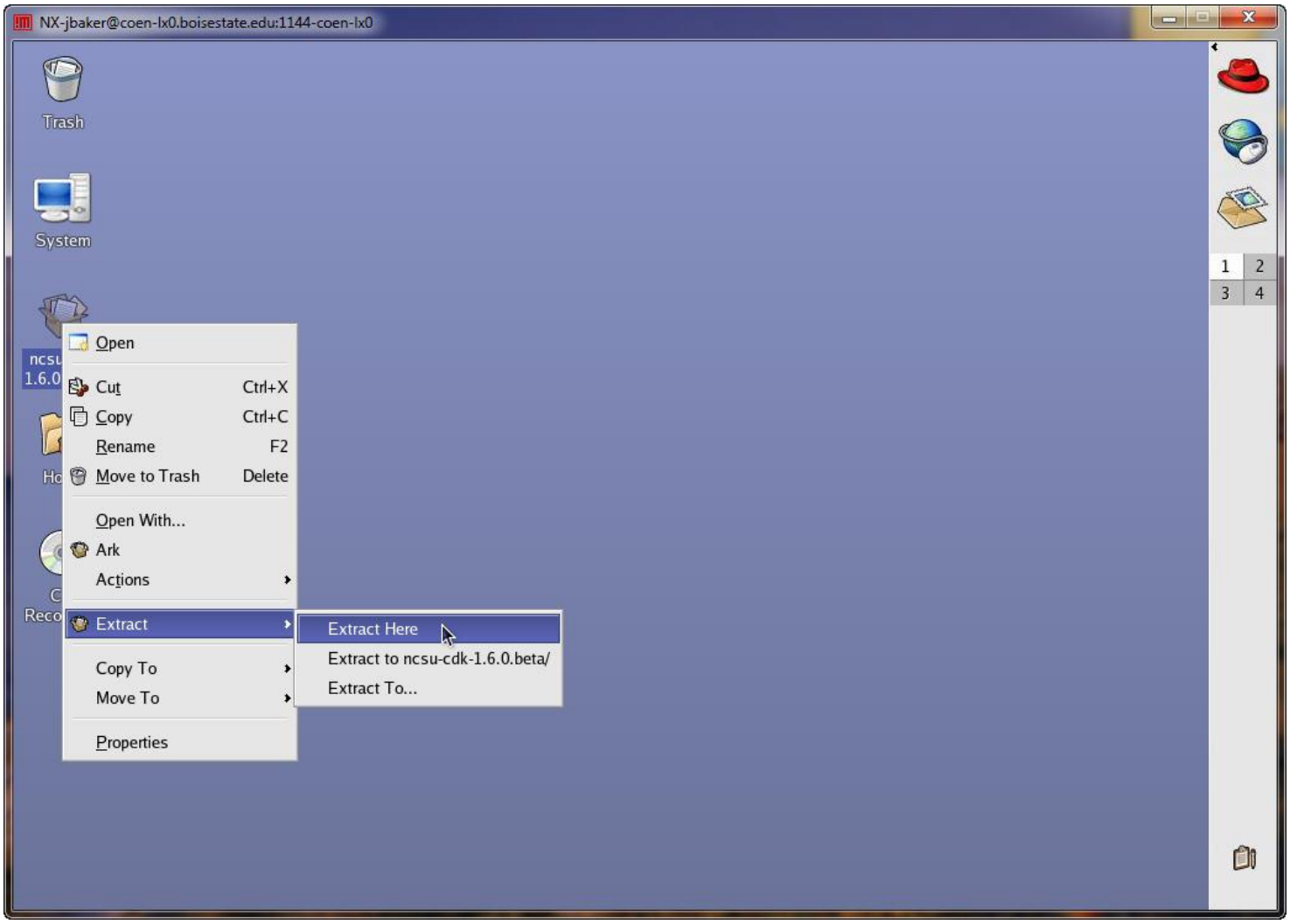
Cadence Design System Tutorials from CMOSedu.com (Return)

Tutorial 1 – Layout and simulation of a resistive voltage divider

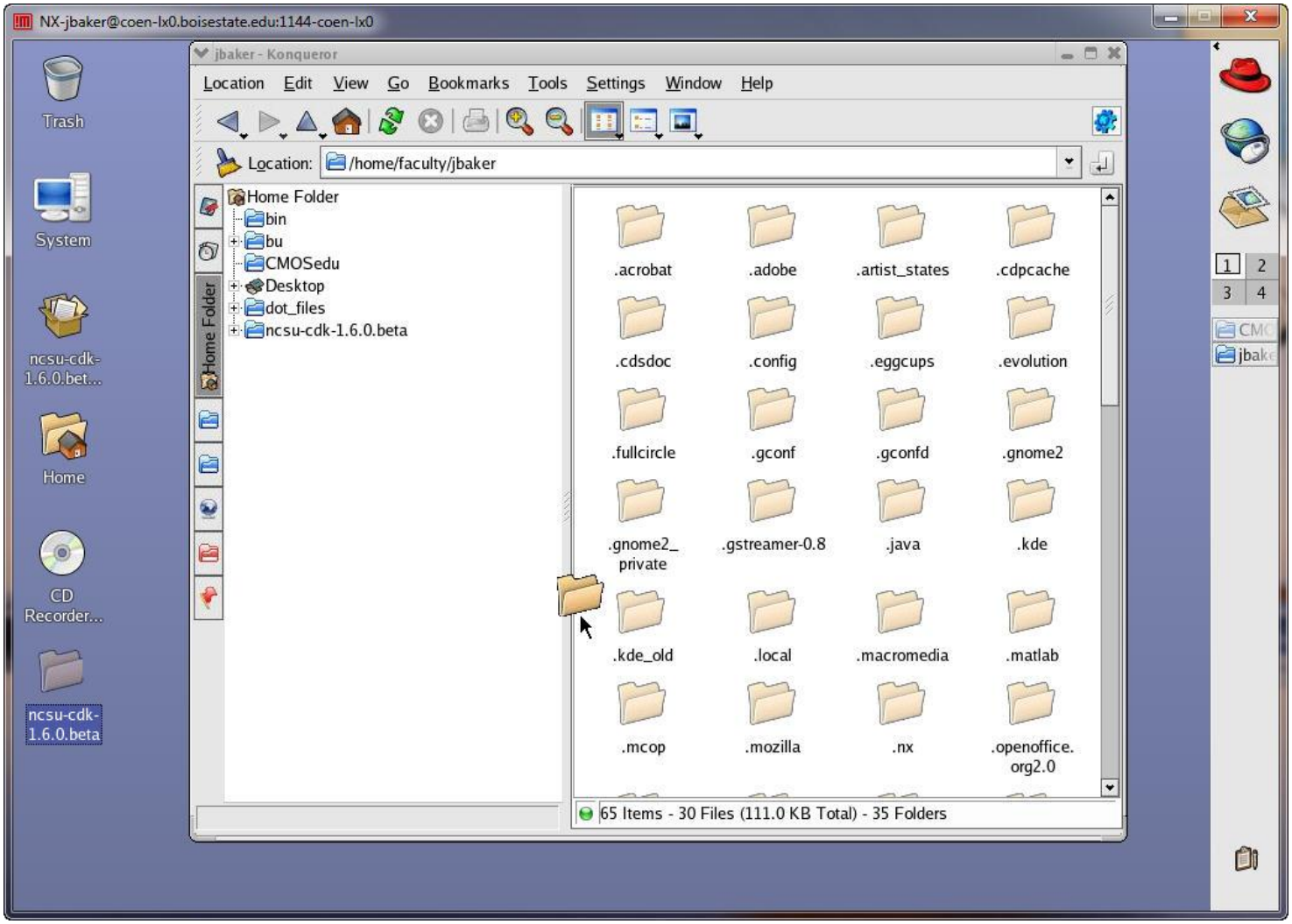
This tutorial will introduce you to Cadence 6.1 for chip design, layout, and simulation. To demonstrate the operation of Cadence we'll set it up for
use with ON's C5 process (formerly AMI's C5 process) and fabrication through MOSIS.
This tutorial uses the MOSIS scalable CMOS (SCMOS) submicron design rules.

This tutorial uses a remote desktop to connect to the server.
Here are links for help installing a remote desktop and Xterm (written and video).
The operation of Cadence isn't dependent on which method is used to log into the server.

To begin, download the NCSU Cadence Design Kit (CDK) version 1.6.0 beta from http://www.eda.ncsu.edu/wiki/NCSU_CDK to your desktop and extract
the download's contents by right+clicking on the icon as seen below.



Next move the extracted folder (**not** the downloaded tar.gz archive), ncsu-cdk-1.6.0.beta, to your home folder (directory), $HOME (my home directory
is /home/faculty/jbaker, this path and $HOME are equivalent), see below.



In your home directory open the **.bashrc** file (ensure that you can view hidden files and assuming that you are using the Bash shell) and add the following lines

```
export SPECTRE_DEFAULTS=-E
export CDS_Netlisting_Mode=Analog
export CDS_LOAD_ENV=CWDElseHome
export CDK_DIR=$HOME/ncsu-cdk-1.6.0.beta
```

When finished, in a terminal window in your home directory, type the command ". .bashrc" (period, space, period bashrc) to re-source the .bashrc file.
Note that if Cadence IC5.141 isn't installed on your system then you likely won't need to add the last line seen above to your .bashrc.

Now, make a working directory in your home account called **CMOSedu**

Copy everything in the directory **$HOME/ncsu-cdk-1.6.0.beta/cdssetup** into **$HOME/CMOSedu**

In the working directory **CMOSedu**, rename **cdsinit**, **simrc**, and **cdsenv** to **.cdsinit**, **.simrc**, and **.cdsenv** (add a period)

In CMOSedu open the file **cds.lib** and add the following lines to point to the built-in Cadence libraries. Note that the path seen below may be different for
your installation of Cadence so verify it's correct before adding the lines.

```
DEFINE analogLib /usr/cadence/IC615/tools.lnx86/dfII/etc/cdslib/artist/analogLib
DEFINE functional /usr/cadence/IC615/tools.lnx86/dfII/etc/cdslib/artist/functional
DEFINE sbaLib /usr/cadence/IC615/tools.lnx86/dfII/etc/cdslib/artist/sbaLib
```

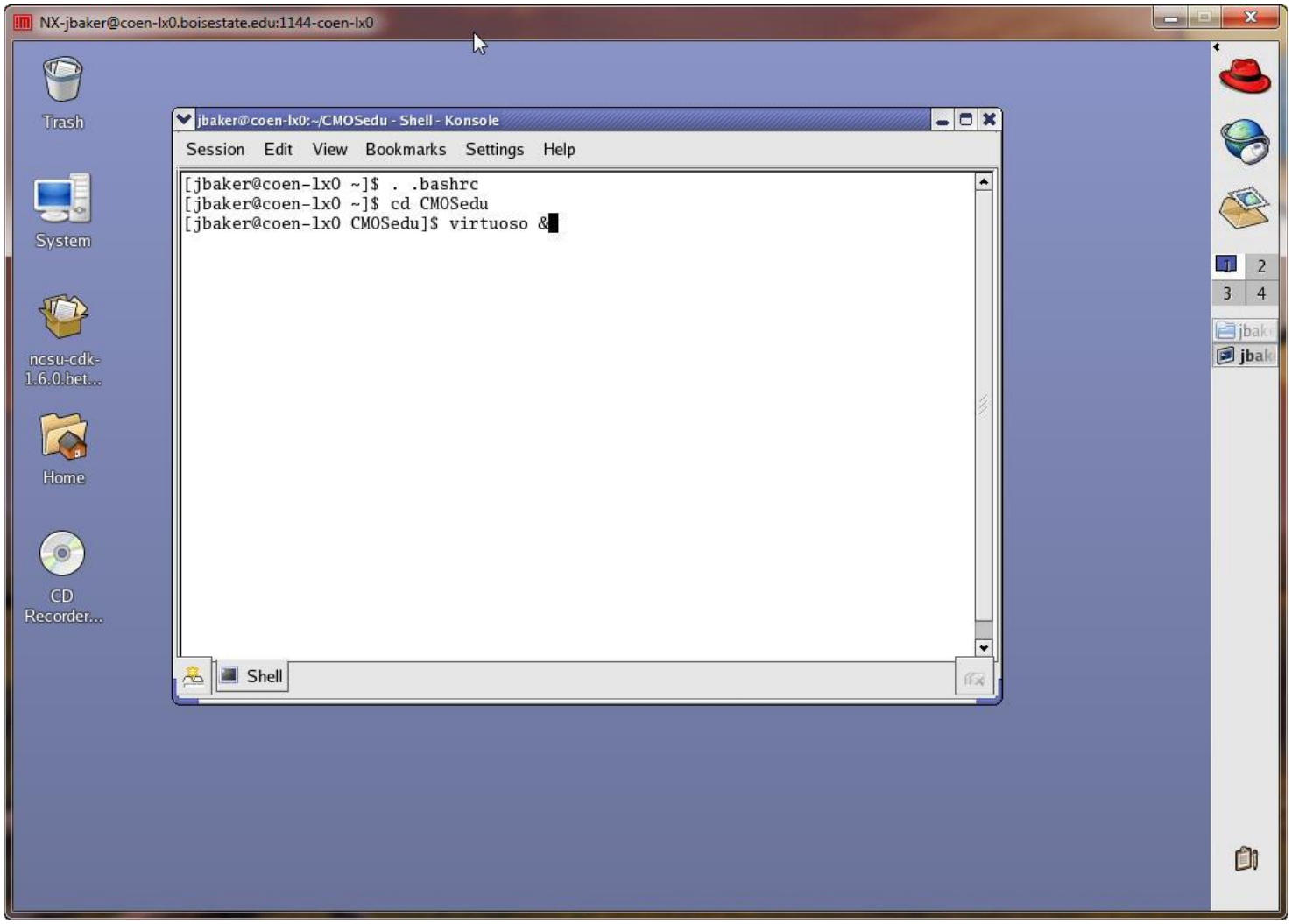
We will use Spectre (Cadence's spice) for the simulations in these tutorials so add

envSetVal("asimenv.startup" "simulator" 'string "spectre") to the bottom of your .cdsinit file in your home directory (later after you've started and exited Cadence).
This makes Spectre the default simulator.
Additional helpful hints for modifying the .cdsinit are found here.

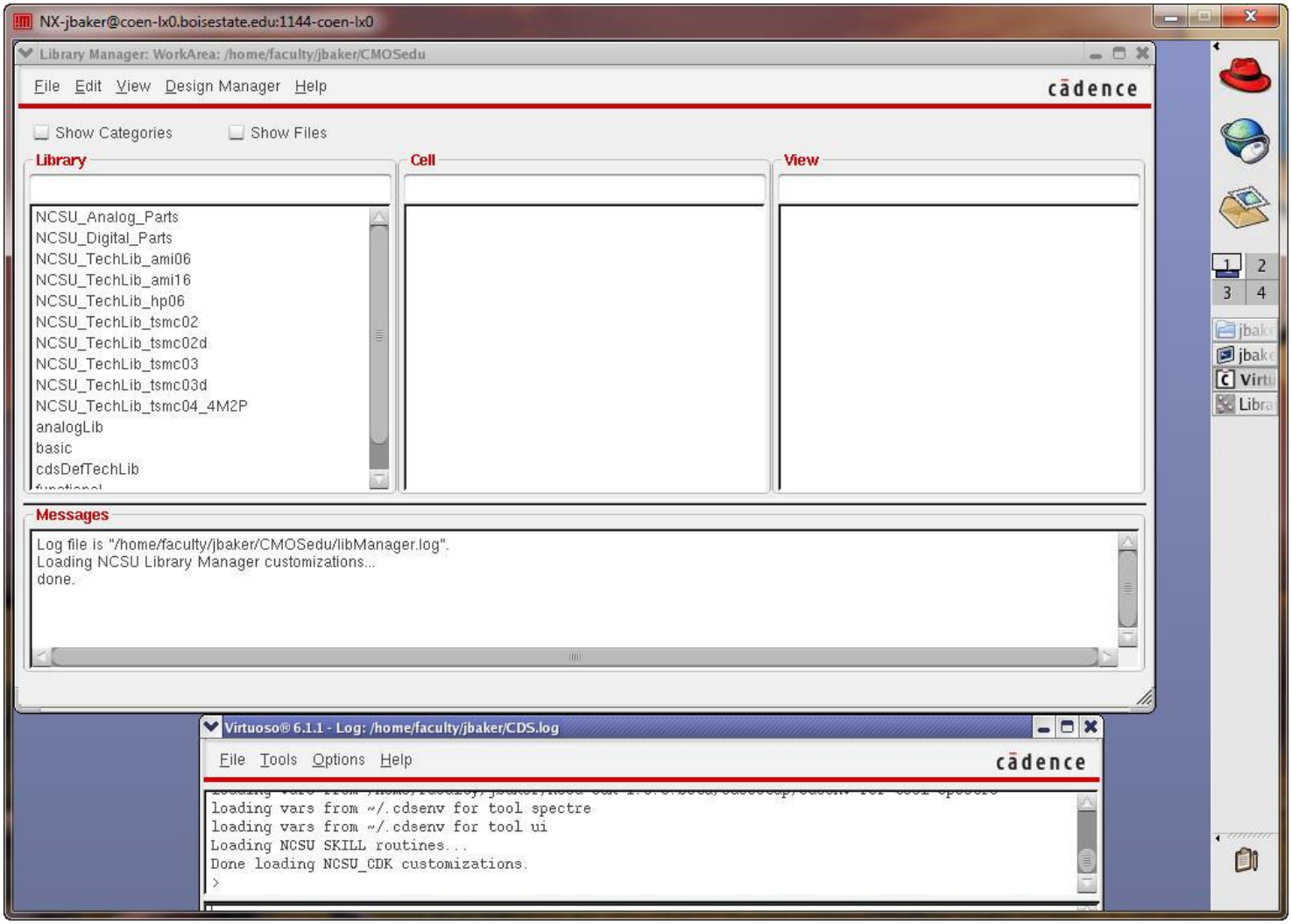
Finally, in the directory **$HOME/ncsu-cdk-1.6.0.beta/lib/NCSU_TechLib_ami06** delete the files **divaDRC.rul**, **divaEXT.rul**, and **divaLVS.rul**.
Save diva_rul_files.zip to your desktop.
Extract the files in this zip to your desktop.
Move the extracted files (**divaDRC.rul**, **divaEXT.rul**, and **divaLVS.rul**) into **$HOME/ncsu-cdk-1.6.0.beta/lib/NCSU_TechLib_ami06** in the deleted files' places.
The deleted files are locked and point to the wrong place (a bug in the beta version that will be fixed in the final release).
Note that these steps are important if you want to DRC, Extract, and LVS your layouts!

