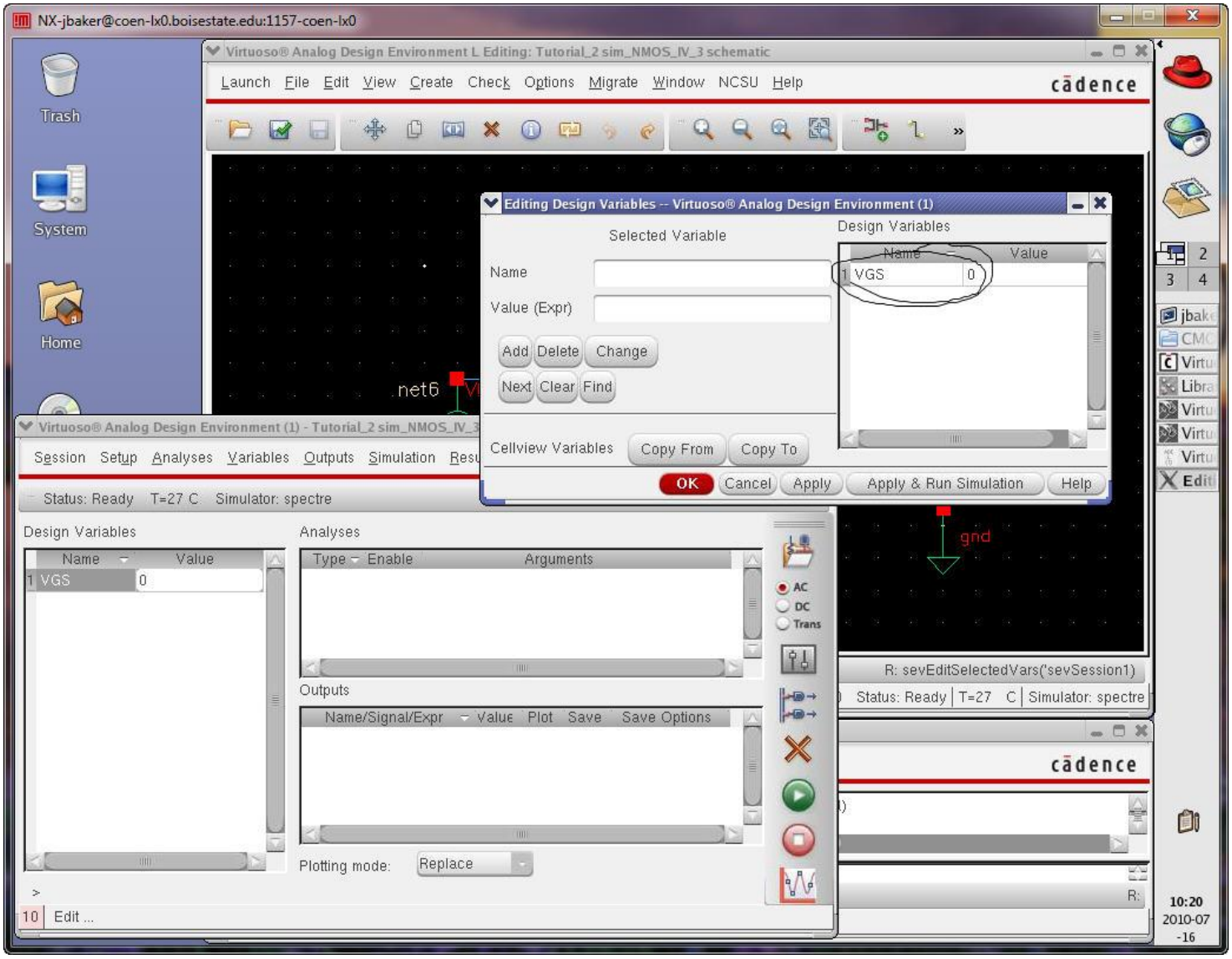
We are almost ready to simulate.
Set the value of V0 to VGS and the value of V1 to 0 as seen below.
Check and Save the schematic.



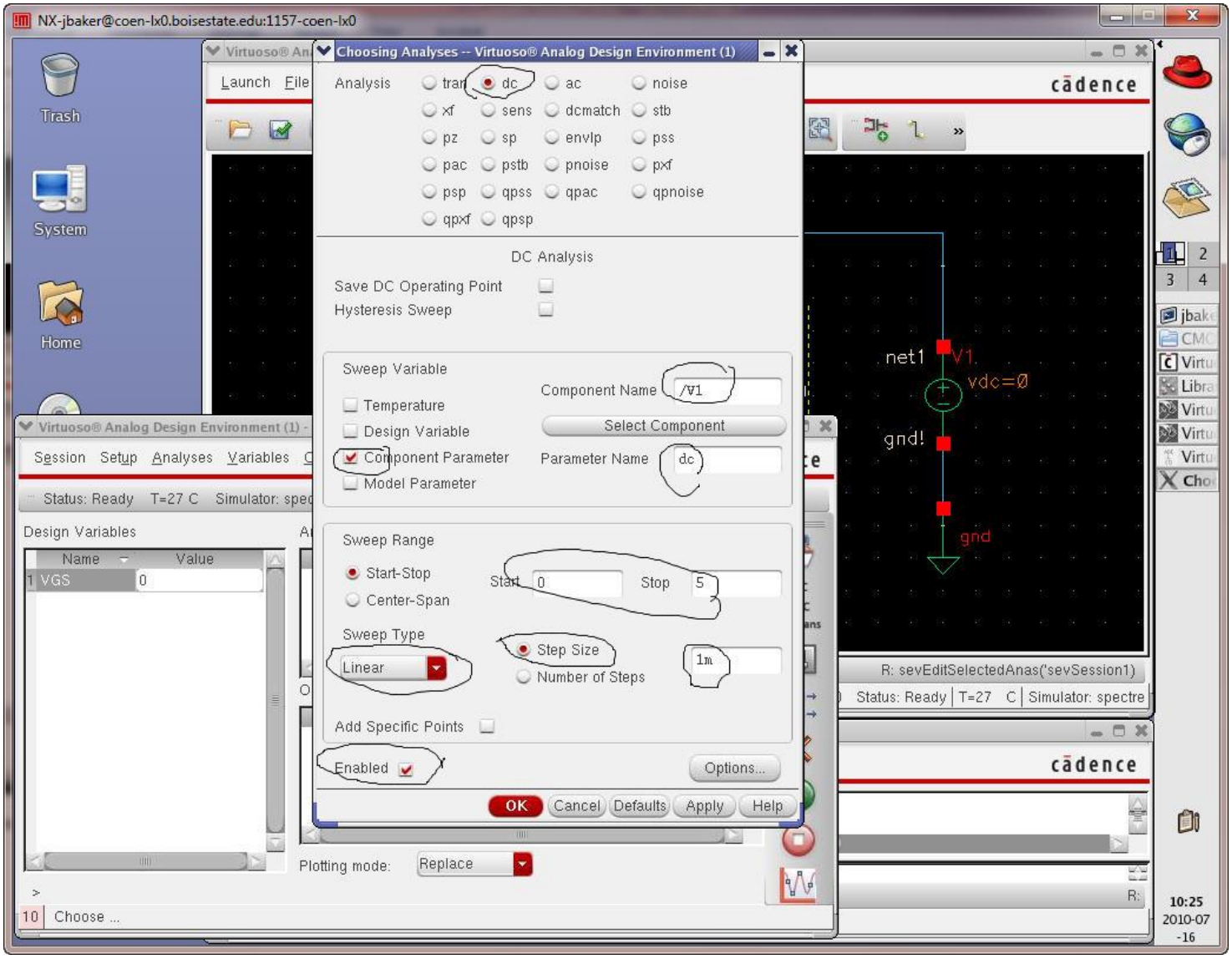
Launch the ADE.
Go to the menu Setup -> Model Libraries press the button circled below and navigate to /$HOME /ncsu-cdk-1.6.0.beta/models/spectre/standalone
Select ami06N.m (NMOS model) then Press Open and OK.
Note that for general purpose simulation we would also select ami06P.m for the PMOS models.



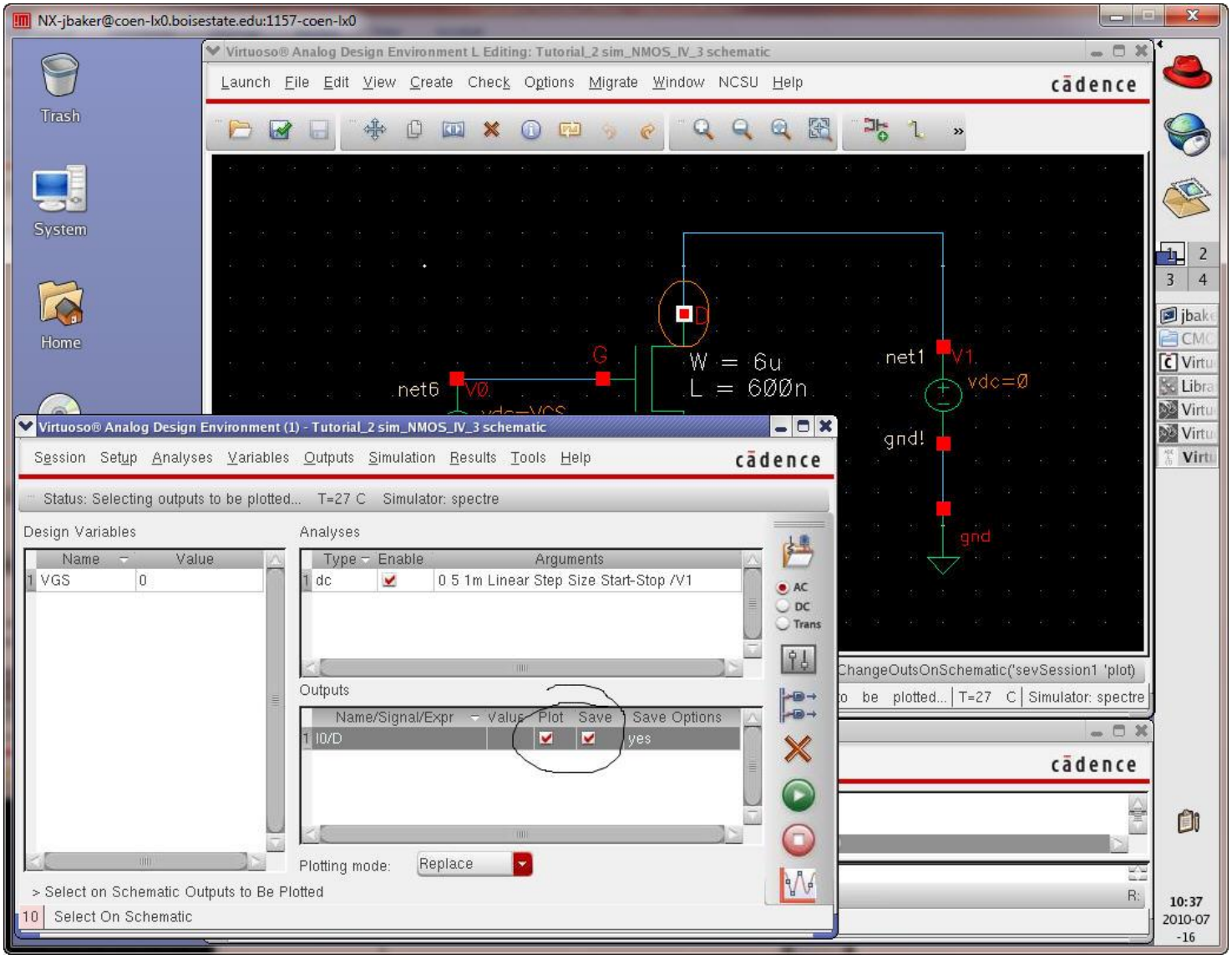
Next, in the ADE window, select Variables -> Edit and add VGS (value for the left DC voltage source in our schematic) and an arbitrary value of 0.
Press OK and notice how the Design Variable shows up in the ADE.



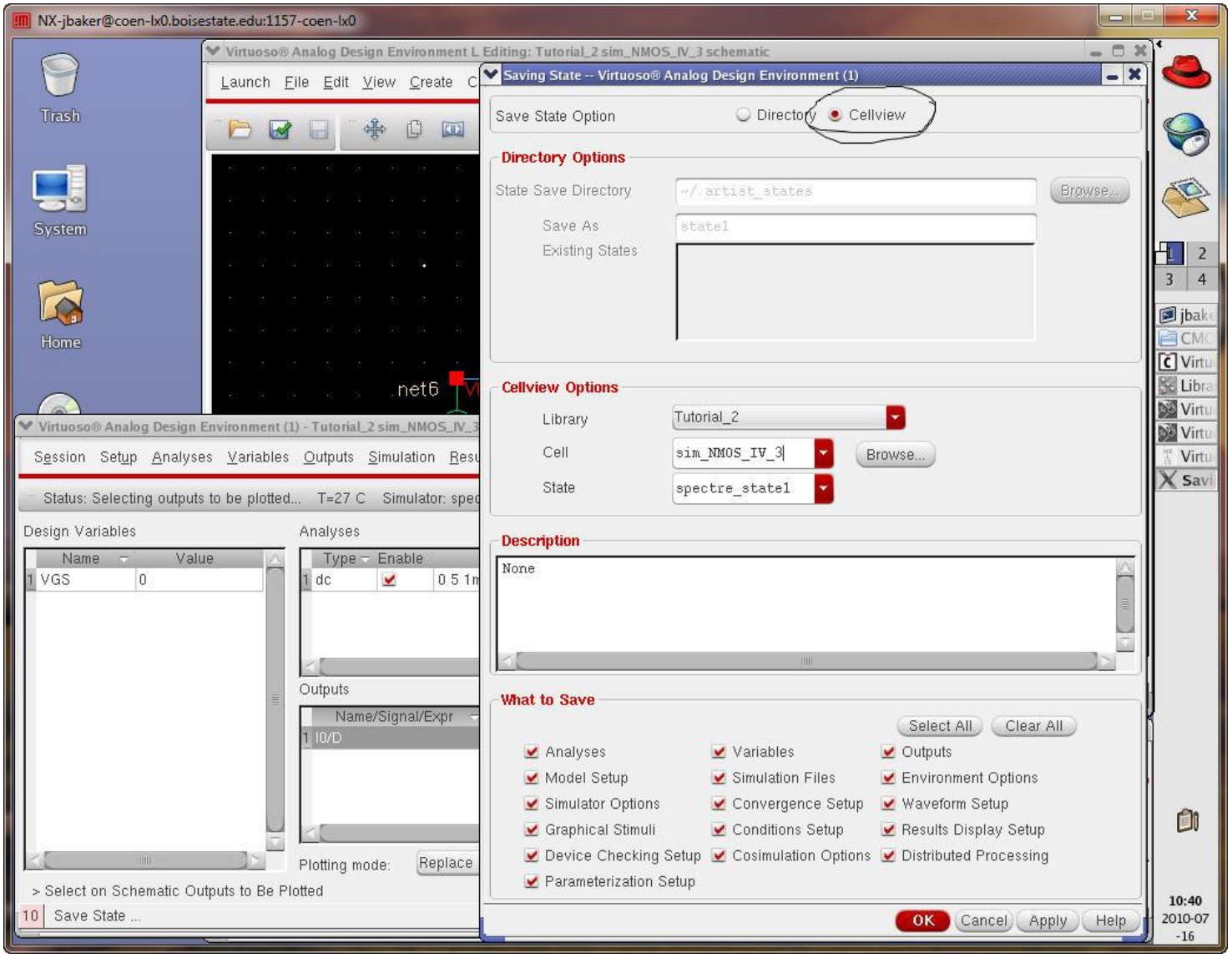
Next select Analyses -> Choose and set the parameters seen below.
Note that V1 is the right DC voltage source in the schematic connected to the Drain.



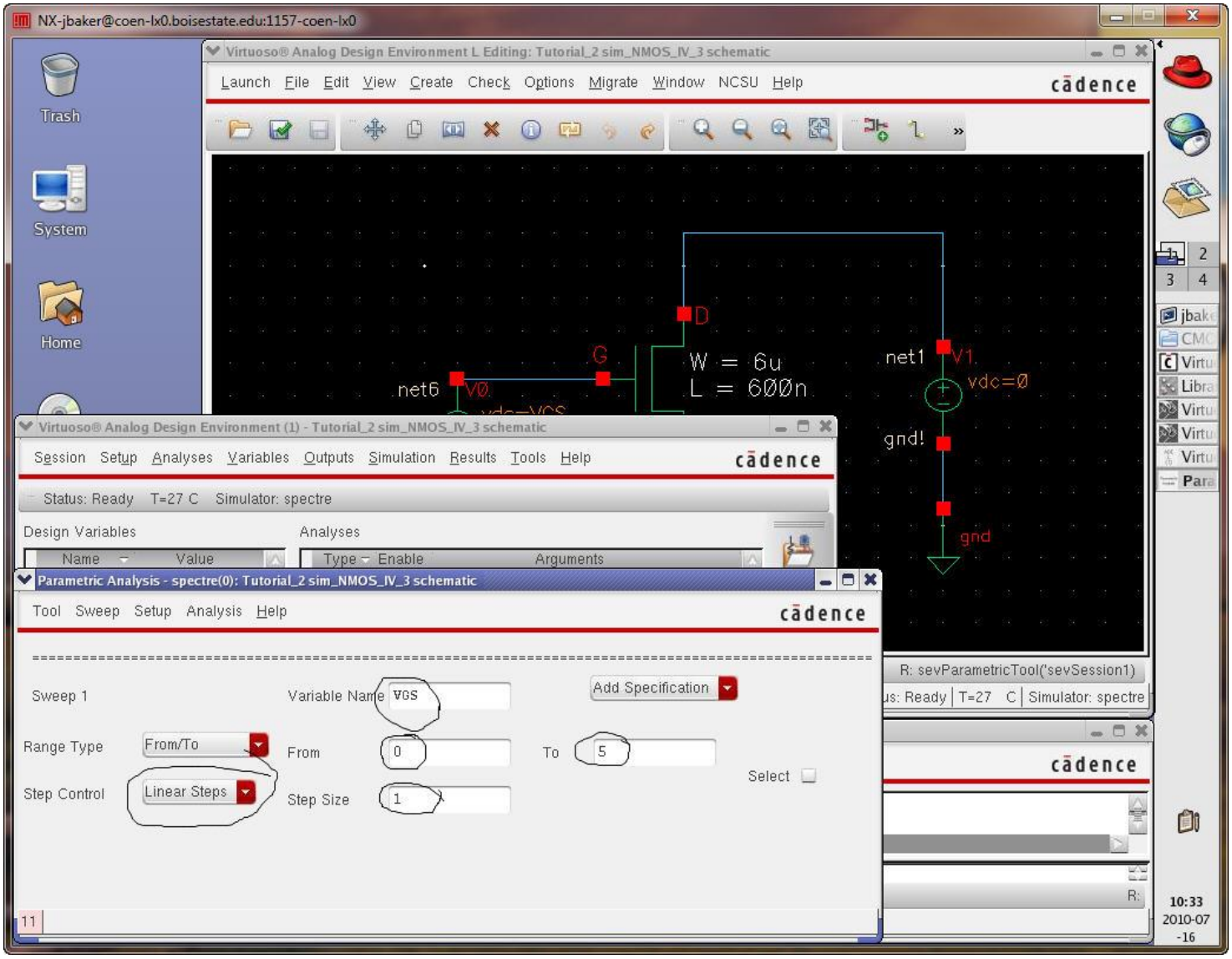
Select Output -> To Be Plotted -> Select On Schematic and click on the D Pin to select the current.
Ensure that both plotted and saved are selected, circled below.



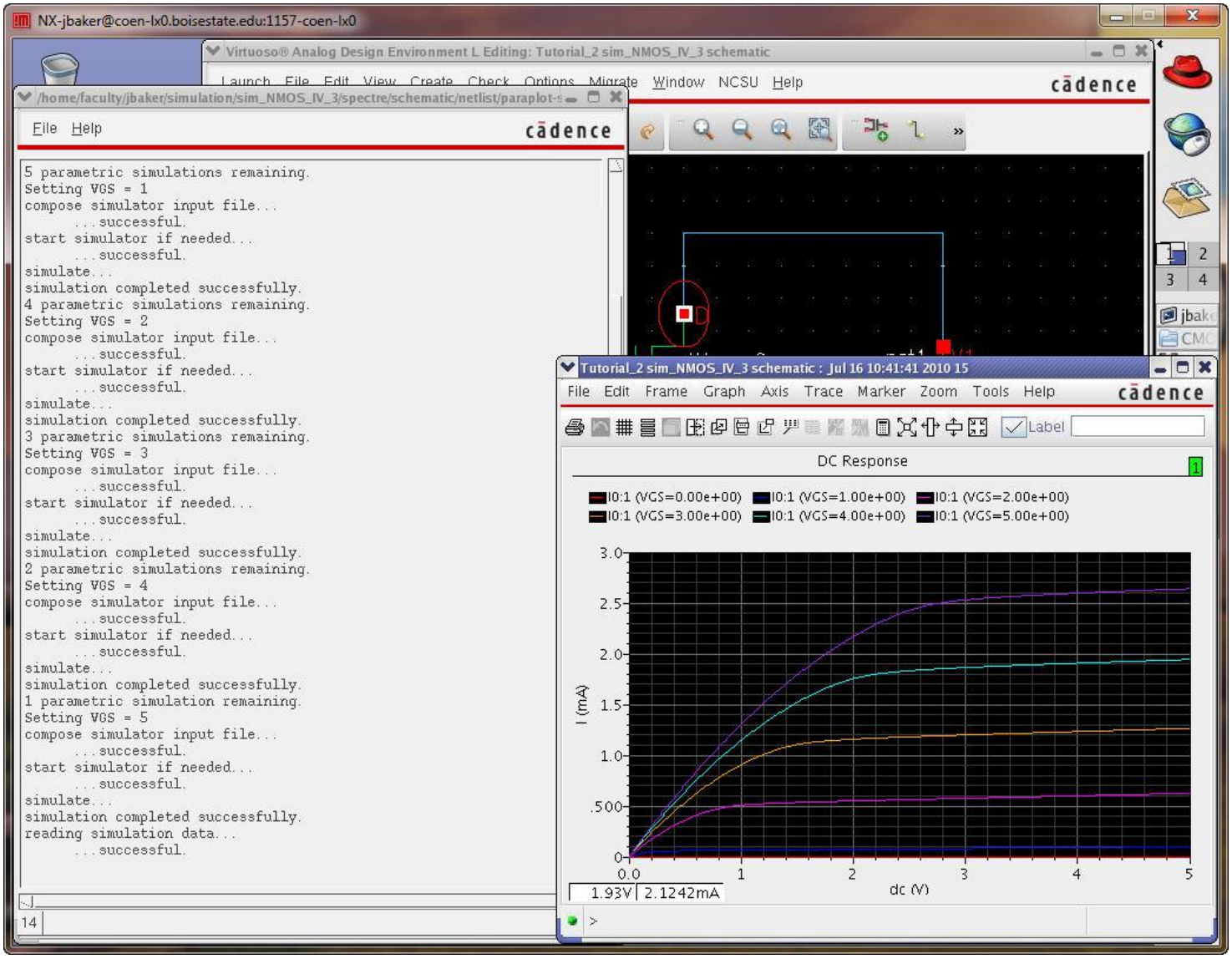
Now let's save the simulation state.
Select Session -> State and save the in the Cell View (you can use the Library Manager to verify it has been saved).



