Electrical and Information Technology LTH

Analog IC Design (ETIN25)

Lab1 Laboratory Manual

Henrik Sjöland
Jonas Lindstrand
Martin Liliebladh
Markus Törmänen

October 2016

Laboratory 1: Cadence, DC parameters and current mirror

The goal of this laboratory is to get acquainted with Cadence, to study the basic DC parameters of the MOSFET and take a closer look at a current mirror. Cadence is a powerful design and simulation tool and it is crucial when it comes to analog and radio circuit design on chip. In this manual the long way through the menus will be shown every time something new will be done, but the short command, if there is any, will also be shown. The short command will be shown in brackets next to the command, e.g. "add instance" (i). Also, notice that Cadence is case sensitive.

Introduction

As you have seen in the lectures the transistor has three regions of operation which are determined by the DC voltages V_{DS}, V_{GS} and V_{th}. The transistor behaves differently in the different regions and the drain to source current can be modeled according to:

$I_{DS} = 0 \text{ for } V_{GS} - V_{th} < 0$ (off) (1)

$I_{DS} = \frac{k'}{2}\frac{W}{L}(2(V_{GS} - V_{th})V_{DS} - V_{DS}^2) \text{ for } 0 < V_{DS} < V_{GS} - V_{th}$ (triode/linear region) (2)

$I_{DS} = \frac{k'}{2}\frac{W}{L}(V_{GS} - V_{th})^2 \cdot (1 + \lambda V_{DS}) \text{ for } 0 < V_{GS} - V_{th} < V_{DS}$ (active/saturation region) (3)

During the laboratory you will simulate the transistor operating in different regions and plot the results. Afterwards should you extract the parameters $k'(W/L)$, V_{th}, g_m and λ from the plots with Matlab. How to extract the parameters is described in the next section.

Parameter Extraction by Local Fitting Method

The method of local fitting means that each of the MOSFET parameters is measured in the working region of the transistor where it dominates. In this way, only a small number of simulations are necessary, which makes the method easy to perform. The main drawback is that it is very model dependent, i.e. a unique measuring program has to be designed for each model.

Let us first have a look at the N-channel transistor in its linear region (small V_{DS}). In the model the drain current is modeled as in equation (2). Here we find that I_{DS} is proportional to the gate-source voltage minus V_{GS0} as defined below.

$$V_{GS0} = V_{th} + V_{DS}/2$$

This means that the threshold voltage can be estimated by, for a fixed value of V_{DS}, plotting an I_{DS} *vs* V_{GS} diagram and estimating the intersection with the V_{GS}-axis, see figure 1.

In order to determine the value of *kW/L* we take the derivative of equation (2) as function of V_{GS}, and get

$$\frac{\delta I_D}{\delta V_{GS}} = k'\frac{W}{L}V_{DS}$$

and since this is the slope of the curve, we can now easily determine *kW/L* by analyzing the linear part of the curve. It is advisable to set a small value for V_{DS} to achieve a wide linear range of operation.

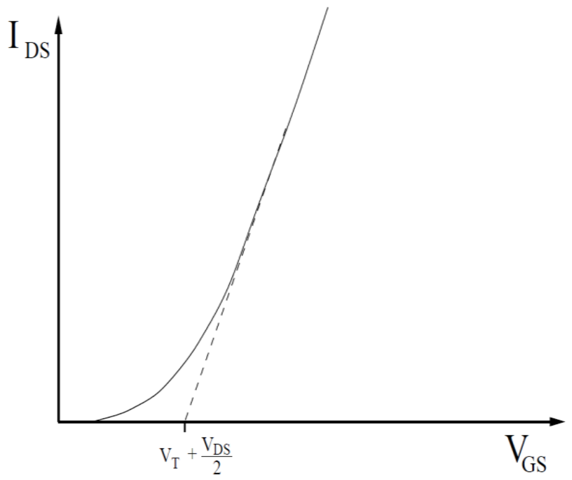


Figure 1: Extraction of the threshold voltage.

The channel length modulation coefficient (λ) is measured in the saturation region of the transistor. See Eq. 1.164 and 1.165 in the textbook. The drain current, in this region, is described by equation (3). If we assume a fixed $V_{GS} > V_{th}$, we can determine λ by looking at the sensitivity of I_{DS} for $V_{DS} > V_{GS}-V_{th}$. By interpolating the slope of the I_{DS} curve in saturation, λ can be determined by finding for which value of V_{DS} the extrapolated line intercepts the V_{DS}-axis. See figure 2.

$$V_{DS0} = -1/\lambda$$

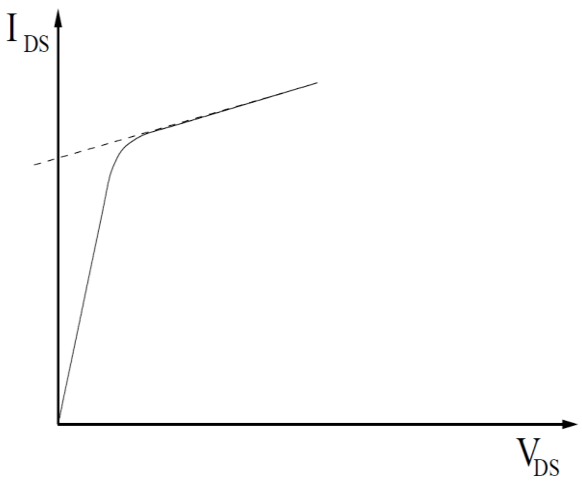


Figure 2: Measuring the channel length modulation factor.

The small-signal transconductance g_m in the saturation region is defined as

$$g_m = \frac{\delta I_D}{\delta V_{GS}} \approx k' \frac{W}{L} (V_{GS} - V_{th})$$

Homework

1. Read this manual carefully. It is recommended that you start Cadence and try to solve the laboratory assignments before the laboratory..
2. Look at the section about Parameter Extraction by Local Fitting Method and prepare a Matlab program that can extract the parameters when you have got the data from the laboratory session.
3. Calculate the following parameters: g_m, C_{GS}, I_{DS} and f_t. Use parameters from the 65nm datasheet used in this course (found on the Analog IC course website) and the following values: W = 10µm, L = 0.2µm, V_{GS} = 600mV, V_{DS} = 50mV and V_{DS} = 600mV
4. Explain in your own words how the current mirror works. Why is the current mirror used?
5. Calculate the reference current (I_{ref}) and width (W) of the transistors in a 1:1 PMOS current mirror. The DC output resistance (r_O) is 500kΩ and the drawn channel length is 1µm. All transistors are in strong inversion with V_{ov}=0.2V. Use the 65nm process parameters.
6. Explain why it is beneficial to have the transistors divided into an even number of fingers?
7. What is the definition for the output impedance of transistor?

Introduction to Cadence and simulation of a MOSFET's DC parameters

Start a terminal window and create a folder for the laboratory sessions. Open the folder and initiate Cadence with the command "inittde ana20xx". When the terminal is ready type "virtuoso &" to start Cadence, see figure 3.

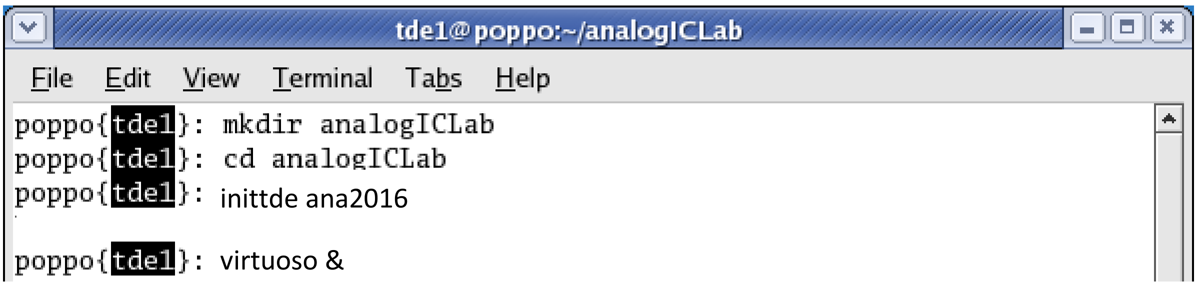


Figure 3. Terminal window with the commands to create a directory and start Cadence.

Cadence will now open two windows, "icfb – Log:/…" and "Library Manager". Cadence uses many windows and it is not unusual to have six windows opened at the same time. To make it a bit easier and to get a good overview one suggestion is to use more than one desktop. "icfb – Log:/…" is the main window in that extent that if you close it or use the menu "File"-> "Exit …" Cadence will shut down. It also logs all the commands and activities you perform, you can always check here if something is not working. "Library Manager", see figure 4, shows the different libraries with their content. A library contains different instances/circuit elements that can be used in your designs. You will now create your own library that will be used during the laboratory sessions of this course. Go to "File - > New -> Library" in the "Library Manger" window. Write a name for your library, e.g. analogIC, and click "OK". In the next window you choose "Attach to an existing techfile" and click "OK". In the list "Technology Library" choose "cmos065" and then "OK". The library will now appear in the library list in "Library Manager".

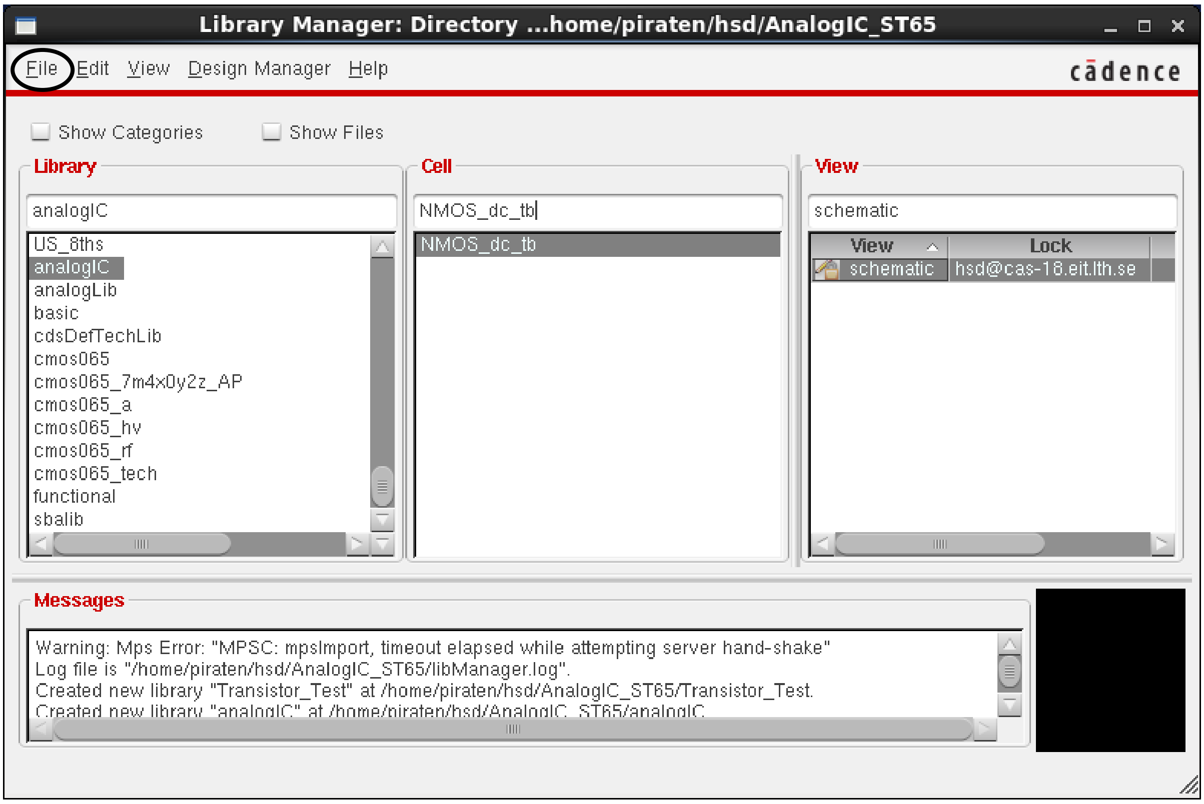


Figure 4. Library manager

You should now create you first cell or circuit in Cadence. Make sure that the library you just created is marked in the list and then go to "File - > New" once again but this time choose "Cell View …". Check that "Library name" is the right one and write a good name for your cell, e.g. NMOS_dc_tb, and choose "Schematics L" in the "Open with" list. See figure 5. Click "OK". Your cell will now be created and a "Schematic Editing" window will open. It is in this window that you will design your circuit.

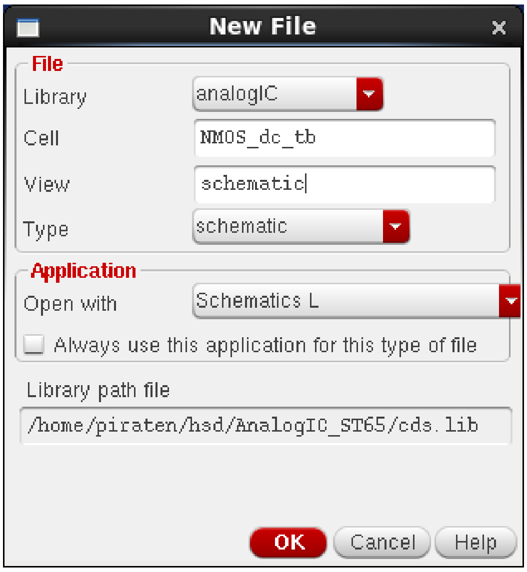


Figure 5. Create schematic.

Let's create a simple schematic like the one in figure 6. This circuit can be used to test the DC operation of a single MOSFET and compare it to the homework assignment. This part of the lab is also intended to get you familiar with Cadence schematic tools.