We are now ready to start Cadence and design a chip using ON's C5 process and the MOSIS scalable CMOS design rules (lambda of 300 nm and a technology
code of SCMOS_SUBM, see here for help on submitting a chip to MOSIS).

Open a terminal window and change directories to CMOSedu (the Unix command is **cd CMOSedu**). To start Cadence's Virtuoso editing tool type, in the
same terminal window, **virtuoso &** (adding **&** runs the process in the background allowing you to continue using the open terminal window) as seen below.

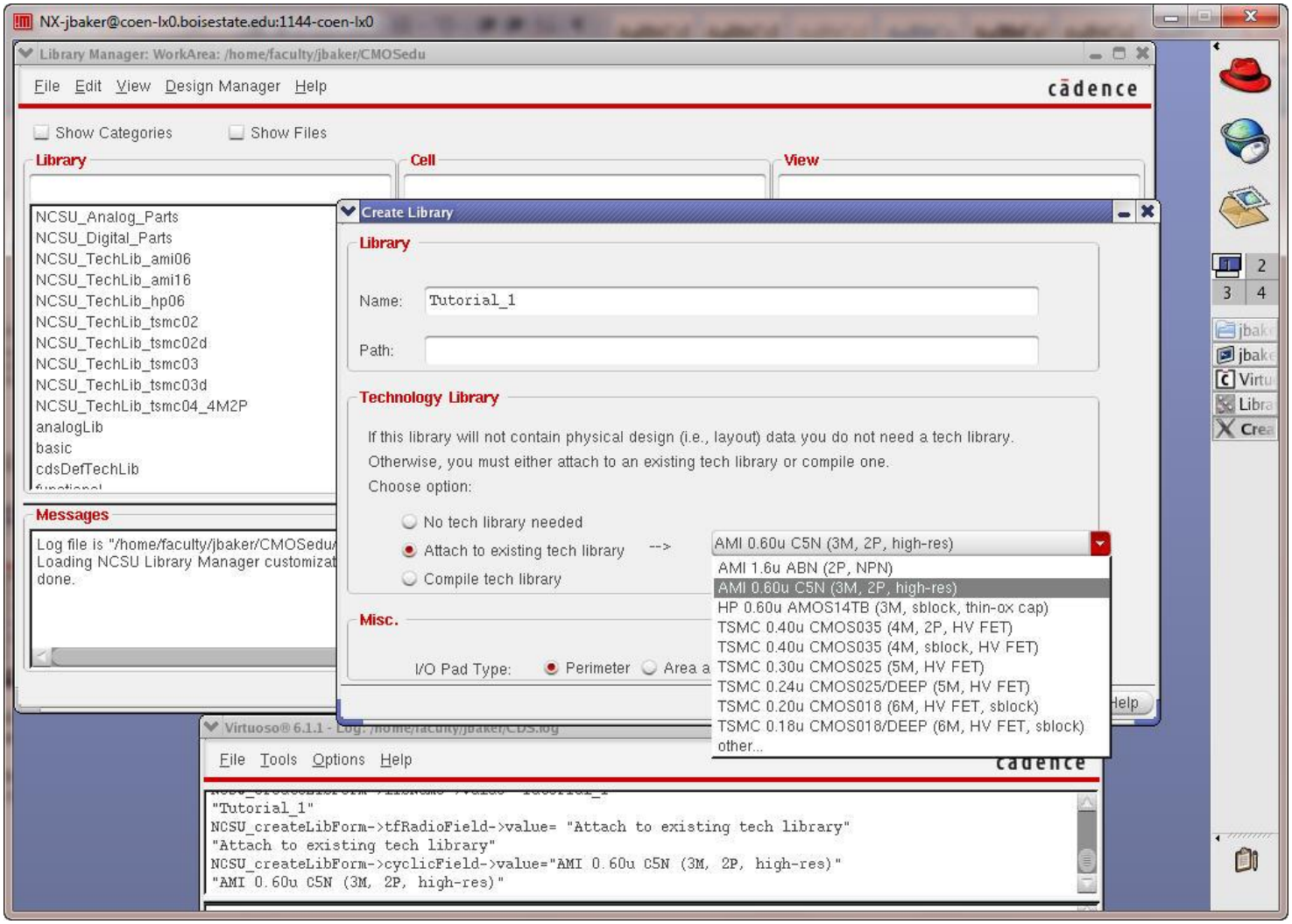


After starting Virtuoso, and re-sizing windows, the following should appear

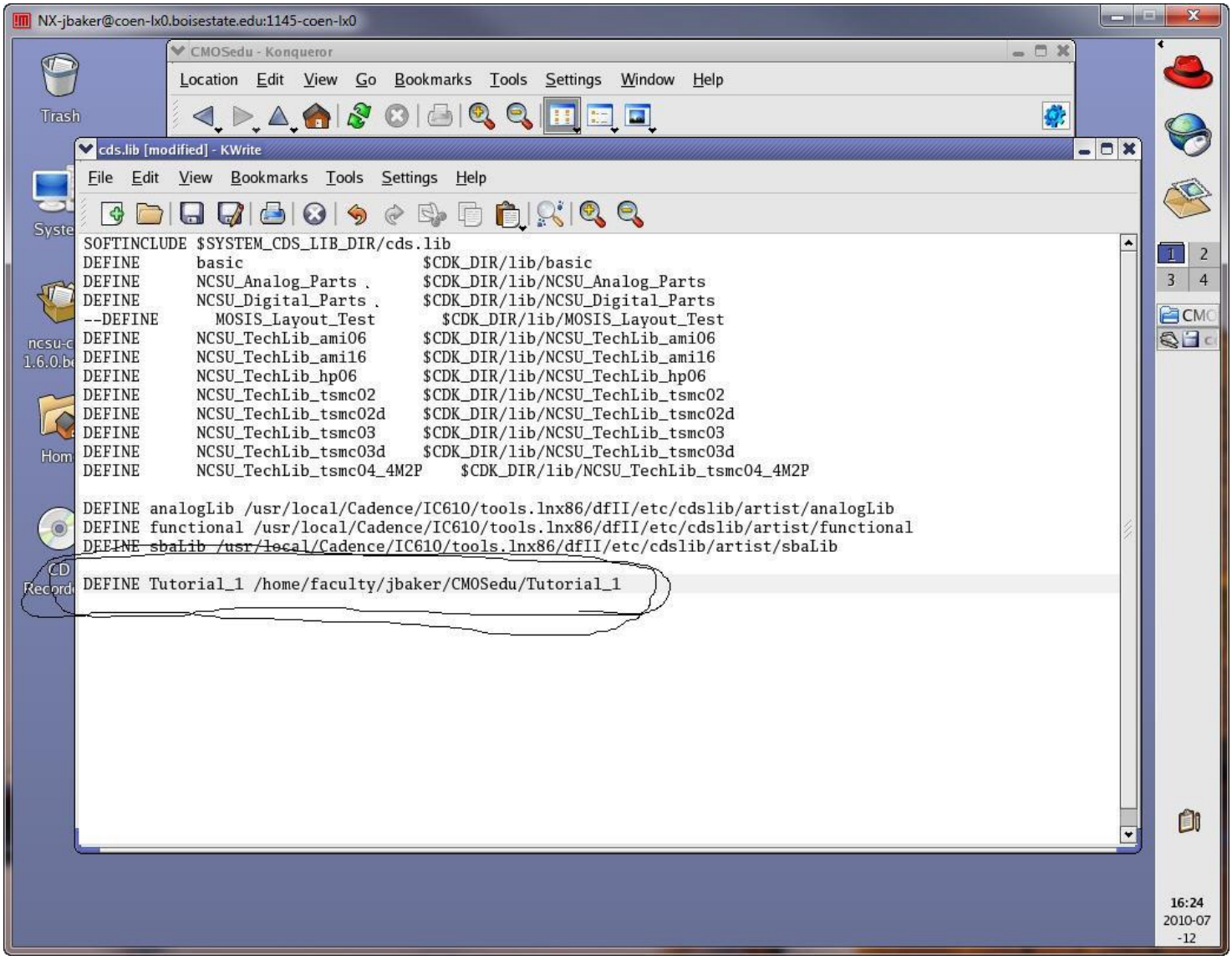


The bottom window is called the Command Interpreter Window or CIW. We need to keep the CIW visible since it tells us what the tools are doing. The
other window is the Library Manager. If this window isn't open, or you close it, it can be opened in the CIW using Tools -> Library Manager.

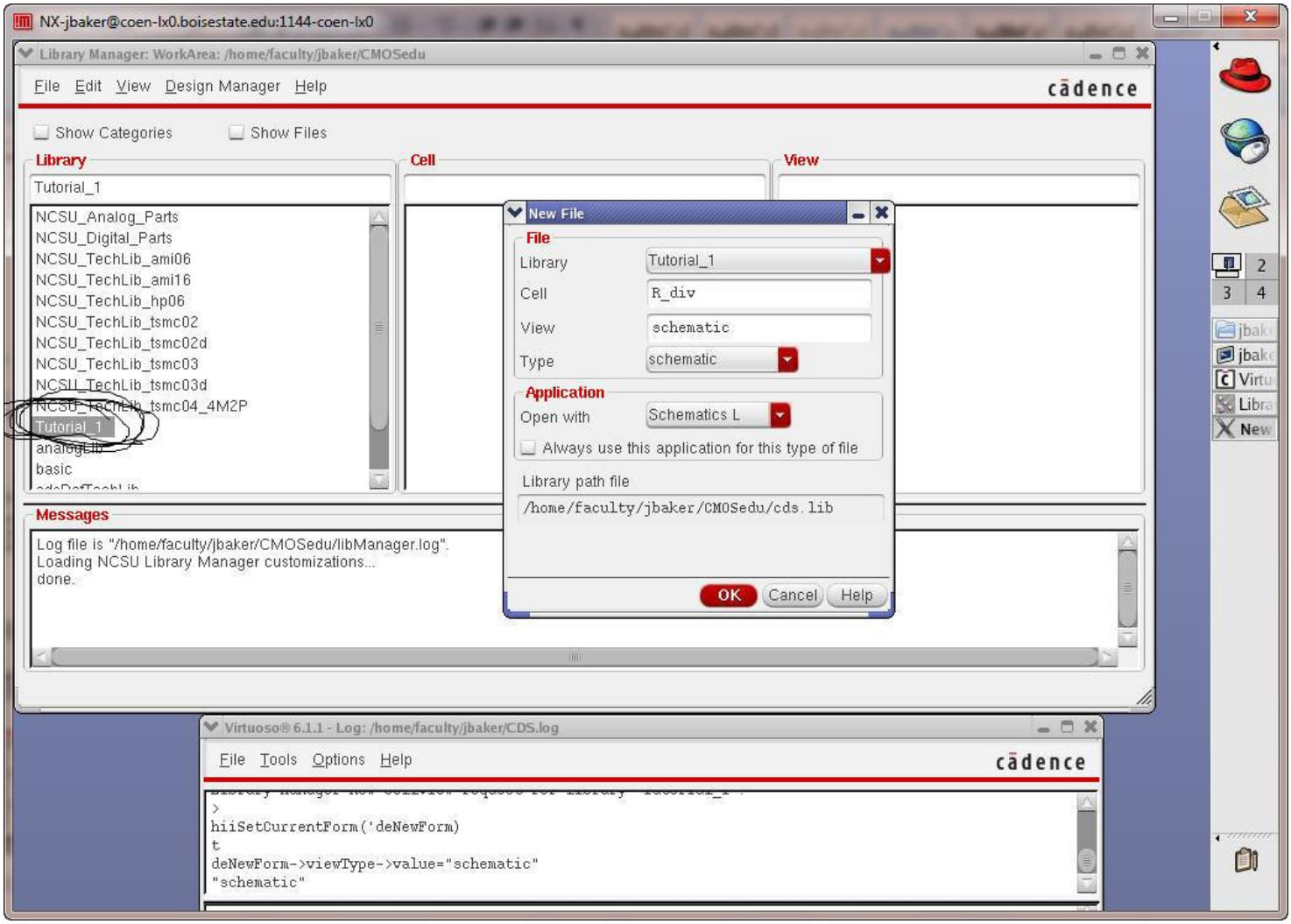
Next let's create a new library by going to, in the Library Manager, File -> New -> Library. The window to create this library, after pressing OK, may be
behind other windows so bring it to the front. Call the tutorial Tutorial_1 and attach the AMI 0.60u C5N process (remember AMI semiconductor is now
On semiconductor) as seen below.



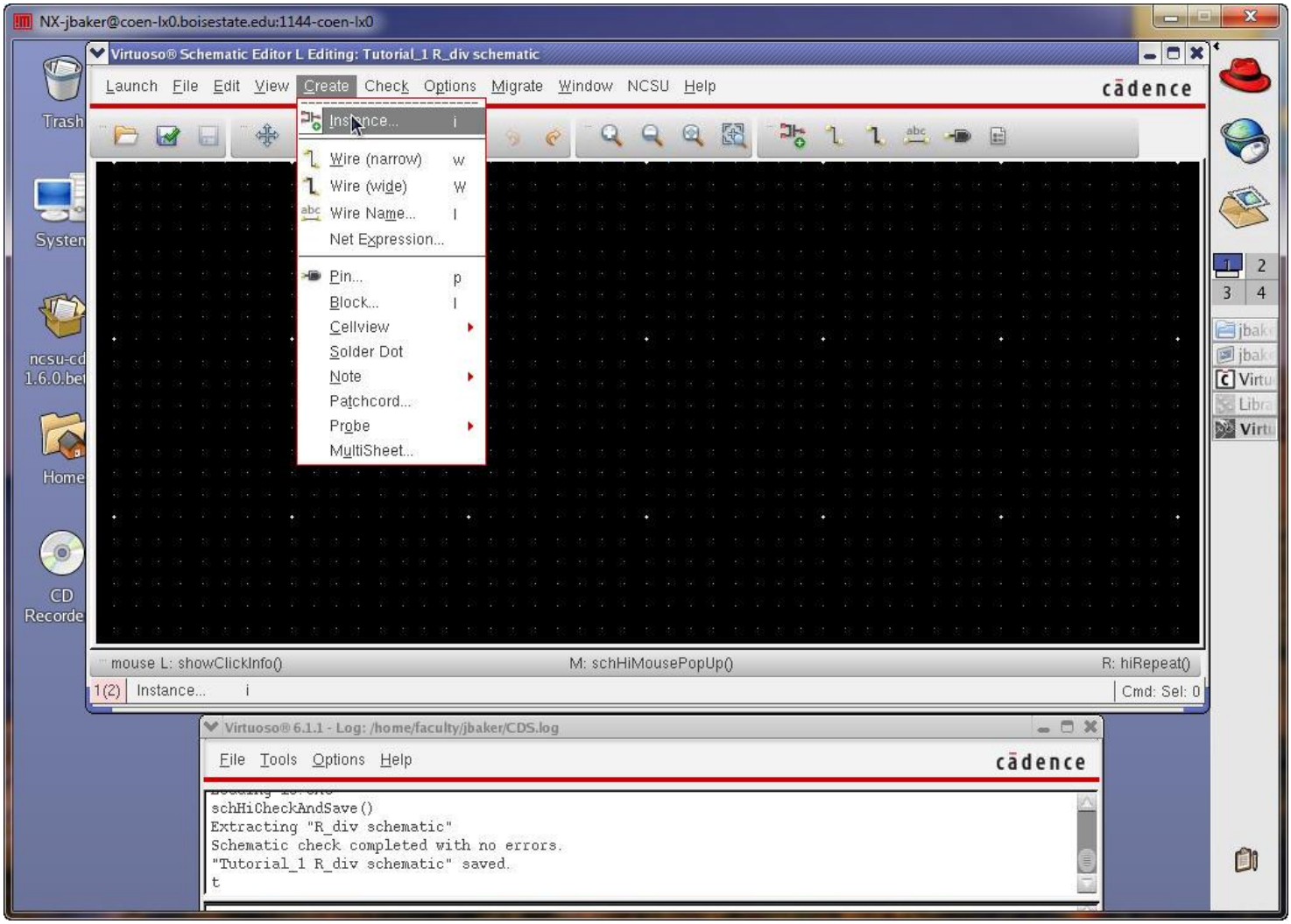
Let's pause for a moment and look at the cds.lib file in the CMOSedu directory. As seen below when a library is created a definition line is added to the
cds.lib file. If we wanted to simulate the IC61 book examples found at CMOSedu.com we simply unzip, for example, Ch1_IC61.zip into the CMOSedu
directory and add **DEFINE Ch1_IC61 $HOME/CMOSedu/Ch1_IC61** to the cds.lib (remembering $HOME = /home/faculty/jbaker or my home directory).