Next choose Tools -> Parametric Analysis and set the parameters seen below.



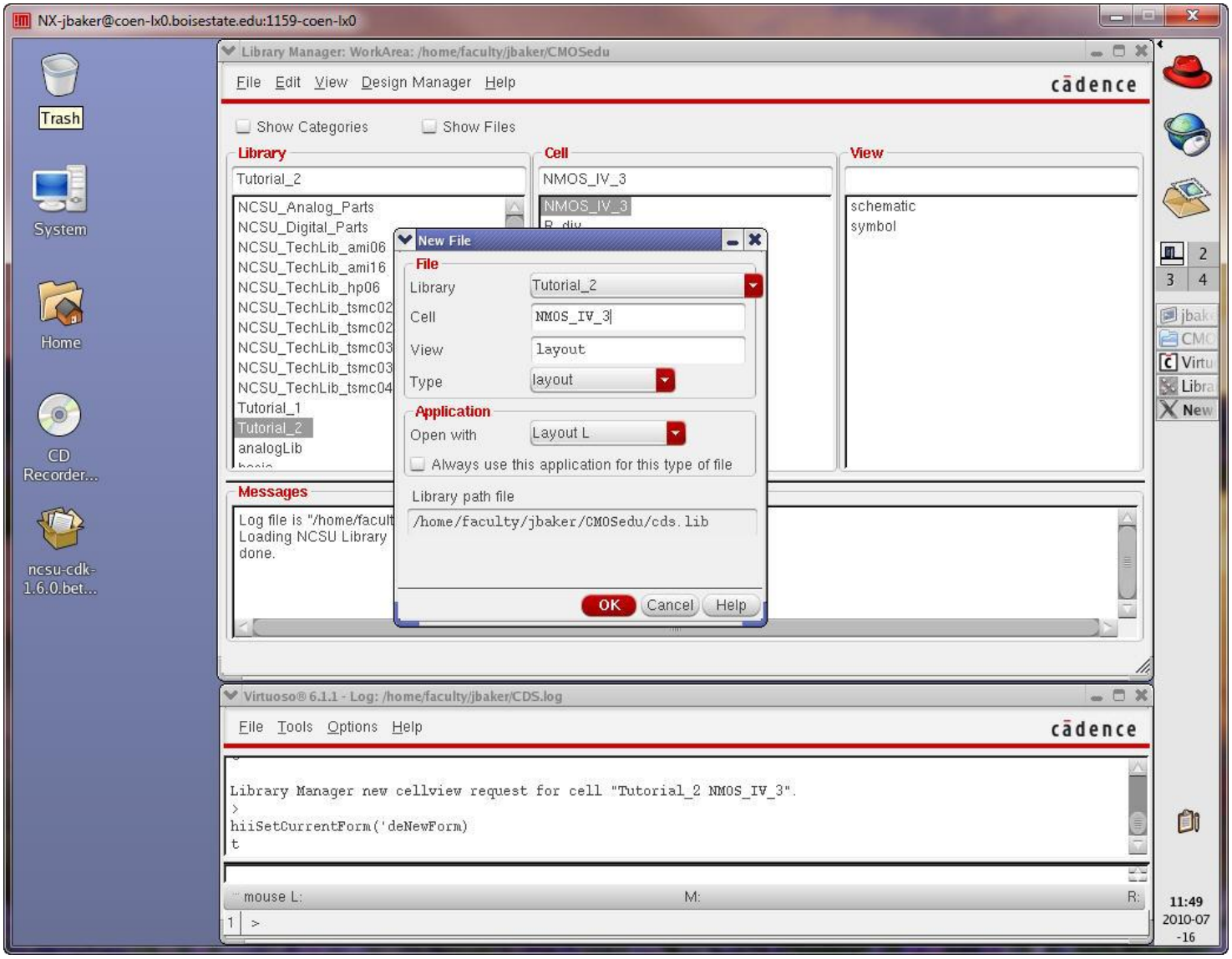
Finally, select Analysis and Start, in the Parametric Analysis window, to see the MOSFET IV curves below.



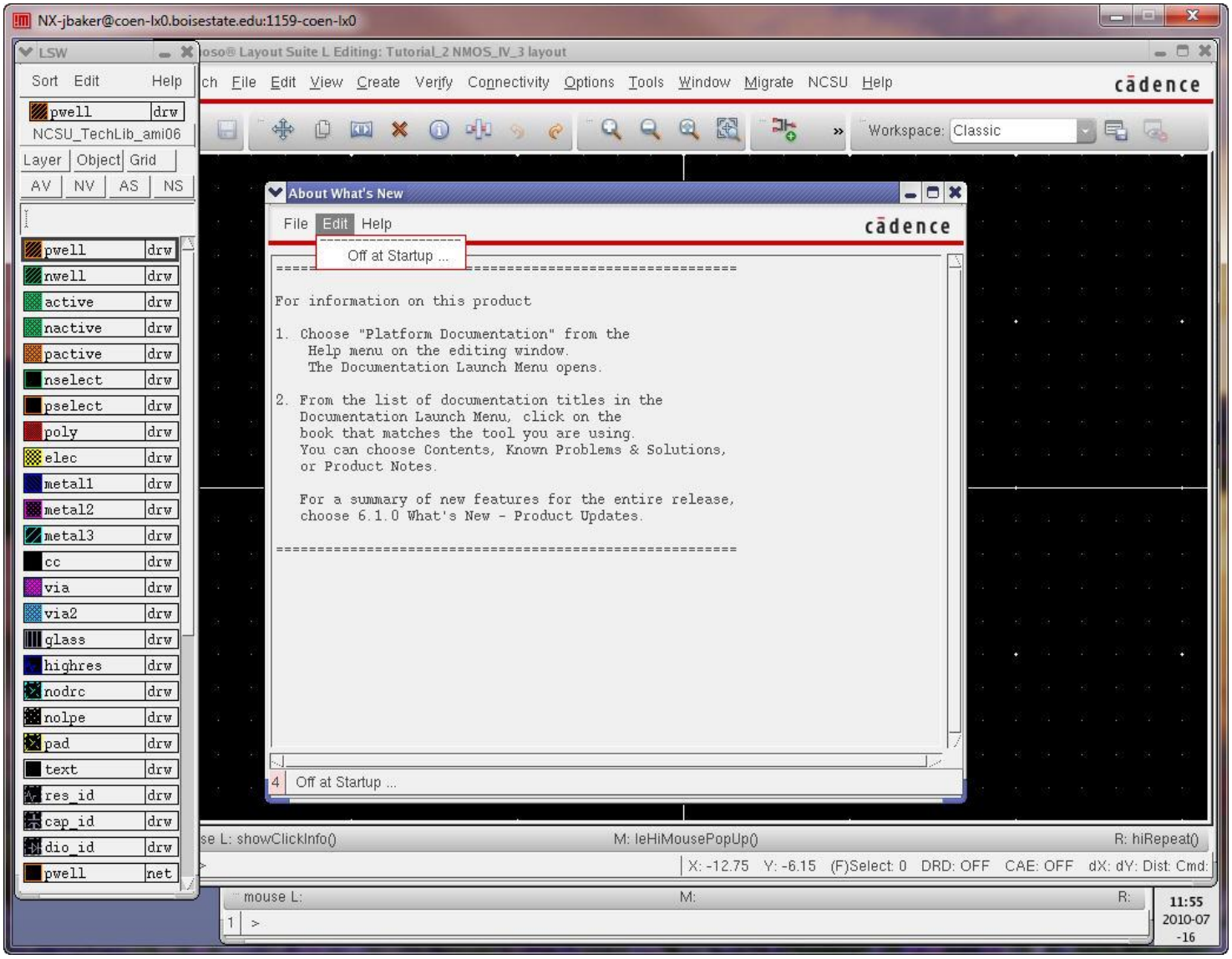
Note that we could have avoided making the symbol view so that plotting these curves would take less effort.
Why we chose this path will become clearer later in the tutorial (having a symbol allows us to simulate either the schematic or extracted, from layout, views)

Quit the ADE and Save the State (okay to overwrite existing state).
Close the schematic of sim_NMOS_IV_3 (close all Cell Views).

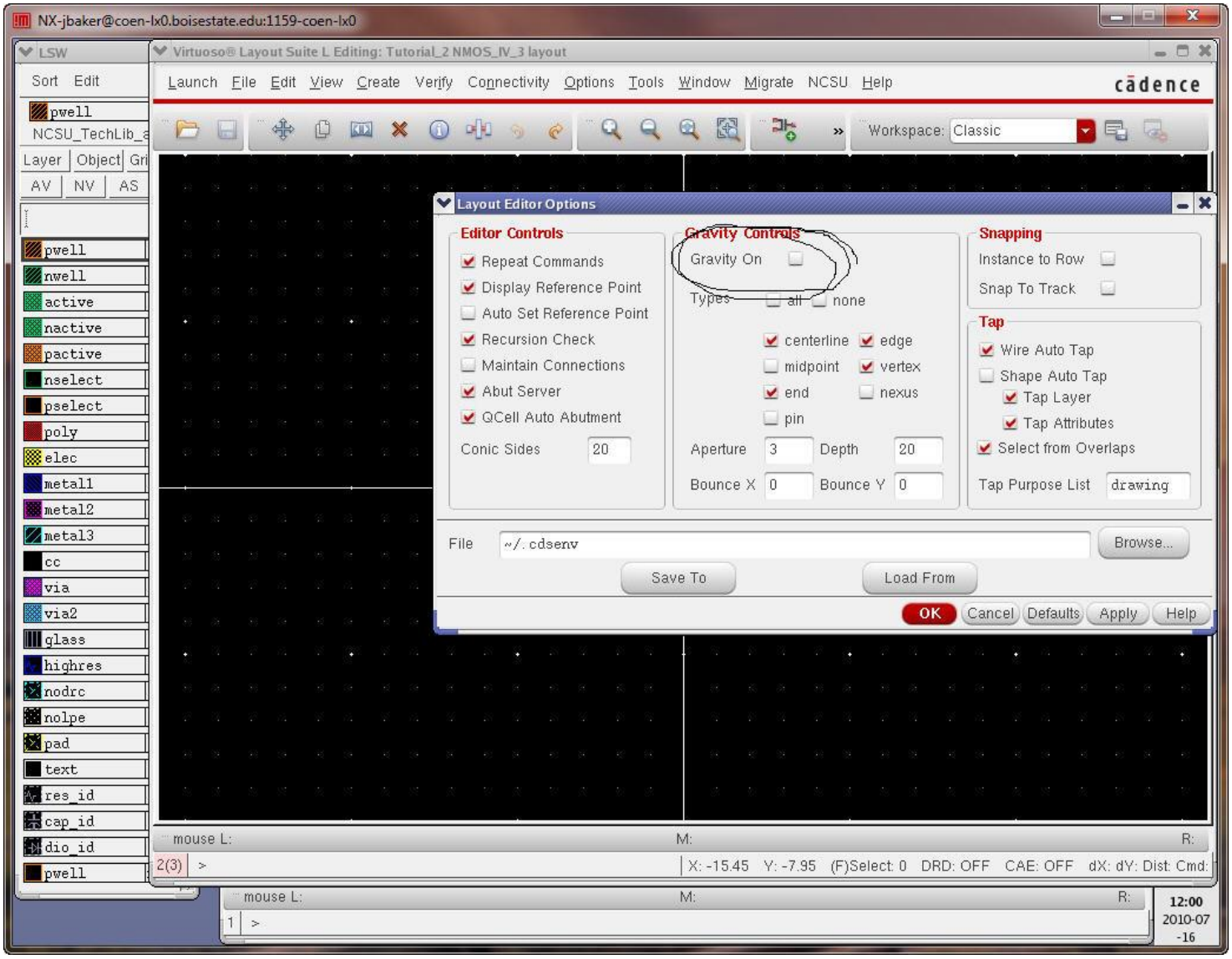
We are now ready to lay out this cell and simulate it.
Create a layout view for NMOS_IV_3 (**not** sim_NMOS_IV_3)



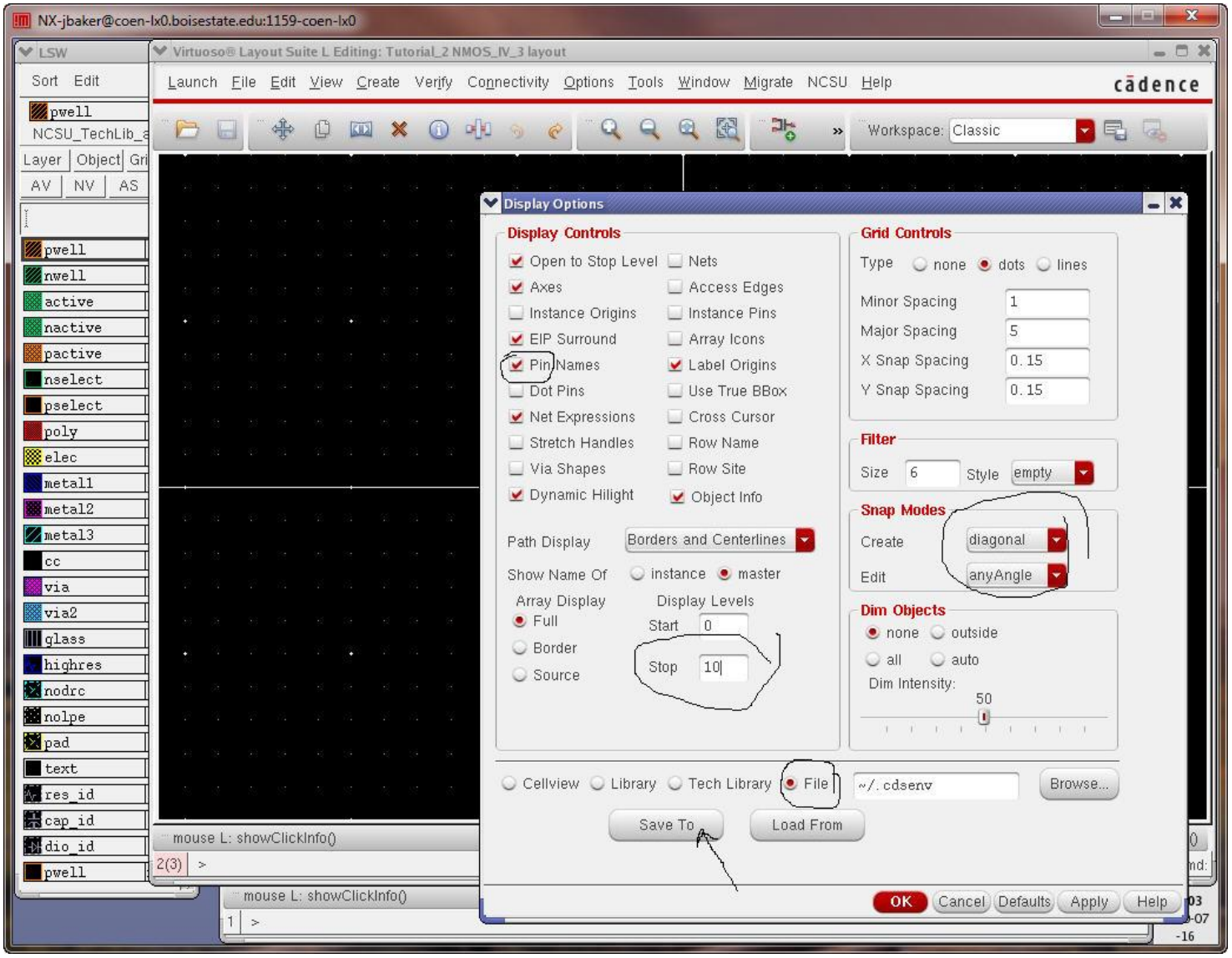
We get that pesky start-up Window again even though we set it to be Off at Start-up before (below).
We'll also see that our Editor Options, **E**, and Display Options, **e**, (also below) have reverted back to the original states.
Change the values back to those seen below.


NX-jbaker@coen-lx0.boisestate.edu:1159-coen-lx0
LSW
Sort Edit Help
pwell drw
NCSU_TechLib_ami06
Layer Object Grid
AV NV AS NS
pwell drw
nwell drw
active drw
nactive drw
pactive drw
nselect drw
pselect drw
poly drw
elec drw
metal1 drw
metal2 drw
metal3 drw
cc drw
via drw
via2 drw
glass drw
highres drw
nodrc drw
nolpe drw
pad drw
text drw
res_id drw
cap_id drw
dio_id drw
pwell net
oso® Layout Suite L Editing: Tutorial_2 NMOS_IV_3 layout
ch File Edit View Create Verify Connectivity Options Tools Window Migrate NCSU Help
cadence
Workspace: Classic
About What's New
File Edit Help
cadence
Off at Startup ...
For information on this product
1. Choose "Platform Documentation" from the
Help menu on the editing window.
The Documentation Launch Menu opens.
2. From the list of documentation titles in the
Documentation Launch Menu, click on the
book that matches the tool you are using.
You can choose Contents, Known Problems & Solutions,
or Product Notes.
For a summary of new features for the entire release,
choose 6.1.0 What's New - Product Updates.
4 Off at Startup ...
se L: showClickInfo()
M: leHiMousePopUp()
R: hiRepeat()
X: -12.75 Y: -6.15 (F)Select: 0 DRD: OFF CAE: OFF dX: dY: Dist: Cmd:
mouse L:
M:
R:
1 >
11:55
2010-07
-16