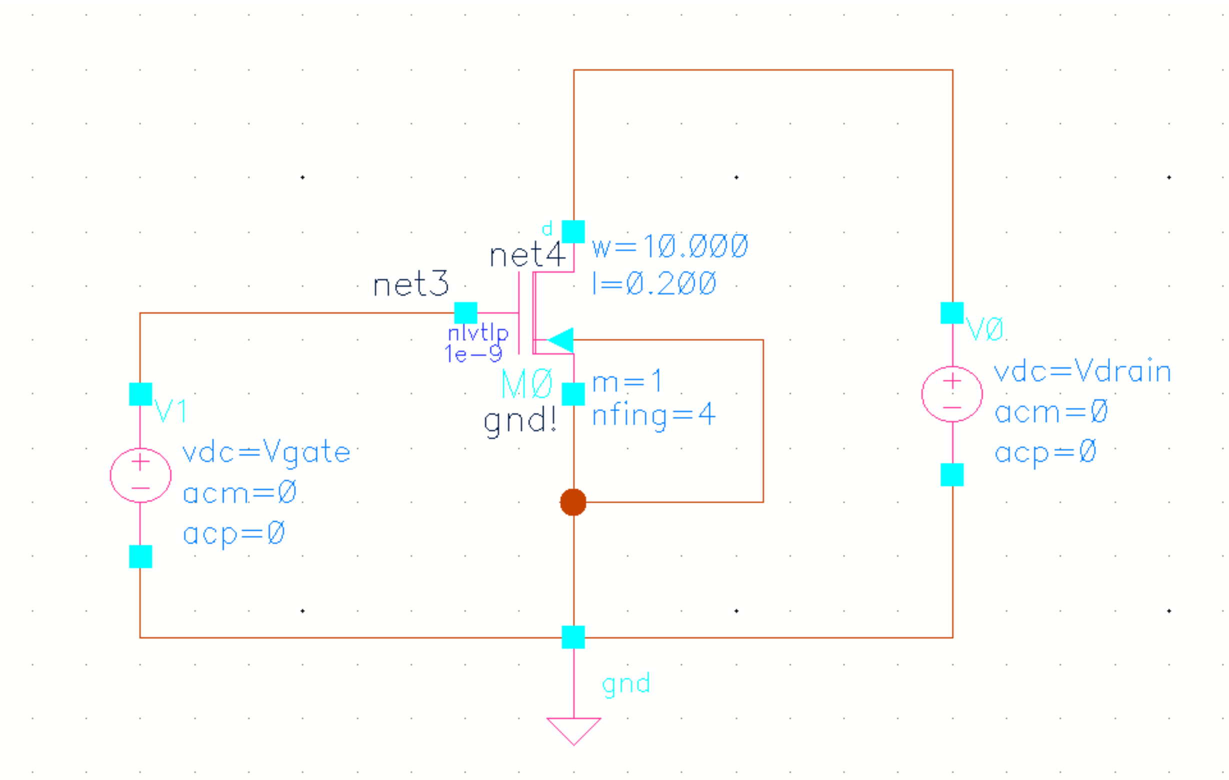


Figure 6. Schematic of the circuit used to test a MOSFETs DC behavior.

To add an instance, circuit element, go to the menu "Create -> Instance…" (i)

In the new window appearing, Add Instance, click on "Browse". Another window opens called "Library Browser – Add Instance" and here you should select Library – cmos065, cell – nlvtlp, view – symbol. Another window "Add Instance" pops up, see figure 7. In this window you can enter the component parameters, but it is not necessary to do so at this point. Move the cursor over schematic window and you will see a MOSFET symbol, left click on the mouse to place it. In nlvtlp, the n stands for NMOS, lvt for low threshold voltage, and lp for low power. It is the NMOS transistor with parameters according to the datasheet, i.e. the one we will use in this course.

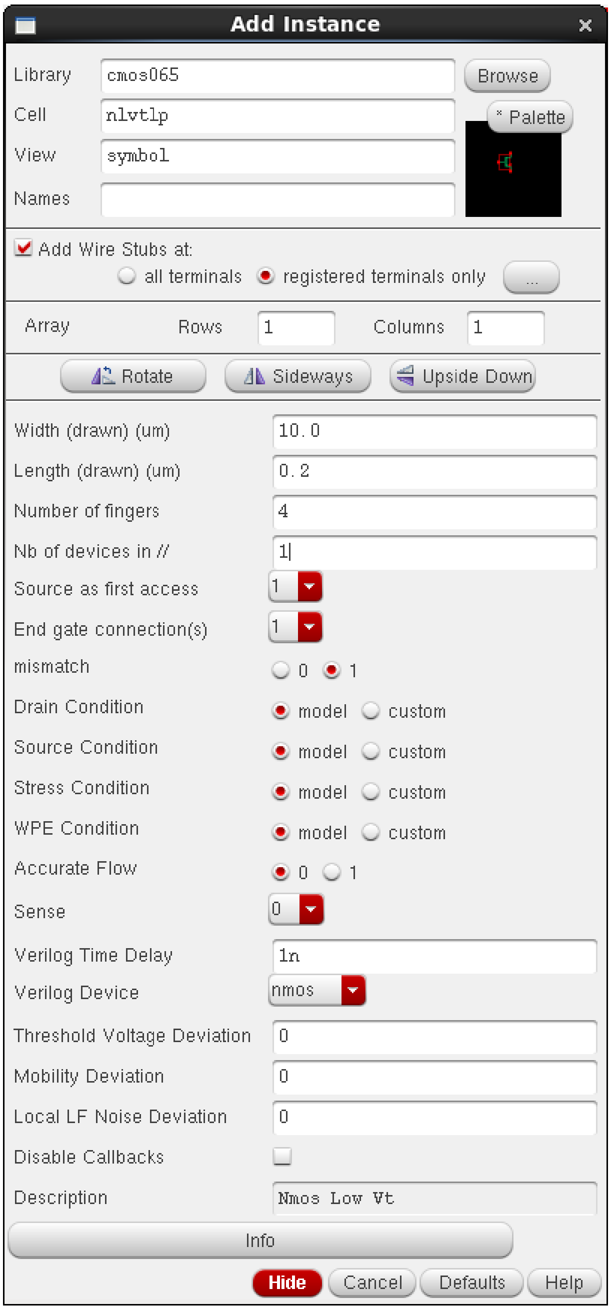


Figure 7. Add an instance, nmos.

Another important library is analogLib. Here you find fundamental circuit elements such as ground and vdd symbols, voltage and current sources, resistors, capacitors, etc. The circuit elements in this library are ideal and have no parasitics, non-linearities, mismatches or other non-idealities.

If the window “Library Browser – Add Instance” is still opened just click on “analogLib” in the “Library” menu and find the following elements: vdc and gnd. Hint: you can write the name of the cell in the box under the headline “Cell” to find your element faster i.e. you don’t have to scroll down. If you already closed the “Library Browser – Add Instance” window just add an instance as described earlier (i). Add the elements to your schematic according to figure 6. When you are done adding your circuit elements just press the escape button on the keyboard to close the add instance windows. Escape is one of your best friends in Cadence as it cancels the command you are currently using, e.g. adding instances, wires or moving objects. Make it a habit to press it when you are done with your latest command and get used to press the Esc button a lot.

Now you should add the wires. Go to the menu “Create->Wire (narrow)” (w)

Connect the elements by left clicking on the red contacts. You can also click anywhere in your schematic window to fasten the wire there before continue to the next contact. If you want to move a wire or element go to the menu “Edit->Move” (M) and then click on the object you want to move. In the Edit menu you will also find useful commands as: undo (u), redo (U), stretch (m), copy (c), delete (Delete) and rotate (right mouse click). The difference between move and stretch is that with stretch the wires connected to the element will follow as you move the element and the connection will not be broken, but with move the element will be moved out of the circuit.

When finished the schematic should look like the one in figure 6.

If you did not enter the proper parameters of the NMOS when creating it using the “Add Instance” window of figure 7, you should now edit the properties of the NMOS. To edit the properties of an instance go to the menu “Edit –> Properties –> Objects…” (q) and then click on the nmos. A new window opens called “Edit Object Properties”. Take a look of the different properties that can be changed. Edit the “Width”, “Length” and “Number of fingers” according to figure 8.

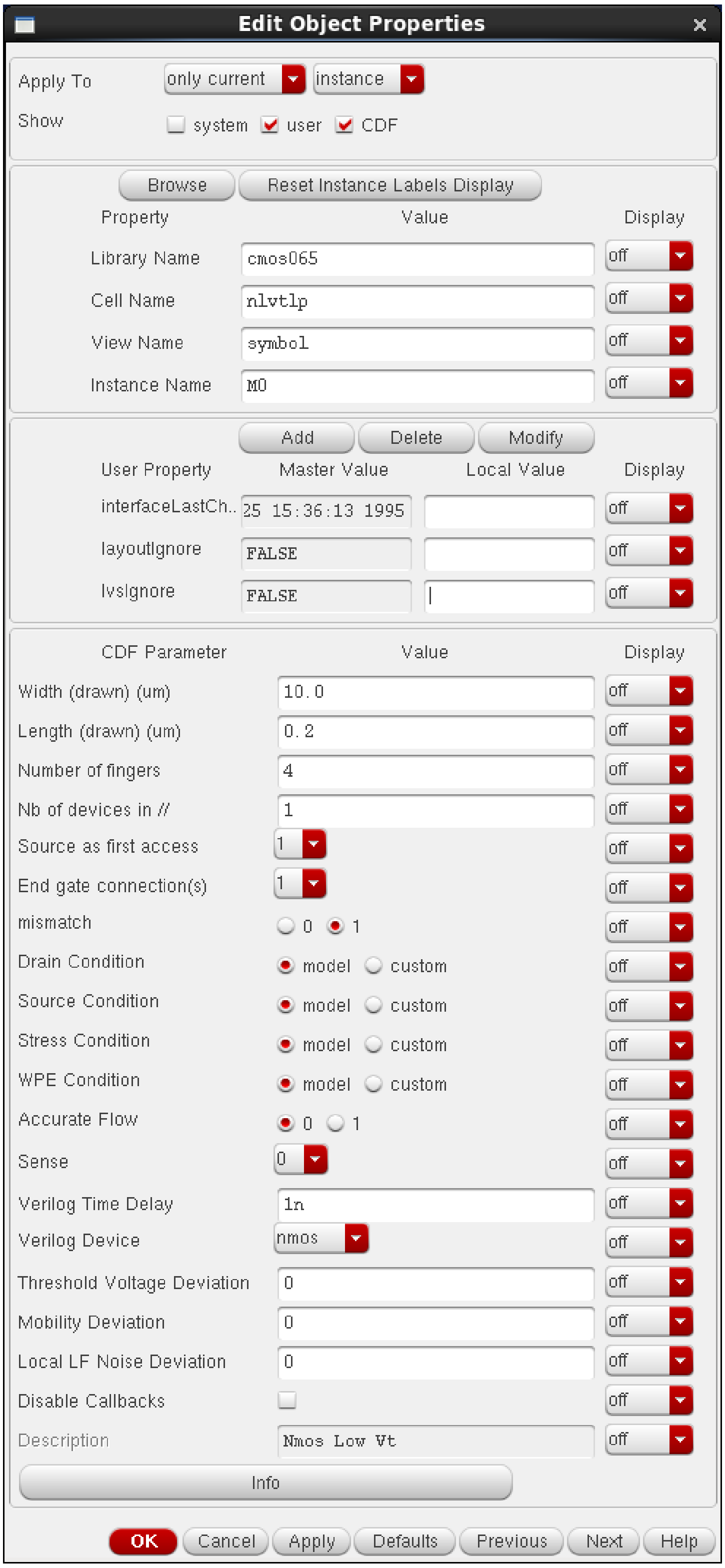


Figure 8. Edit object properties.

When you are designing and simulating a circuit you often want to try different widths of a transistor or different bias voltages to find an optimal design. To make this easier you can use “Design Variables” in your schematic. Edit the property “DC voltage” of the two DC voltage sources and write Vgate and Vdrain respectively as their DC voltage, see figure 9.

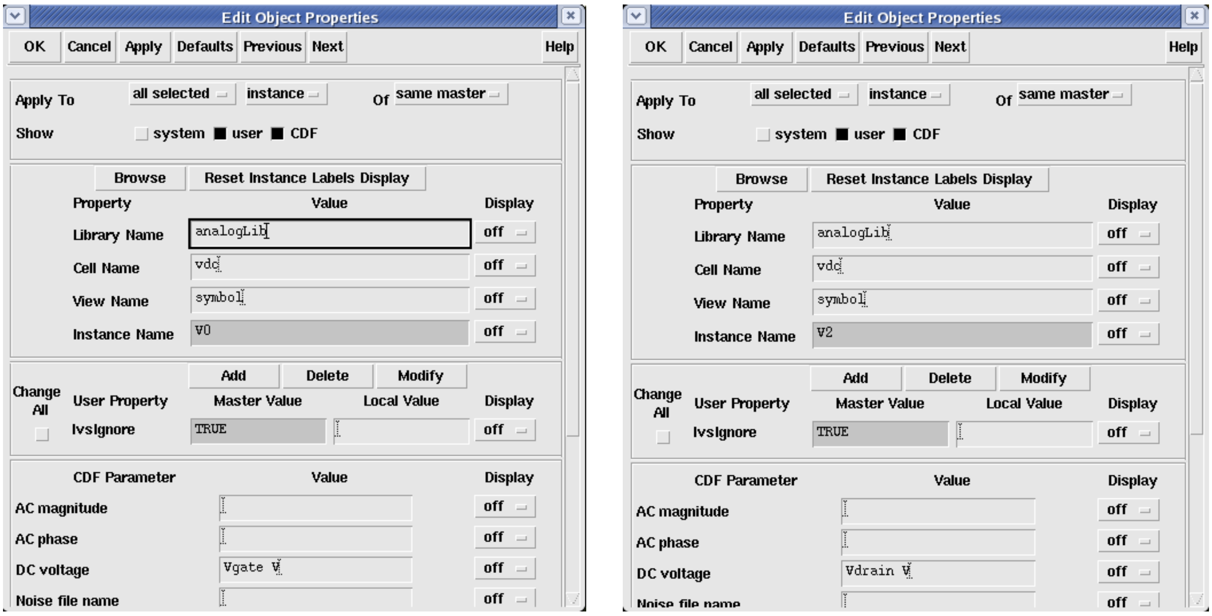
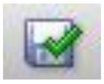


Figure 9. Edit properties of the two DC voltage sources with variables.