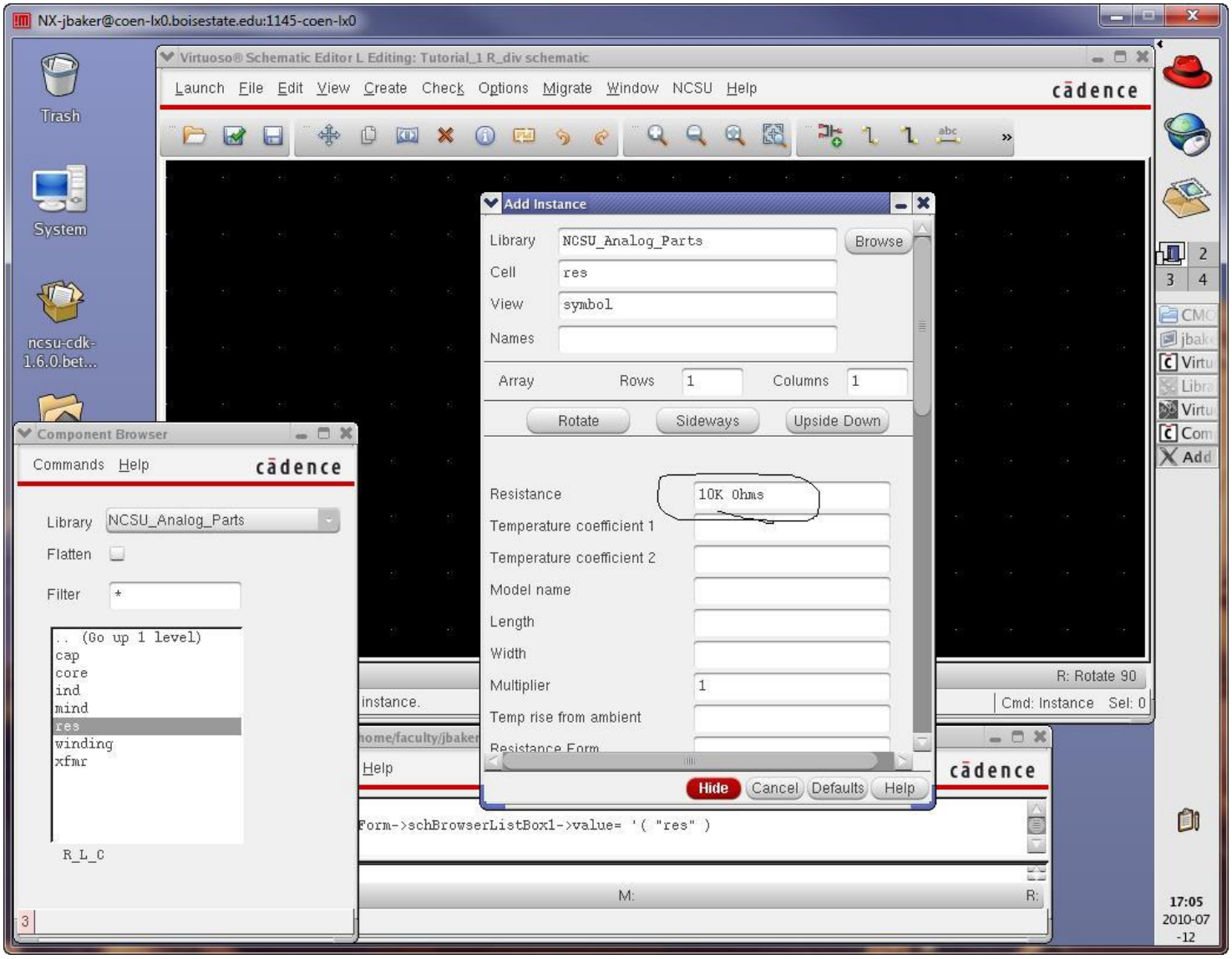
Next select the Tutorial_1 library in the Library Manager and then the menu items File -> New -> Cell View and enter the information seen below. Again
note that the window may be behind some other window.



After selecting OK and resizing the window we can add a component (an instance) by going to Create -> Instance (or just pressing the Bindkey **i** or use the
menu item above the drawing display) as seen below.

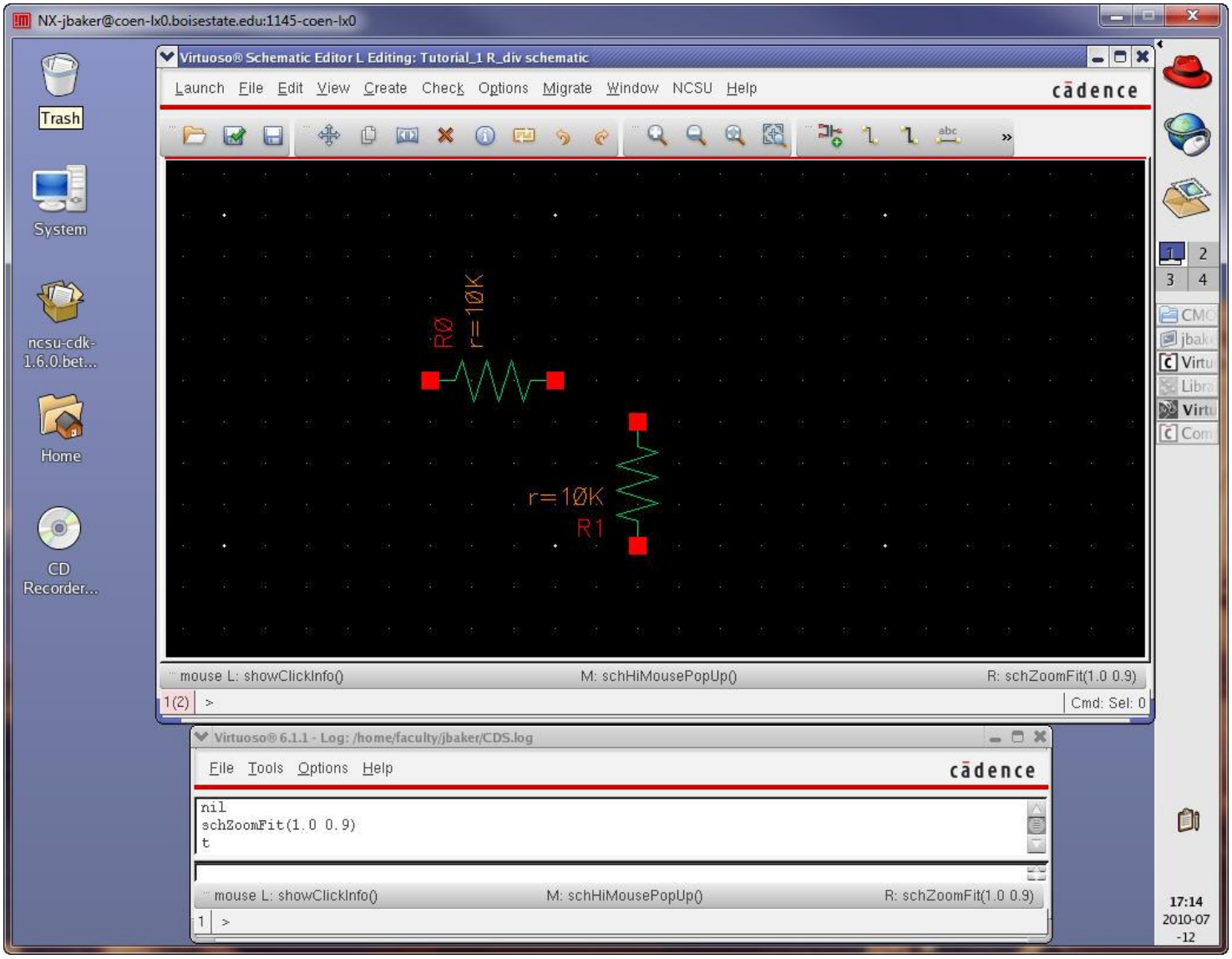


After re-sizing the window, selecting NCSU_Analog_Parts (in the Component Browser window), R_L_C, and res the following appears. Set the resistance
value to 10k as seen below.

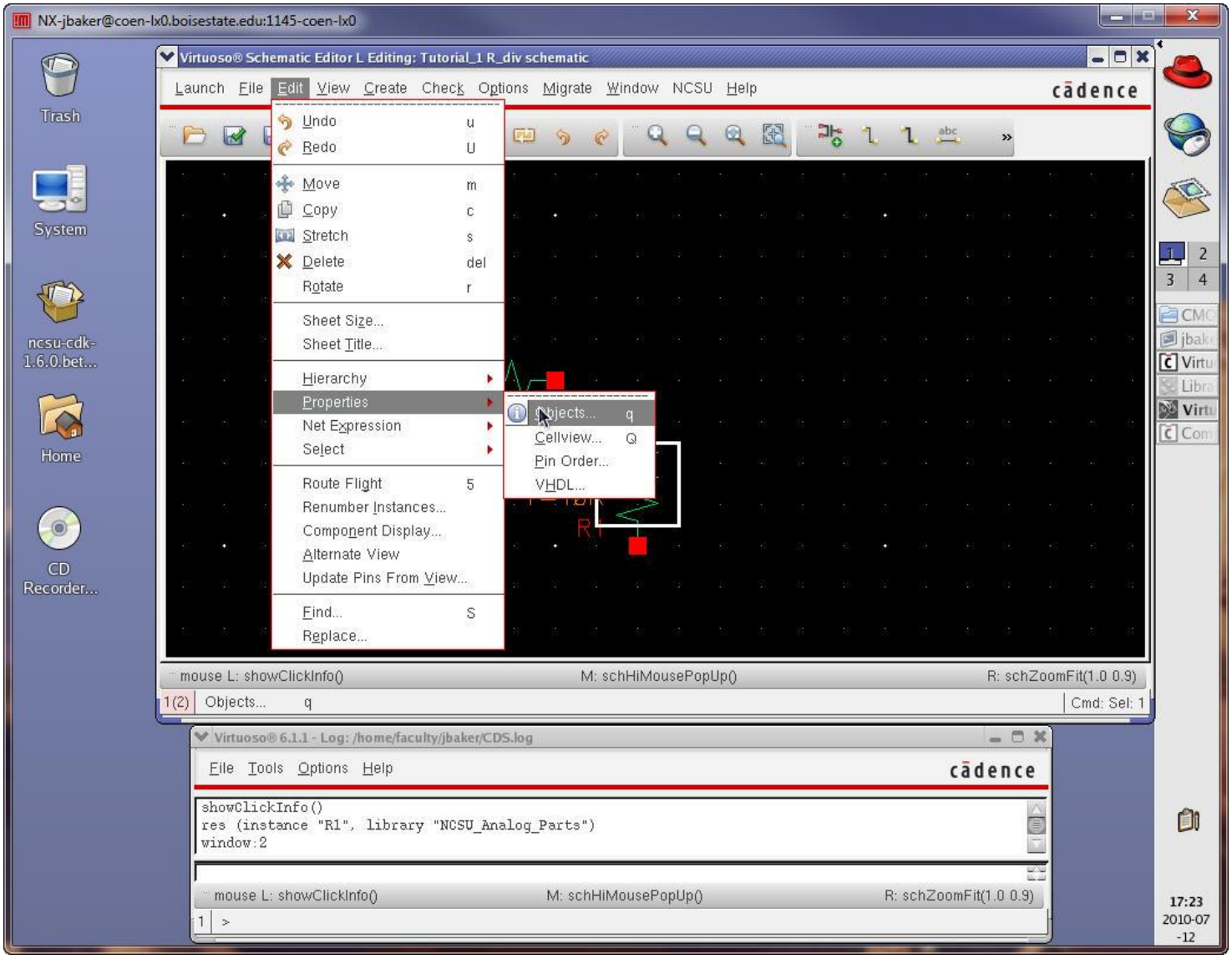


Hide the Add Instance window and minimize the Component Browser to the task bar. Add the two resistors as seen below. Right clicking the mouse button
rotates the symbol. Pressing Esc leaves the "Add Instance" mode.

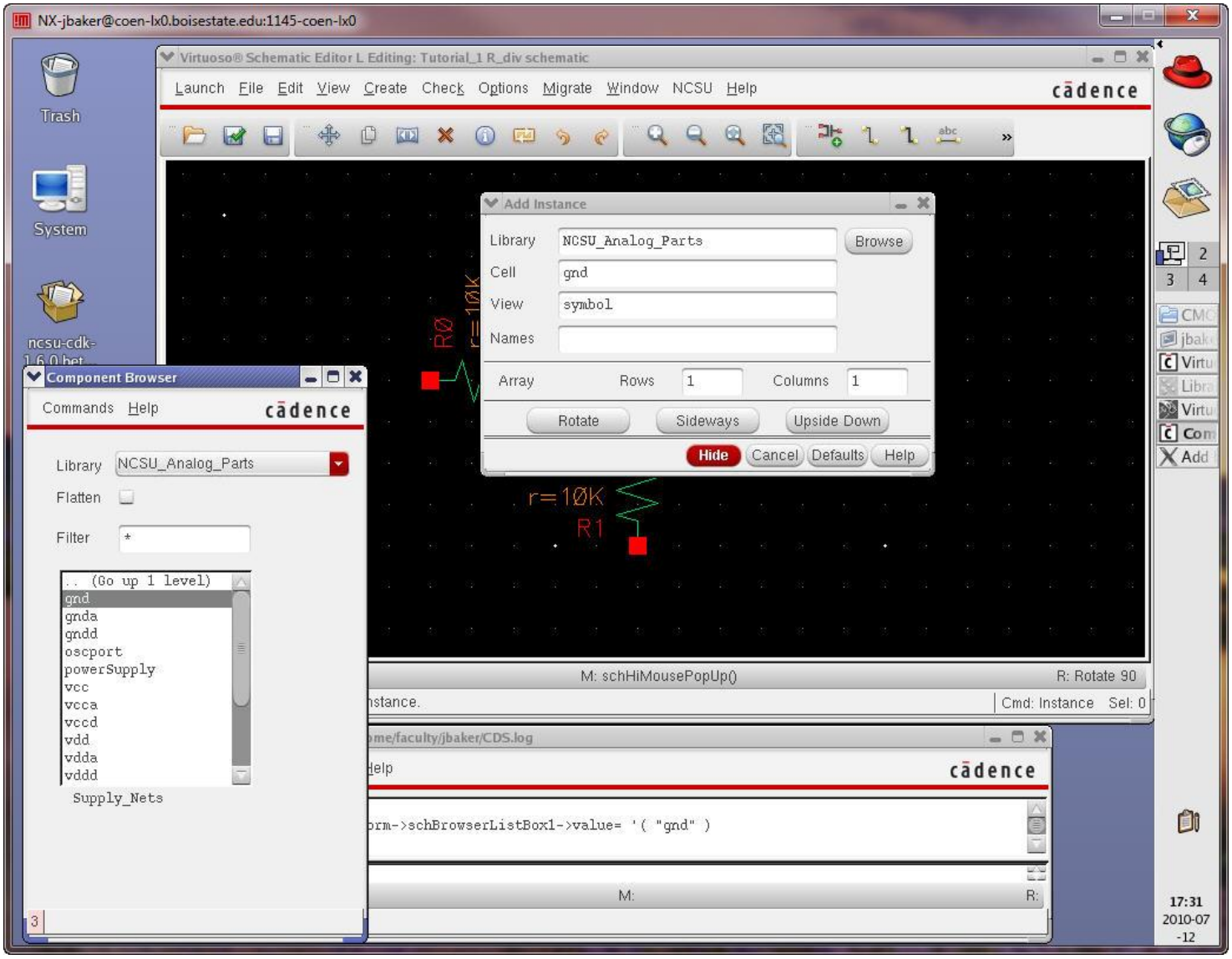
Note that the Bindkey **f** fits the display. A listing of the Bindkeys is found here.



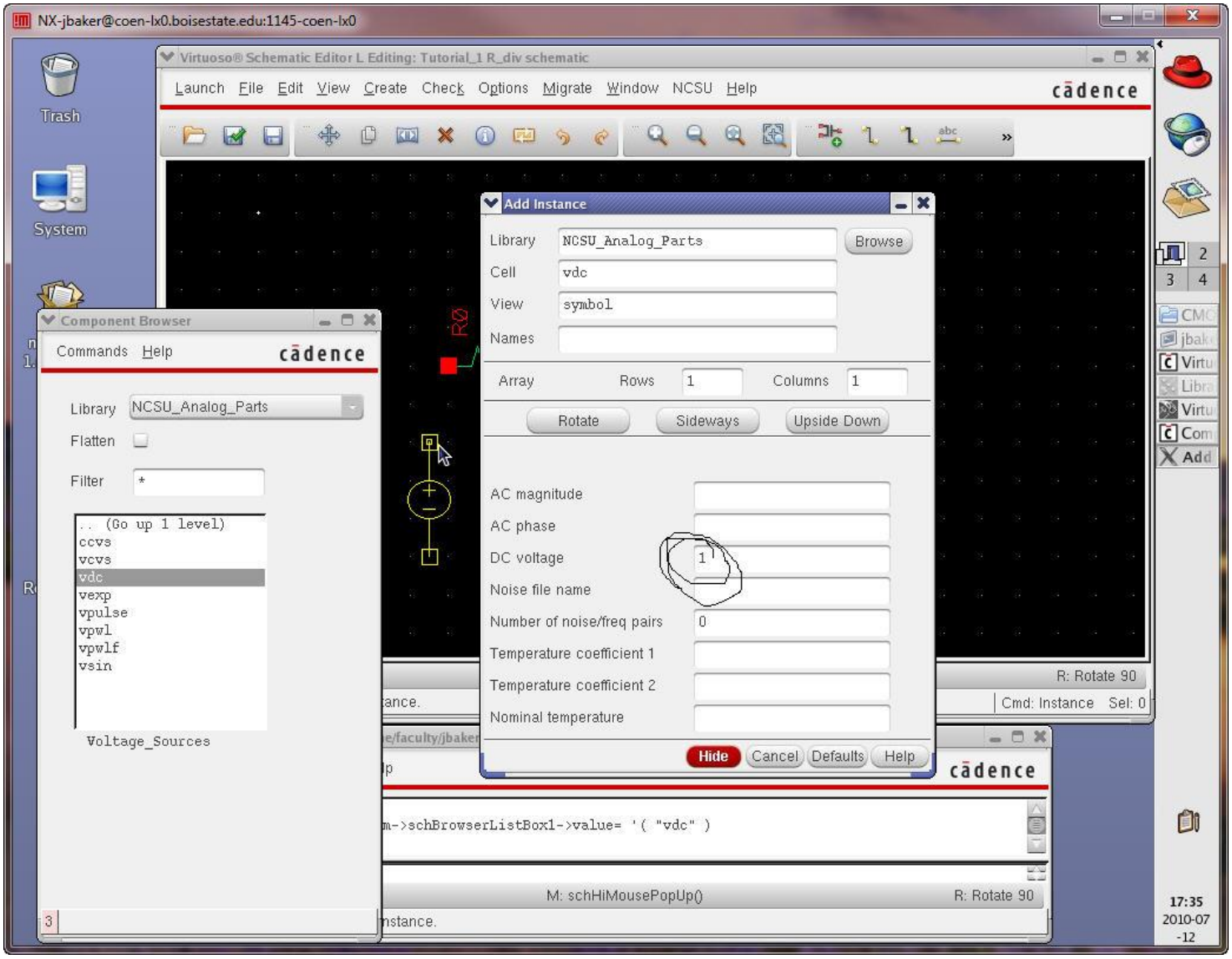
To change the resistor's value select the resistor and use Edit -> Properties -> Objects (or just use the Bindkey **q**) as seen below. We'll use this command often.
Click your mouse in the drawing area and press Esc a few times so that no commands are active.