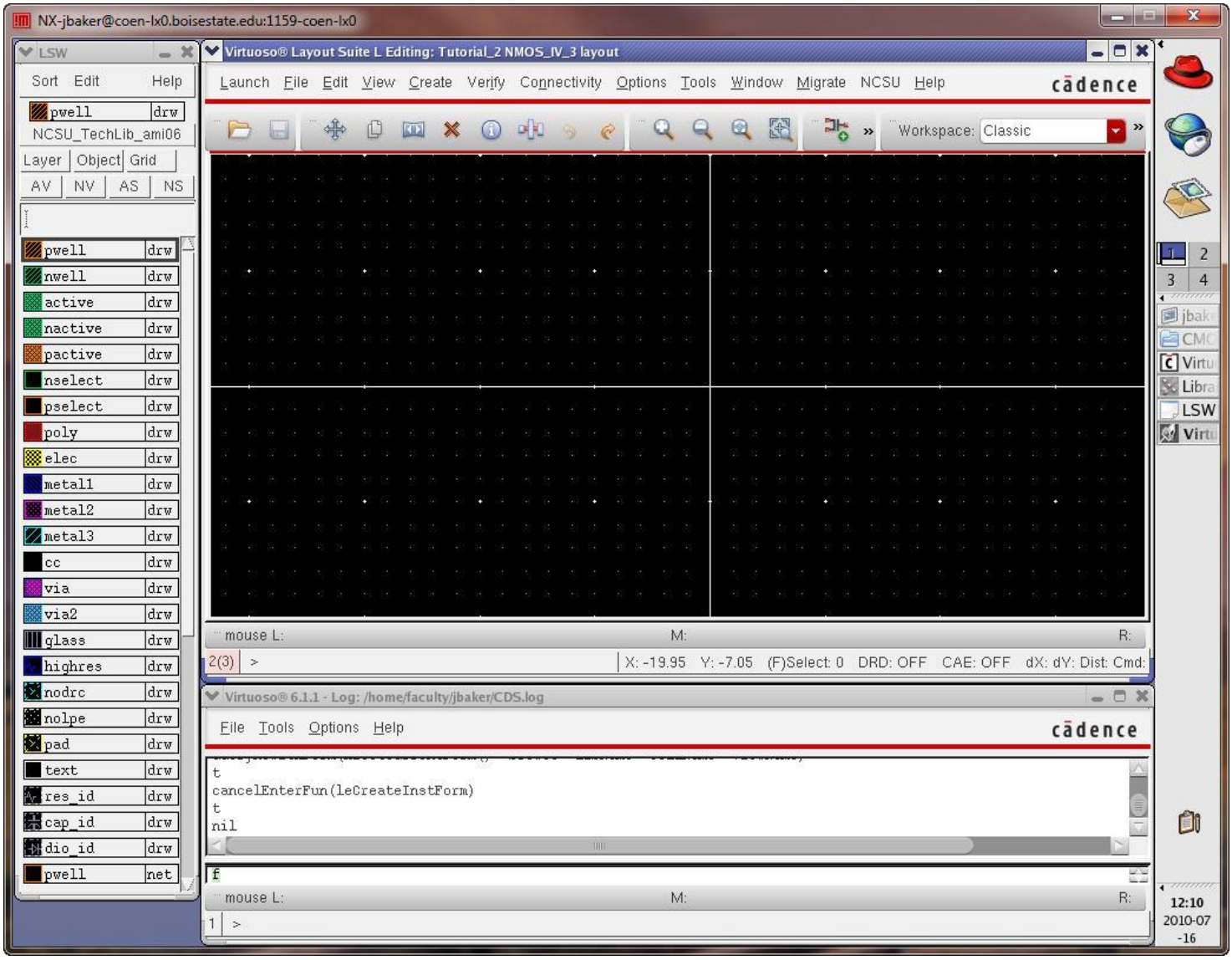
e



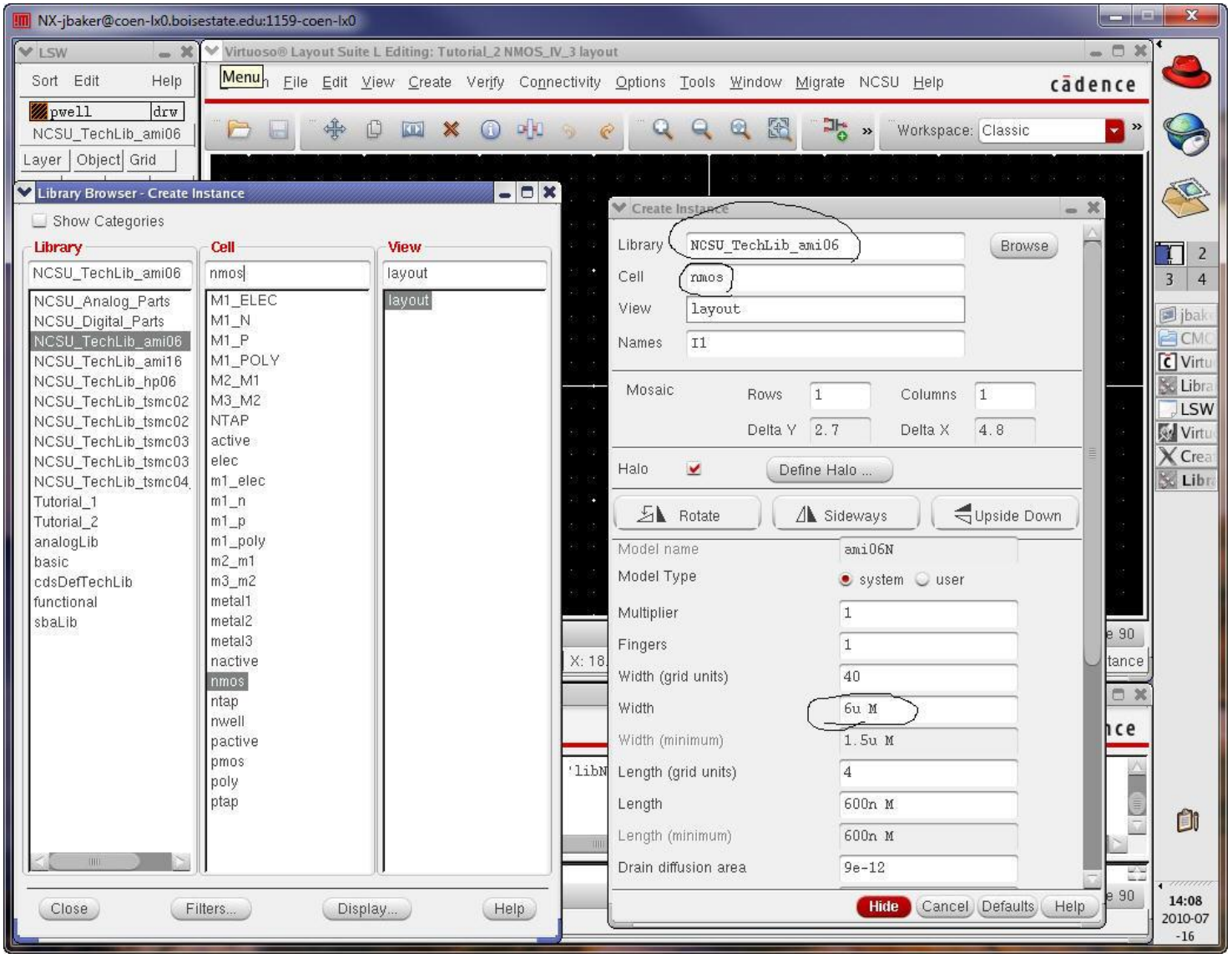
After pressing **e** we get the following.
To ensure that the options are remembered Save To File (the CDS, Cadence Design System, Environment, .cdsenv) as seen below (saved in $HOME, aka, ~/).
Note that to see the Save To button you may need to re-size the Display Options dialog window.



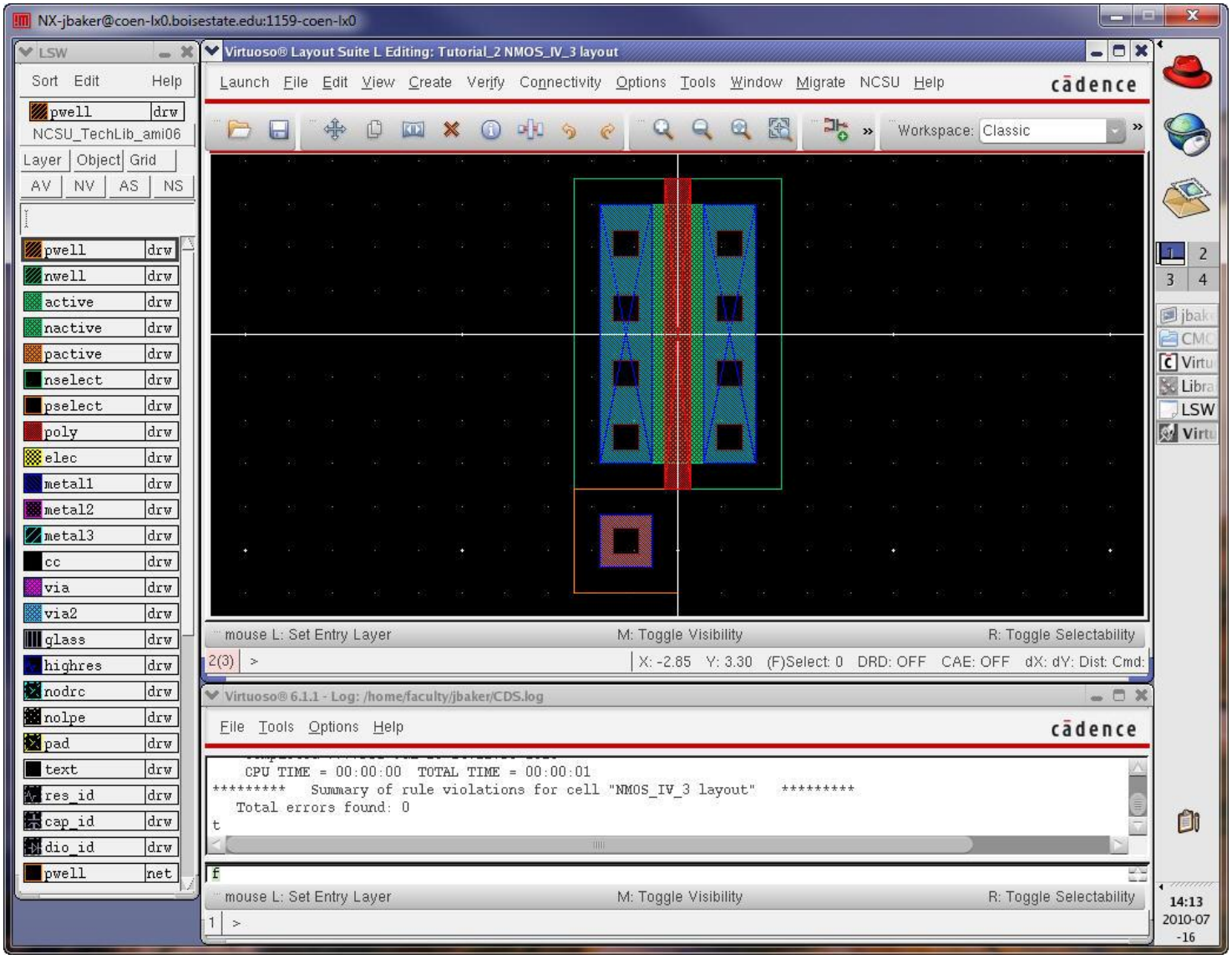
When finished hit OK and re-size the layout window as seen below.



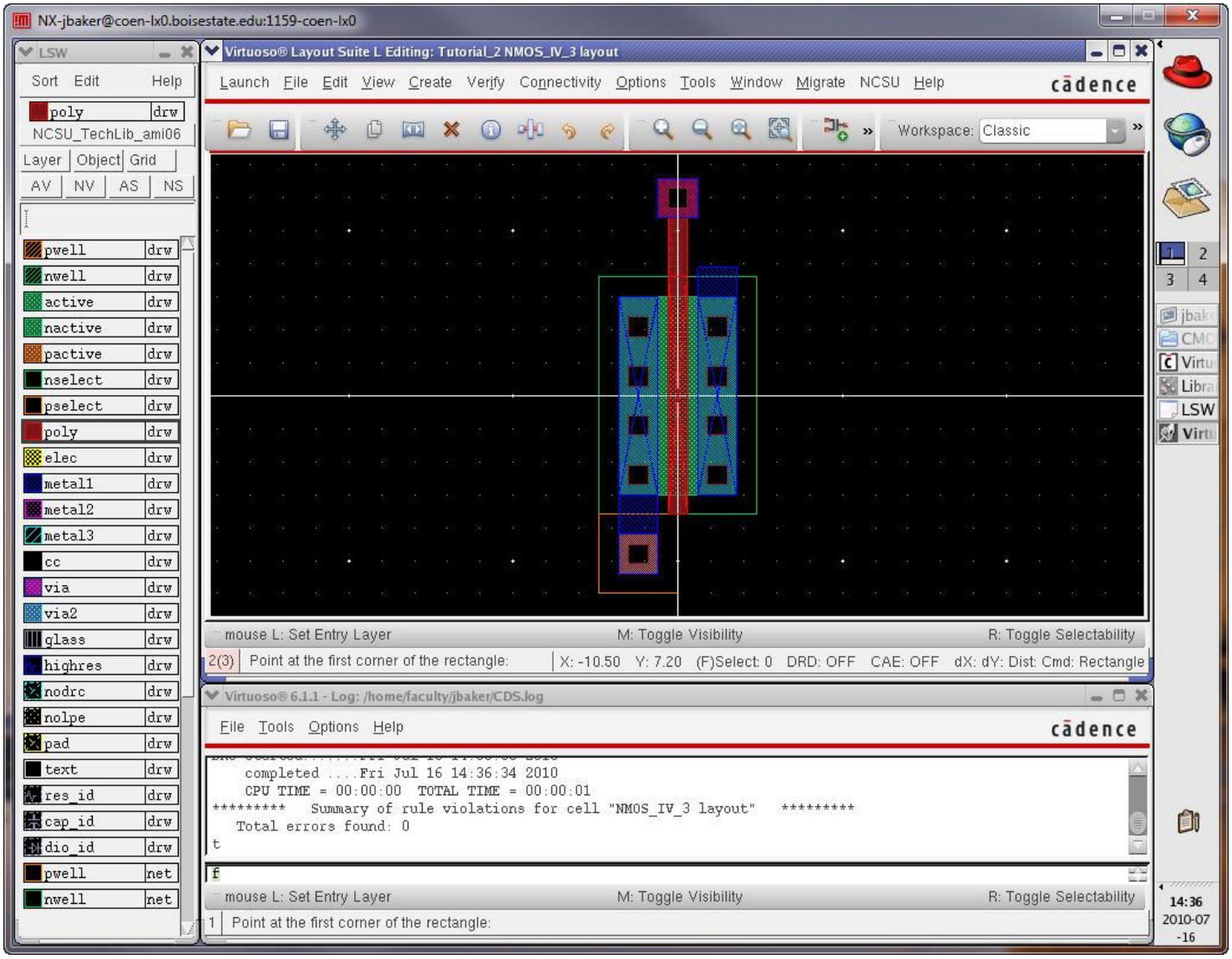
Next instantiate an nmos device that is 6 um wide and 600 nm long.



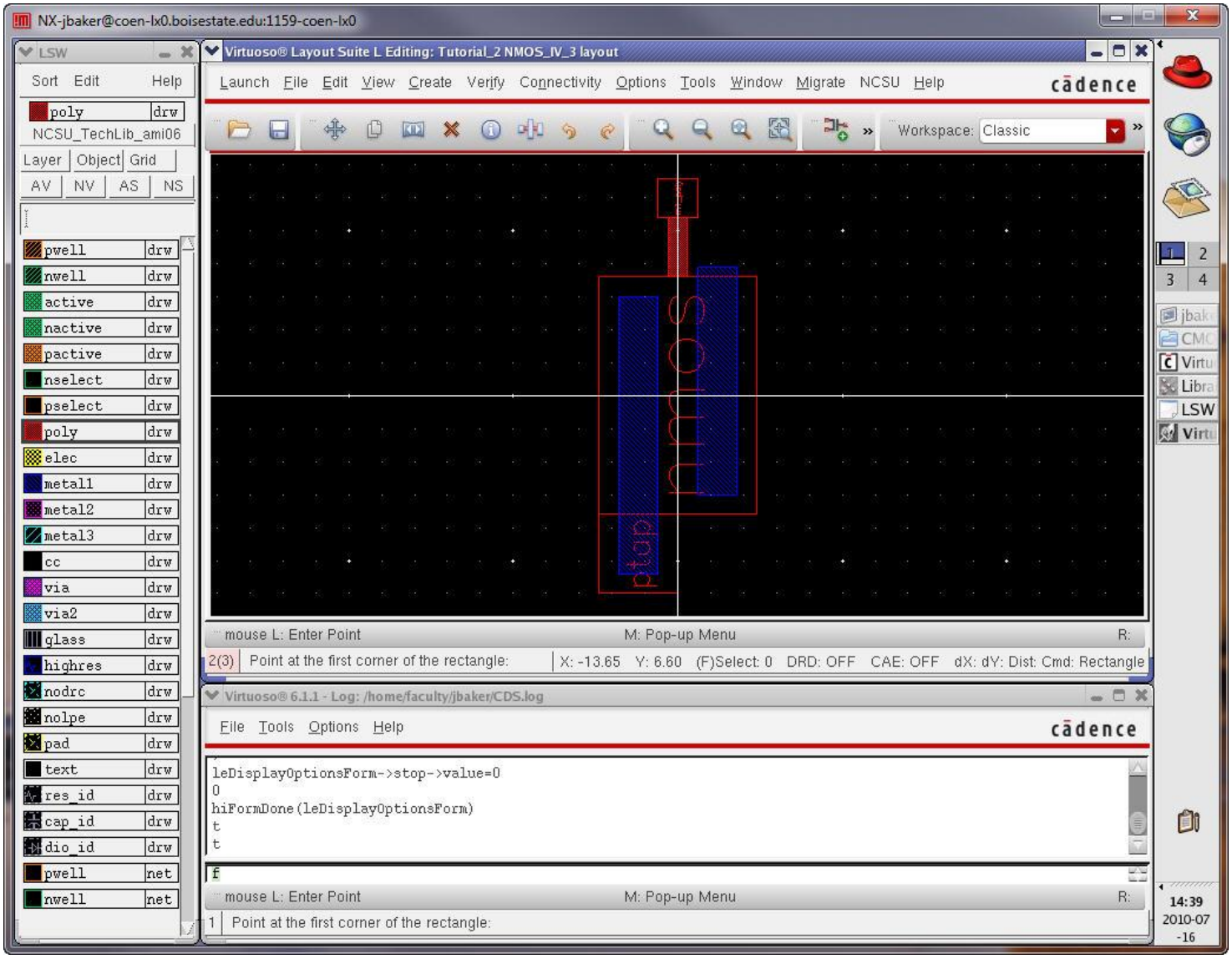
Next instantiate a ptap cell (metal1 connection to p+) in the p-substrate so you get something that looks like the following. DRC the layout (Verify -> DRC then OK).



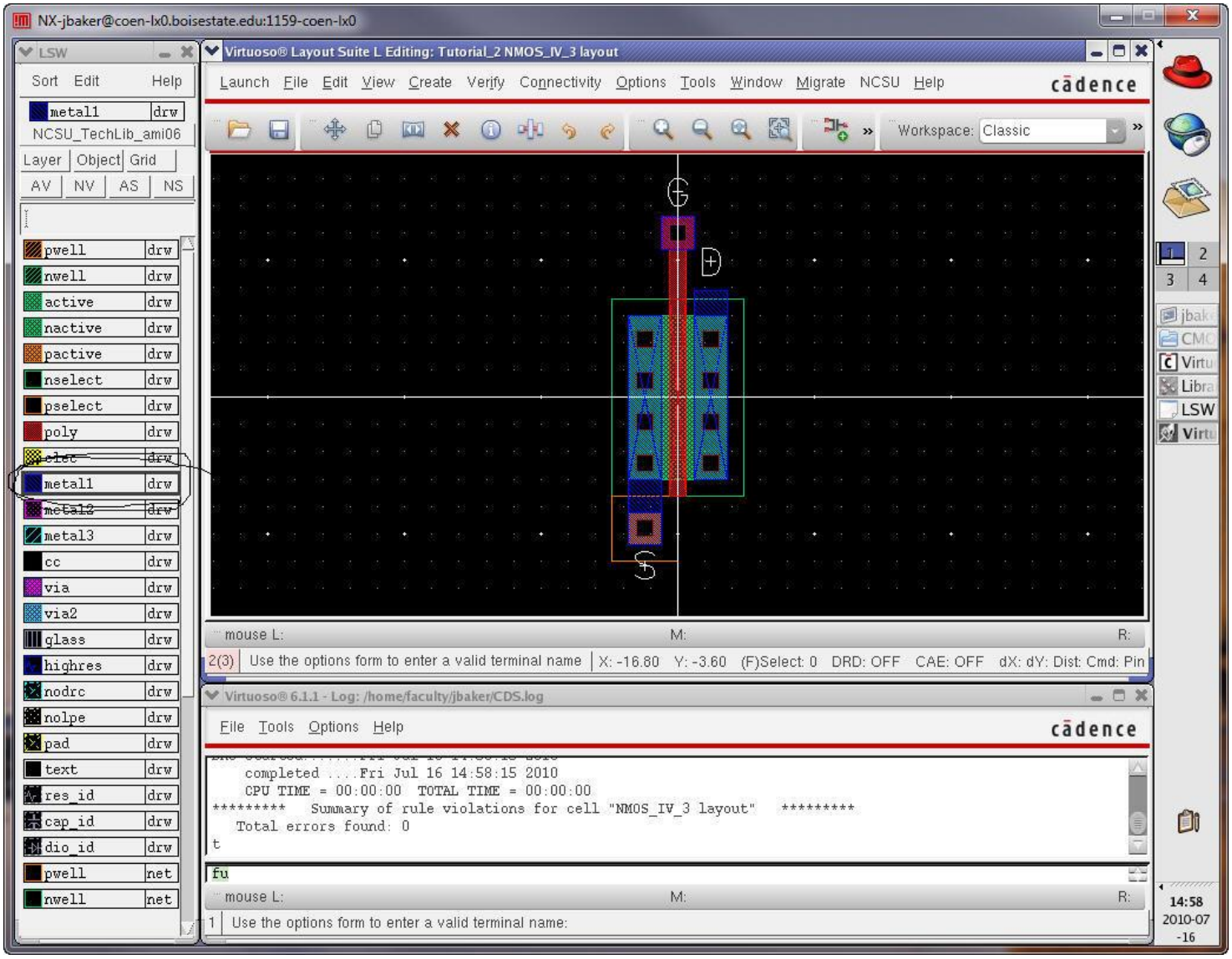
Next add rectangles on the metal1 layer to connect the source to the p-substrate and to the drain.
Also add a metal1-connection-to-poly cell (m1_poly) and a poly rectangle to connect the gate to m1_poly.
DRC your layout.