Your schematic is now ready for simulation and should be saved. Press the button "Check and Save" in the top menu bar. Every time you change something in your schematic you have to press "Check and Save" before you run your next simulation. Most of the time when a simulation won't start, you have forgot to press "Check and Save".


Check and Save

To start the simulation tool go to the "Launch" menu at the top of the schematic window and then choose "ADE L". A new window will appear called "Virtuoso Analog Design Environment". First open "ArtistKit -> Setup Corners". A window called "Setup Corners" pops up. Click on "Save Scenario" and "Save Model File" in that window, then "Session -> Close". This selects default (nominal) transistor models and closes the window. Using different corners the effects of variations in fabrication can be simulated.

To setup a simulation go to the "Analyses -> Choose …". The new pop up window contains all different simulations that can be performed and now we will do a dc simulation. Click on "dc" and then check the box "Save DC Operating point", it should look like figure 10. Click OK.

In order to set the two variables we added in the schematic go to "Variables -> Copy from Cellview". The variable names will now appear on the left side under the headline "Design Variables" in the "Analog Design Environment" window. To edit the value double click on the variable name and type the value of the variable in the box, do not forget to click on the "Change" button to save the change. Set Vgate to 600m (this represent 600mV) and Vdrain to 50m. It should look like it does in figure 11. Then press "OK" to close the "Edit Design Variables" window.

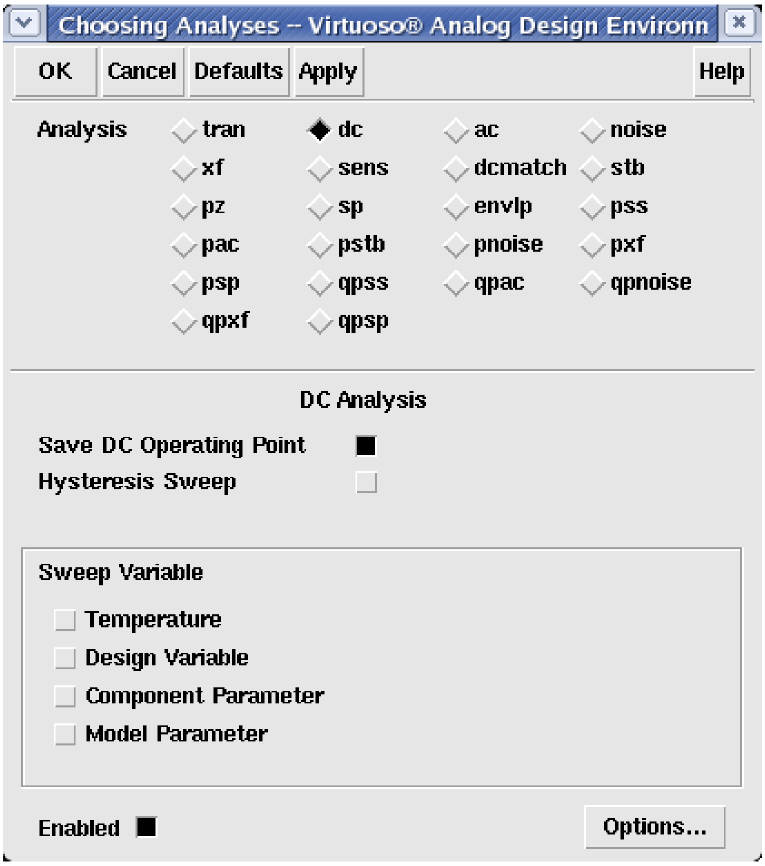


Figure 10. Analyses that can be performed in Cadence.

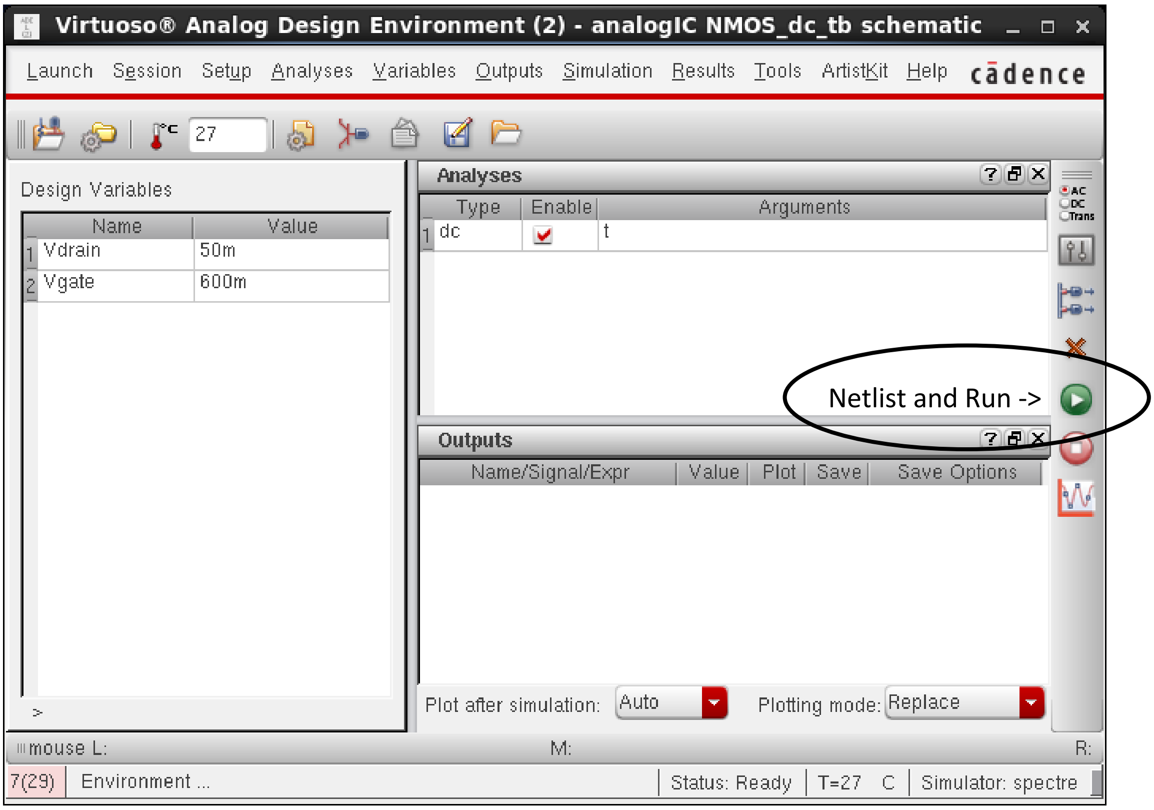


Figure 11. Analog Environment with design variables set and a dc analisys ready to go.

Now you are ready to run the simulation. Go to “Simulation -> Netlist and Run” or click on the icon in the right side menu. When the simulation is done go to the menu “Results” -> “Print” - > “DC Operating Points” and the click on the NMOS in the schematic window. A new window will appear, “Results Display Window”, with a list of parameter values from the NMOS-transistor. You can find the value of e.g. the gate-to-source capacitance cgs, the channel resistance ron in the linear region, the threshold voltage vth and the drain-to-source current ids. Look through the list and write the value of the following variables:

cgs = ____________. ron = ____________. vth = ____________. ids = ____________. gm =____________.

In which region dose this NMOS operates with these bias voltages? ____________________________.

How do you know that? ____.

Now to make a variable sweep go to the “Analog Design Environment” window, menu “Tools - > Parametric Analysis …”. A new window “Parametric Analysis” appears, press “Add Variable” and choose Vgate in the list that appears under “Variable Name” and sweep from 0 to 1.2. Use 25 steps with automatic step control. See figure 12. This will overwrite the value you have given Vgate in the “Analog Design Environment” window and sweep it from 0-1.2 Volt while Vdrain will remain unchanged. Go to the menu “Analysis -> Start All” in the Parametric Analysis window to start the simulation with the variable sweep.

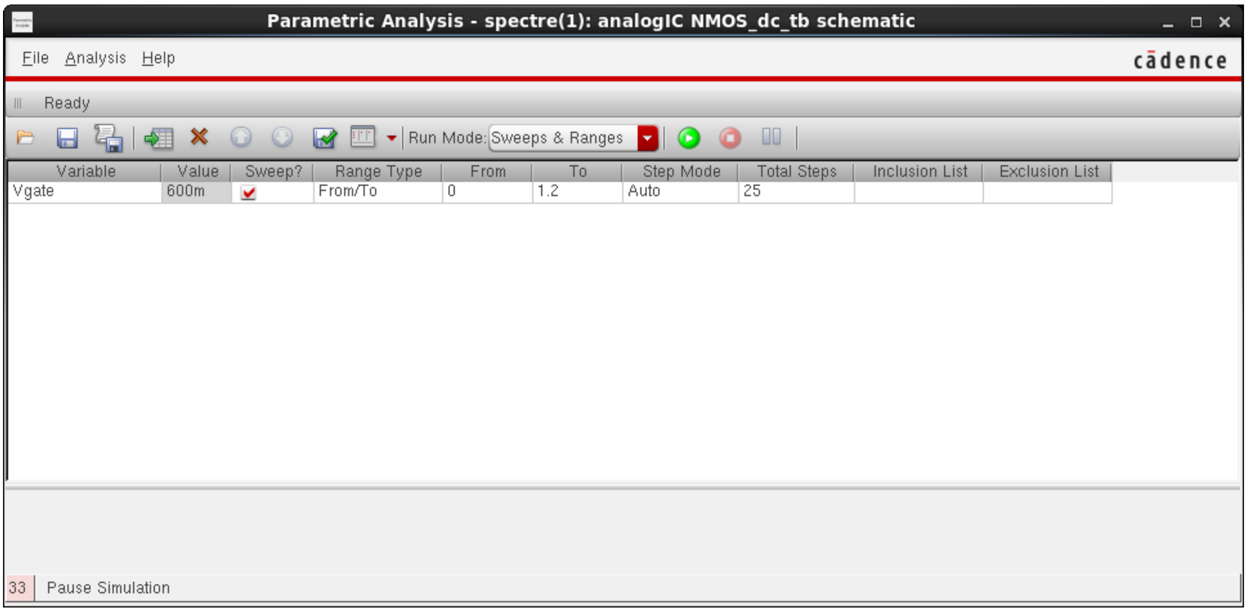


Figure 12. Parametric analysis, variable sweep.

You will now take a look at another important tool, the calculator. Once again go to the “Tools” menu in “Analog Environment”, where you use “Calculator” to open the calculator. It can perform a lot of operations and plot the results. Now click on the “op” button, stands for operating point, and a new window named “Select an instance” will appear. Click on the NMOS in the schematic, the name of the window will now change to something like “OP parameters for M0”, see figure 13. Select ids in the list and click “OK”, a line of text will appear in the calculator indicating the ids current of the NMOS you selected, see figure 14. Now press the plot button (encircled in figure 14) and Cadence will plot how the drain-to-source current varies with gate voltage (because source is grounded gate voltage = Vgs and drain voltage =Vds). Back to the calculator and press the “printvs” button (also encircled in figure 14). The simulated values used to plot the graph will now be presented in a list, in a new window called “Visualization & Analysis”. Choose ”File -> Save As CSV” and write a good “File Name”. The file will be saved in the directory where you initiated and started Cadence. You can open the text file by writing

emacs filename.txt &

in the terminal window (not in Cadence). Use this data in Matlab to plot the curve and extract the threshold voltage and $k'(W/L)$ according to this manual and the homework assignment. Compare your k’ with the one from the datasheet. You do not have to do it now but the plot and extraction should be included in the lab rapport. Note the threshold voltage from Cadence operating point as before to compare it with your extracted value later.

Is the curve a straight line as predicted by eq. (2)? If not, think carefully about what region of the curve to use for the parameter extraction.

Redo the sweep but sweep Vdrain from 0-1.2 V instead of Vgate and plot ids once again. Save the data and use it in Matlab to find λ as described in this manual.

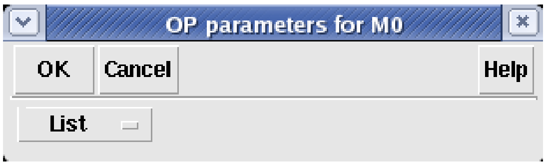


Figure 13. OP parameters.