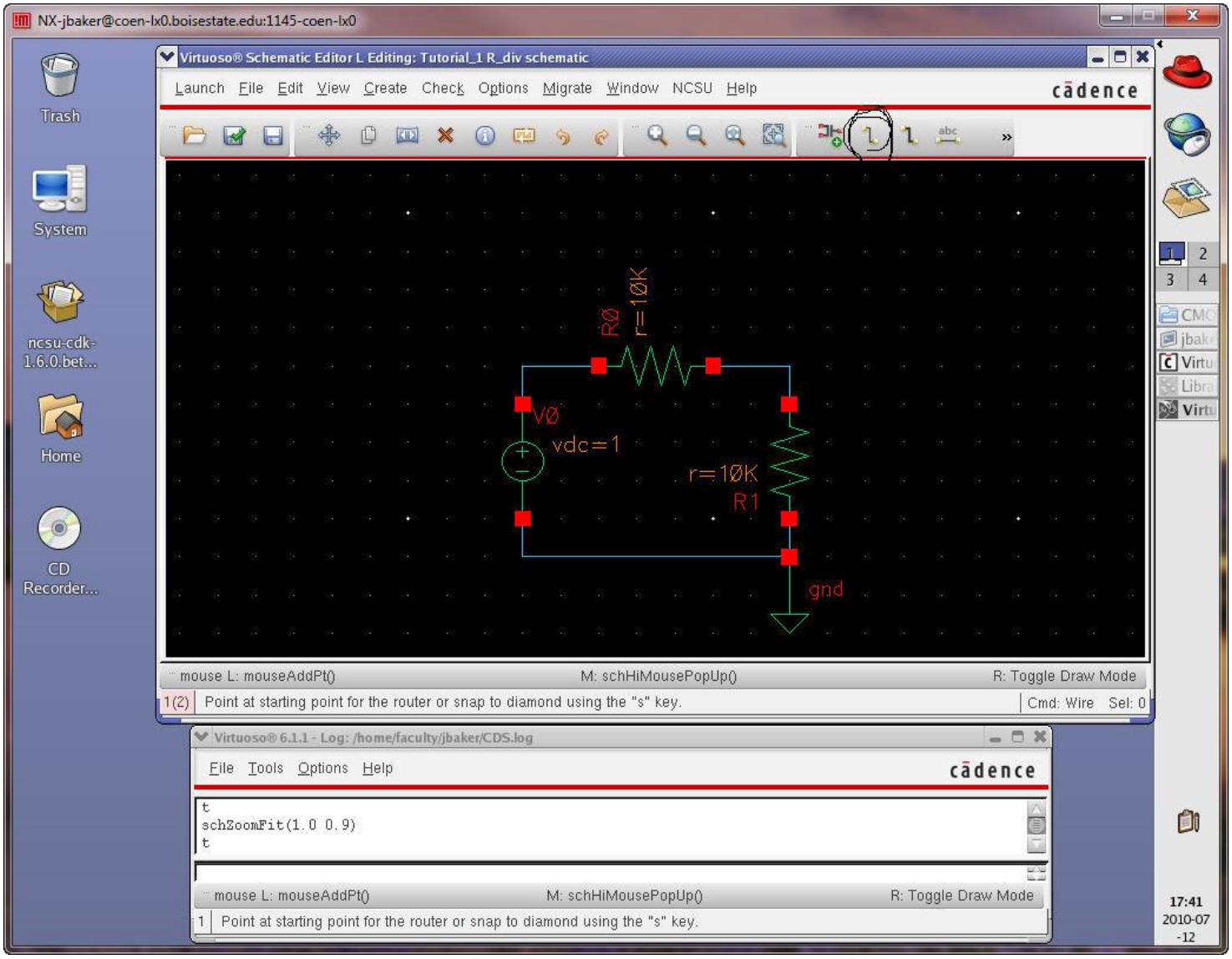
Next add ground to the schematic by pressing i (add instance) and maximizing the Component Browser, selecting Supply_Nets and gnd as seen below.
If you know the name and Library of the instance you want to add you can type them into the fields directly in the Add Instance window.



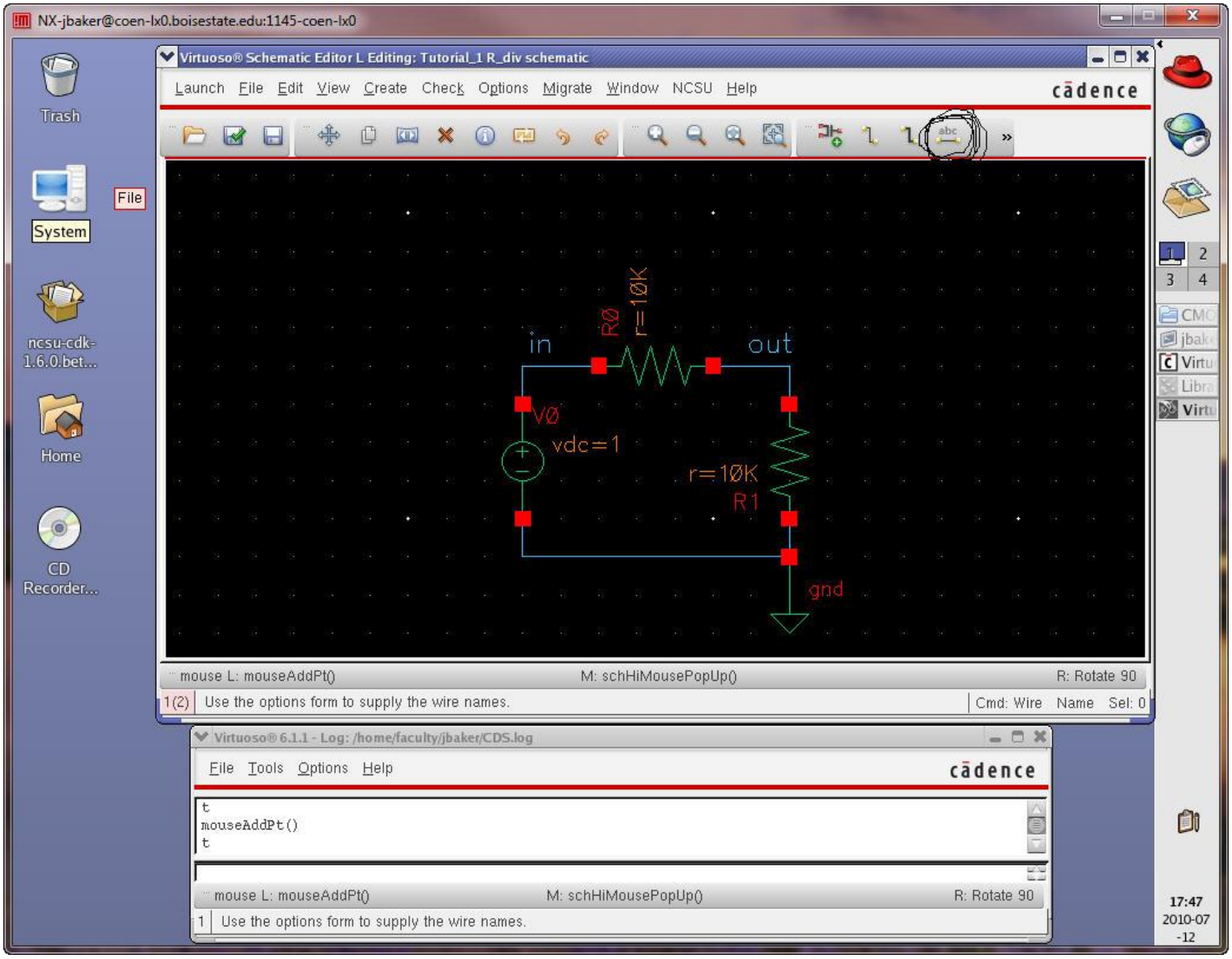
Add the ground symbol and then add a 1-V DC source, symbol name of vdc under Voltage_Sources as seen below.



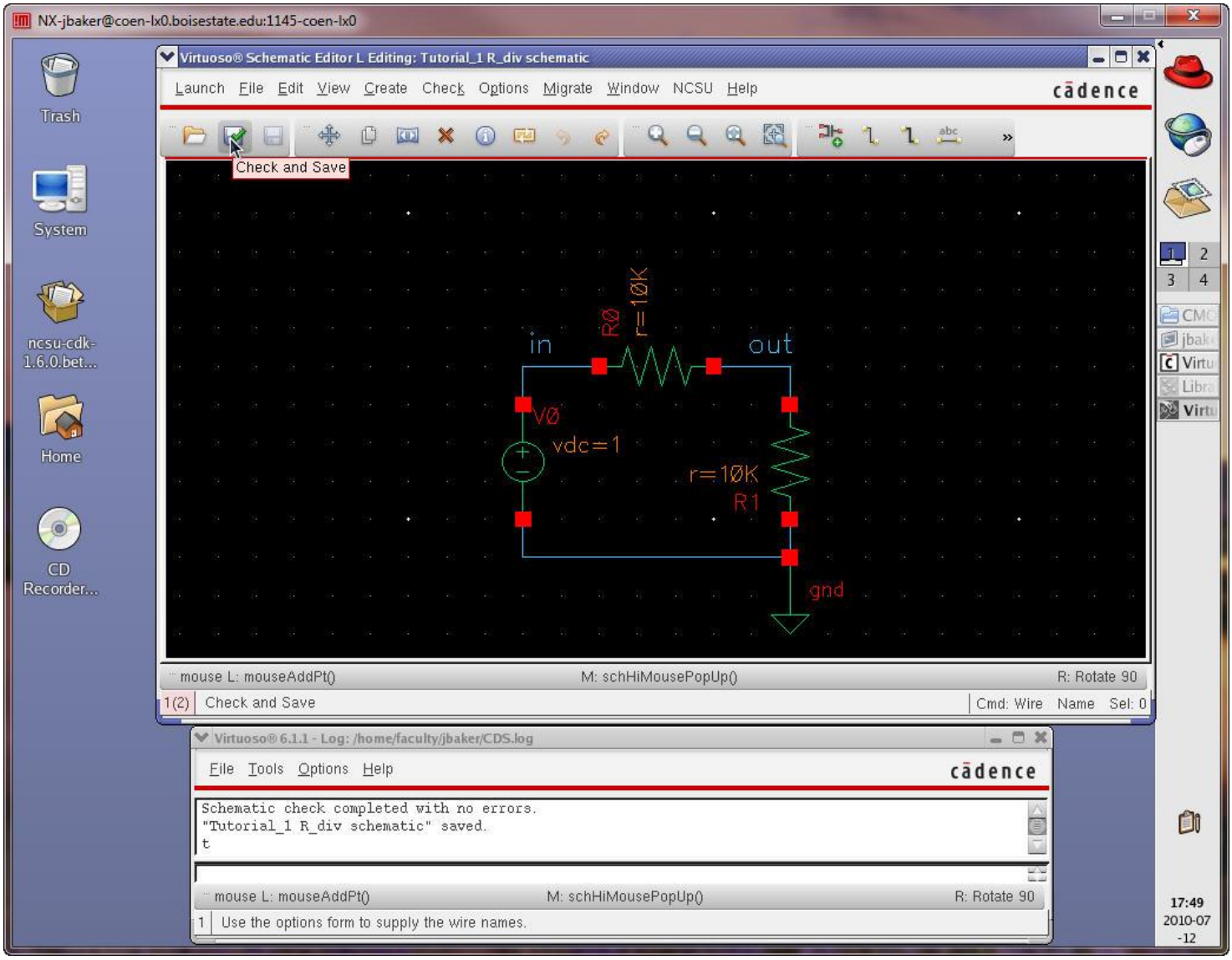
After placing the symbol we need to wire the circuit together. This can be done by using the Bindkey **w** (for wire) or the menu item as seen below.



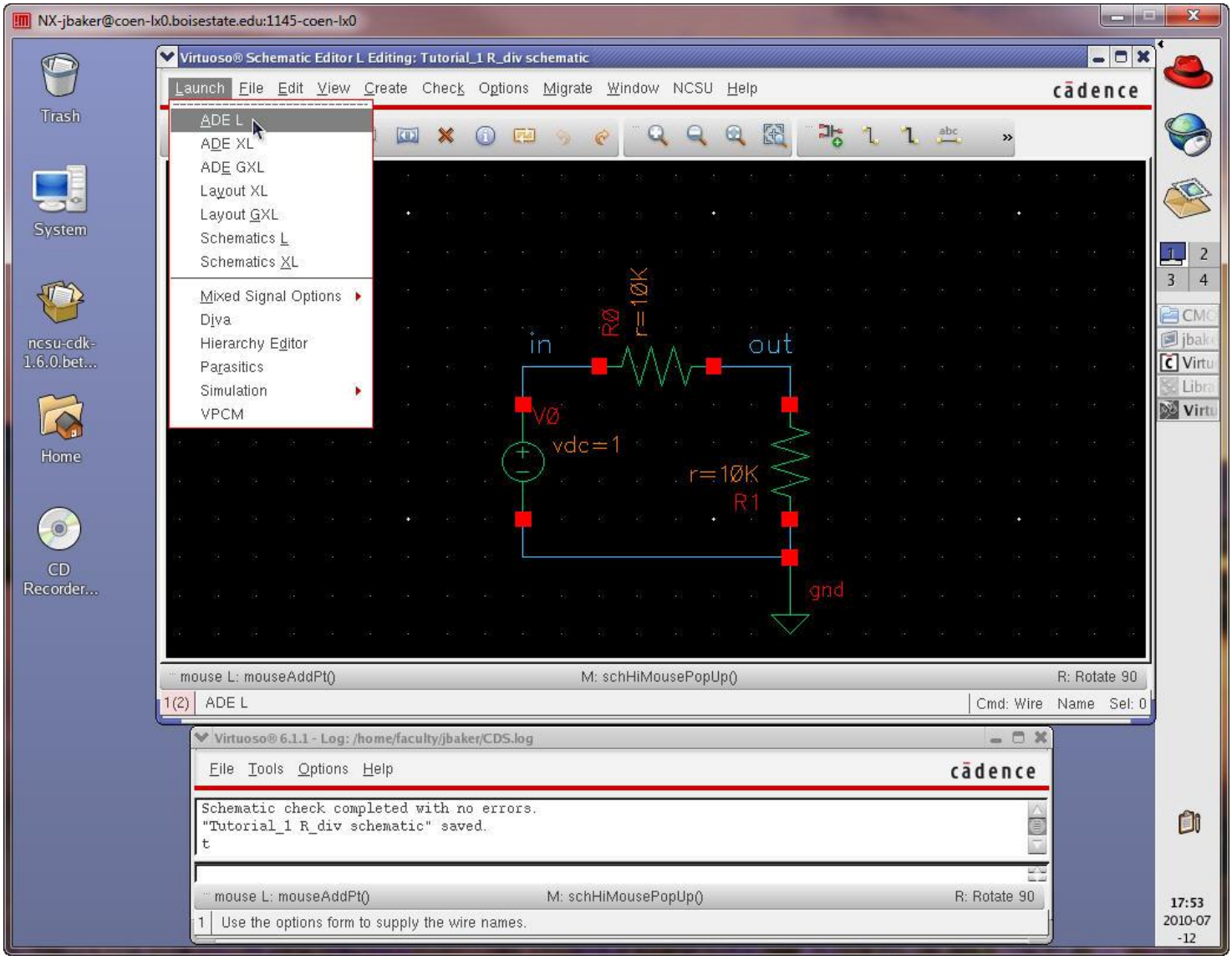
It's often useful to label wires with signal names. Using the Bindkey l (lowercase L) or the menu item enables naming wires. Let's do this as seen below.



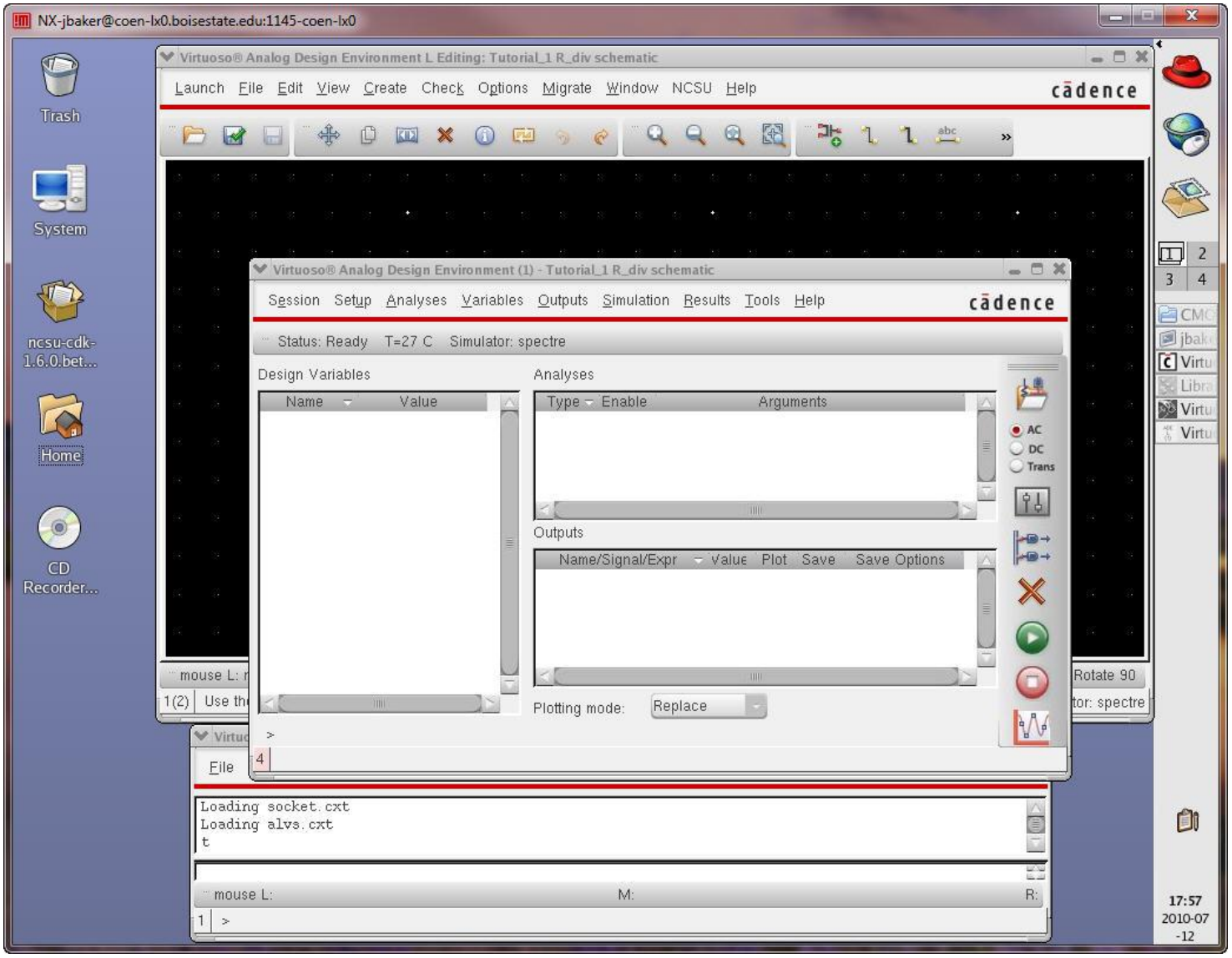
We are about ready to simulate the operation of this circuit. Let's do a "Check and Save" first. If we have edited the schematic and try to simulate
without checking and saving first the simulation will fail.