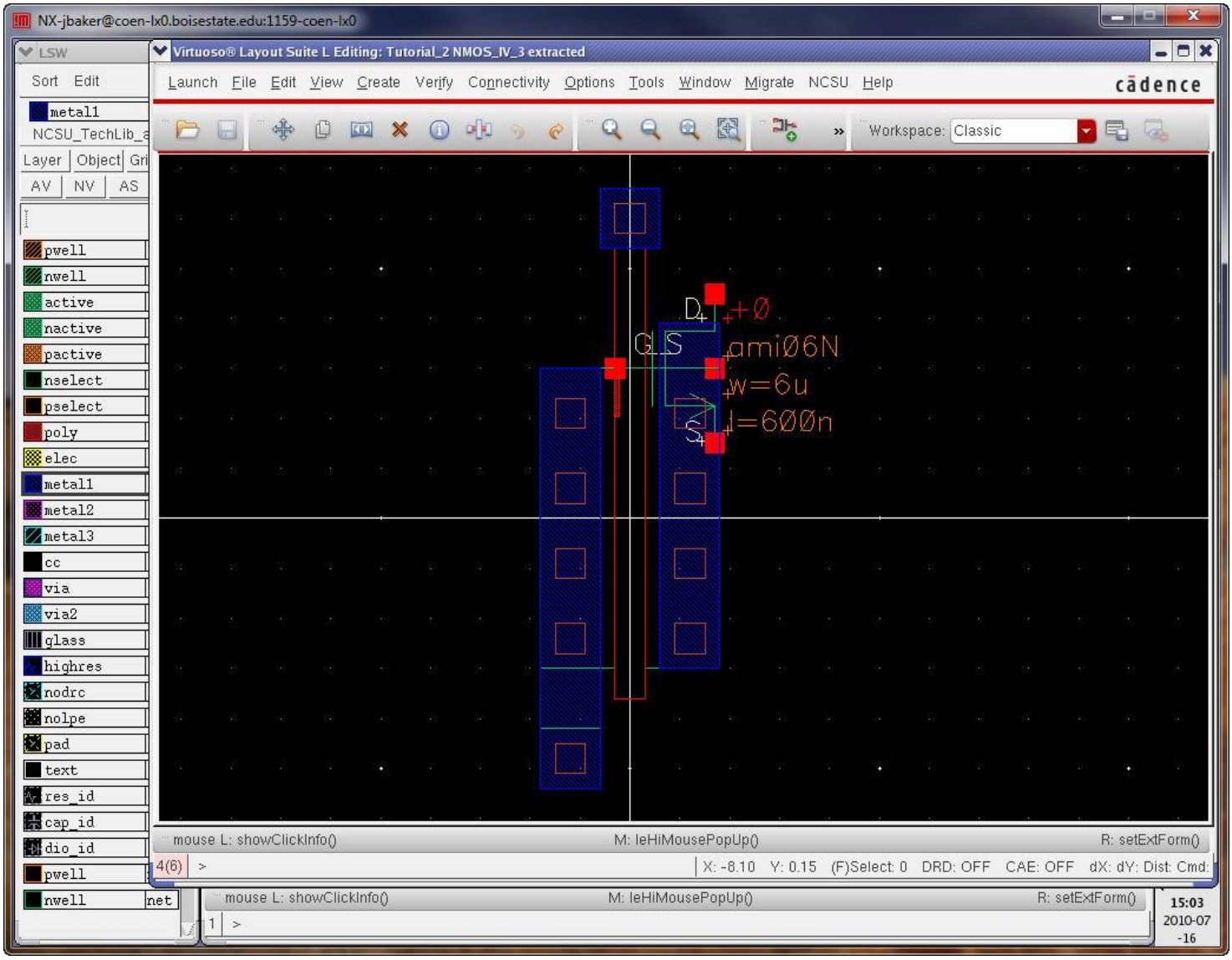
Pressing **e** and then setting the display level stop to 0 we can see the rectangles and cells.



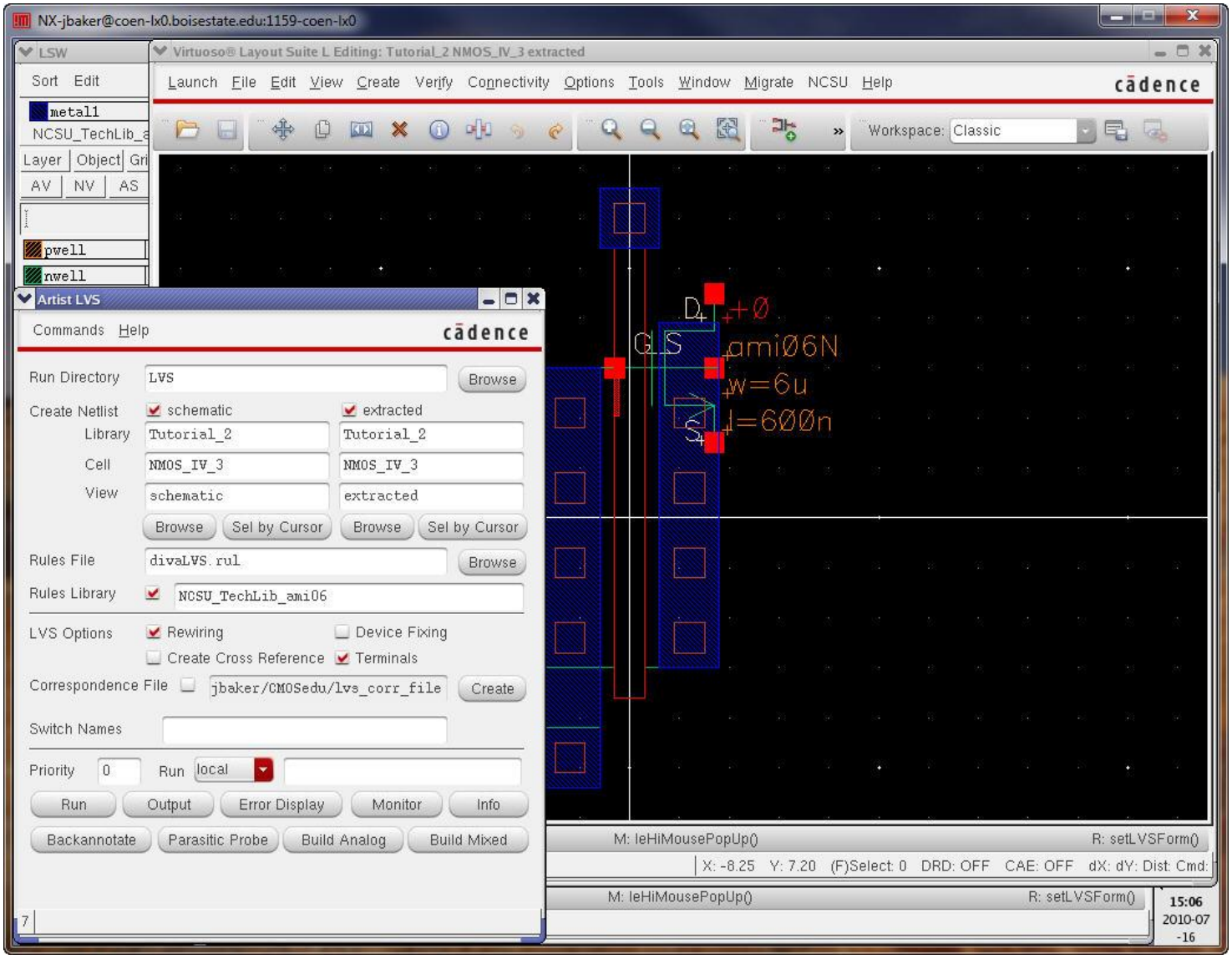
Set the display stop level back to 10.
Add D, G, and S Pins on metal1 as seen below.
Ensure "Display Pin Name" is selected and I/O type is inputOutput to match the pins in the schematic.
Draw the pins so they overlap the metal1.
DRC the layout to ensure no errors (fix errors if you find them).
Save the cell view.



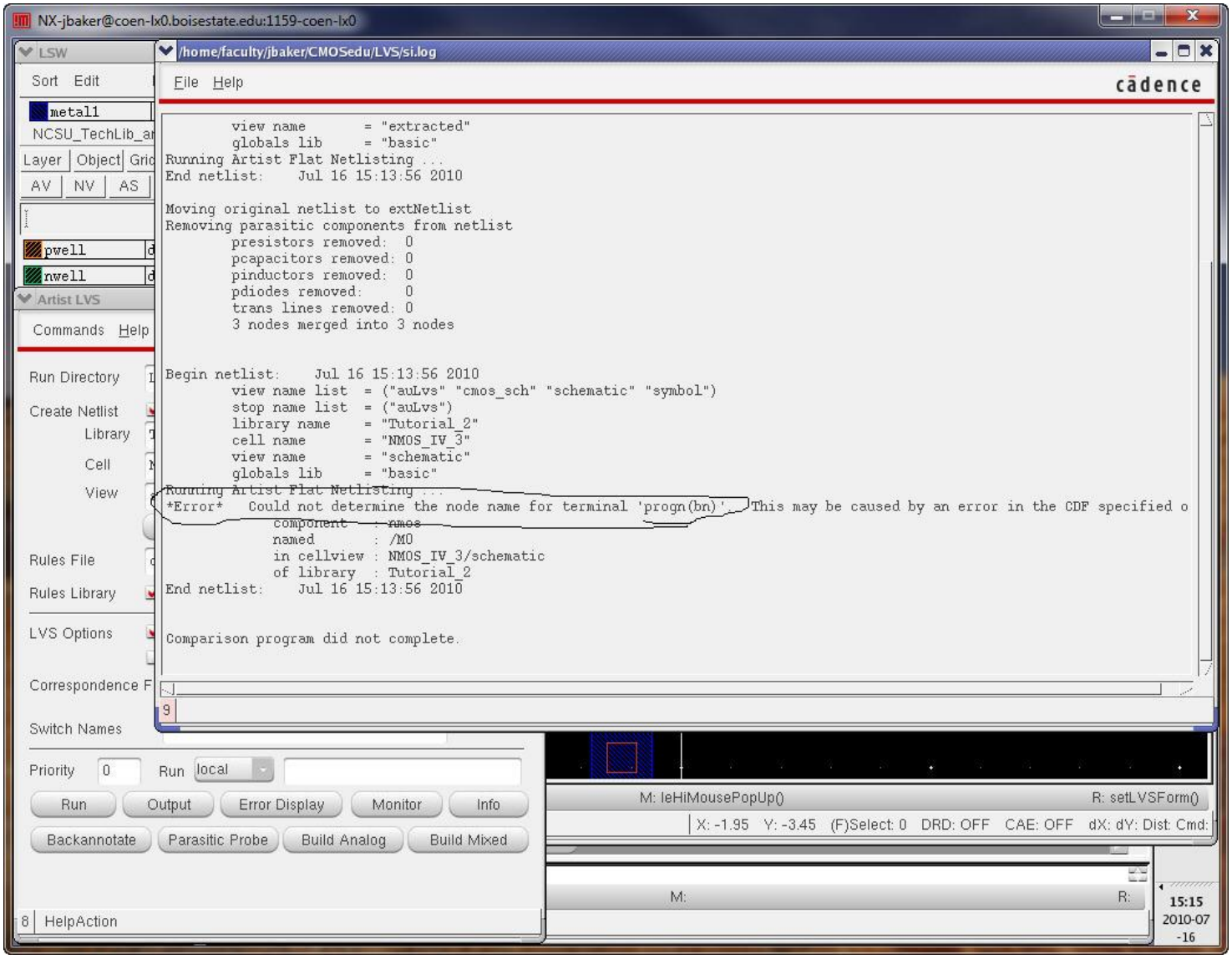
Next Extract the layout (Verify -> Extract).
Using the Library Manager open the extracted cell view (NMOS_IV_3).



Notice that the symbol displayed here has 4 terminals, not the 3-terminal transistor we used in the schematic (where it's assumed the p-substrate is tied to gnd!)
Next run an LVS (Verify -> LVS).

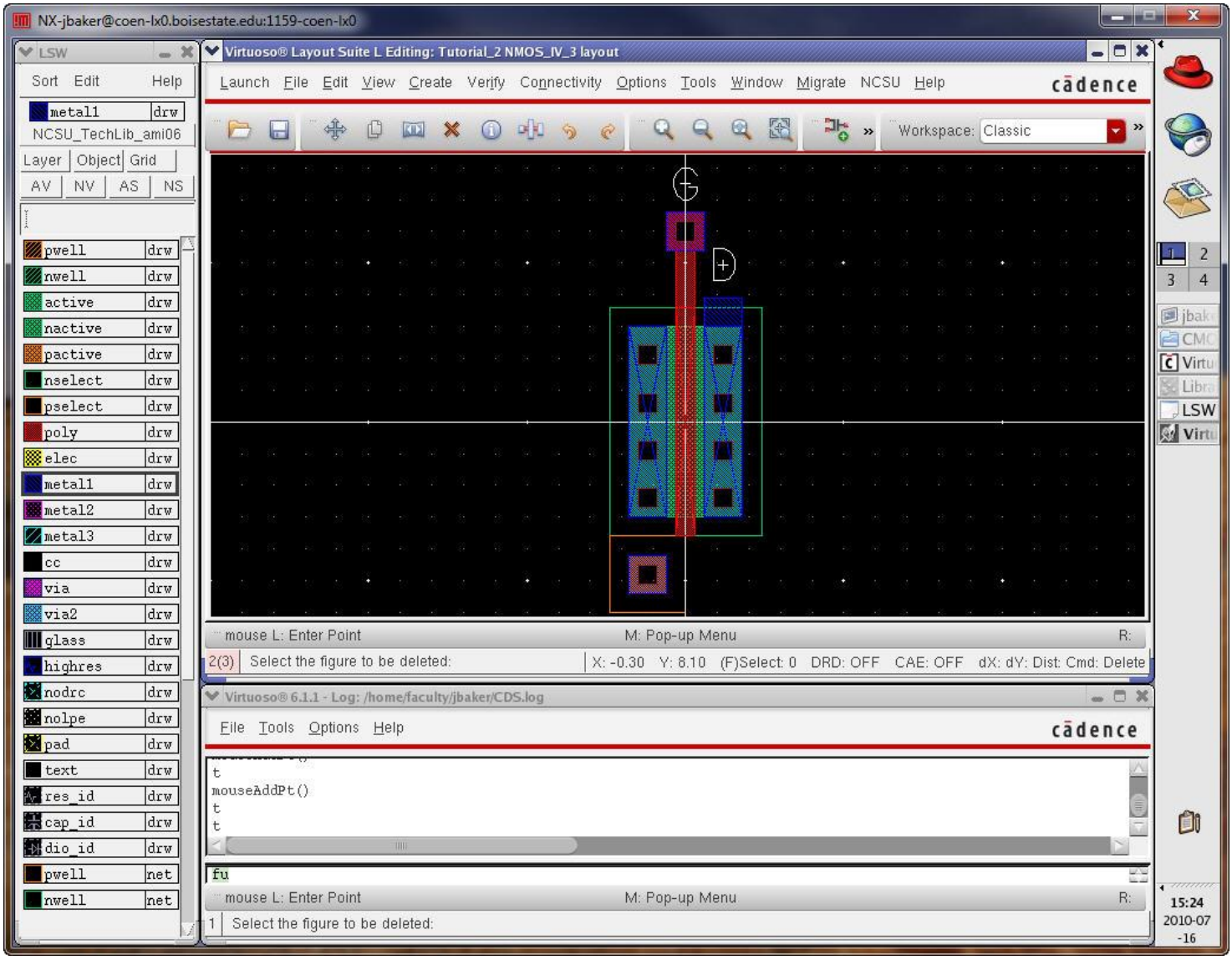


After hitting Run you will be notified that the job failed. Click OK.
In the Artist LVS window click on Info and then Log File to get, after scrolling down, the following.

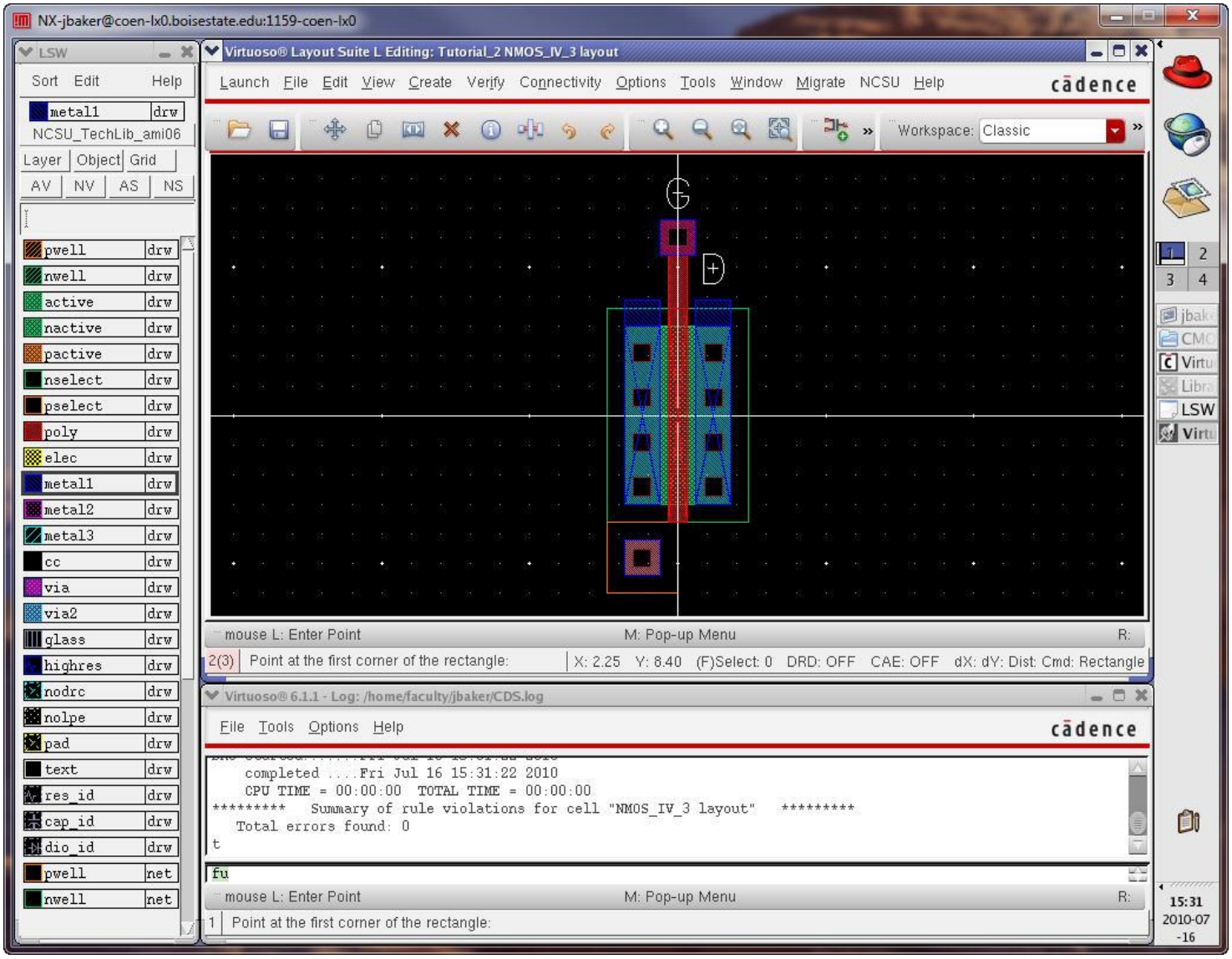


The error is related to the bulk (p-substrate) connection for the NMOS. Again, as mentioned above when the 3 terminal MOSFET symbol is used it's assumed
that the bulk is tied to gnd! for an NMOS device and vdd! for a PMOS device.

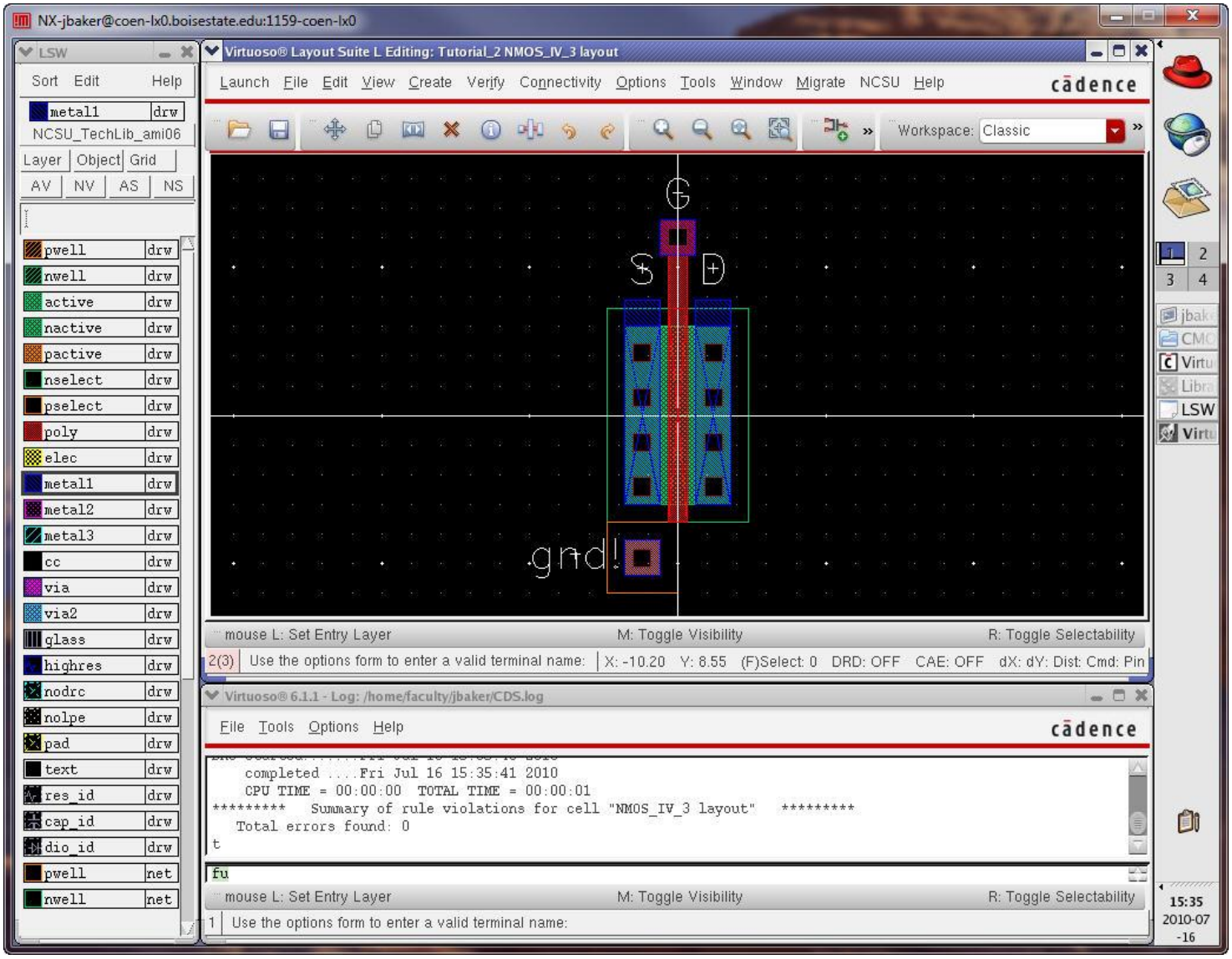
Let's modify the layout as seen below.
Delete the metal1 rectangle and pin connecting the S to the bulk (ptap) as seen below.