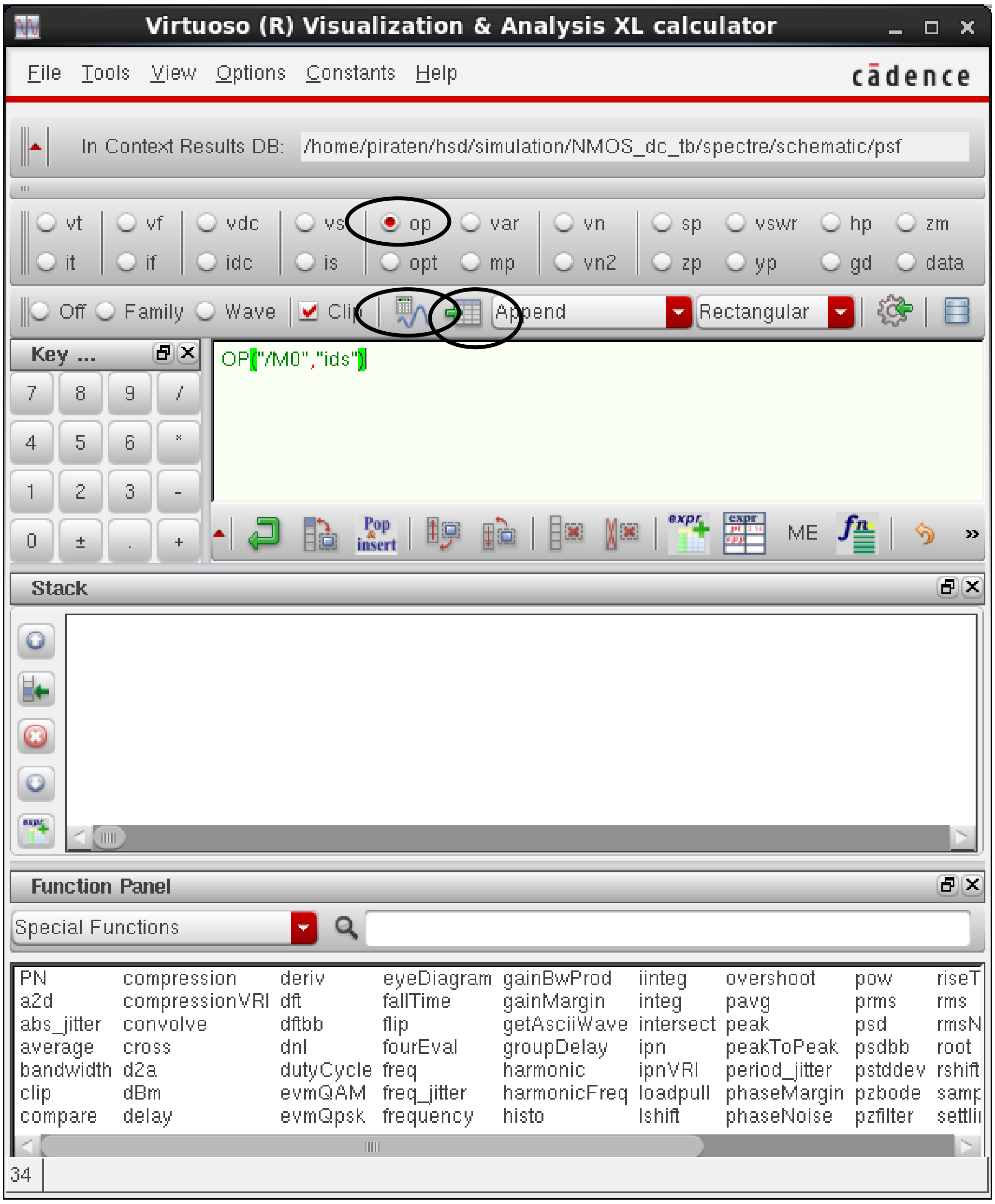


Figure 14. Calculator

HINT: If you want to make a picture of a schematic in Cadence you don't have to "print screen" and get the black background instead you can use a plotting tool from Cadence. You find it in the "File -> Print" menu in the Schematic Editing window. A new windows opens, "Submit Plot", and in the lower right corner there is a button called "Plot Options …", click it for another window with the same name to open. Here you should choose "Plotter Name" to be "EPS" and don't forget to change "Scale" to 1 (or some other value that makes a reasonable large picture). Choose "Send Plot Only To File" and write the full file path. When you are in the desired directory in your Unix terminal window just write "pwd" to get the path to that directory. Now just copy that path and add "filename.eps". Click OK in both "Plot Options" and "Submit Plot". The schematic will be saved as an EPS picture in black and white and can be used for reports and presentations. See figure 15 and 16.

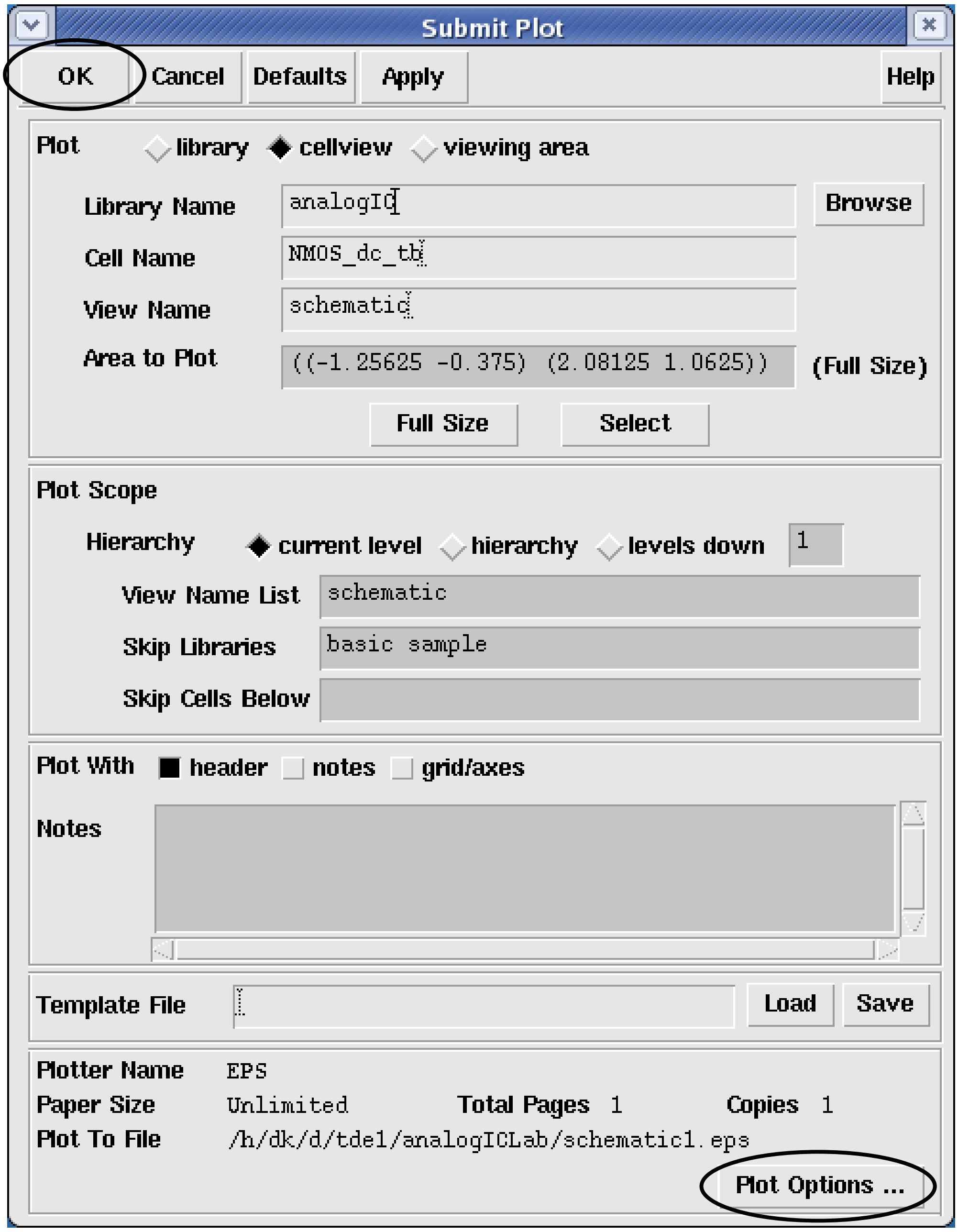


Figure 15. Submit Plot. How to plot a schematic in Cadence.

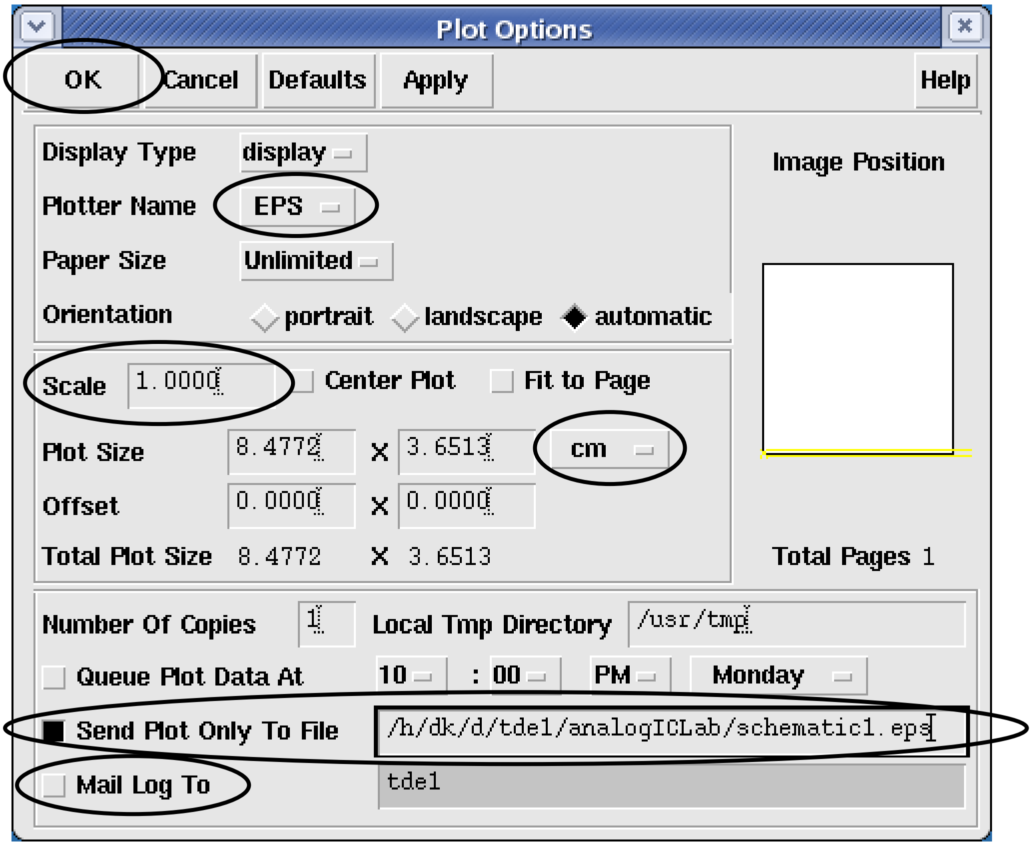
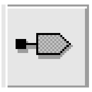


Figure 16. Plot Options. How to plot a schematic in Cadence.

Design and simulation of a PMOS current mirror in Cadence

In this part of the lab a PMOS current mirror schematic will be built and the schematic will be used to create a symbol view. This is a box representation of the current mirror schematic, which will be used to simulate the DC output resistance of the current mirror using a testbench. Create a new schematic cellview with the name “CurrentMirror” in the Library created in previous exercises i.e. analogIC. Build the current mirror using the PMOS transistor *plvtlp*. To add the transistors, go to the menu “Create -> Instance…” (i).

Give the transistors the width and length from homework assignment 5; divide the transistors into 2 fingers. The Vdd terminal is found in analogLib and cellview name is vdd. To add some I/O pins to the designed current mirror go the menu “Create-> Pin…” (p) or click on the icon in the top menu.


Create Pin

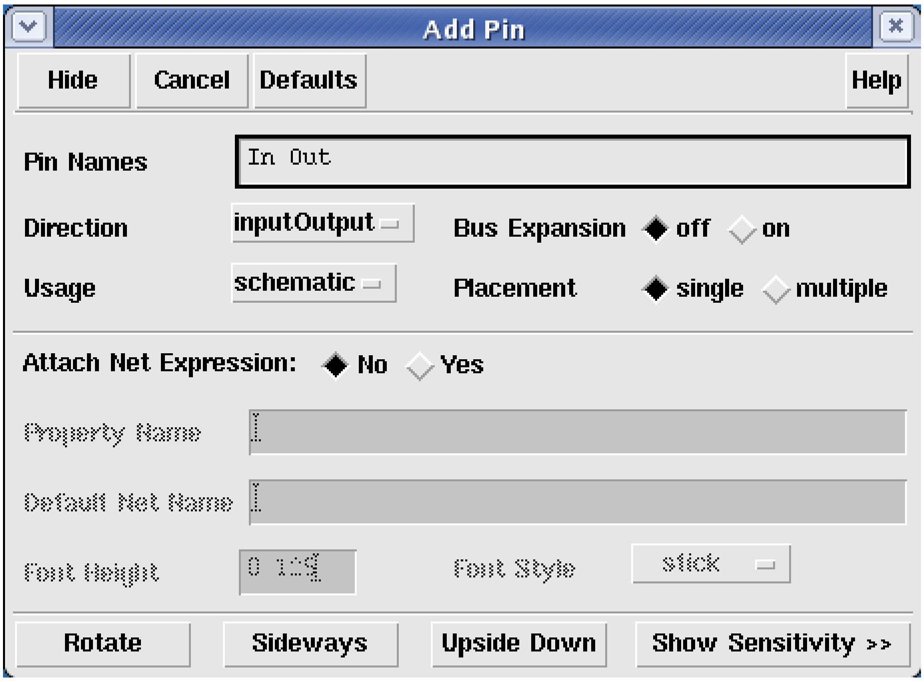


Figure 17. Add Pin Window

The I/O pins should have the direction inputOutput as seen in figure 17. The complete current mirror should look like figure 18. Now a cellview should be created, go to the menu “Create -> Cellview -> From Cellview…”. Click OK. Setup the Pin specification: “In” should be a Left Pin and menu “Out” should be a Right Pin (see figure 19 for reference). Click OK.