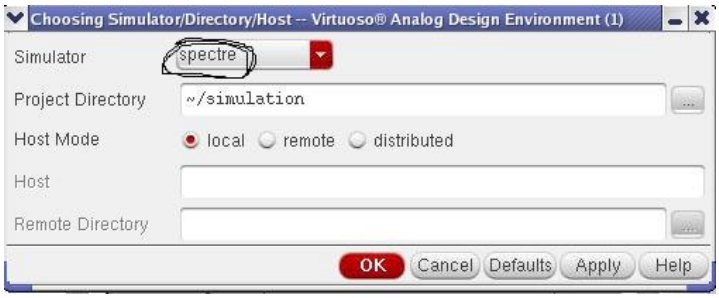
To simulate with Spectre (Cadence's SPICE simulator) go to the menu Launch -> ADE L as seen below.



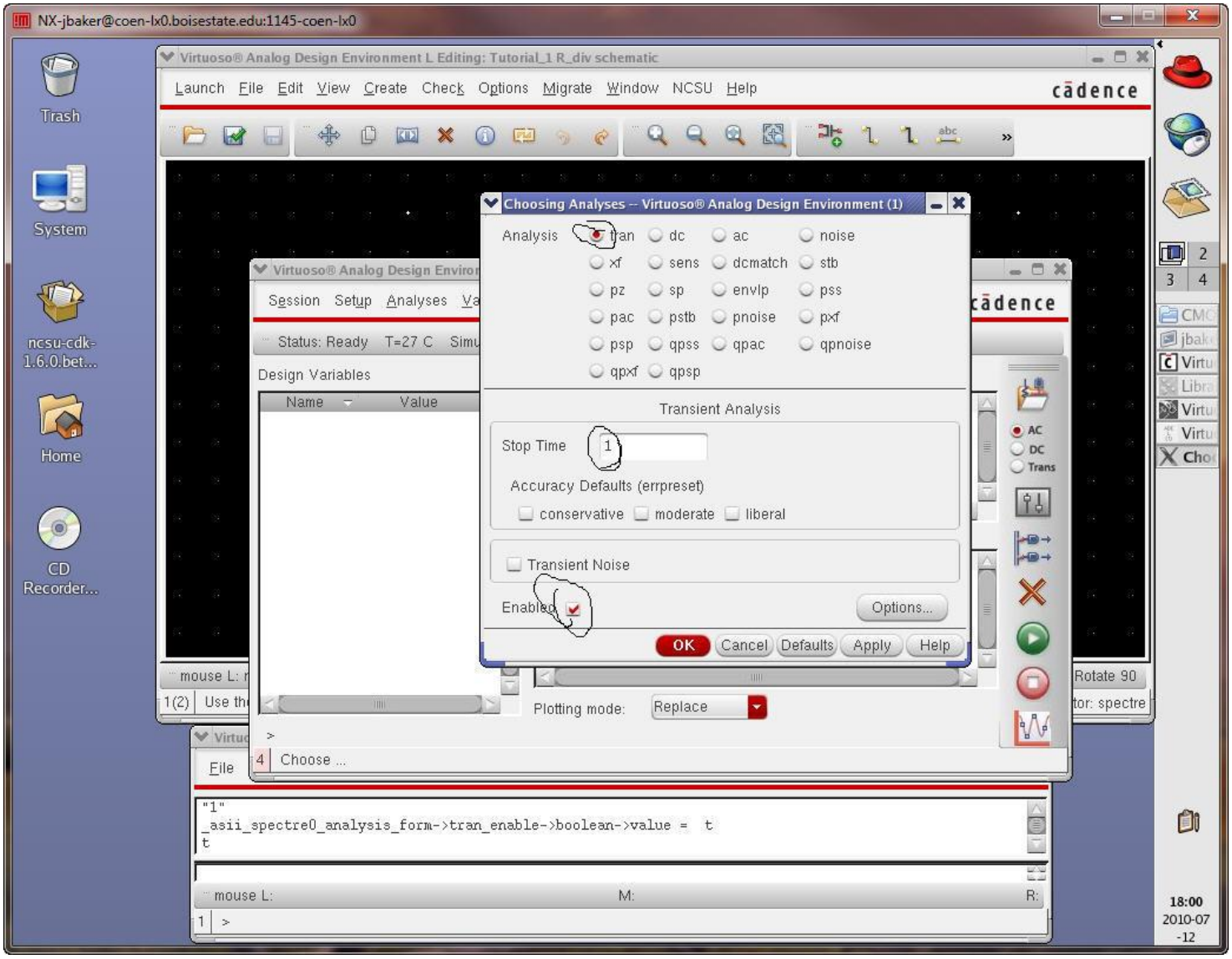
The Virtuoso Analog Design Environment (ADE) window should appear as seen below.



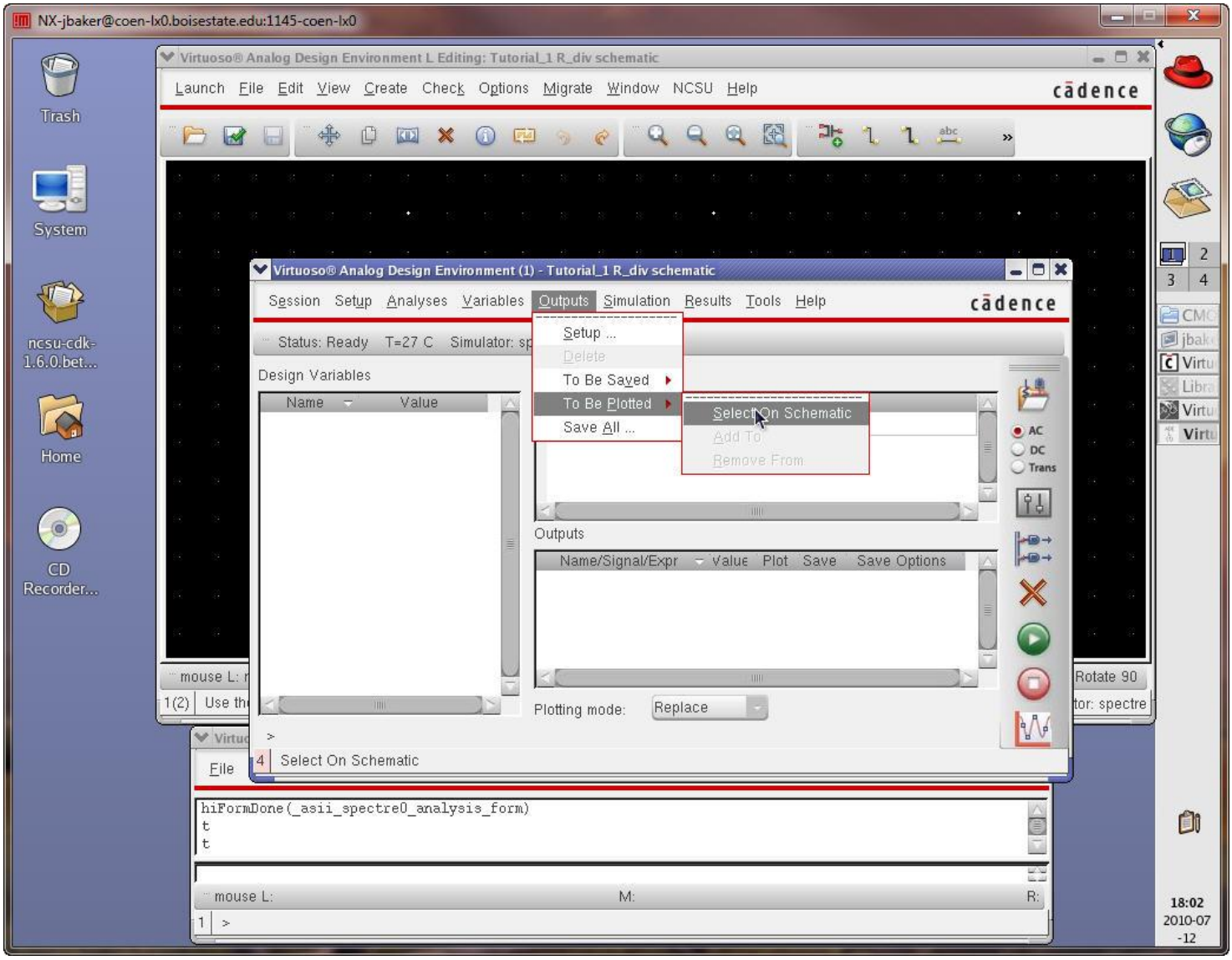
Use the menu in the ADE and go to Setup -> Simulator/Directory/Host to verify the simulator is set to Spectre.
We'll assume throughout these tutorials that Spectre is used. If it's not the default then please re-visit above instructions.



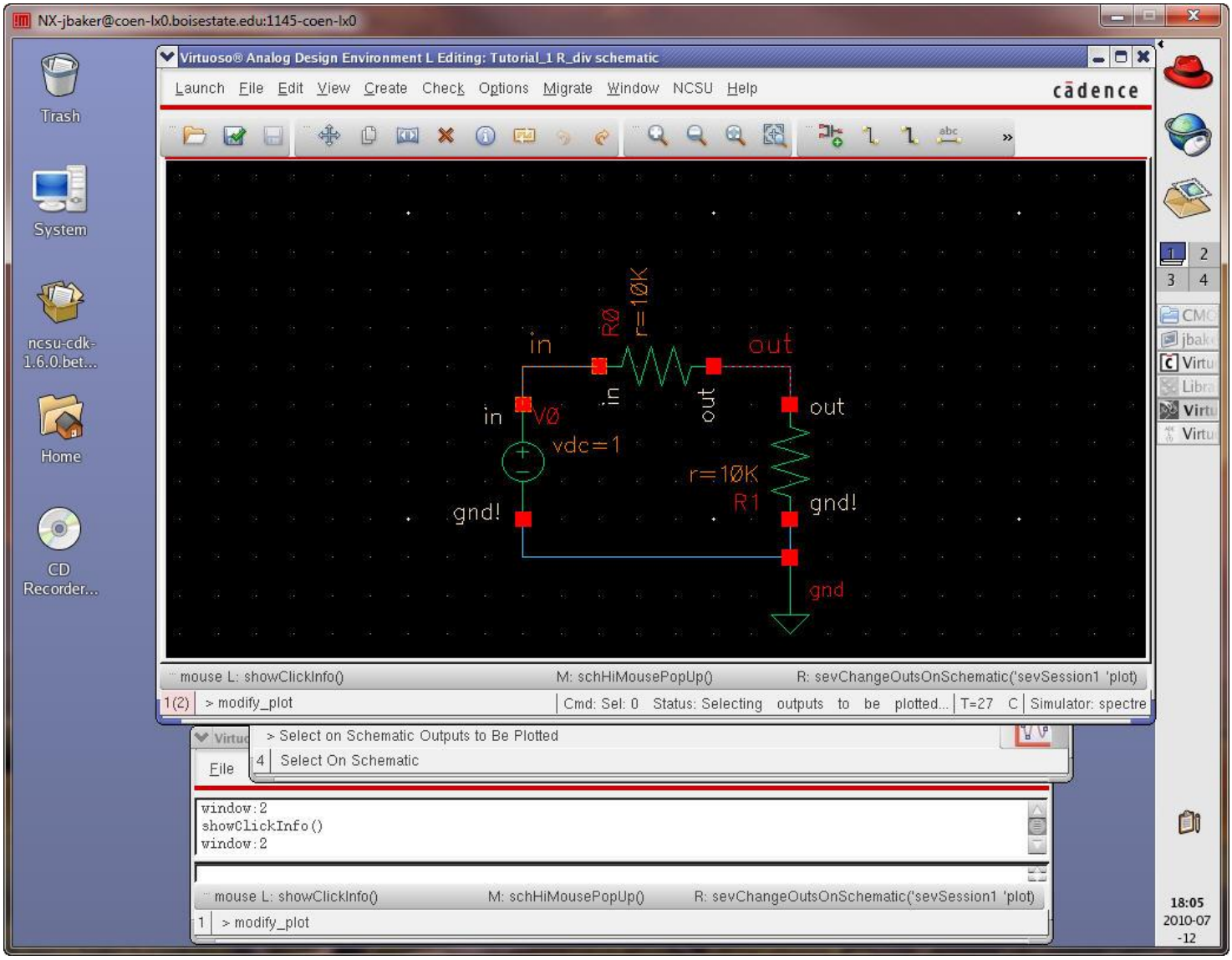
Next go to Analyses -> Choose and select a transient analysis (tran), a stop time of 1 second, and Enabled as seen below.



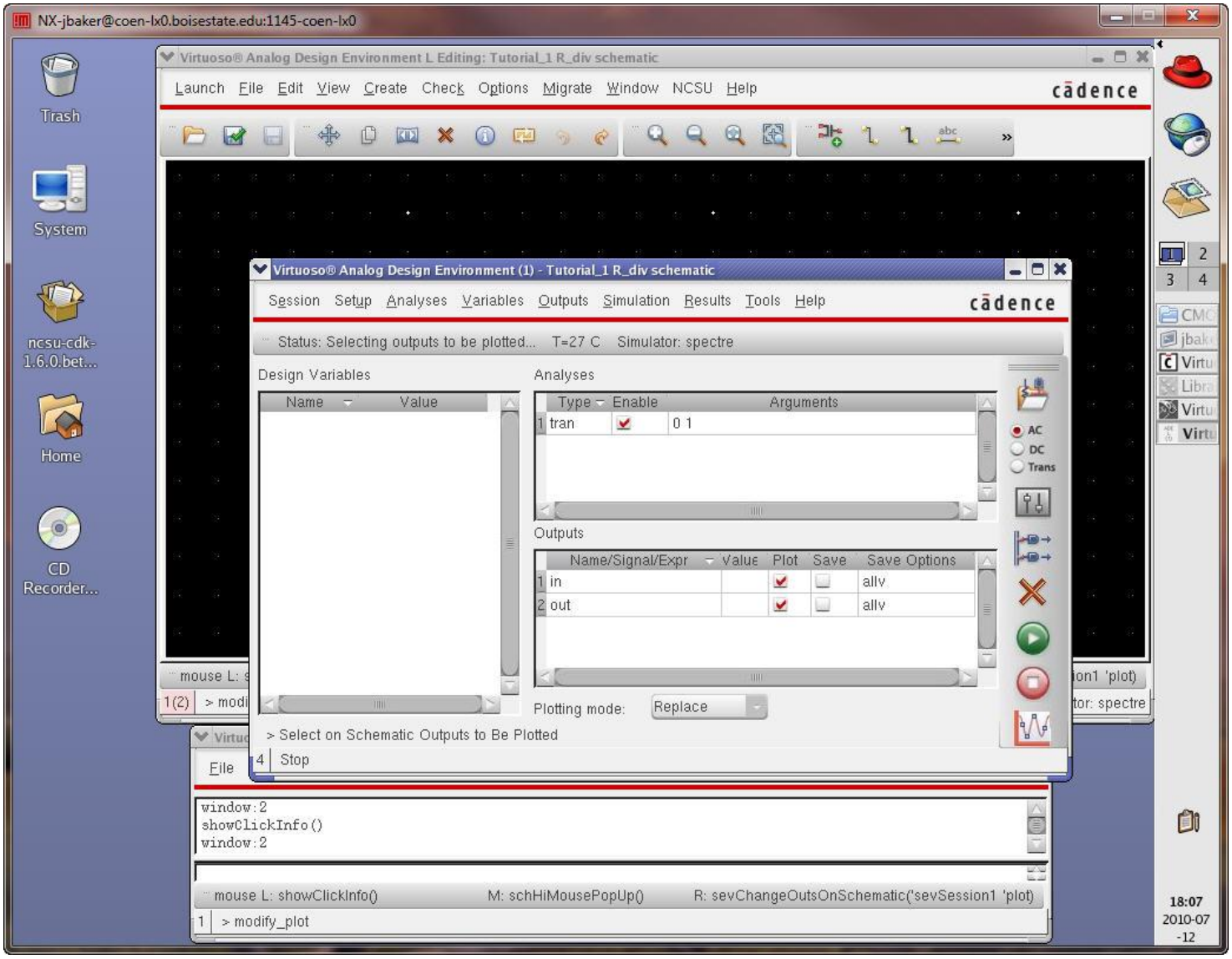
Next we need to select the signals we want to plot. Follow the menu items seen below selecting "Select On Schematic".



Select the wires as seen below.