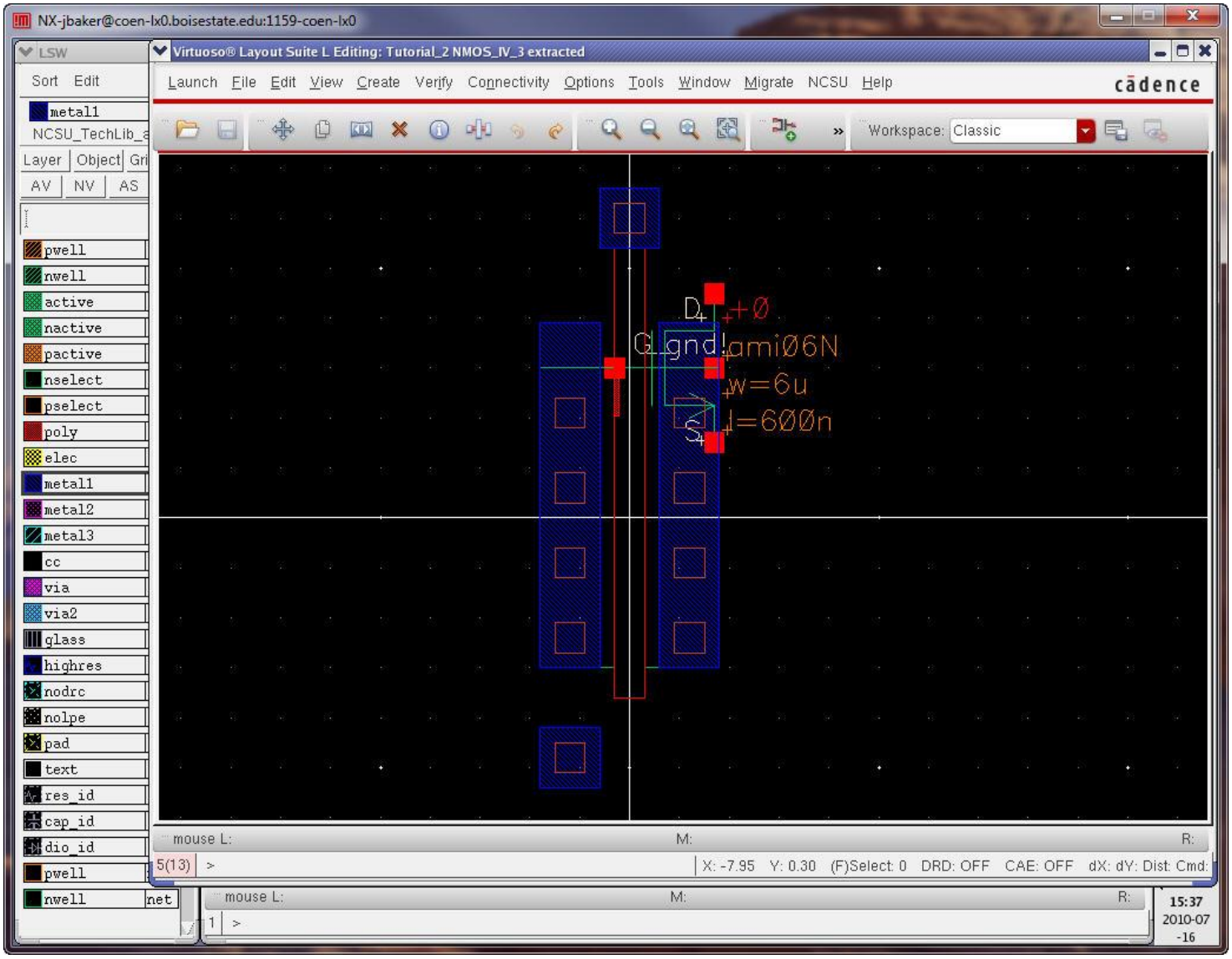
Next, add metal rectangles on metal1 over the S and B metal.



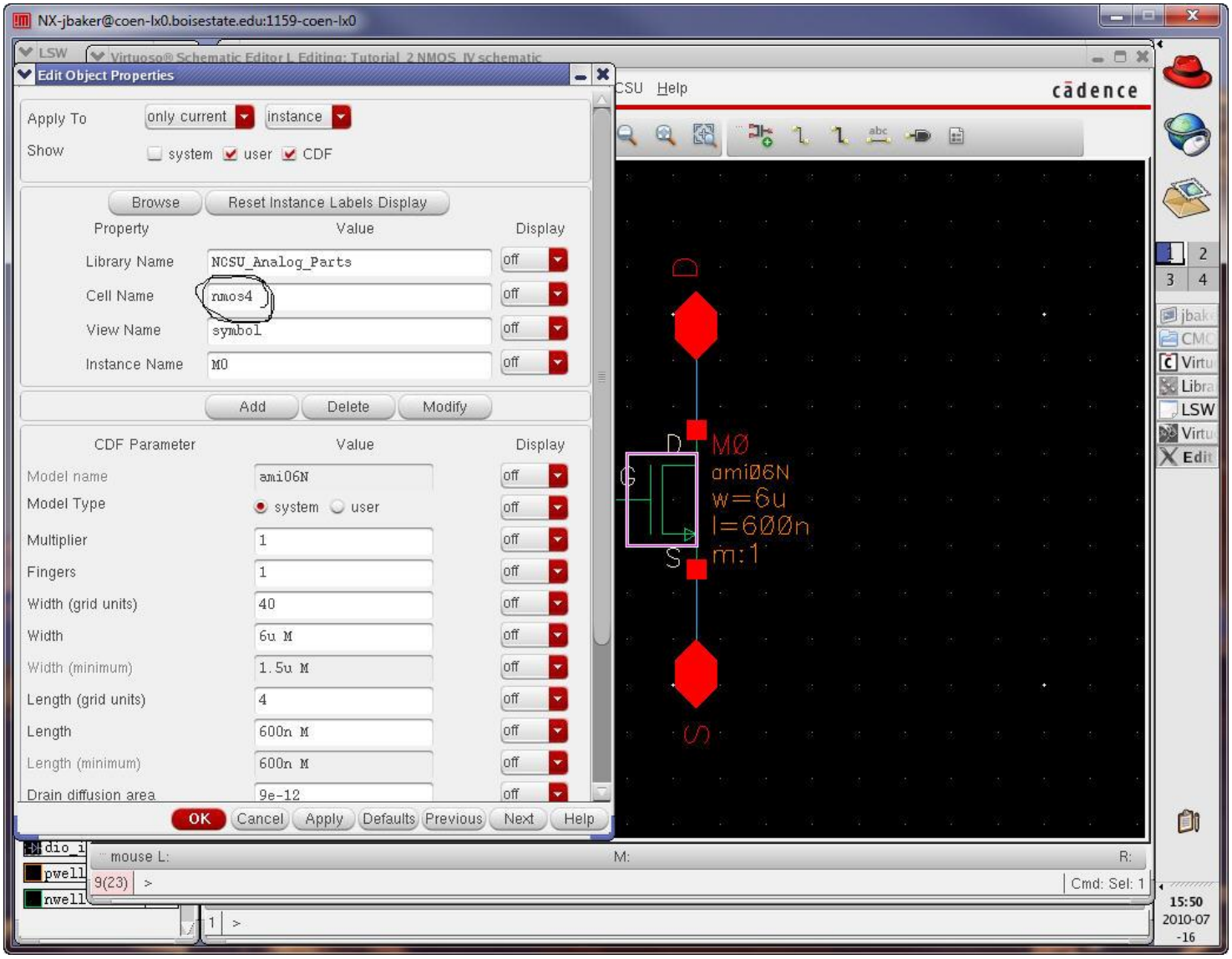
Next add Pins, again make the Pin the same size as the metal1 rectangles (note that that making the Pins the same size as the metal1 rectangles is not a requirement).
Save your layout and DRC it.



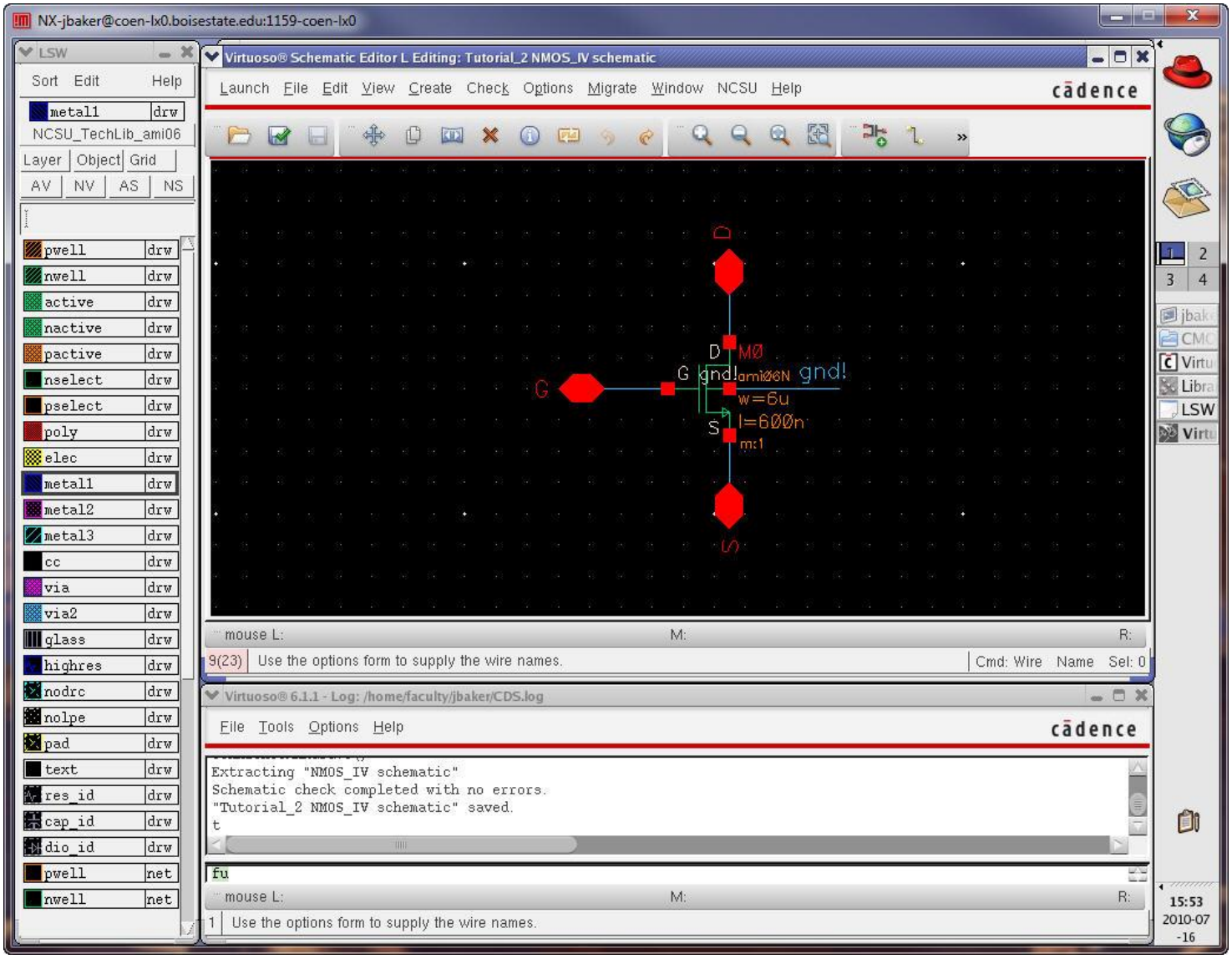
Once again run the Extractor and open the extracted view using the Library Manager.
Notice that it's basically the same view except that now the bulk of the NMOS devices is connected to gnd!



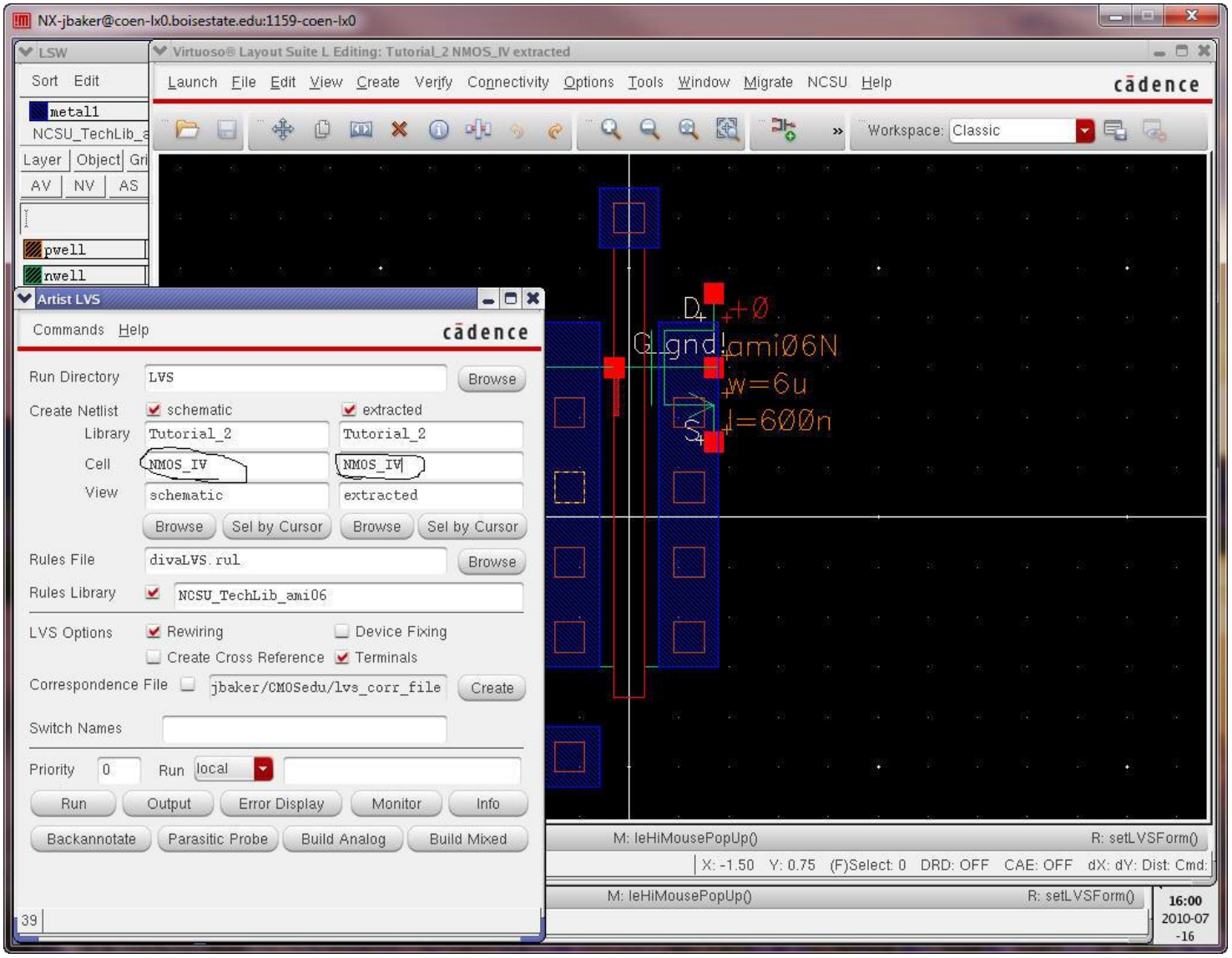
Again we get a failed LVS run and the error "*Error* Could not determine the node name for terminal 'progn(bn)'." It looks like LVS wants us to use
a 4-terminal transistor in our schematics! Let's try this.
Close all Cell views.
Open the Library Manager and rename NMOS_IV_3 to NMOS_IV.
Also, rename sim_NMOS_IV_3 to sim_NMOS_IV
Now open the schematic view of NMOS_IV and change the NMOS symbol as seen below.



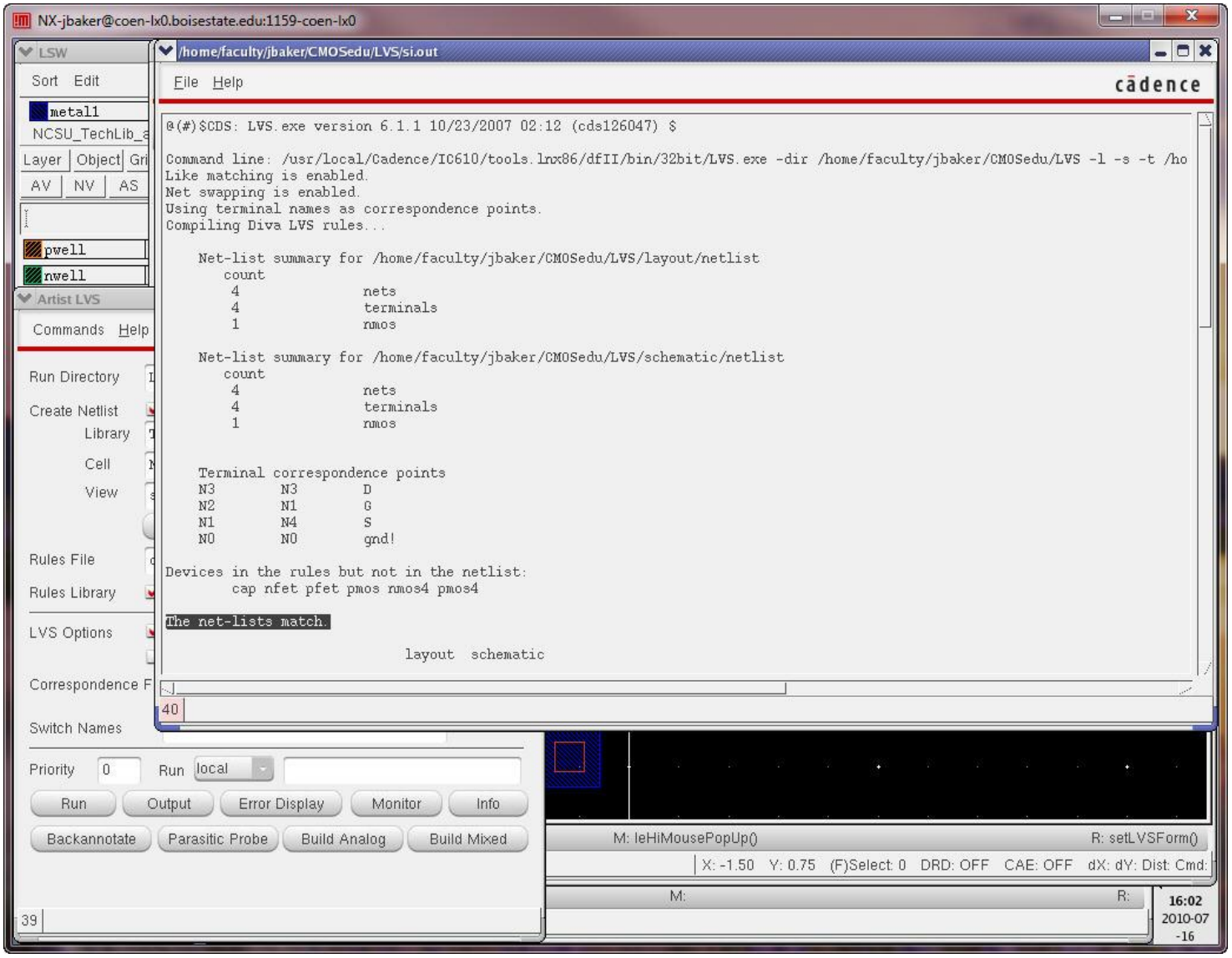
Move the symbol and then add the wire and label as seen below.
It may also be good to simulate this schematic (using the sim_NMOS_IV cell), following the steps seen above, to ensure that we still get the NMOS's IV curves.



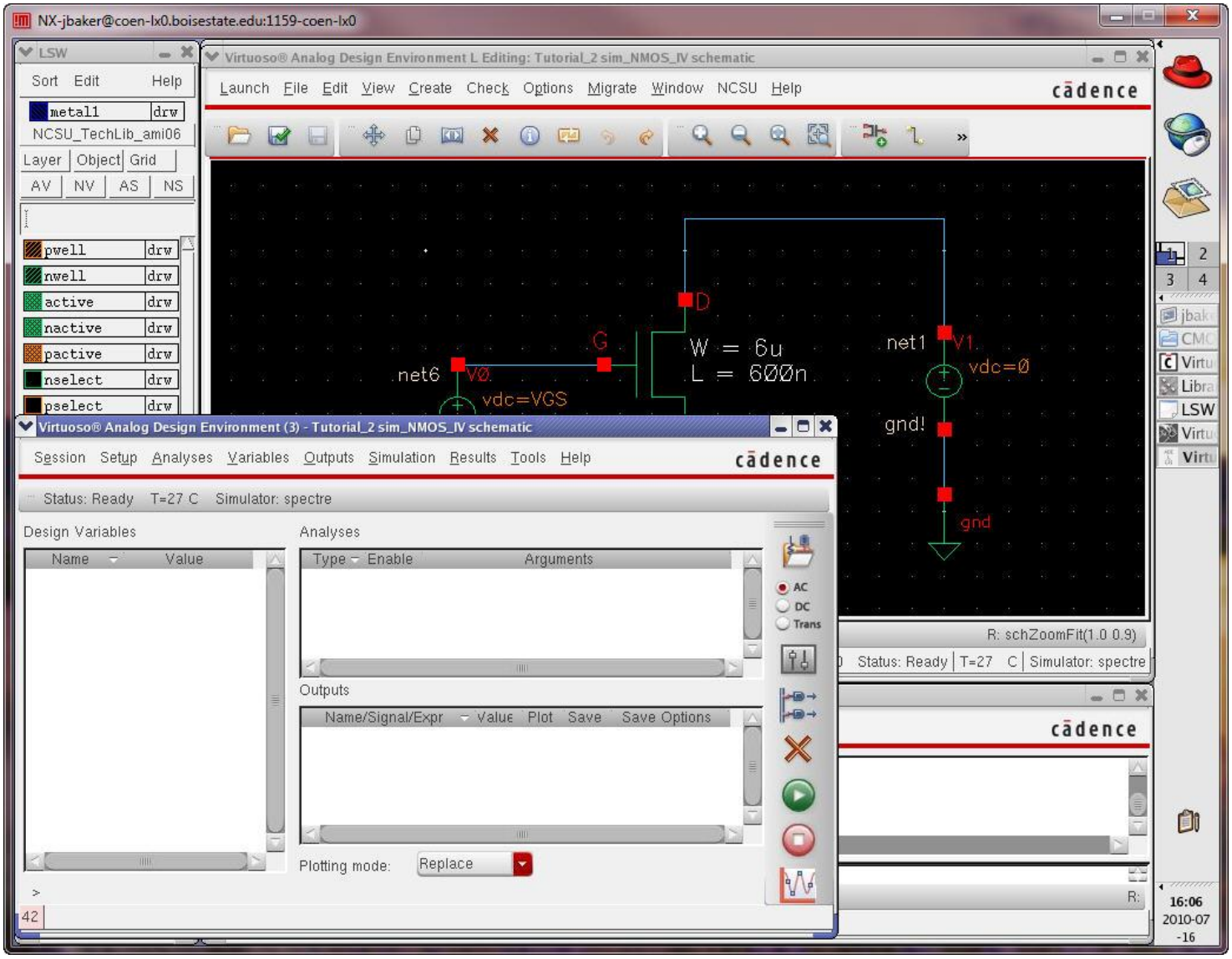
Open the extracted view and perform an LVS.
Ensure that you select the correct cell, see circled text below.