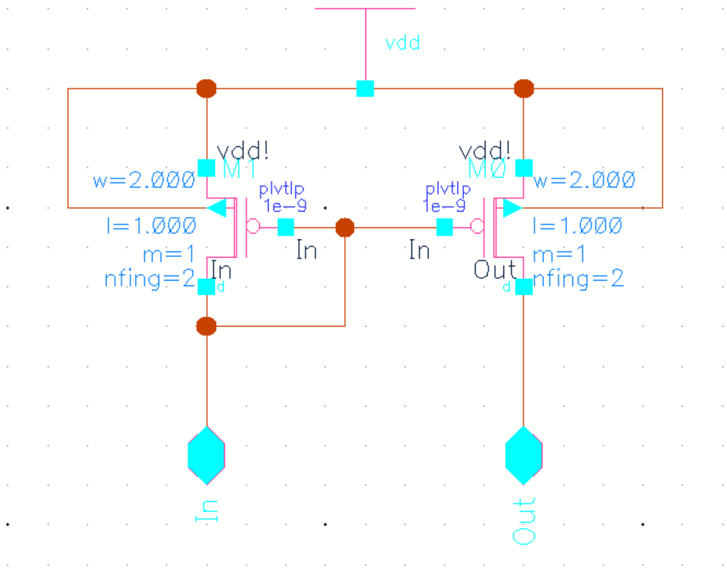


Figure 18. Complete Current Mirror Schematic

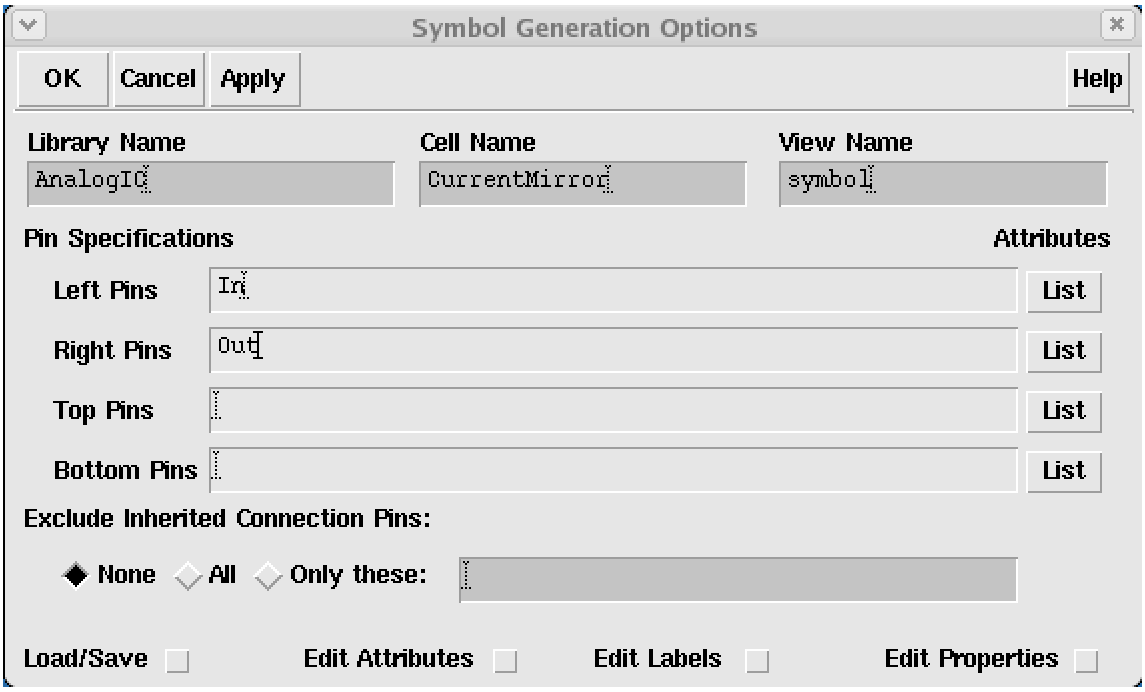


Figure 19. Symbol Generation Options Window

Save the symbol (Check and Save) and close the symbol window (“File –> Close”) and the Schematic window (“File –> Close”). Now a symbolic view of the current mirror has been created. This means that the current mirror can be instantiated in a schematic like any of the other components.

Next is to create at test bench for the current mirror. Create a new cellview in the analogIC library named CurrentMirror_tb. Add the current mirror instance, go to the menu “Create -> Instance…” (i), click on the Browse button. Library: analogIC, Cell: CurrentMirror and View:Symbol. Place the symbol in the CurrentMirror_tb schematic by clicking the left mouse button.

The additional instances needed (Vdd, gnd, Idc and Vdc) are found in analogLib. Start by building the power supply, see figure 20. Set the DC voltage of the DC source to 1.2V.

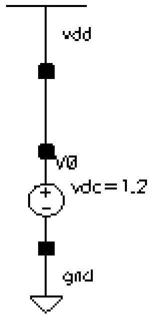


Figure 20. Schematic of the power supply

Connect the DC current source (idc) between gnd and the In pin of the current mirror cellview. name the dc current parameter Iref. Connect a DC voltage source (vdc) between gnd and the Out pin, name the DC voltage of the source to 1.2 + Vds (In order to get the signs right). The test bench should look like figure 21. When done press Check and Save

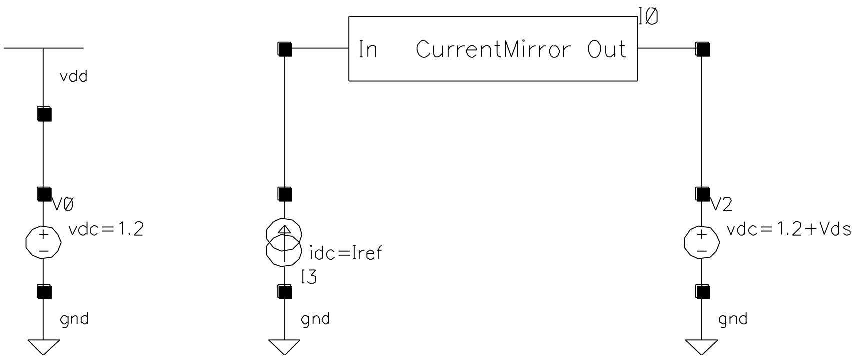


Figure 21. Test bench of the current mirror

Start the analog design environment, set up the corners (ArtistKit -> Setup Corners), import your design variables (Variable -> Copy From Cellview), double- click on the Design variables and set them to 0. Select the current of the output pin Out to be saved, go to Outputs -> To Be Saved -> Select On Schematic (click on the red terminal of the cellview and press Esc). Setup a DC simulation (Analyses -> Choose…), check the sweep variable -> Design variable box and Vds is the variable to be swept. Sweep from -1.2 to -0.1 with a linear step size of 0.1, the form should look like figure 22.

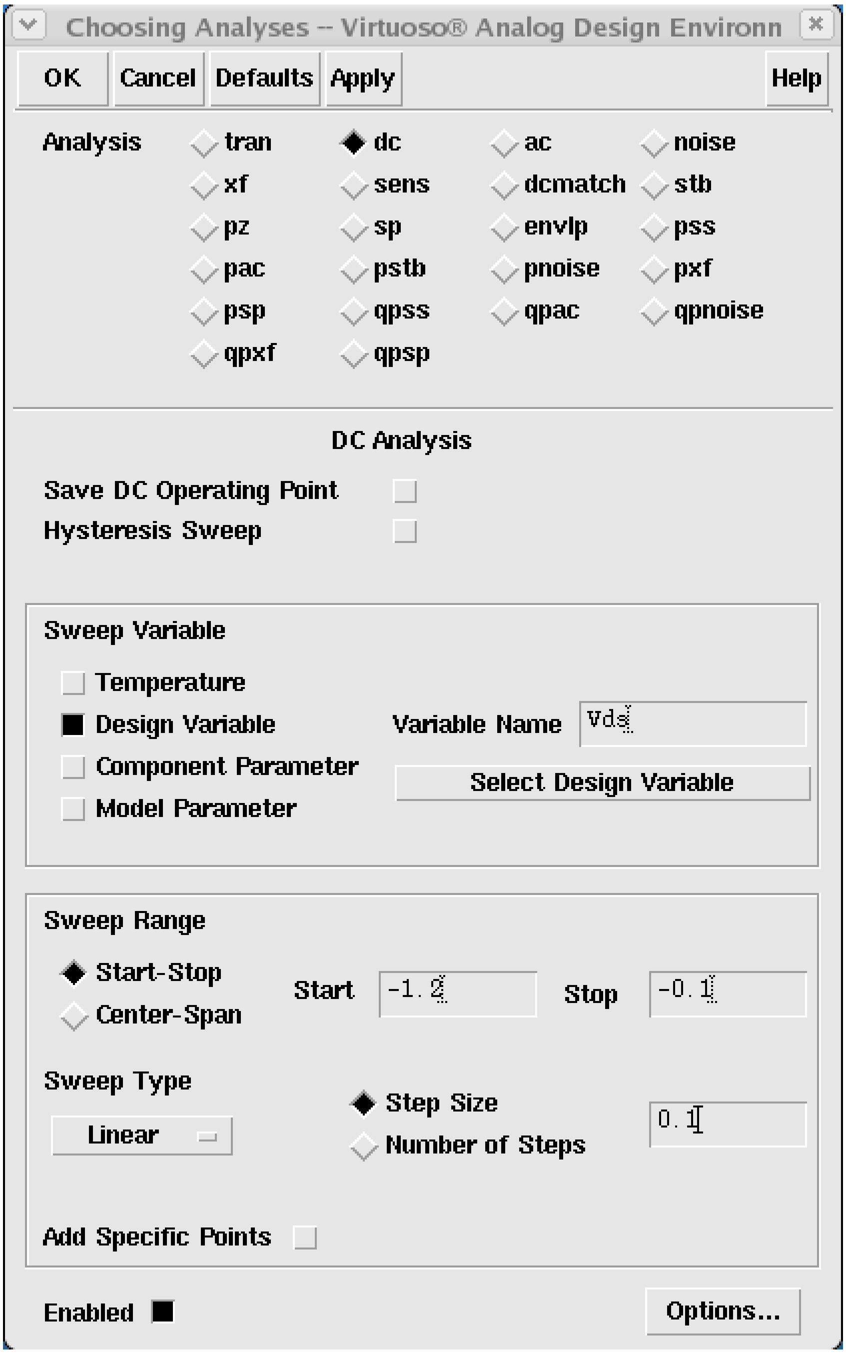


Figure 22. Setup for the DC simulation of the Current Mirror

Now setup a parametric sweep for the other variable (Iref) with a linear sweep from -200µ to-40µ with a step size of 40µ, see figure 23 for reference. Start the simulation; go to (Analyses -> Start All).

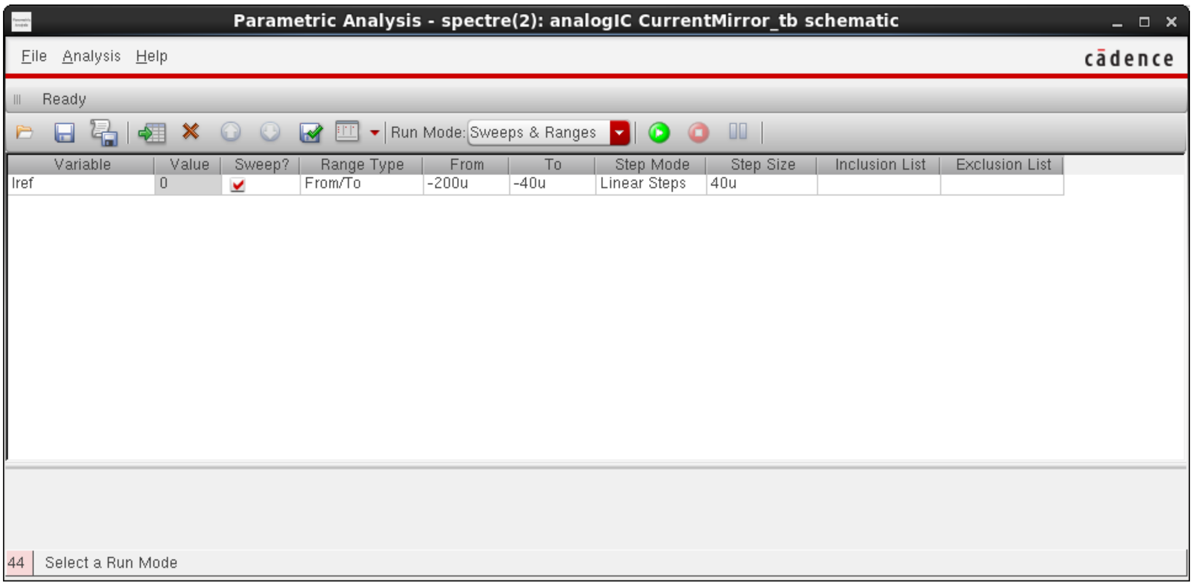


Figure 23. Setup for the Parametric Analysis

When the simulation is complete, start the calculator (Tools -> Calculator…). Plot the output current of the current mirror using the idc function in the calculator. In addition save the current as a CSV file as well. In order to calculate the output conductance you can derive the drain source current (Ids) with respect to drain source voltage (Vds). Use the derive function under special functions, see figure 24. Finally plot the results and print them to a CSV file. The derived result is the channel conductance which is the inverse of resistance, so use the calculator to invert the result and plot this result as well and print it to a CSV file. Compare your results with the home work and discuss your findings.

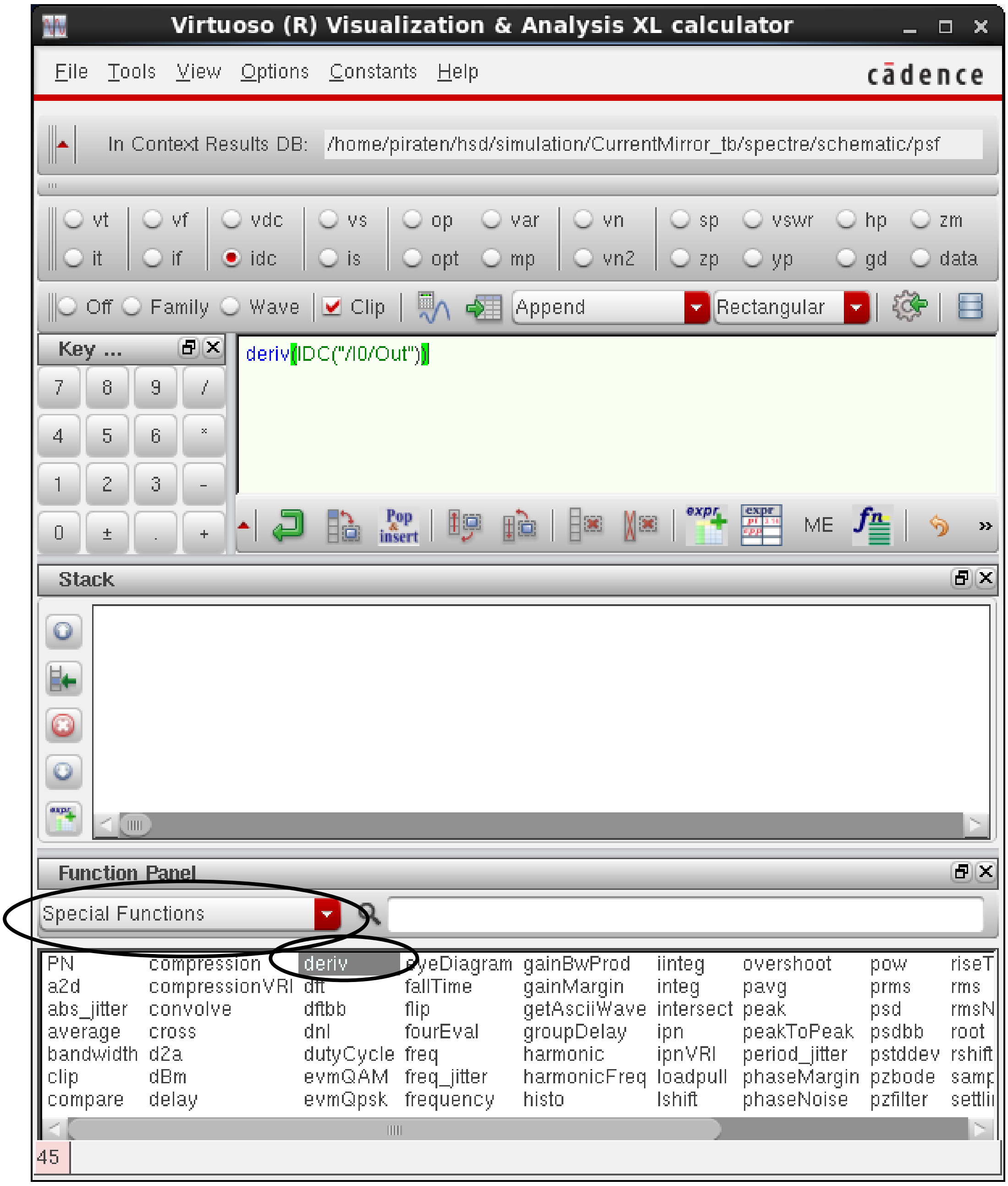


Figure 24. Functions need in the calculator to plot the output resistance of the current mirror

Layout of a PMOS current mirror in Cadence

In the final part of the lab you will finalize a layout of the same Current Mirror you simulated in the previous section. Create new cell view (“File -> New -> Cell View” in the Library Manager). The name should be the same as for the schematic, i.e. CurrentMirror, the view and type should be layout, open with Layout L. An empty Layout Editing Window will pop up, see figure 25.