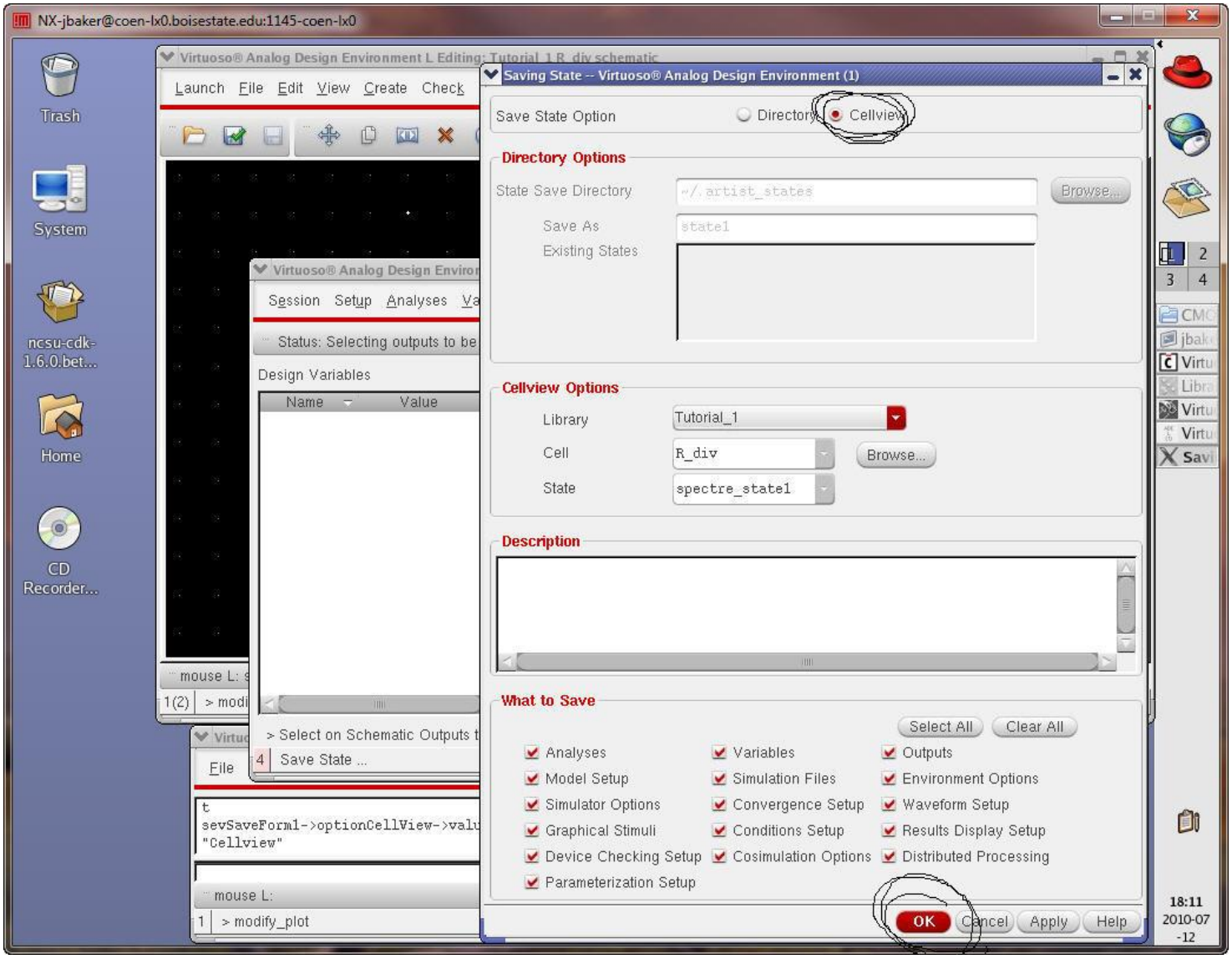


The ADE should now look like the following (bring the window to the front via the task bar).



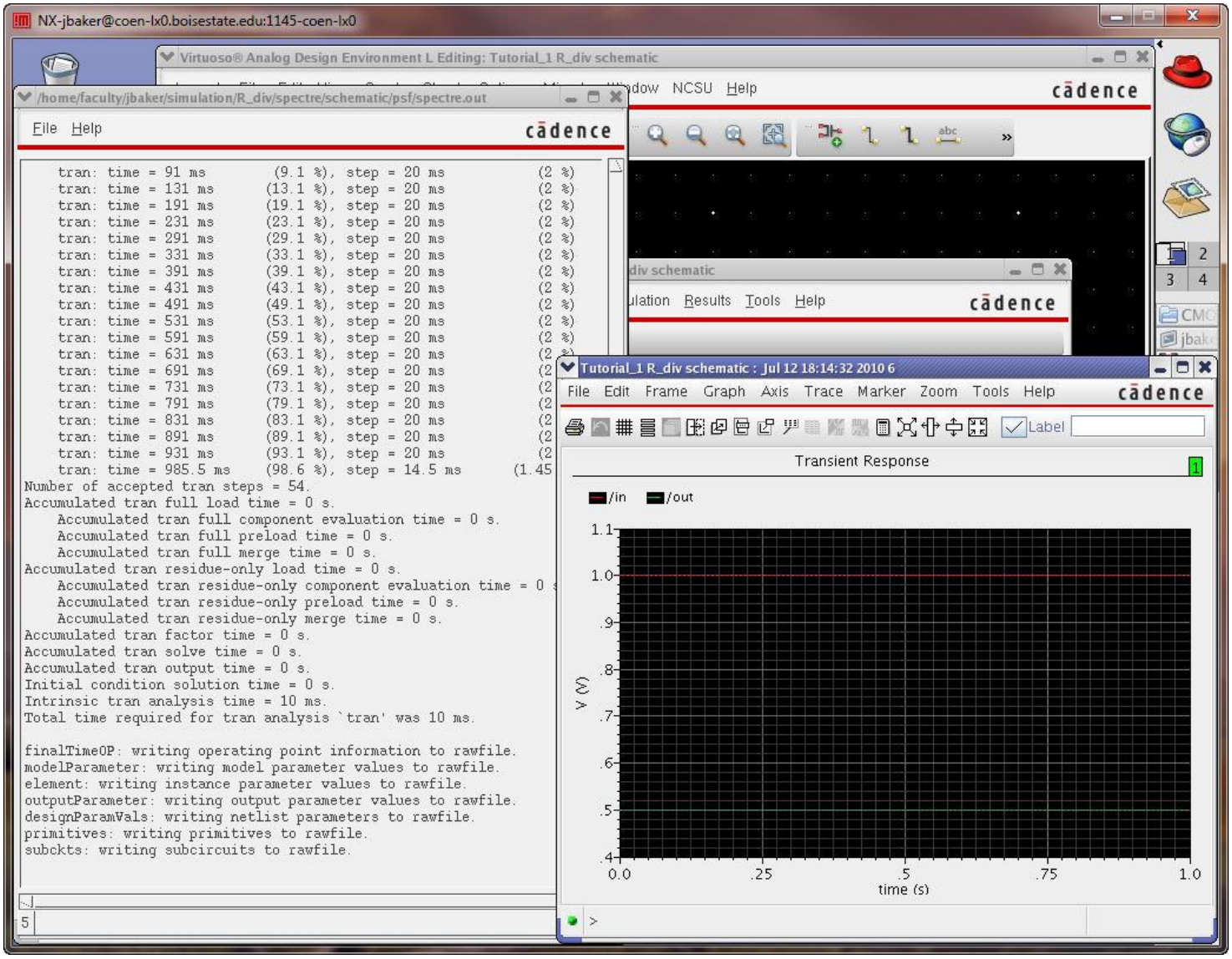
We can now simulate the schematic by pressing the green "Netlist and Run" button. However, before we do this let's save this information so that next time
we want to simulate this circuit we don't have to go through these steps again.

In the ADE window use the menu items Session -> Save State and select Cellview then OK as seen below.



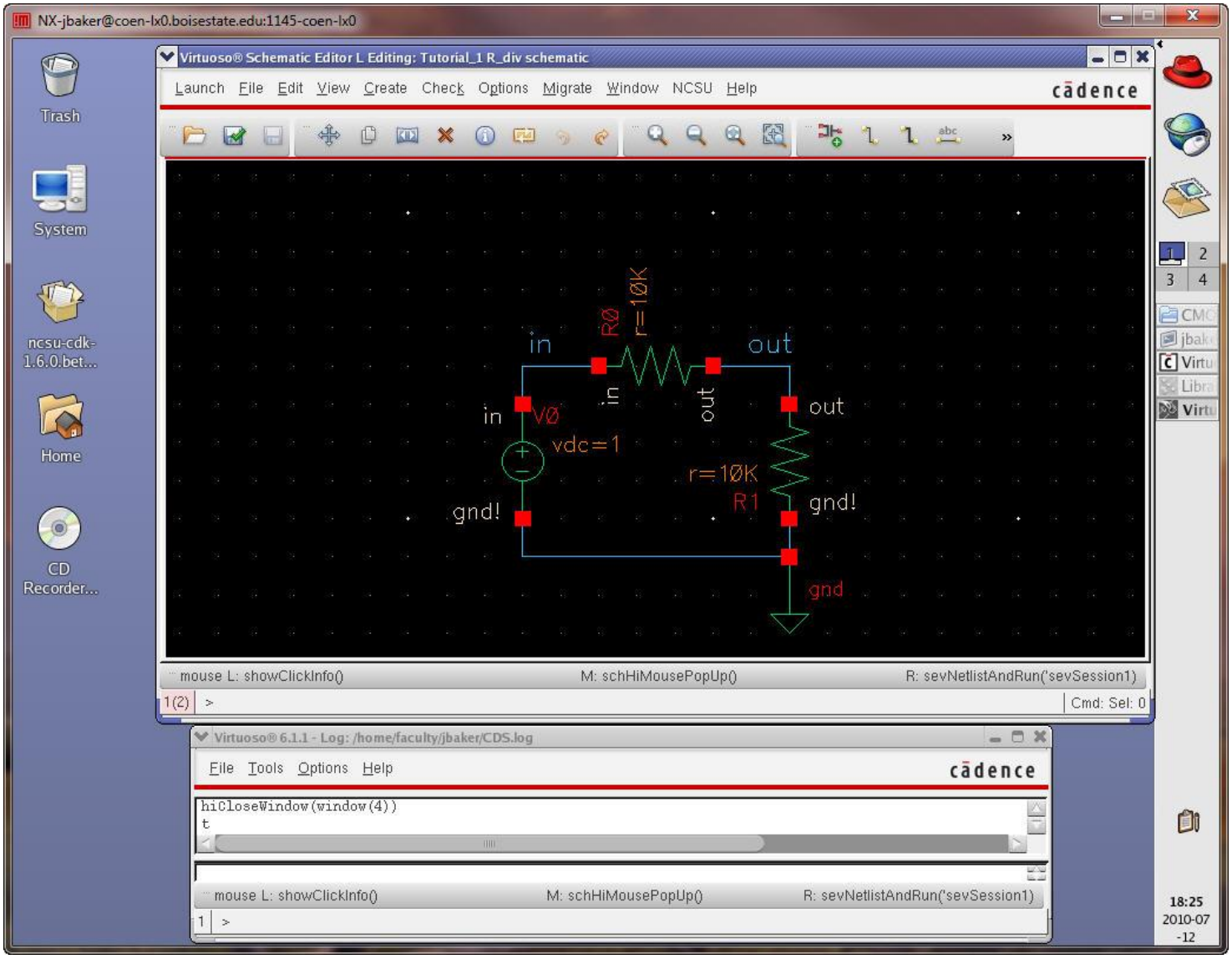
To load this state we select, in the ADE window, Session -> Load State and Cellview then OK. This is important to save time and used throughout the examples
from CMOSedu.com.

Now pressing the green button and running the simulation results in the following.



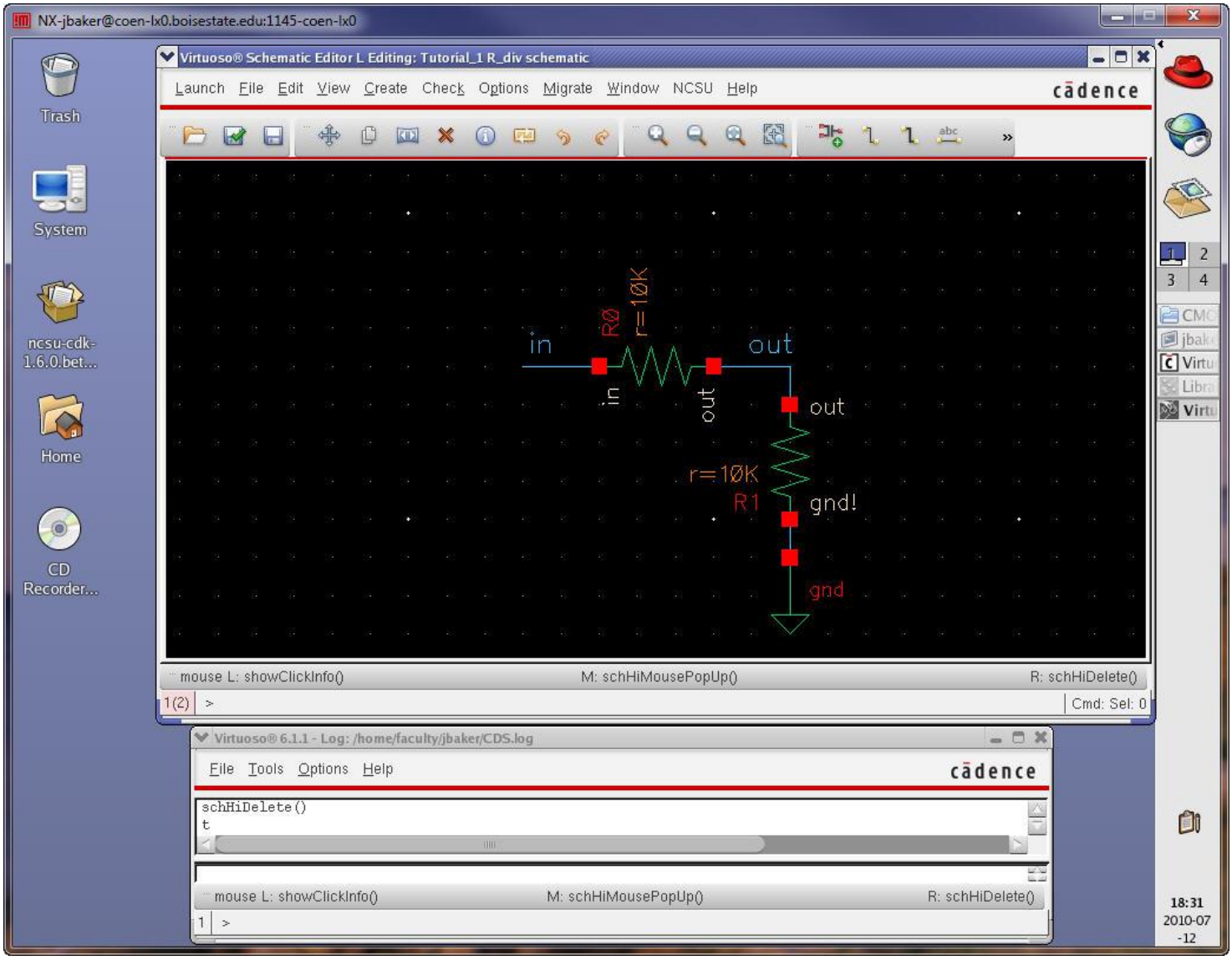
Which is, hopefully, what the reader expected ;-)

Let's close the ADE and go back to the schematic.

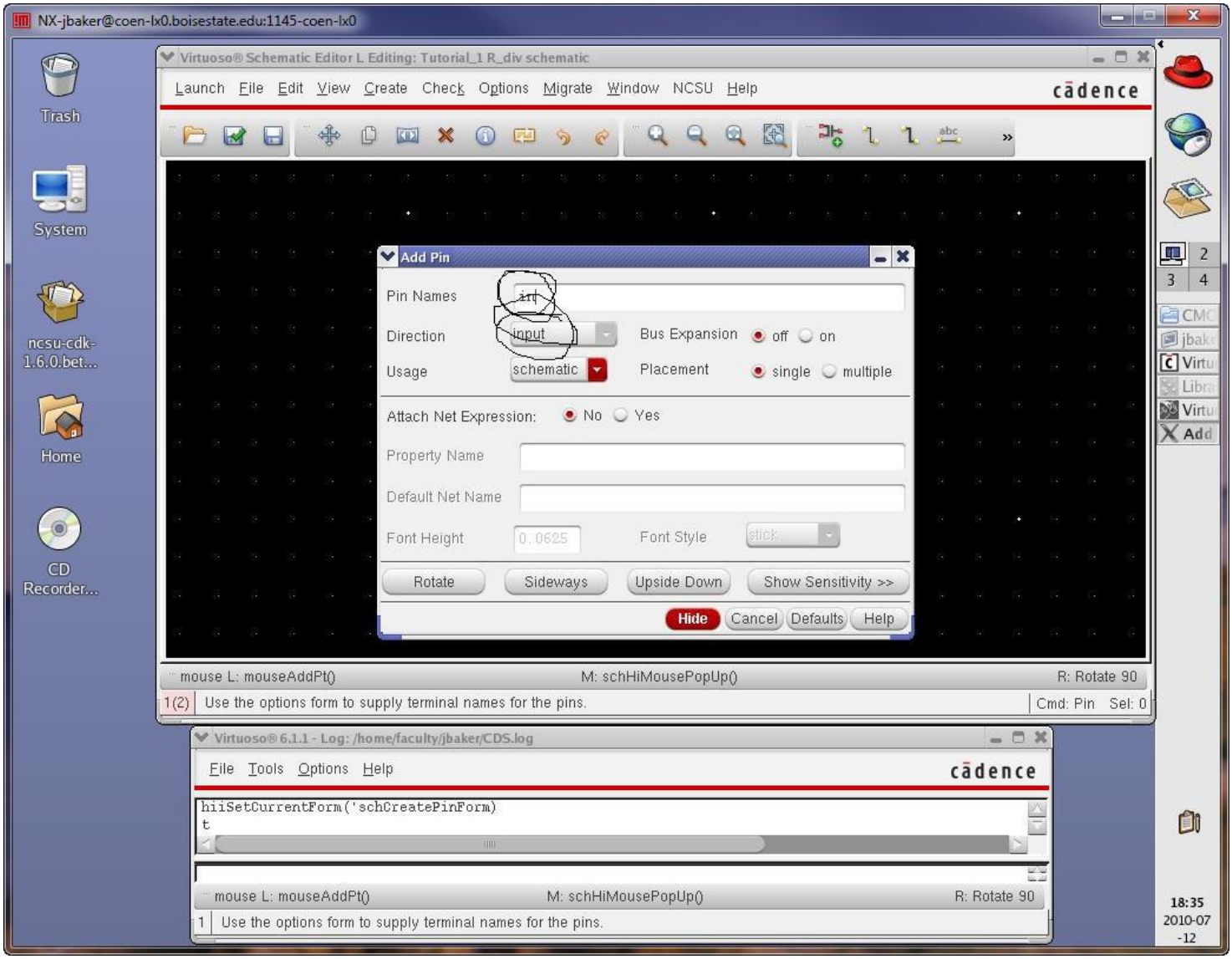


While this is a general purpose schematic let's make it more useful as a cell in an integrated circuit. Select, and delete, the bottom wire and voltage source.
Note that once you select and delete one wire or component Virtuoso is in the delete mode so all you have to do is click on the next item you want to delete.
To exit this mode press Esc. To undo an action press **u**.