NX-jbaker@coen-lx0.boisestate.edu:1159-coen-lx0
LSW
Sort Edit
metal1
NCSU_TechLib_a
Layer Object Gri
AV NV AS
pwell
nwell
Virtuoso® Layout Suite L Editing: Tutorial_2 NMOS_IV extracted
Launch File Edit View Create Verify Connectivity Options Tools Window Migrate NCSU Help
cadence
Workspace: Classic
Artist LVS
Commands Help
cadence
Run Directory LVS Browse
Create Netlist schematic extracted
Library Tutorial_2 Tutorial_2
Cell NMOS_IV NMOS_IV
View schematic extracted
Browse Sel by Cursor Browse Sel by Cursor
Rules File divaLVS.rul Browse
Rules Library NCSU_TechLib_ami06
LVS Options Rewiring Device Fixing
Create Cross Reference Terminals
Correspondence File jbaker/CMOSedu/lvs_corr_file Create
Switch Names
Priority 0 Run local
Run Output Error Display Monitor Info
Backannotate Parasitic Probe Build Analog Build Mixed
39
D
G
gnd!
ami06N
w=6u
l=600n
S
M: leHiMousePopUp()
R: setLVSForm()
X: -1.50 Y: 0.75 (F)Select: 0 DRD: OFF CAE: OFF dX: dY: Dist: Cmd:
M: leHiMousePopUp()
R: setLVSForm()
16:00
2010-07
-16

And now!

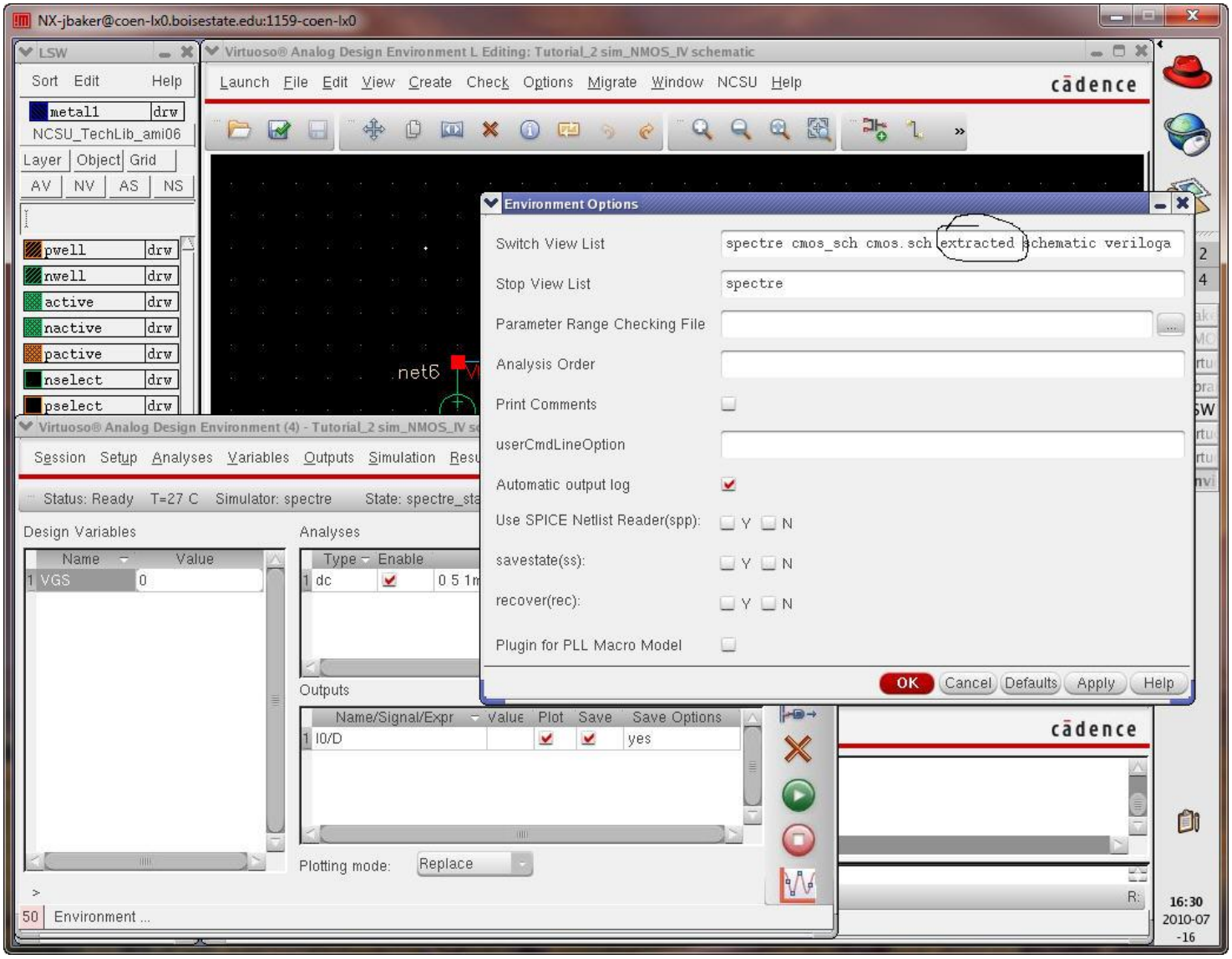


Okay, let's simulate the extracted layout.
Open the schematic view of sim_NMOS_IV and then Launch the ADE L.

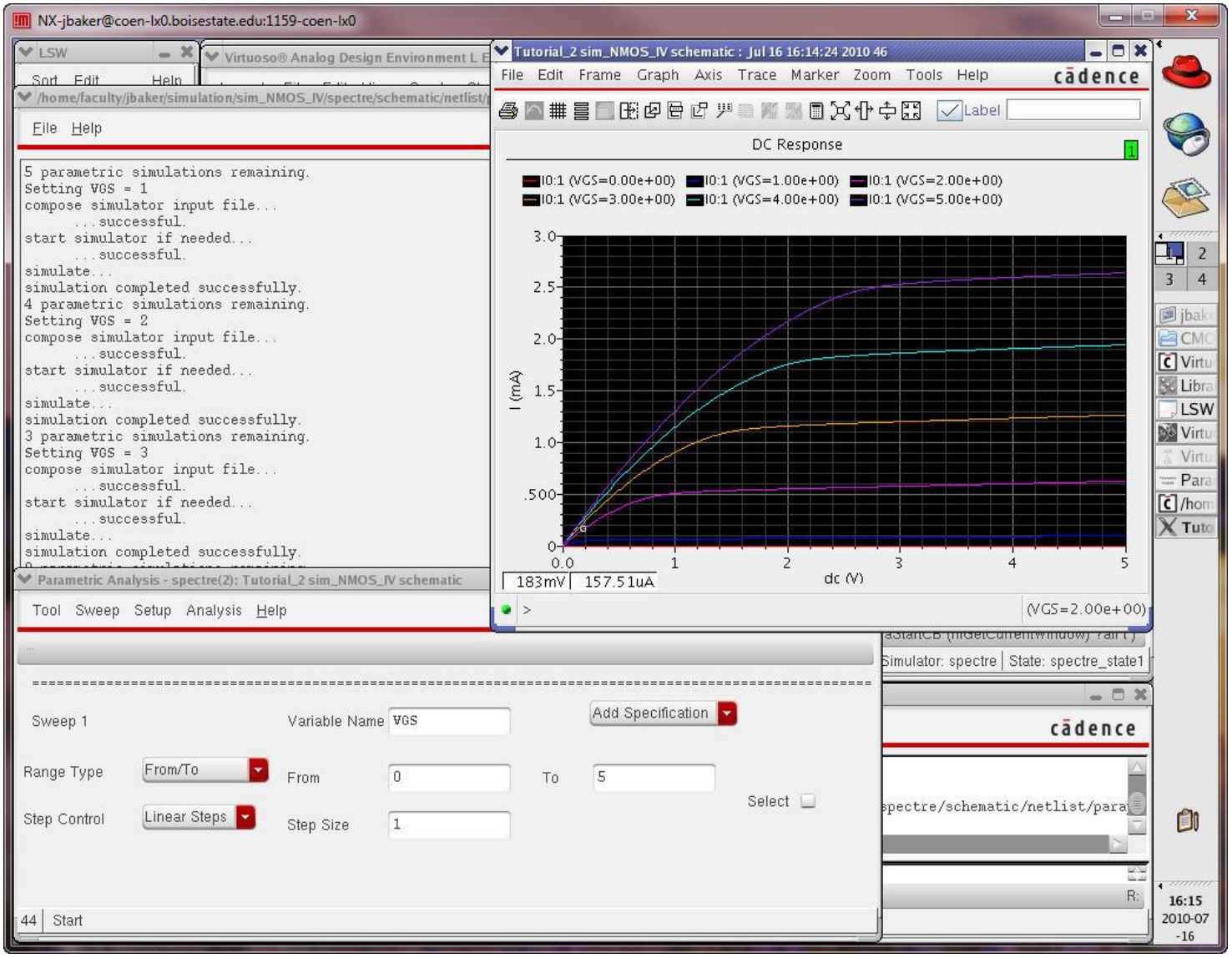


Go to Session -> Load State then select Cellview and OK.
Then, on the ADE menu, select Setup -> Environment and enter extracted before schematic as seen below (so the extracted view is used before the schematic
view when running the sim).

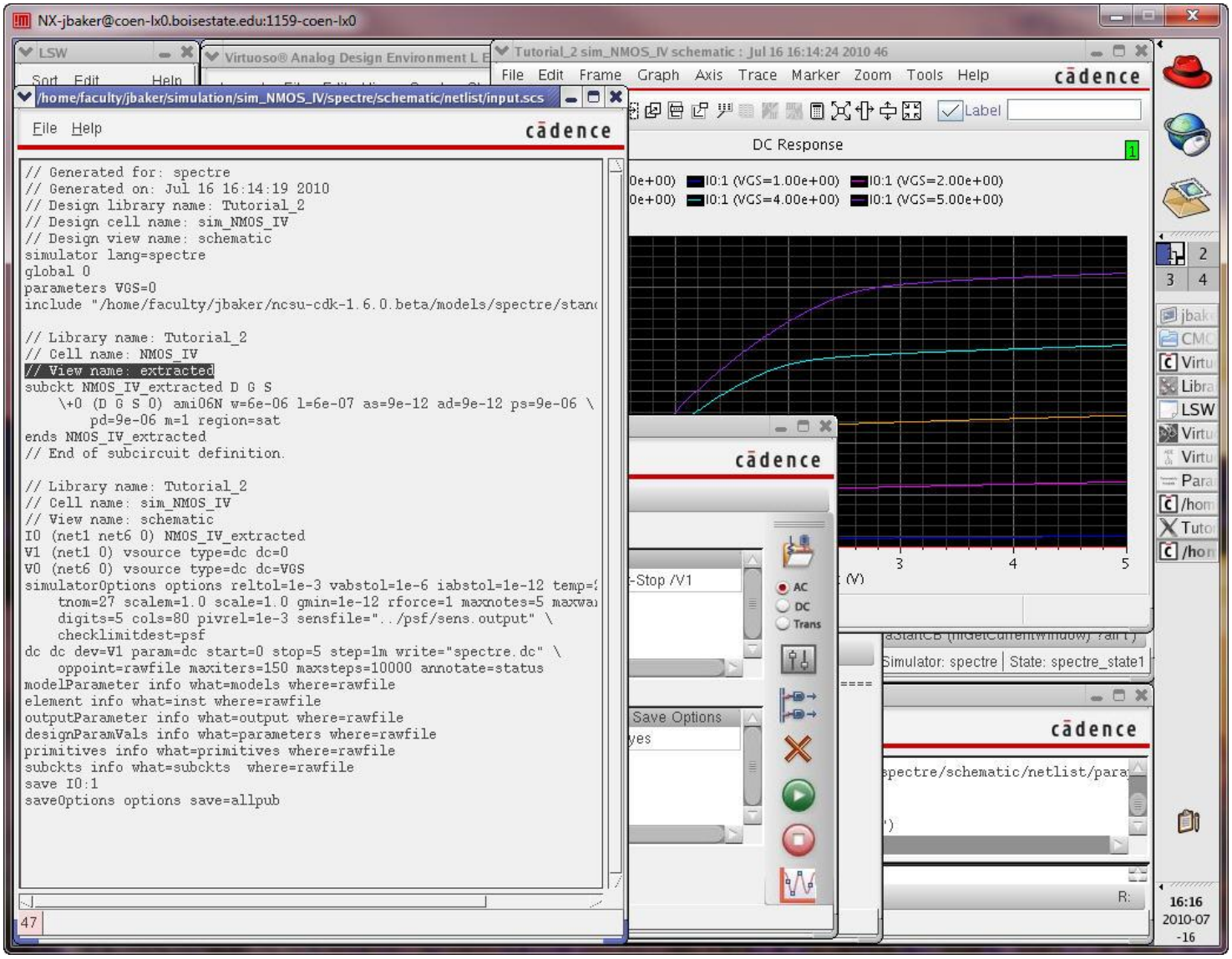
An important note: if you save the state with extracted view in front of schematic view you may get frustrated when you change your schematic and the
simulation doesn't change!



Next, again as we did earlier, select Tools -> Parametric Analysis, enter the values seen below and then select the menu items Analysis -> Start to simulate.

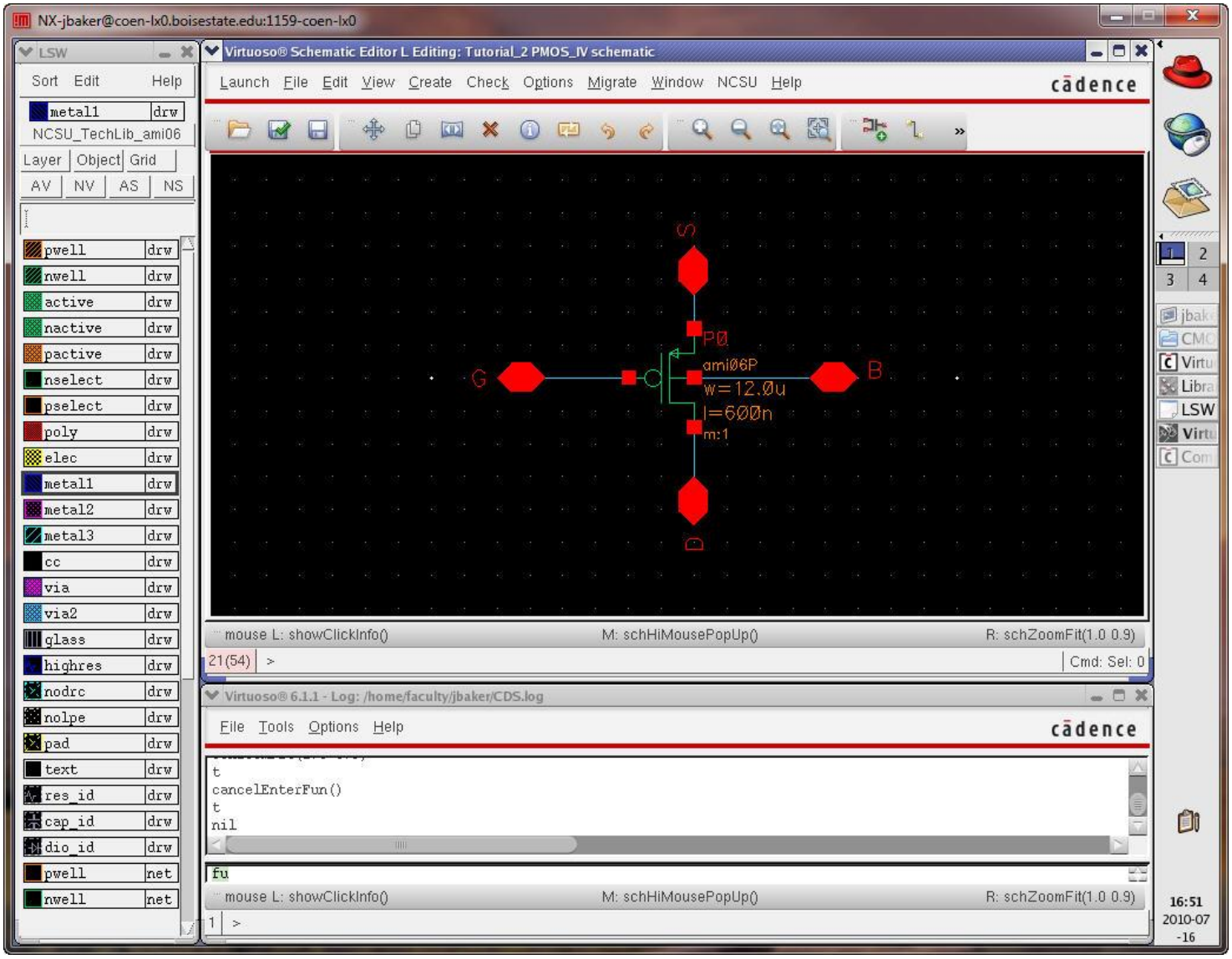


Just to ensure that we are simulating the extracted view and not the schematic view go to the ADE and select Simulation -> Netlist -> Display (see below).



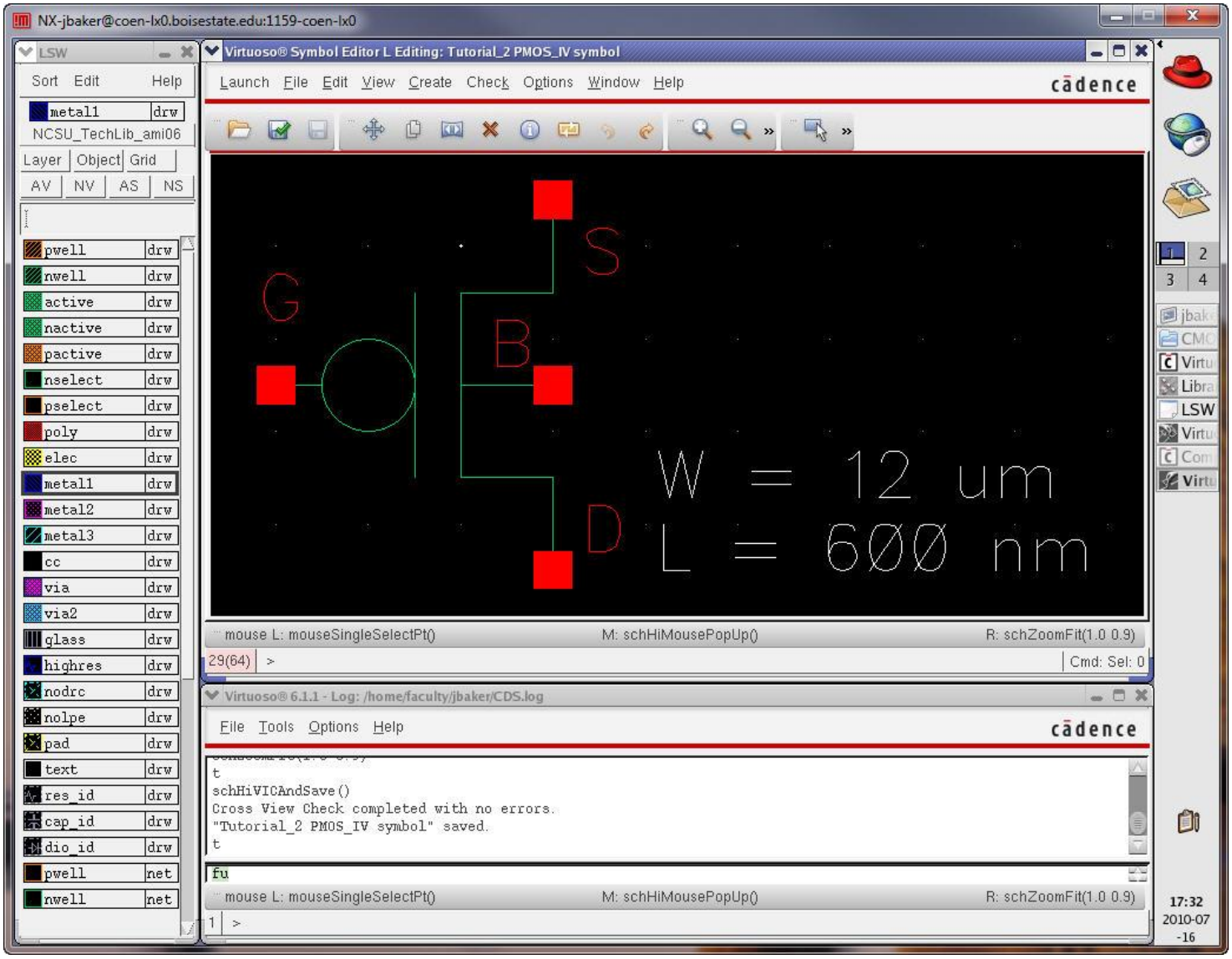
Let's repeat these steps for the PMOS device but in a much more concise manner (since now we are getting the hang of things and have made mistakes and fixed them).