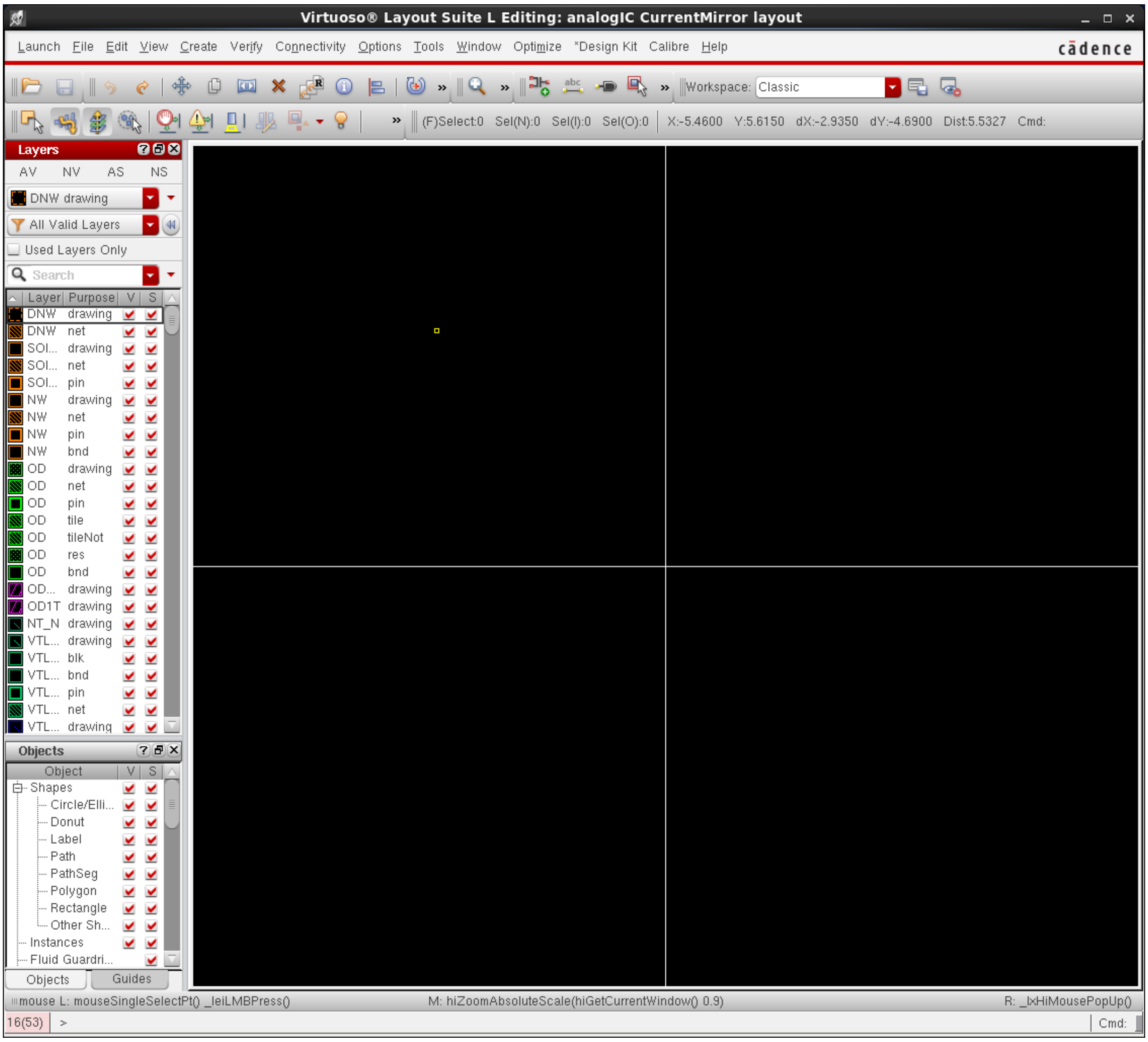


Figure 25. Layout editing window

Start by instantiating the PMOS transistors, “Create -> Instance”. If you have a transistor marked in the schematic you will get the plvtlp device with the parameters according to the schematic. Otherwise browse for the plvtlp and fill out the numbers manually. Instantiate two transistors, rotate them 90 degrees, either using the rotate button in the Create Instance window, or use “Edit -> Move” (m), left click the instance to move, and then the middle mouse button to rotate, and the left click at the new position. The two rotated transistors can be seen in the layout window in figure 26.

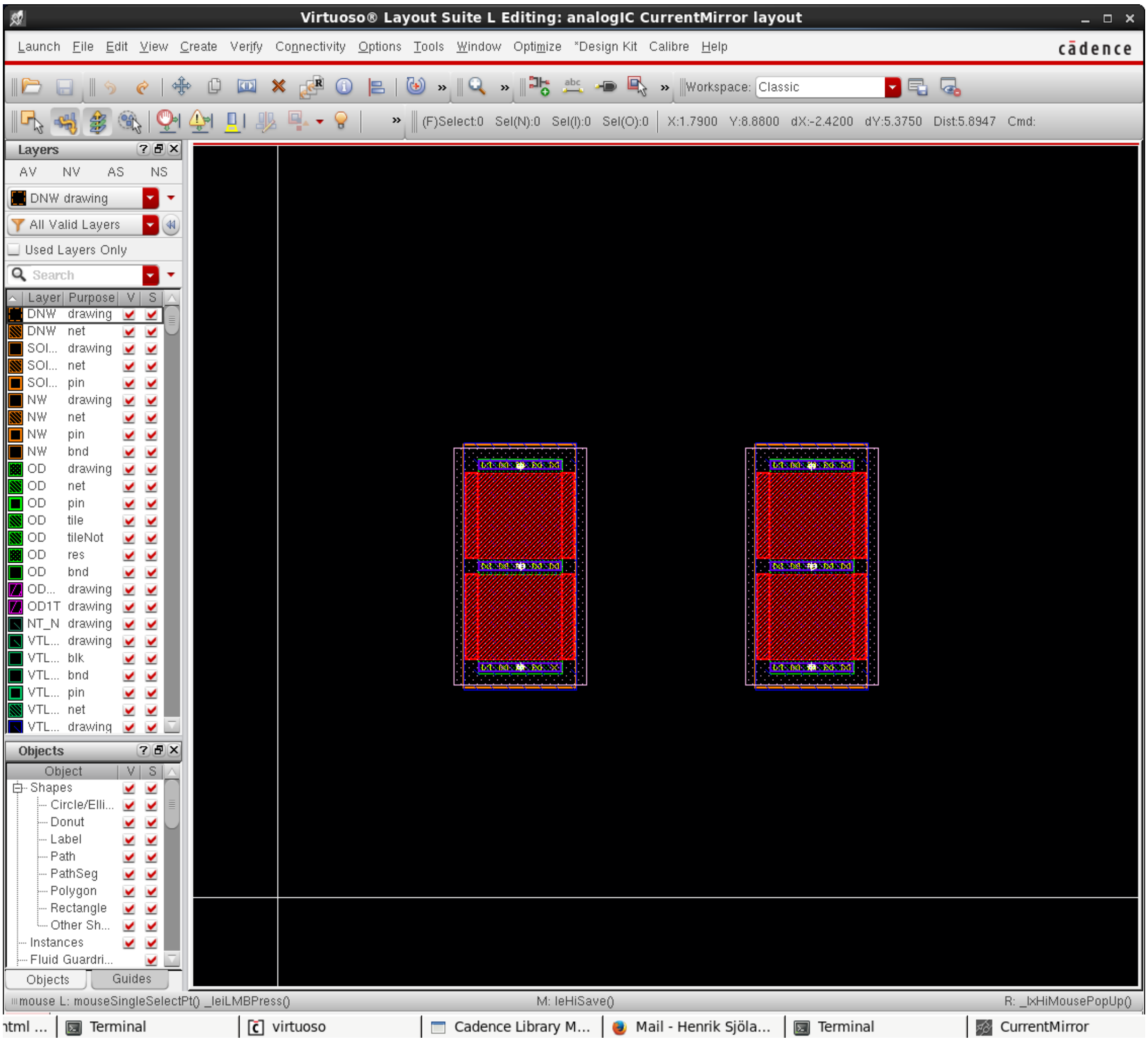


Figure 26. Layout editing window with two PMOS devices

In the layout editing window, to zoom in you just press z on the keyboard and click and drag over the area which you want to zoom into. In order to zoom out you just press shift+z and to fit the layout to the screen/window simply press f on your keyboard.

We start by looking at the left side of the layout editing window, where you find the Layers Sub-Window. The LSW is used to select each layer in the process stack i.e. the different layers provided by the foundry; the layers are both metal and different layers to build the actual semiconductors. Start by marking the OD drawing layer in the LSW, then click on the NV icon button in the LSW. By pressing the NV (None Visible) only the Diff layer is visible. To get all layers visible again, push the AV (All Visible) button in the LSW. This works for all the different layers. One can also make a few layers visible by clicking them in the LSW.

Next is to learn how the ruler works, the ruler is a frequently used tool during layout. Now we should measure the width and length of the transistor channel. Start by making the OD drawing and PO drawing layers only visible. Zoom in on one transistor. “Tools -> Create Ruler” (k) brings up the ruler tool. Click the pointer in one of the corners of the rectangle you want to measure, then just drag the pointer to the corner on the opposite side of the rectangle and click. Where do you measure the transistor channel length, and the width? Use the ruler to measure both. Finally remove the rulers by "Tools -> Clear All Rulers" (shift+k).

First we should connect the gate fingers of the 2 transistors; we start by adding a contact from the poly-silicon to the metal 1 layer by creating a contact. To generate a contact, go the menu "Create -> Via" (o). Choose 12 Rows in the Create Via form, Via Definition M1_PO, show enclosures and set all enclosures to 0.04 (figure 27). Place the contact on the edges of the poly-silicon as displayed in figure 28. Save the layout!