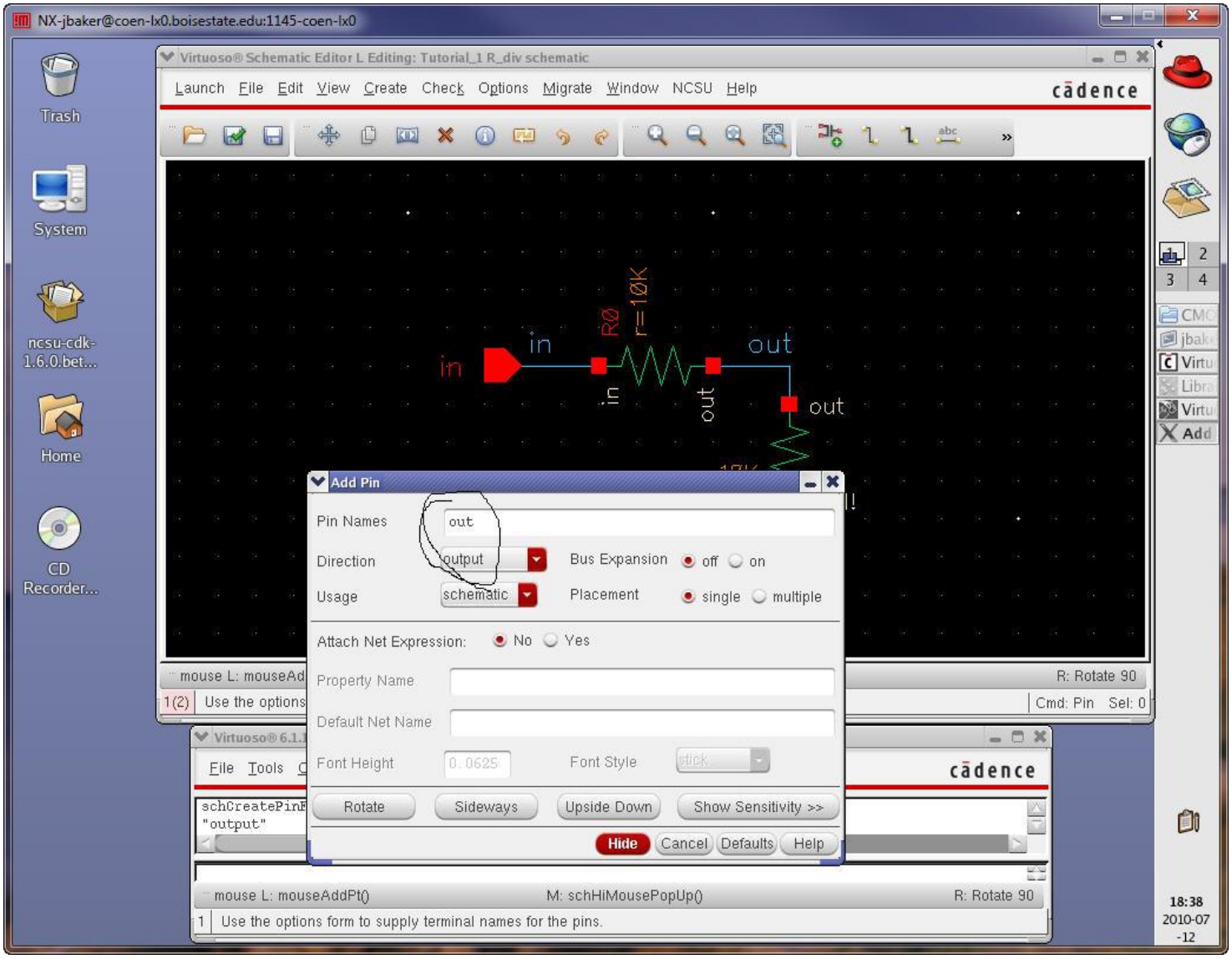
Ensure you have the following schematic.



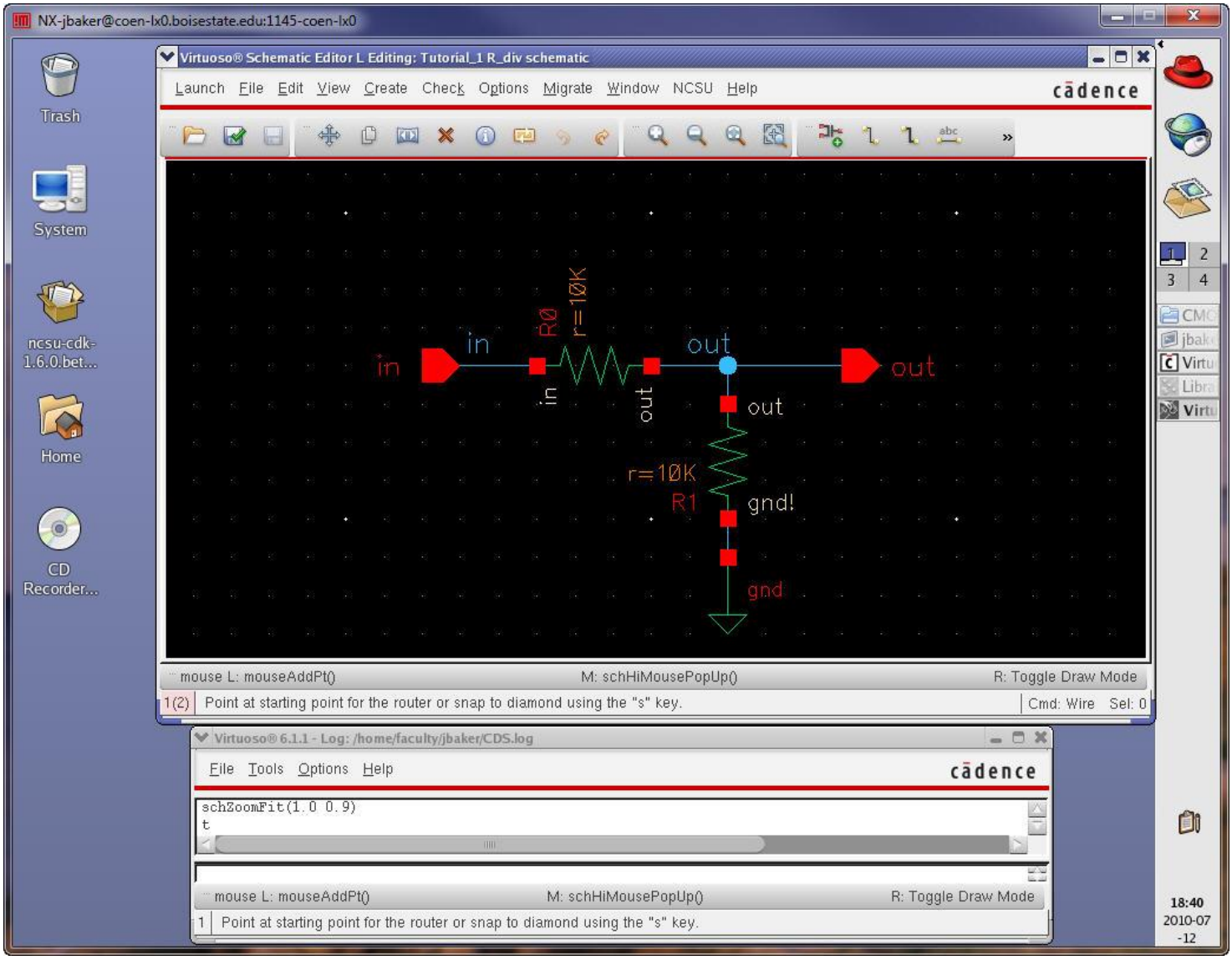
Let's add pins to the schematic using the menu Create -> Pin (or the Bindkey **p**). Pins are what is used to connect the schematic symbol, which will make shortly,
up in a higher-ranking schematic view. Add an input pin called in (so it matches the wire name, useful but not necessary).



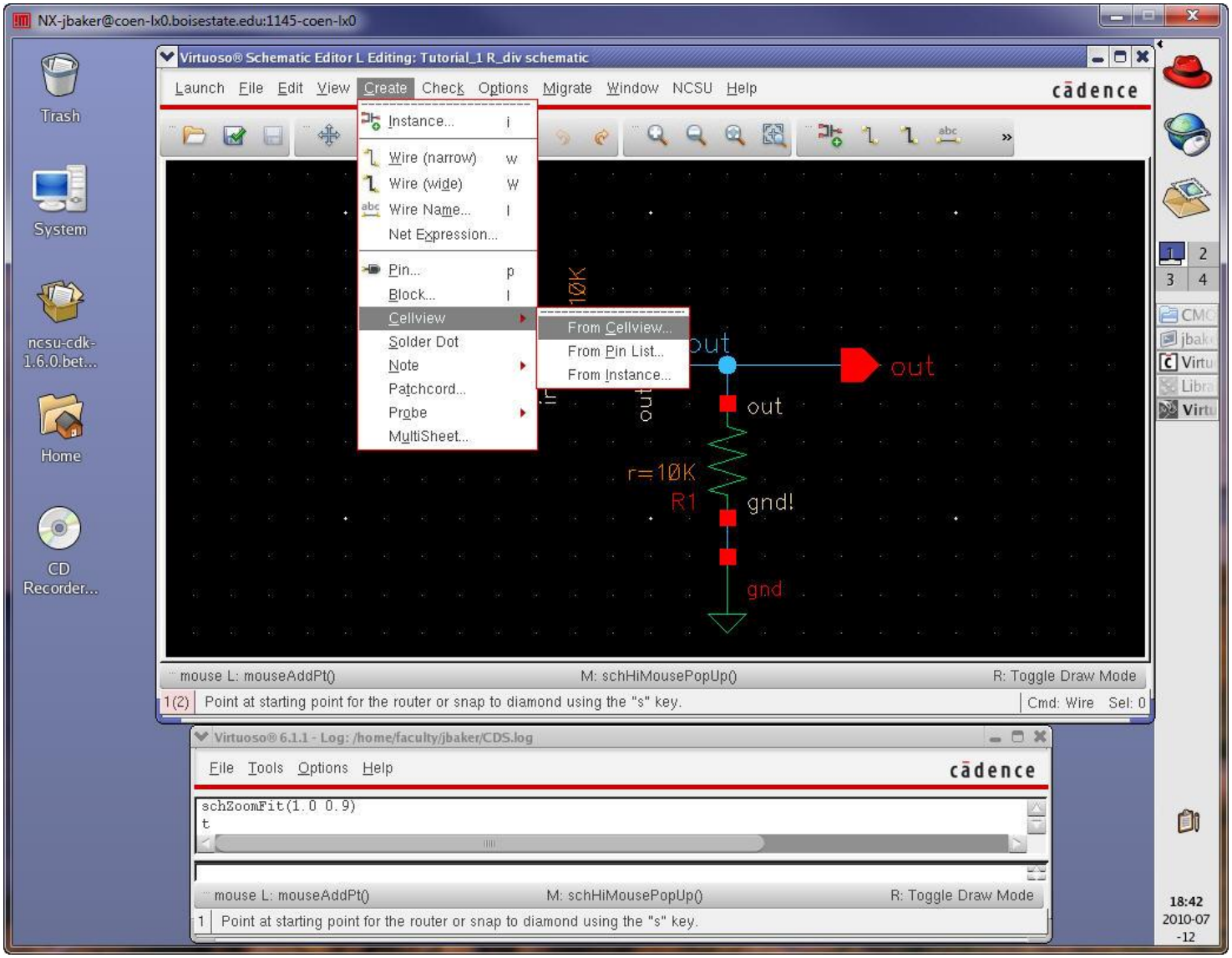
And an output pin.

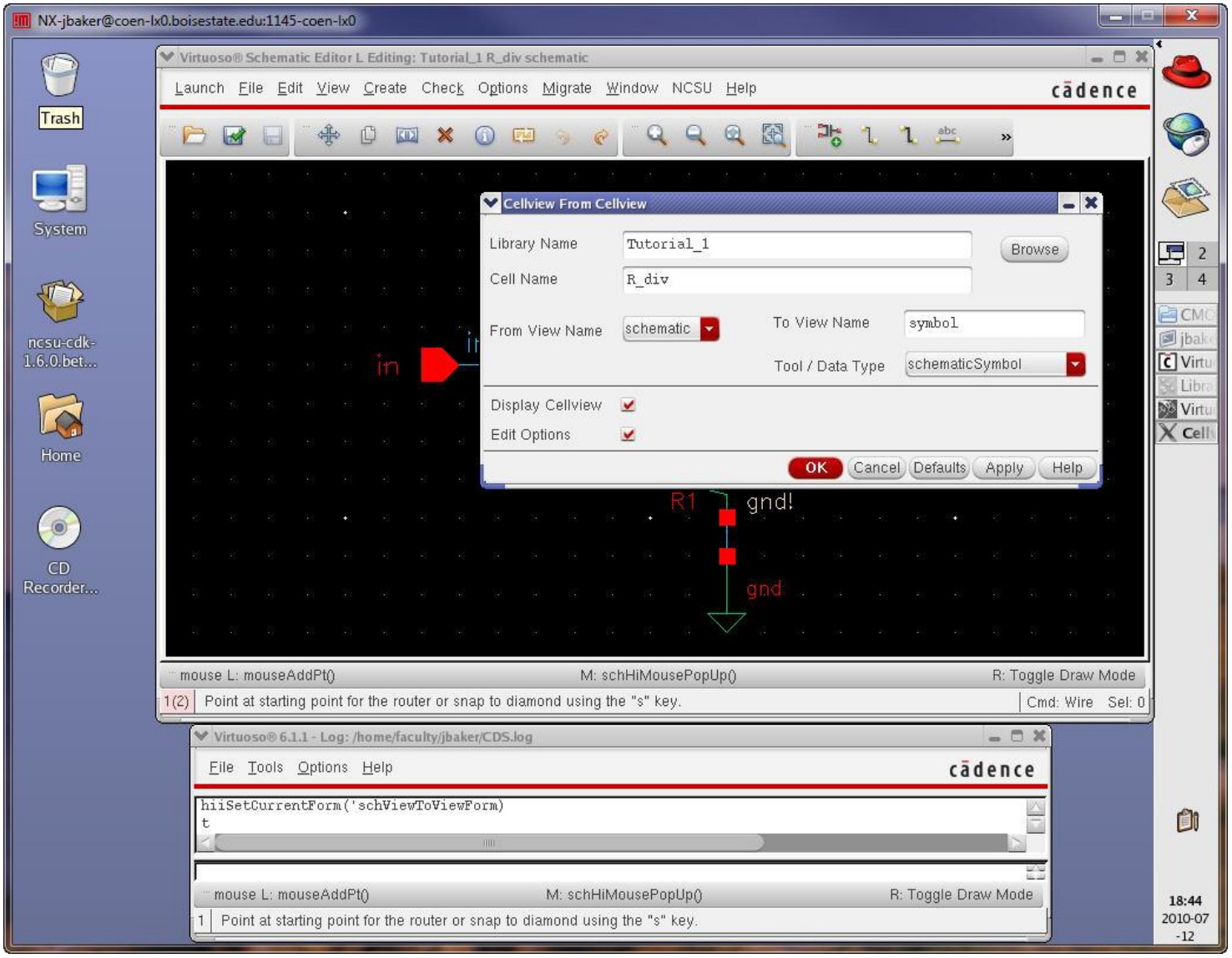


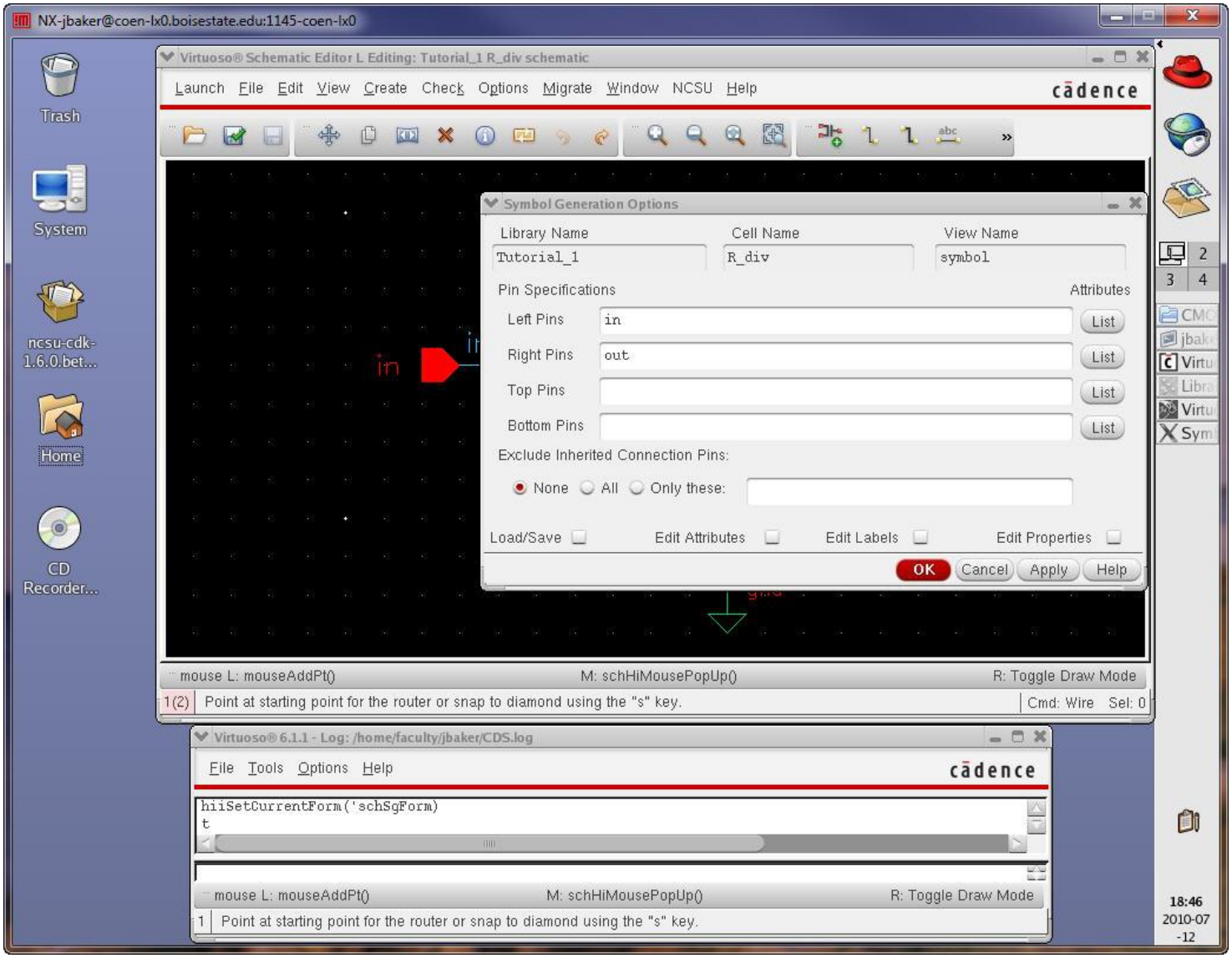
Adding a wire to the pin and "Checking and Saving" results in the following.