Create a schematic Cell View called PMOS_IV with 4 pins as seen below. Note the W and L of the PMOS device.



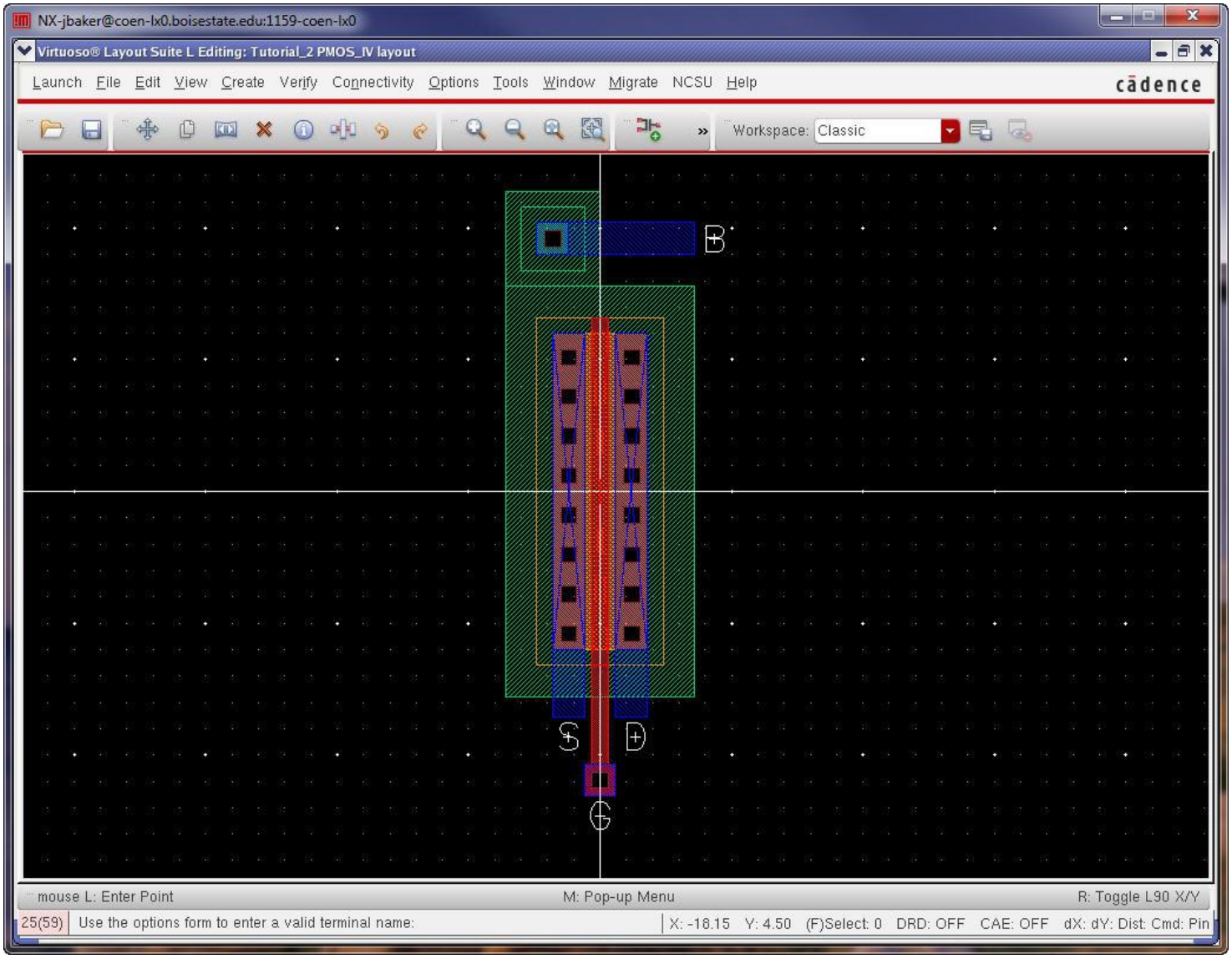
Check and Save the schematic.

Next create a symbol for the schematic.

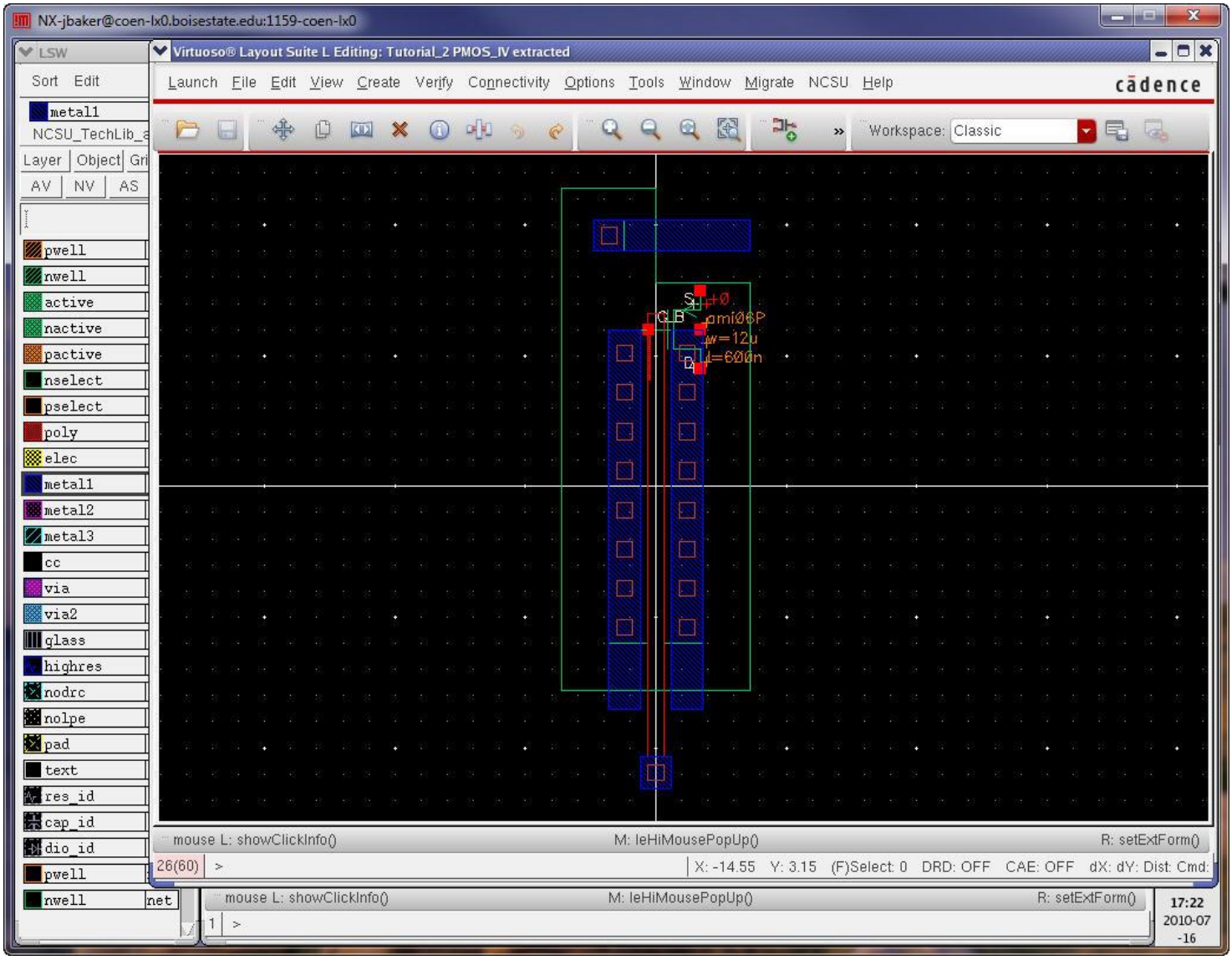


Check and Save the symbol.

Next create the layout (pmos cell and ntap).
DRC and Save the layout.

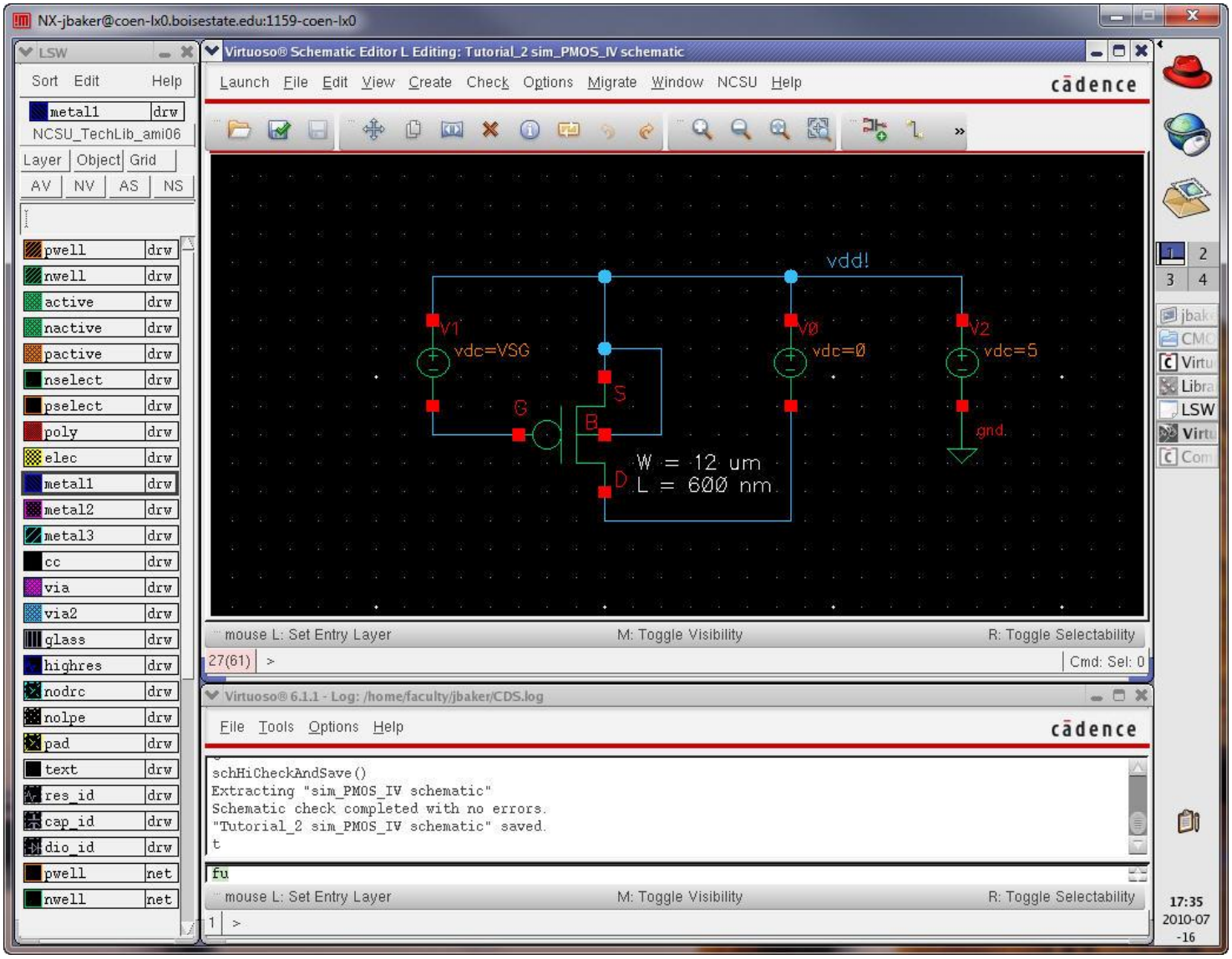


Next Extract the layout.
Open the extracted view in the Library Manager.

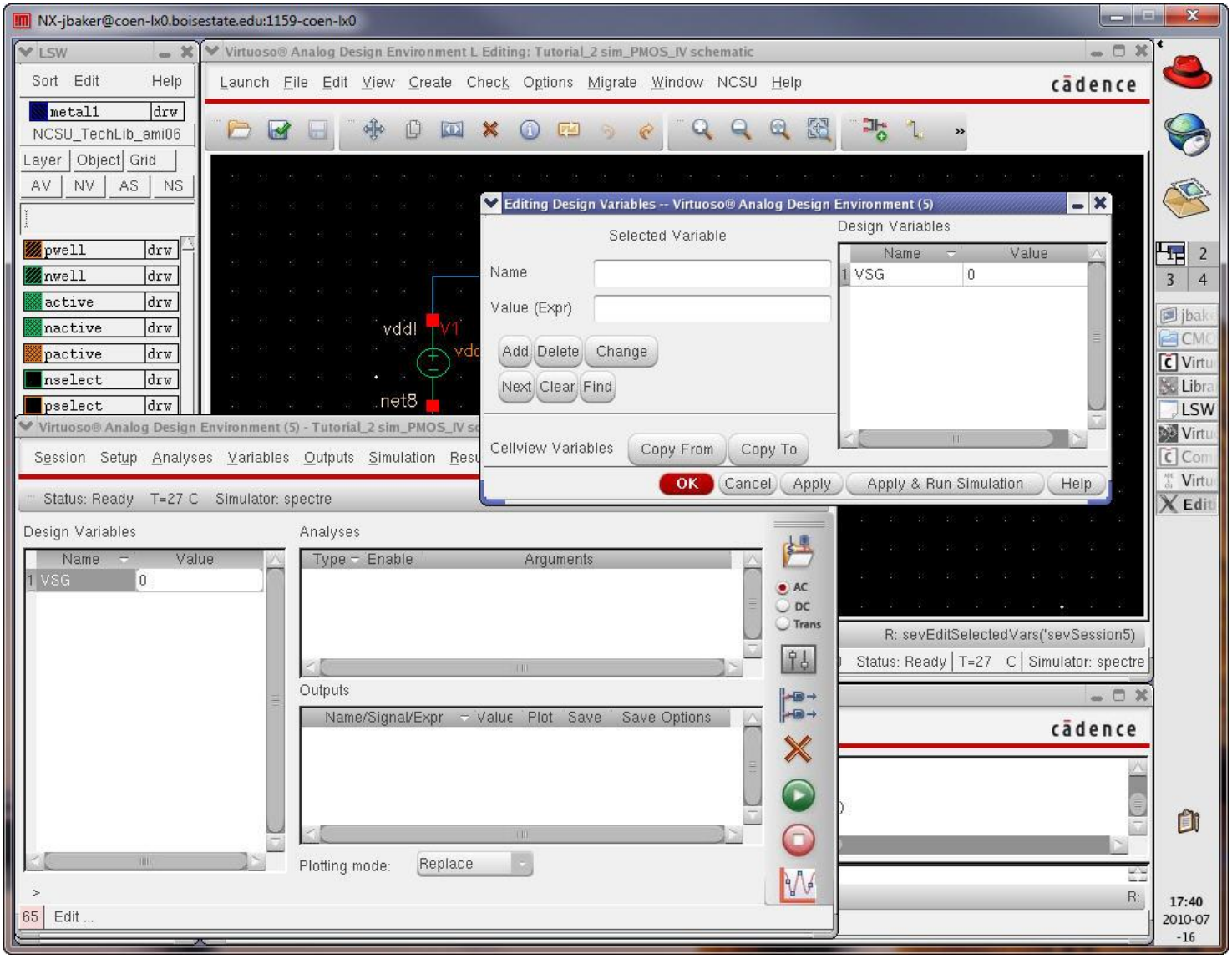


Save and close all open Views.

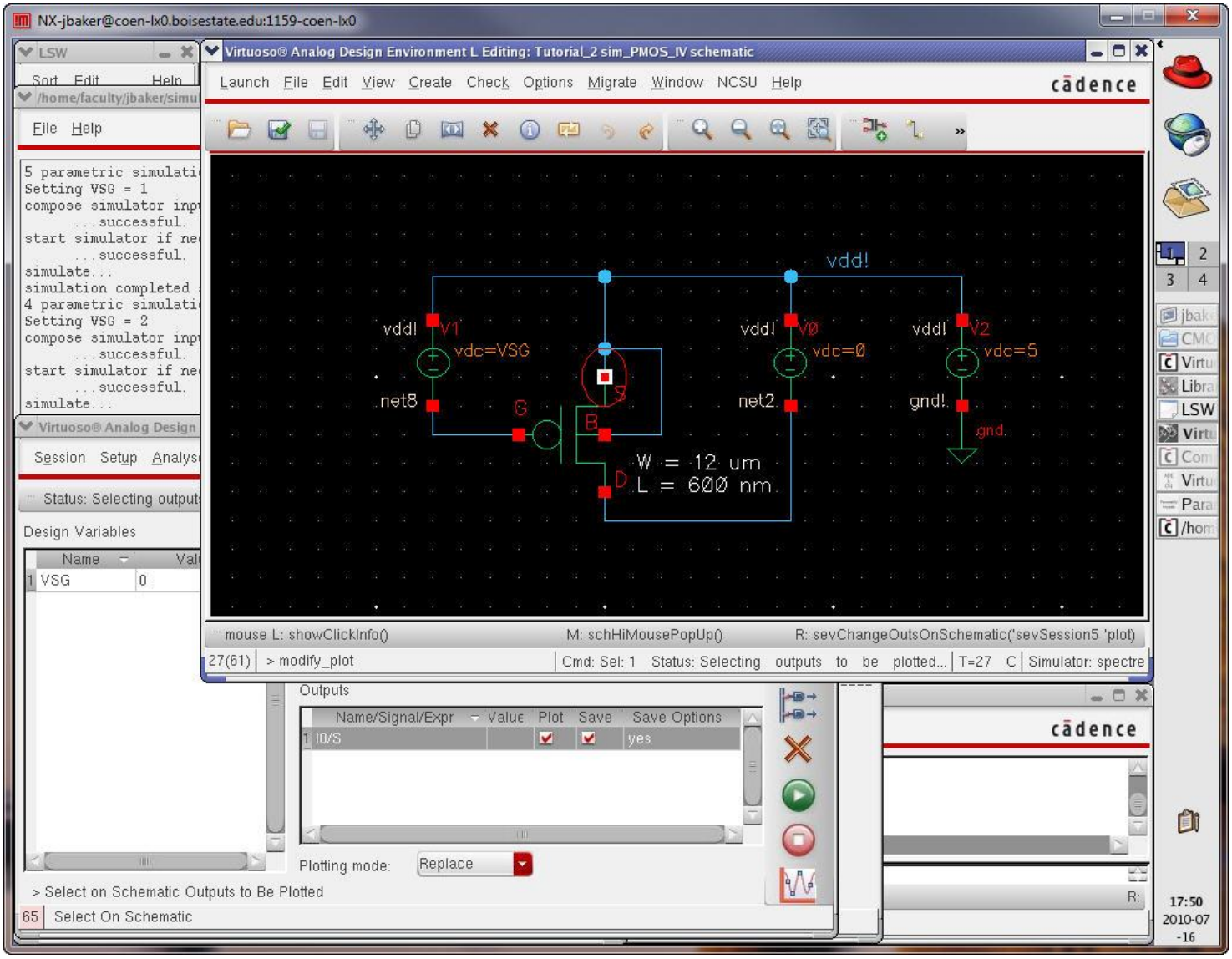
Next create a cell called sim_PMOS_IV and draft the following schematic.
Note the value of /V1 is VSG (not VGS as we used in the NMOS sim).
When finished Check and Save.