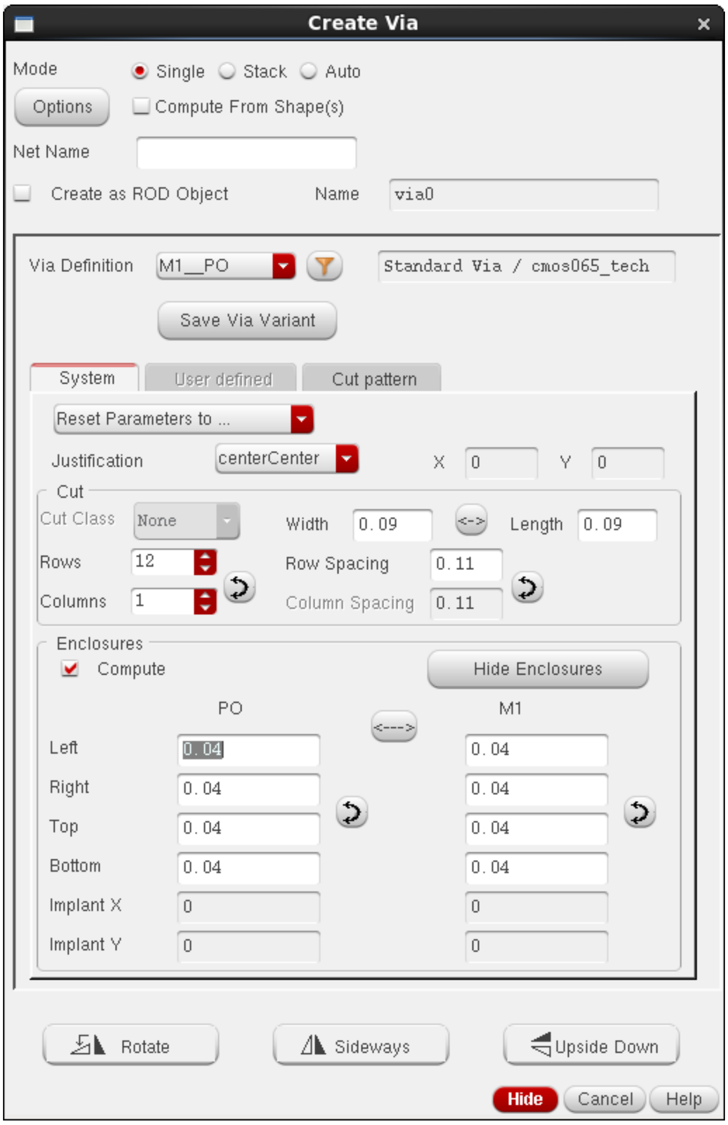


Figure 27. The Create Via Form

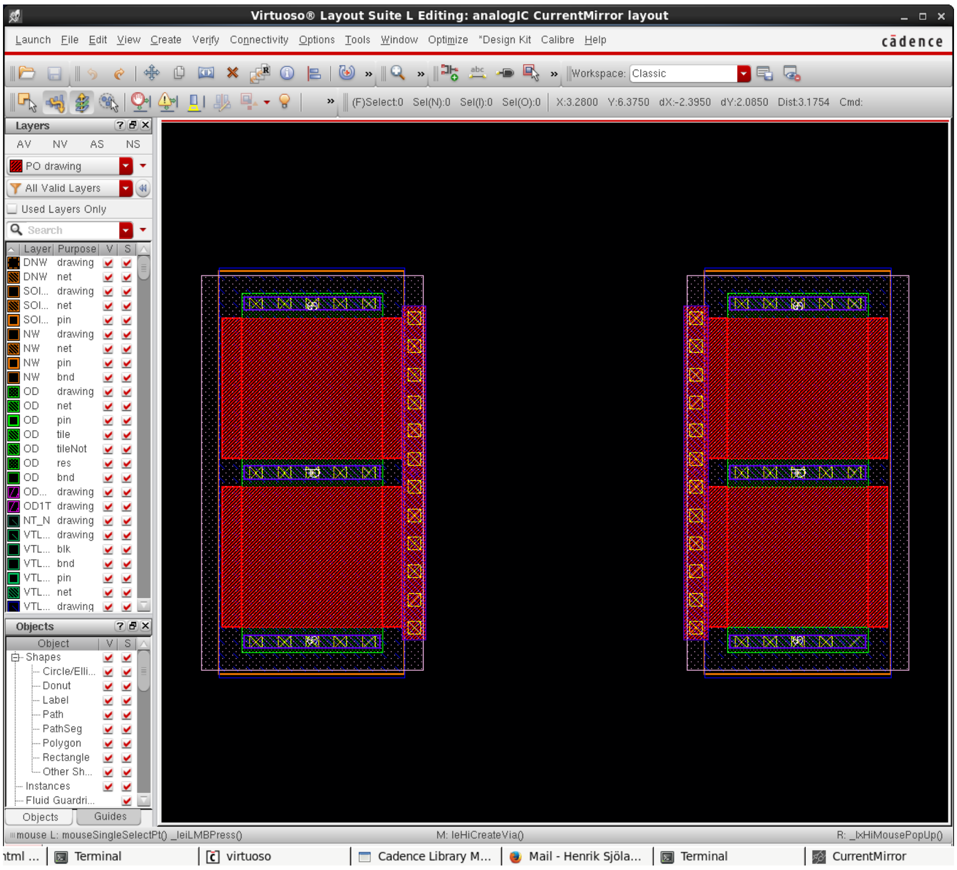


Figure 28. Placement of the gate contacts

Now we have to add contacts to the source areas, the instantiated transistors (p-cells) already have contacts going from the diffusion (OD) to metal 1. In order to go above metal 1(M1), we have to go up one metal layer i.e. to metal 2(M2). Go to the menu “Create -> Via...” (o), in the form select M2X_M1 as Via Definition. Choose all enclosures to 0.04. Finally choose 5 Columns and place the contact as centered as possible on the upper left source region. Now “Edit -> Copy” (c) the contacts to all the other source terminals. The final figure should look like figure 29.

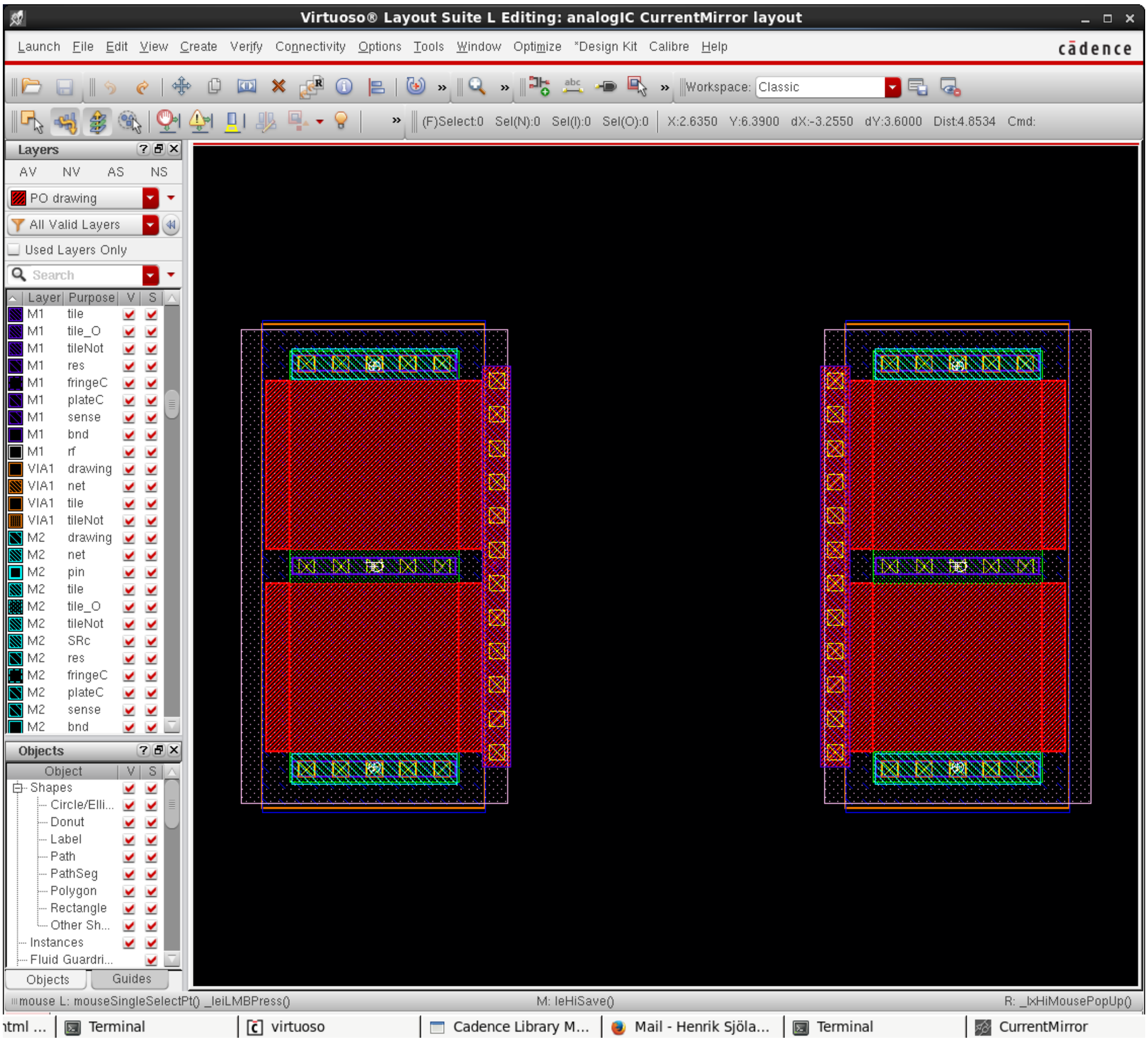


Figure 29. Placement of the source contacts

We will now add Nwell contacts and add the Nwell around the PMOS transistors. Go to the menu "Create -> Via…" (o), then select M1_NW as Via Definition. Use 0.04 enclosures according to Figure 30. Choose 20 Columns and place the contact above the 2 transistors and another one underneath.

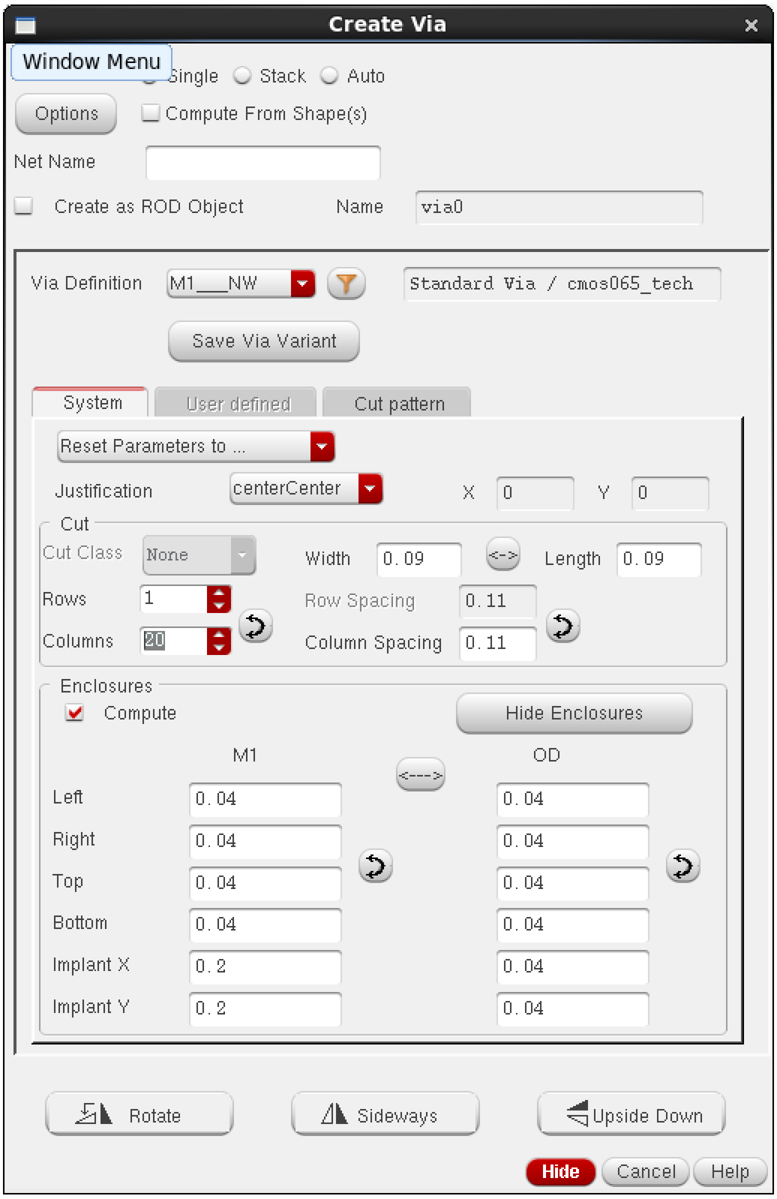


Figure 30. Nwell contact form

Select NW drawing in the LSW, and then draw a rectangle around both the transistors and the Nwell contacts. To draw a rectangle, go to the menu “Create -> rectangle…” (r). Furthermore add two metal 1 to metal 2 contacts on top of the Nwell contacts, the layout should look something like figure 31. Save the layout!

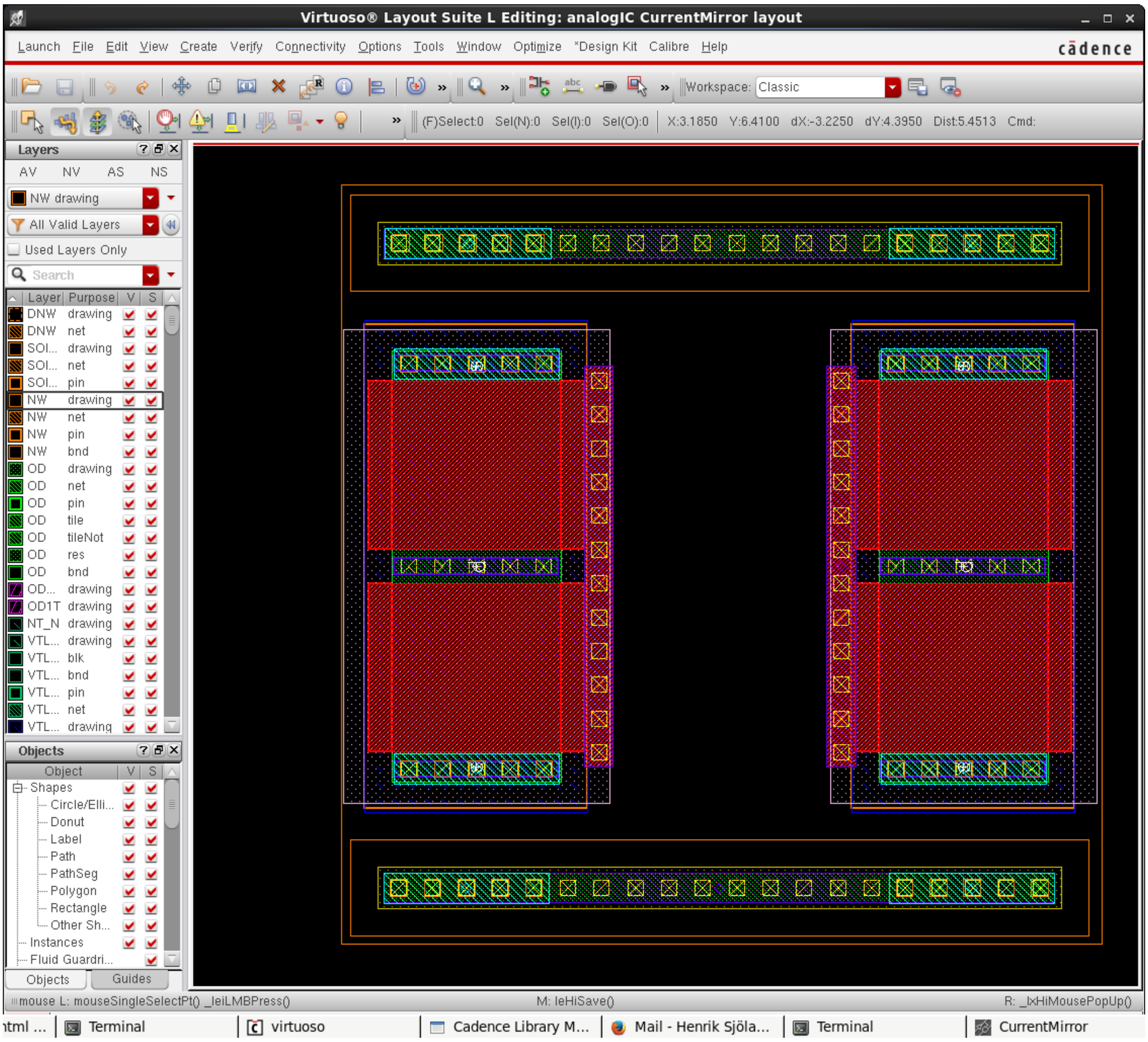


Figure 31. Placement of the bulk contacts and NWell

Now we have done some work on the layout, then it is time to run the Design Rule Check (DRC). The DRC does just what it says; it checks if all the rules for the process are fulfilled. These rules are of different kinds, for examples it is checked if two metal wires are too close, or if the wires are too narrow. To find the DRC tool, go the menu “Calibre -> Run DRC…”, choose *run DK DRC*, press OK and then Yes in dialogs popping up. A window Calibre Interactive appears. Press the Run DRC button on its left side. When DRC is finished a new window with DRC results comes up. Click Show Unresolved to see the list of errors. You can move back and forth between the errors using the arrows in the top tool bar of the window. Errors to look out for are notches in the metal (where the width abruptly changes just a little bit), minimum distance of poly contacts to the transistor, and minimum PP overlap of Poly. Fix the errors. Do not forget to save the layout! Then run the DRC again to check that there are no errors left. When the DRC is clean you can continue.