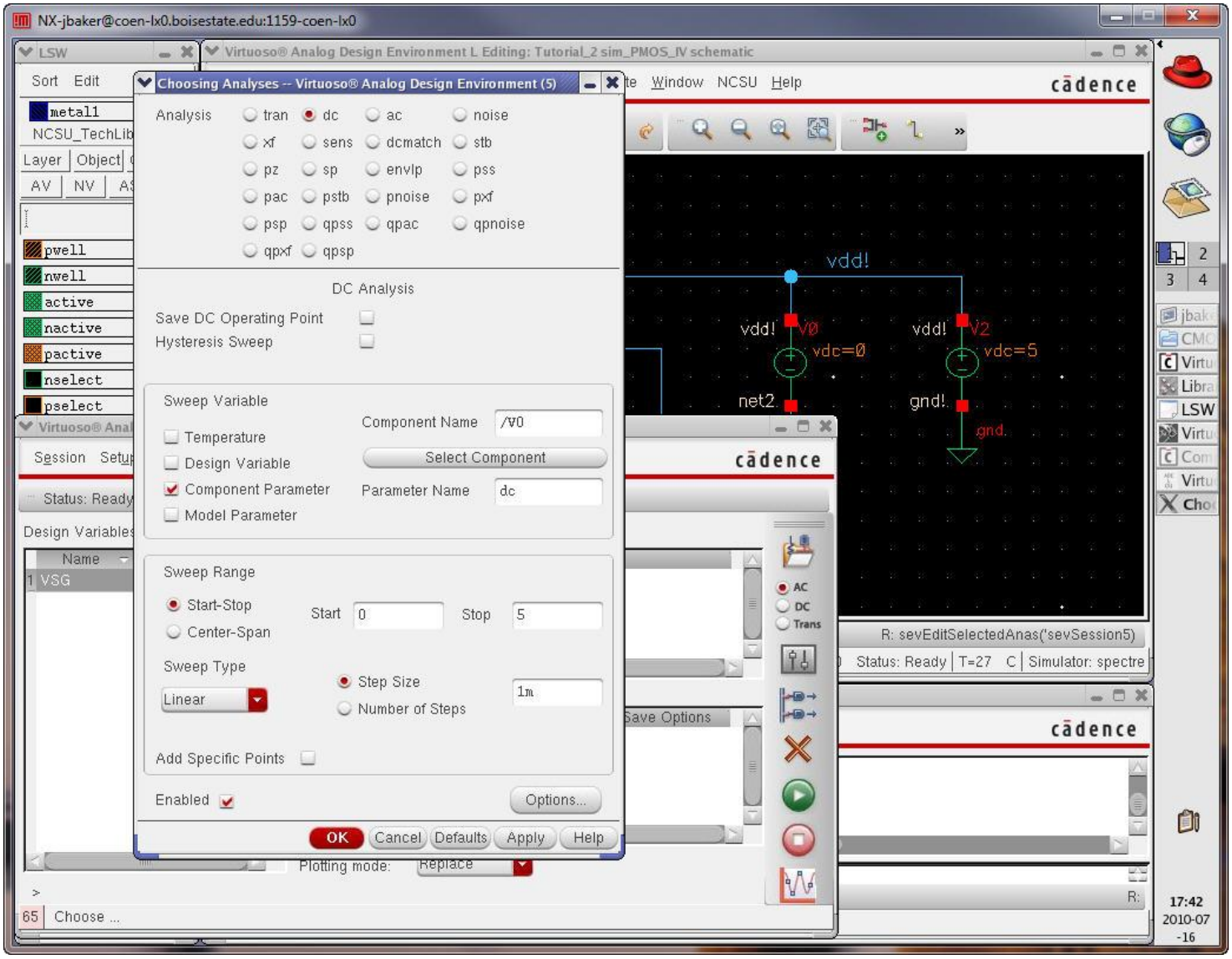
We are now ready to simulate this design.
Launch the ADE then go to Setup -> Model Libraries and select the PMOS models for AMI06 (setting the models is important!).
Variables -> Edit and Add VSG with a value of 0 (not VGS)



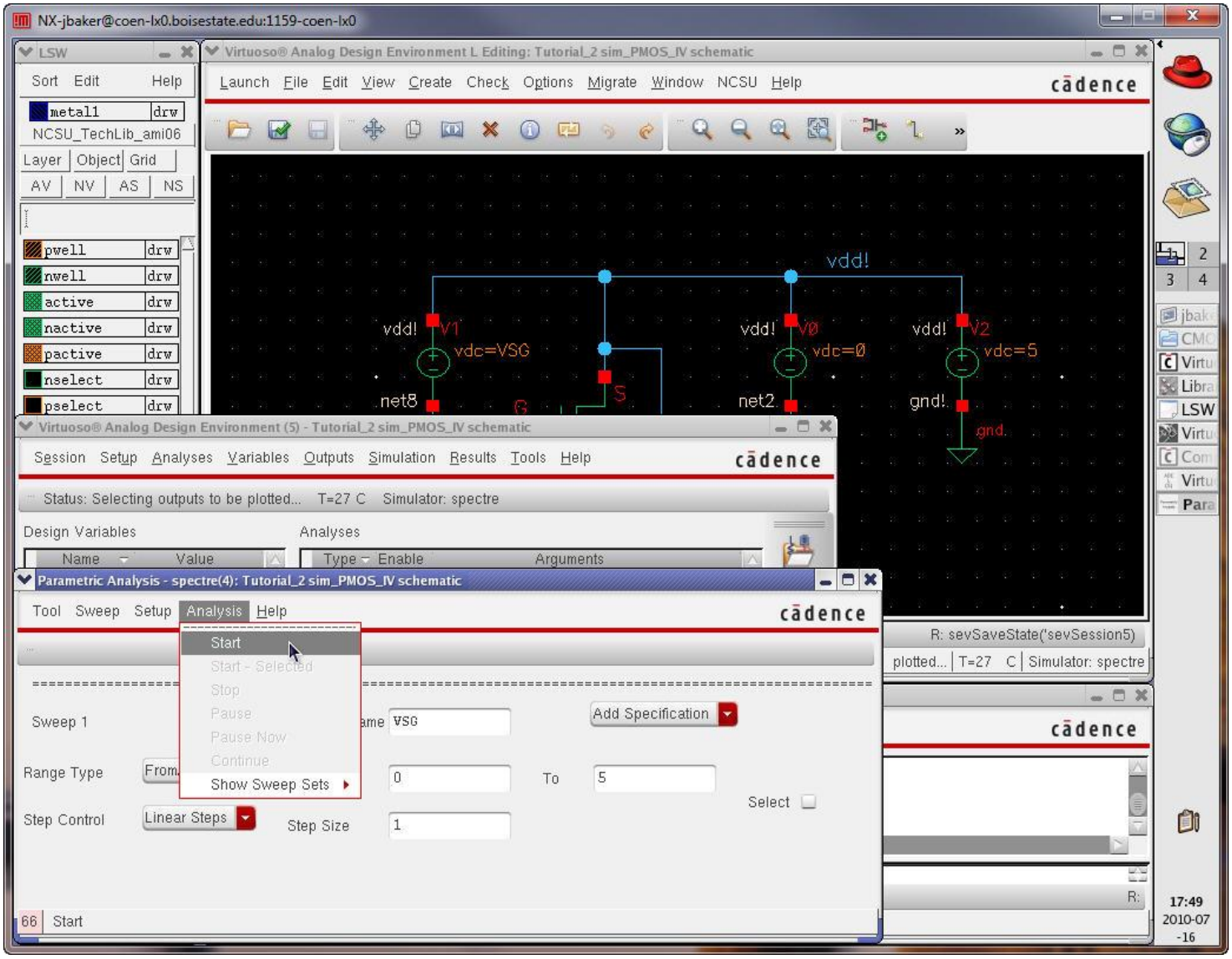
Next select Outputs and the source terminal of the transistor.
Ensure, as seen below, that both Plot and Save are selected.



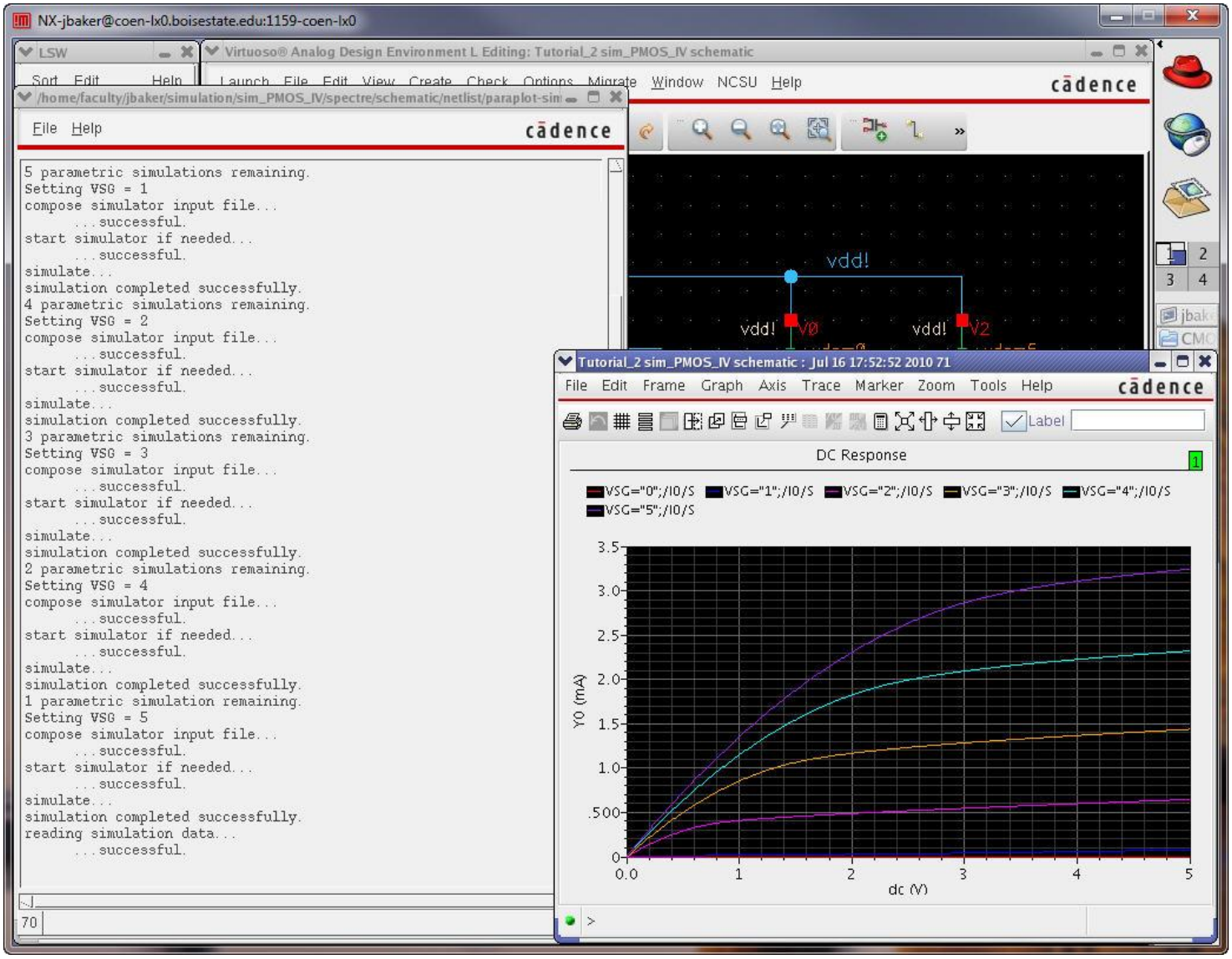
Select Analyses and enter the following.
Note that /V0 is VSD.
Also note that the vdd! Source, that is /V2, really doesn't do anything. It could be zero but we include it since it's common to connect the S/B to vdd!



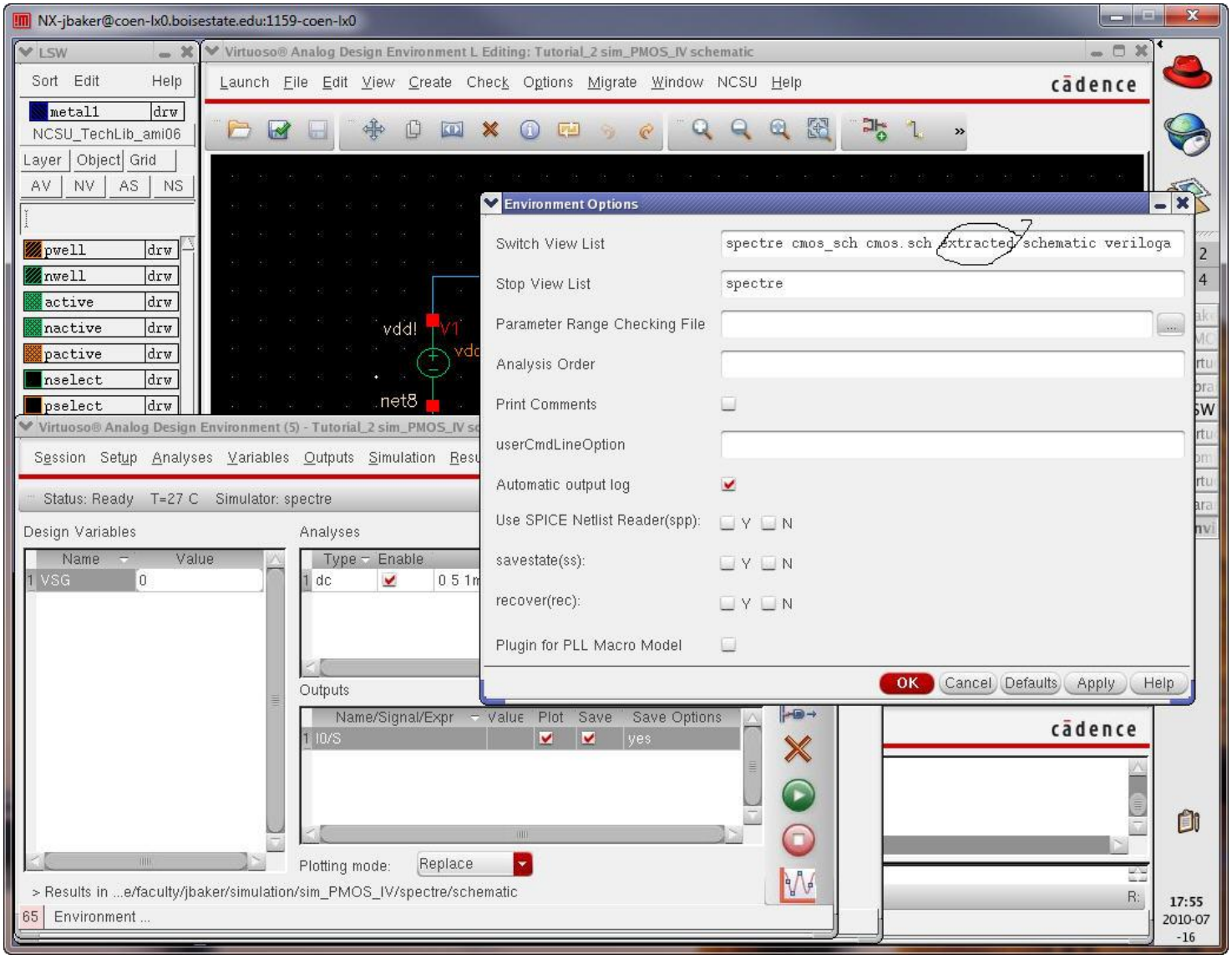
Save the state in the cellview.
Then go to Parametric Analysis and enter the values seen below (again, it's VSG not VGS).



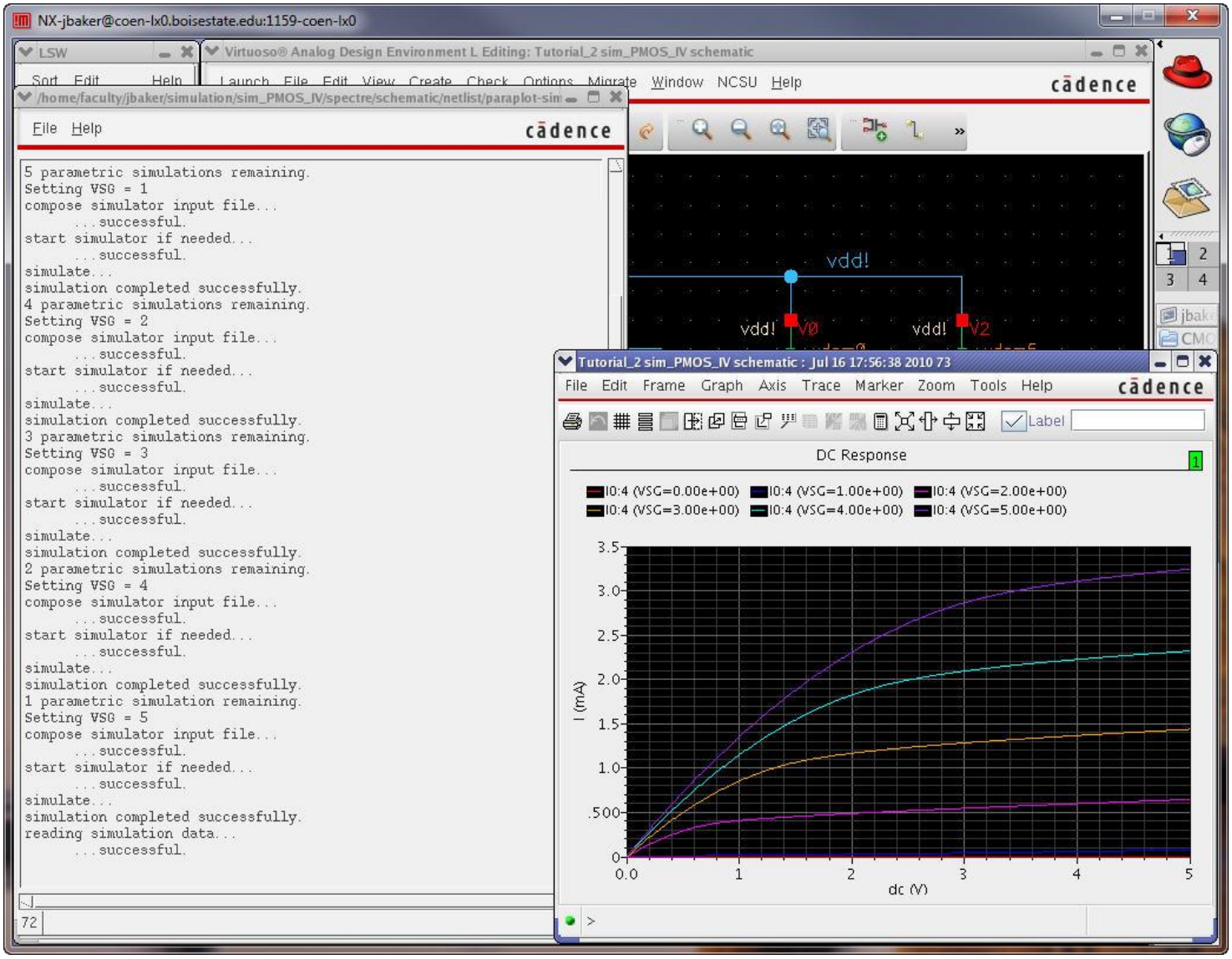
Starting the analysis results in the following. Note that this is the simulation of the schematic.



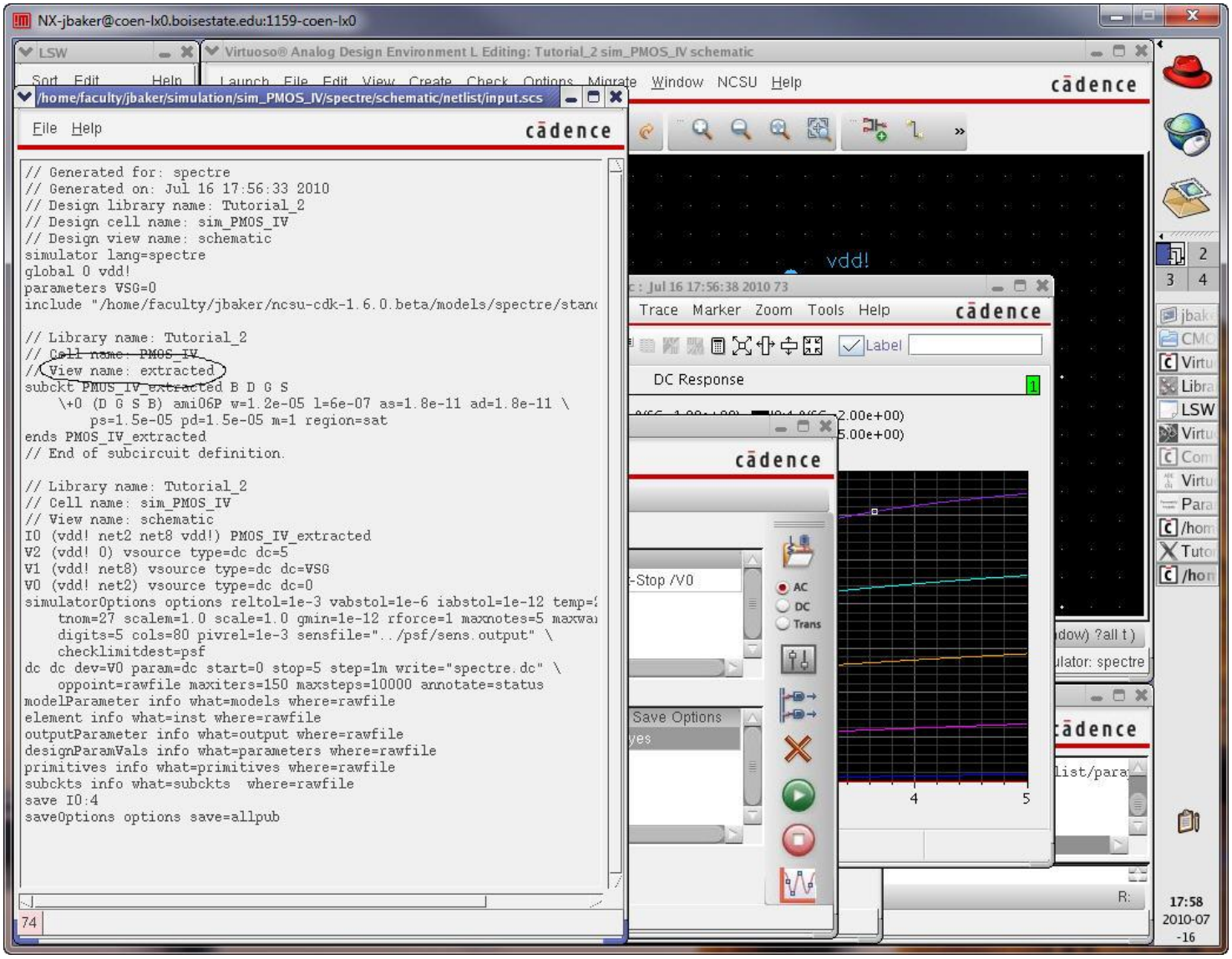
Now let's simulate the extracted layout.
In the ADE go to Setup -> Environment and add extracted before schematic.



And we get the same thing as with the schematic.



Just to verify that we are simulating the extracted view go to ADE Simulation -> Netlist -> Display.



An important note: if you save the state with extracted view in front of schematic view you may get frustrated when you change your schematic and the
simulation doesn't change! The simulator simulates your old extracted view instead of your modified schematic view.

Save the state and close everything. This concludes Tutorial 2.

For your reference the Tutorial_2 directory is available in Tutorial_2.zip.

Return