Now it is time to connect all the individual terminals together. First we will connect the gates together. Start by selecting M1 drawing in the LSW, now we will add a metal1 path between the metal1 to poly contacts of the two transistors. Go the menu "Create -> Shape -> Path…" (p), click on one of the middle contacts connected to the gate of the left transistor and drag the path over to the right transistor gate contact and hit enter on the keyboard. Press Esc. Now click on the path and press q for properties, change the width of the path to 1µm and click OK. Now connect the drain of the left transistor to the gates by using the path (p) in metal 1 (M1). The left transistor should look something like figure 33.

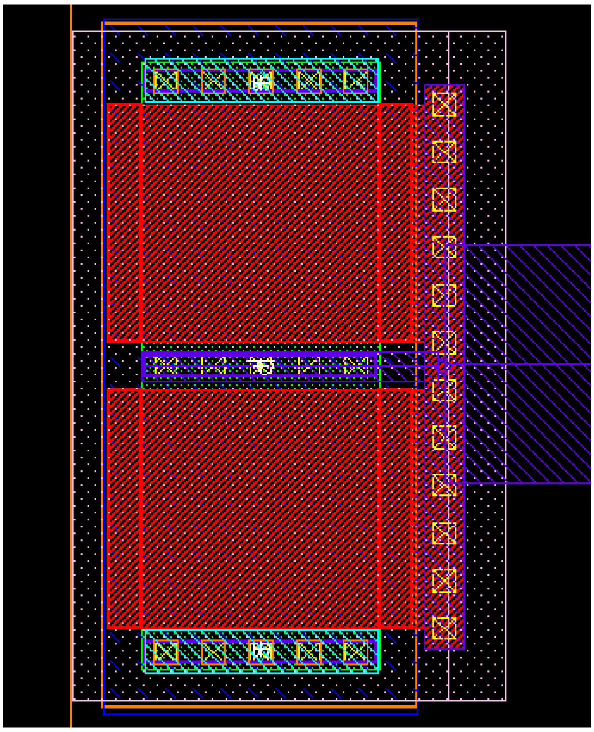


Figure 33. The Diode-Connected Transistor

Now draw a metal2 (M2) rectangle above the transistors and nwell contact. This will by the supply voltage terminal (Vdd). Draw a meta2 path from the bottom nwell contact (over the left transistor) up to the metal 2 rectangle. Draw a similar path also over the right transistor. Your layout should now look something like figure 34. Adjust the width of the paths so they can handle a current of 0.6mA (use the data sheet of the process) by selecting the path and pressing q on the keyboard.

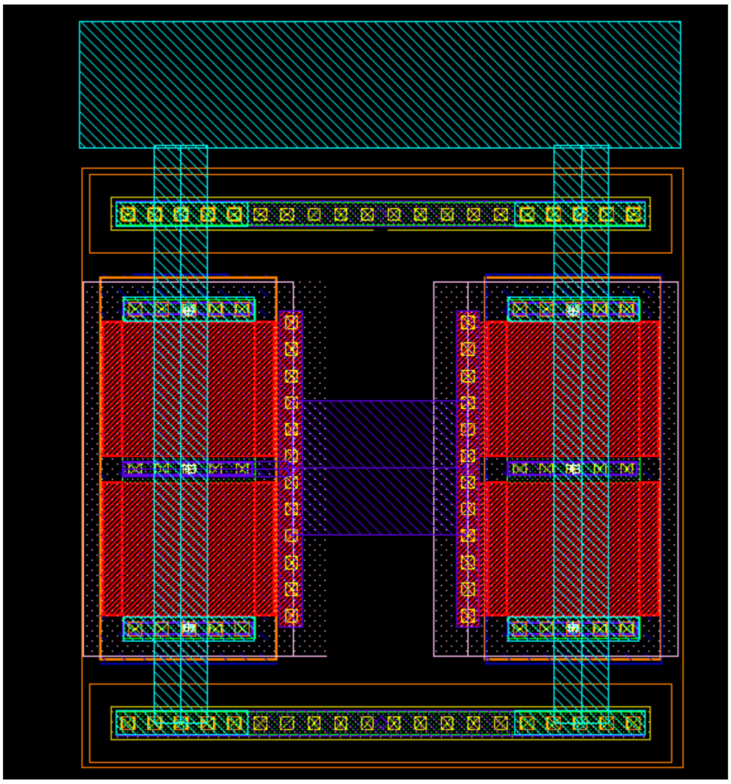


Figure 34. Sources and bulks connected to the VDD plane

Now let’s draw a path from the drain of the left transistor, we start by drawing a path (p) in metal1 from right to left extending a length of about 1 µm outside the left transistor. Then draw a path of equal length from the drain of the right transistor, after this step the layout should look something like figure 35. Save your layout and run DRC.

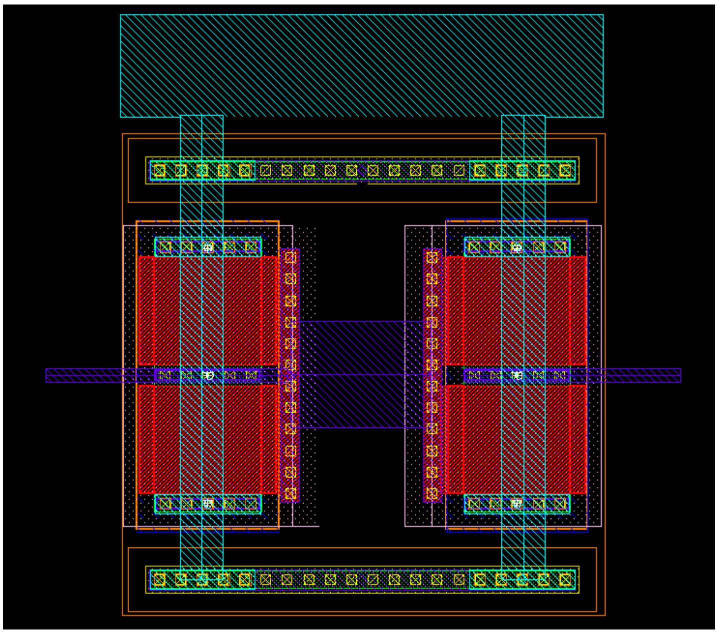


Figure 35. Drains Connected to a Metal 1 path

Now we should define all nodes in the layout, by placing labels. Select M2 pin in the LSW, go the menu “Create -> label…” (l). A form like figure 36 should appear. Start by naming the first label vdd! (! makes the net global), place the label on the metal 2 rectangle.

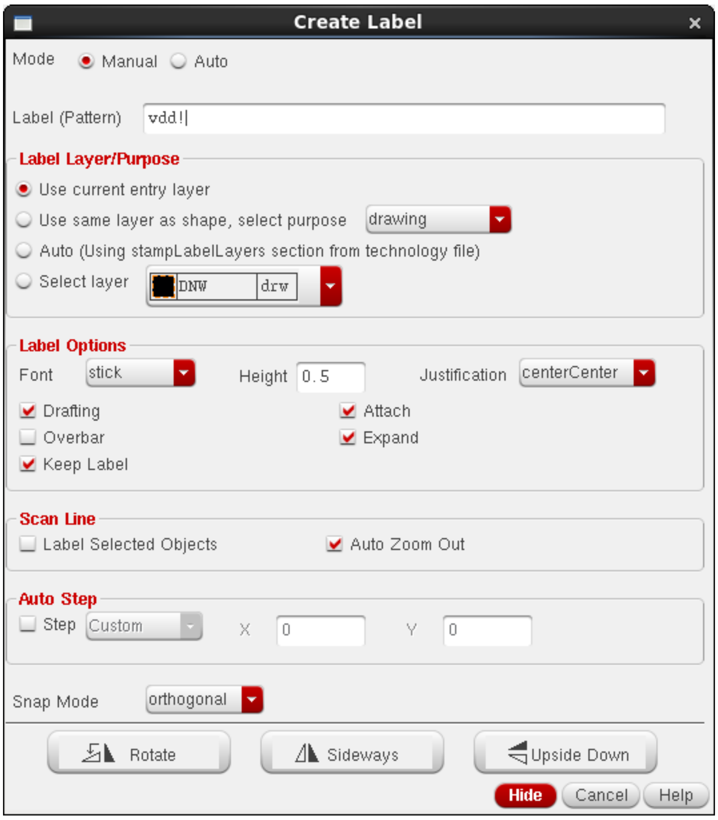


Figure 36. Create Label Form

Now add the remaining Labels (In, Out), the only thing that should be changed is that M1 pin should be selected in the LSW. The final layout should look something like figure 37. Save your layout and run the DRC.

When the layout of the design is DRC clean we should continue with an additional check which is too verify that the layout and the schematic of the current mirror are the same. For this test we run what is called Layout Versus Schematic (LVS). Go the menu “Calibre -> Run LVS”. Use default settings in the dialogs that appear, except that “Skip ERC1 to ERC12 rules check (Wells biasing) (sign off)” should be checked. A window Calibre Interactive appears, get acquainted with that. Enter the “Setup -> LVS Options” menu, then under the Supply tab select “Ignore layout and source pins during comparison”. This avoids false errors due to the vdd! global supply terminal. Then press Run LVS.

When the LVS has completed a display window emerges, if there are some errors scroll down the log and try to solve the errors. Check if all contacts are present and verify that there are no short circuits in your layout.

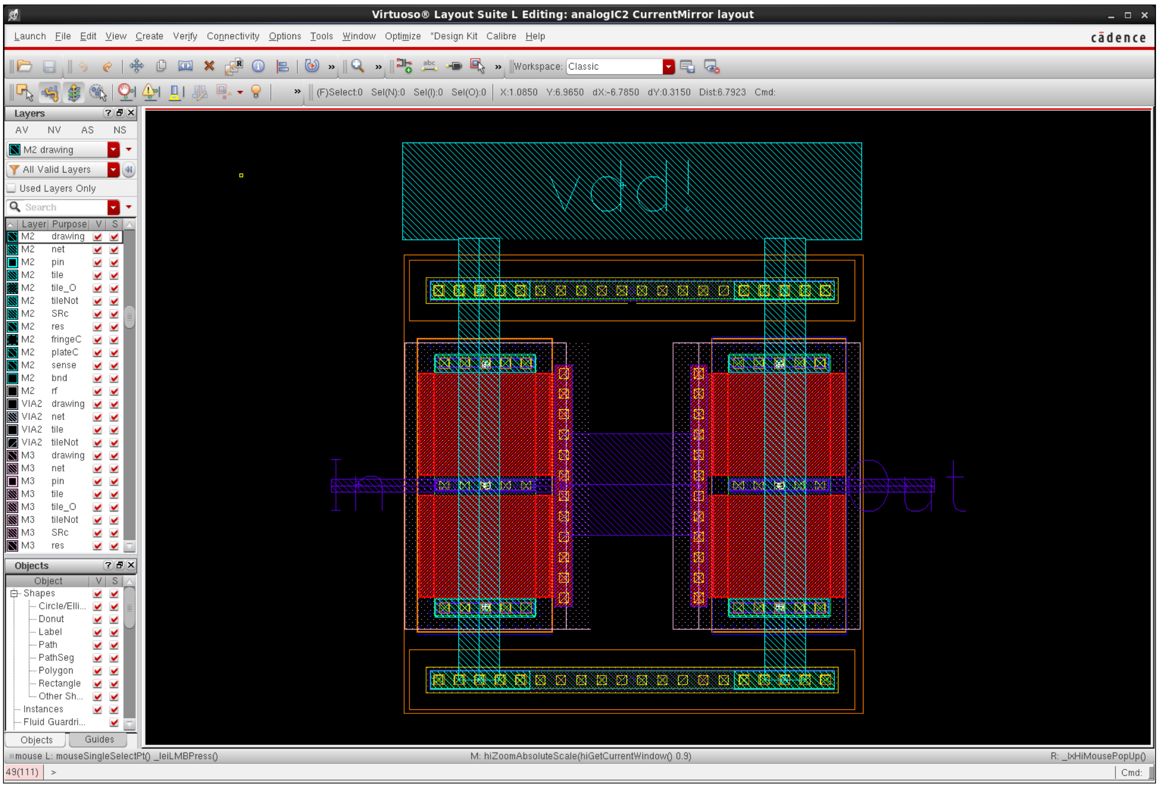


Figure 37. Final Layout

Laboratory Report

Compose a report containing the difference of calculated and simulated values. The report should also answer the questions in the homework section, and explain the results from the laboratory. Include plots of the simulated figures.