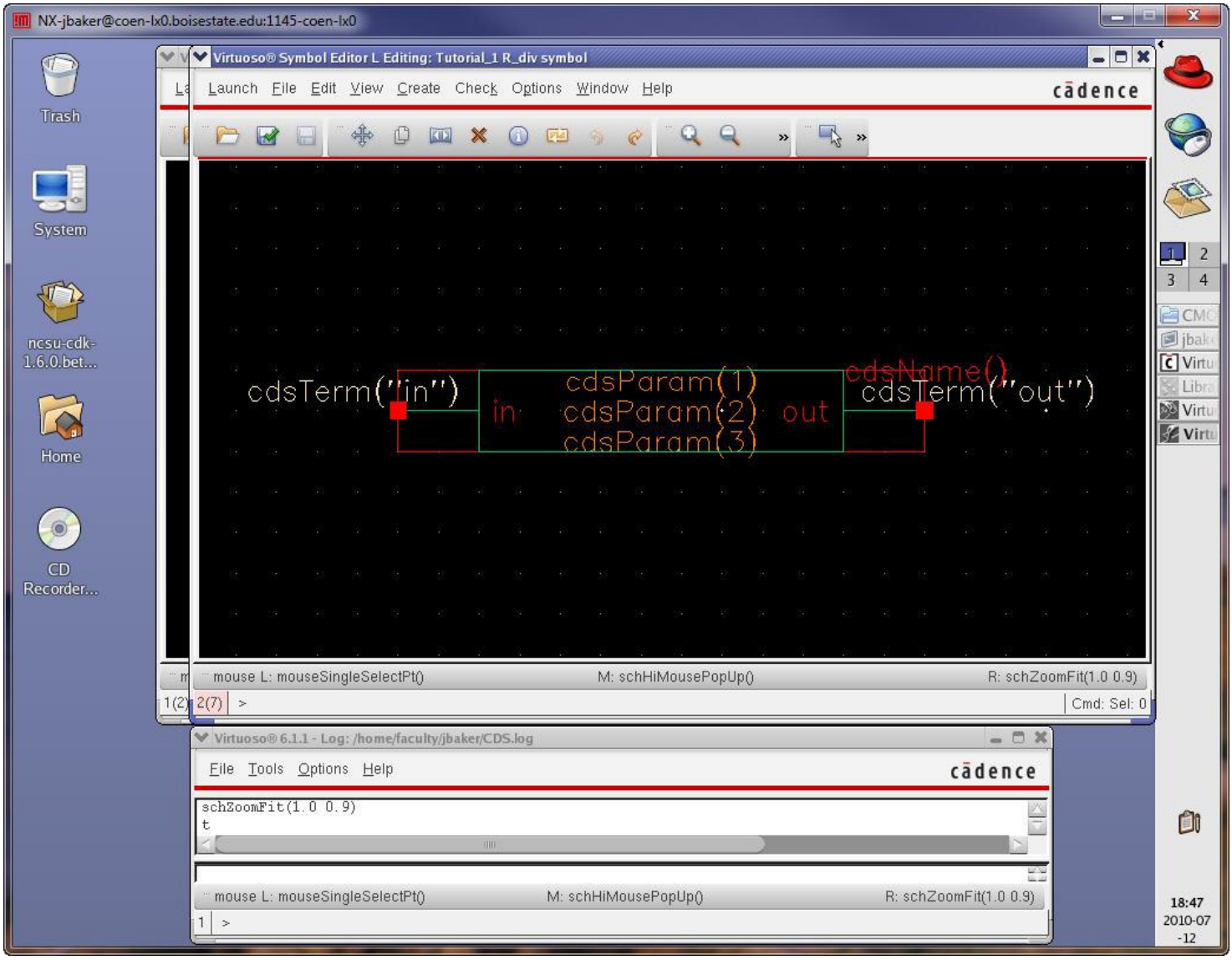
Let's create a symbol for this schematic following the steps seen below.


NX-jbaker@coen-lx0.boisestate.edu:1145-coen-lx0
Trash
System
ncsu-cdk-1.6.0.bet...
Home
CD Recorder...
Virtuoso® Schematic Editor L Editing: Tutorial_1 R_div schematic
Launch File Edit View Create Check Options Migrate Window NCSU Help
cadence
Instance... i
Wire (narrow) w
Wire (wide) W
Wire Name... l
Net Expression...
Pin... p
Block... l
Cellview
Solder Dot
Note
Patchcord...
Probe
MultiSheet...
From Cellview...
From Pin List...
From Instance...
out
out
r=10K
R1
gnd!
gnd
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Toggle Draw Mode
1(2) Point at starting point for the router or snap to diamond using the "s" key.
Cmd: Wire Sel: 0
Virtuoso® 6.1.1 - Log: /home/faculty/jbaker/CDS.log
File Tools Options Help
cadence
schZoomFit(1.0 0.9)
t
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Toggle Draw Mode
1 Point at starting point for the router or snap to diamond using the "s" key.
18:42
2010-07-12

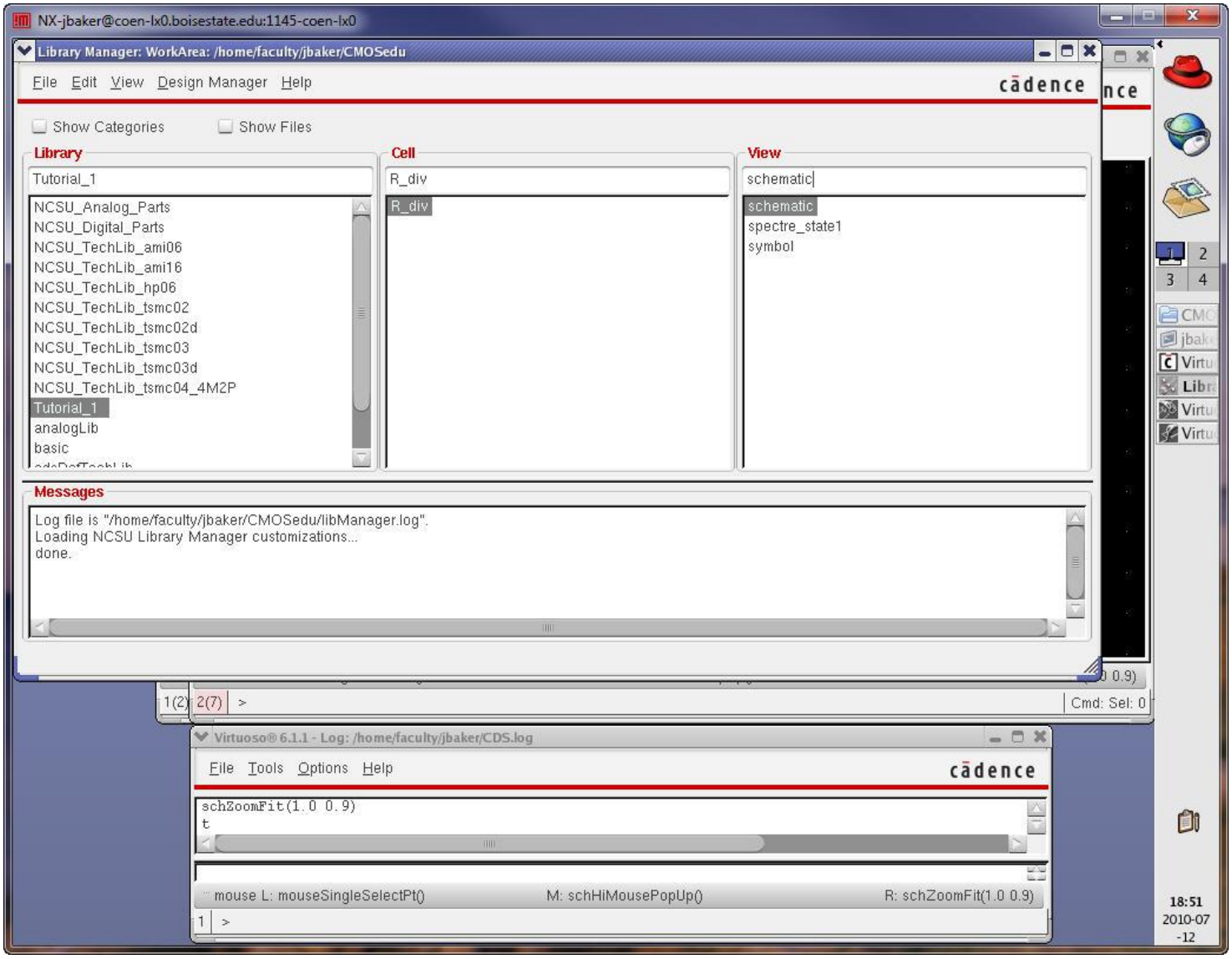

NX-jbaker@coen-lx0.boisestate.edu:1145-coen-lx0
Trash
System
ncsu-cdk-1.6.0.bet...
Home
CD Recorder...
Virtuoso® Schematic Editor L Editing: Tutorial_1 R_div schematic
Launch File Edit View Create Check Options Migrate Window NCSU Help
cadence
Cellview From Cellview
Library Name
Tutorial_1
Browse
Cell Name
R_div
From View Name
schematic
To View Name
symbol
Tool / Data Type
schematicSymbol
Display Cellview
Edit Options
OK Cancel Defaults Apply Help
in
R1
gnd!
gnd
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Toggle Draw Mode
1(2) Point at starting point for the router or snap to diamond using the "s" key.
Cmd: Wire Sel: 0
Virtuoso® 6.1.1 - Log: /home/faculty/jbaker/CDS.log
File Tools Options Help
cadence
hiiSetCurrentForm('schViewToViewForm)
t
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Toggle Draw Mode
1 Point at starting point for the router or snap to diamond using the "s" key.
18:44
2010-07-12


NX-jbaker@coen-lx0.boisestate.edu:1145-coen-lx0
Trash
System
ncsu-cdk-1.6.0.bet...
Home
CD Recorder...
Virtuoso® Schematic Editor L Editing: Tutorial_1 R_div schematic
Launch File Edit View Create Check Options Migrate Window NCSU Help
cadence
Symbol Generation Options
Library Name
Cell Name
View Name
Tutorial_1
R_div
symbol
Pin Specifications
Attributes
Left Pins
in
Right Pins
out
Top Pins
Bottom Pins
List
Exclude Inherited Connection Pins:
None All Only these:
Load/Save
Edit Attributes
Edit Labels
Edit Properties
OK Cancel Apply Help
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Toggle Draw Mode
1(2) Point at starting point for the router or snap to diamond using the "s" key.
Cmd: Wire Sel: 0
Virtuoso® 6.1.1 - Log: /home/faculty/jbaker/CDS.log
File Tools Options Help
cadence
hiiSetCurrentForm('schSgForm)
t
mouse L: mouseAddPt()
M: schHiMousePopUp()
R: Toggle Draw Mode
1 Point at starting point for the router or snap to diamond using the "s" key.
18:46
2010-07-12

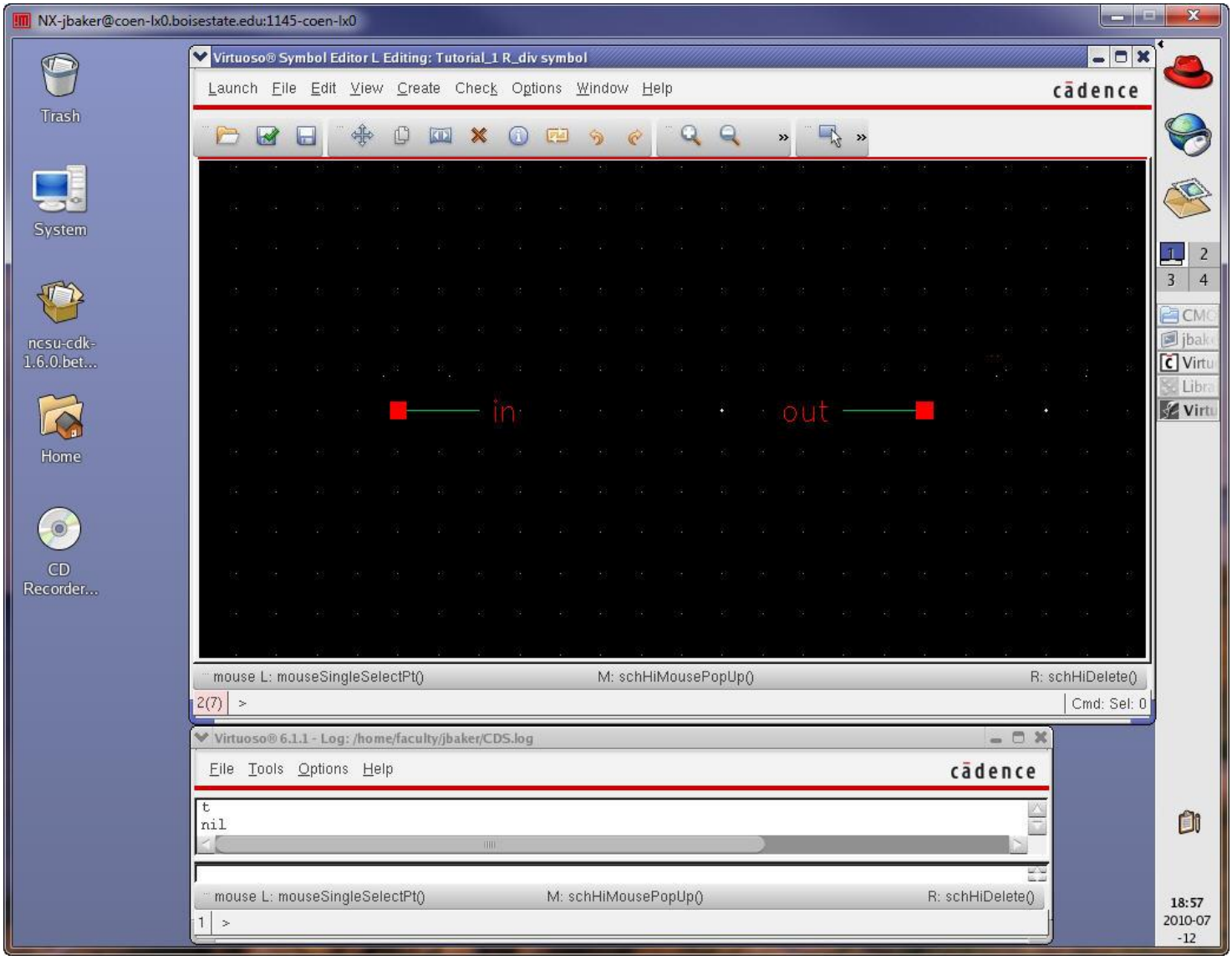
And finally,



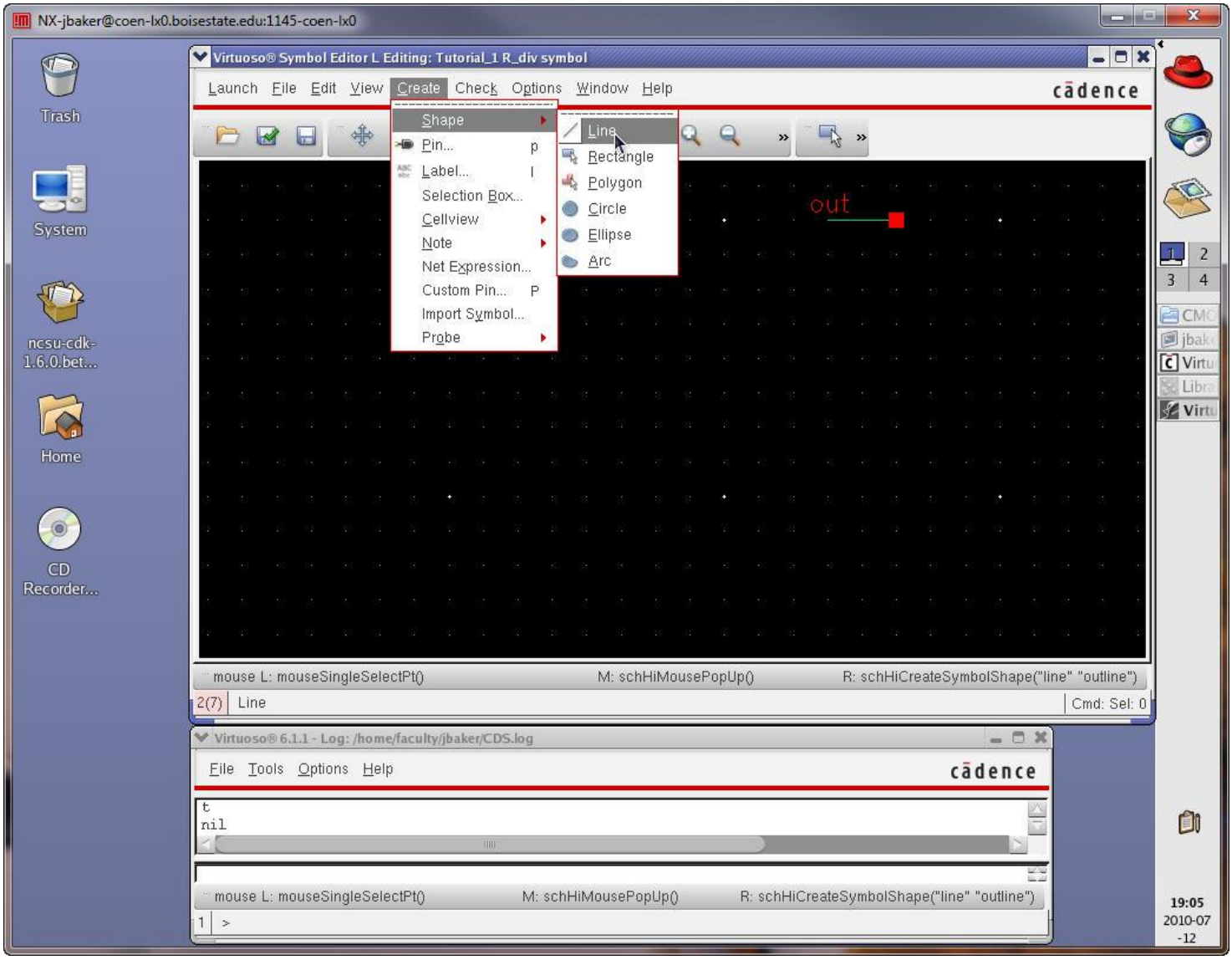
If we look at the Library Manager we see three Cell Views for the R_div group remembering that the spectre_state1 Cell View contains our simulation parameters
which we load when the ADE is started as mentioned above.



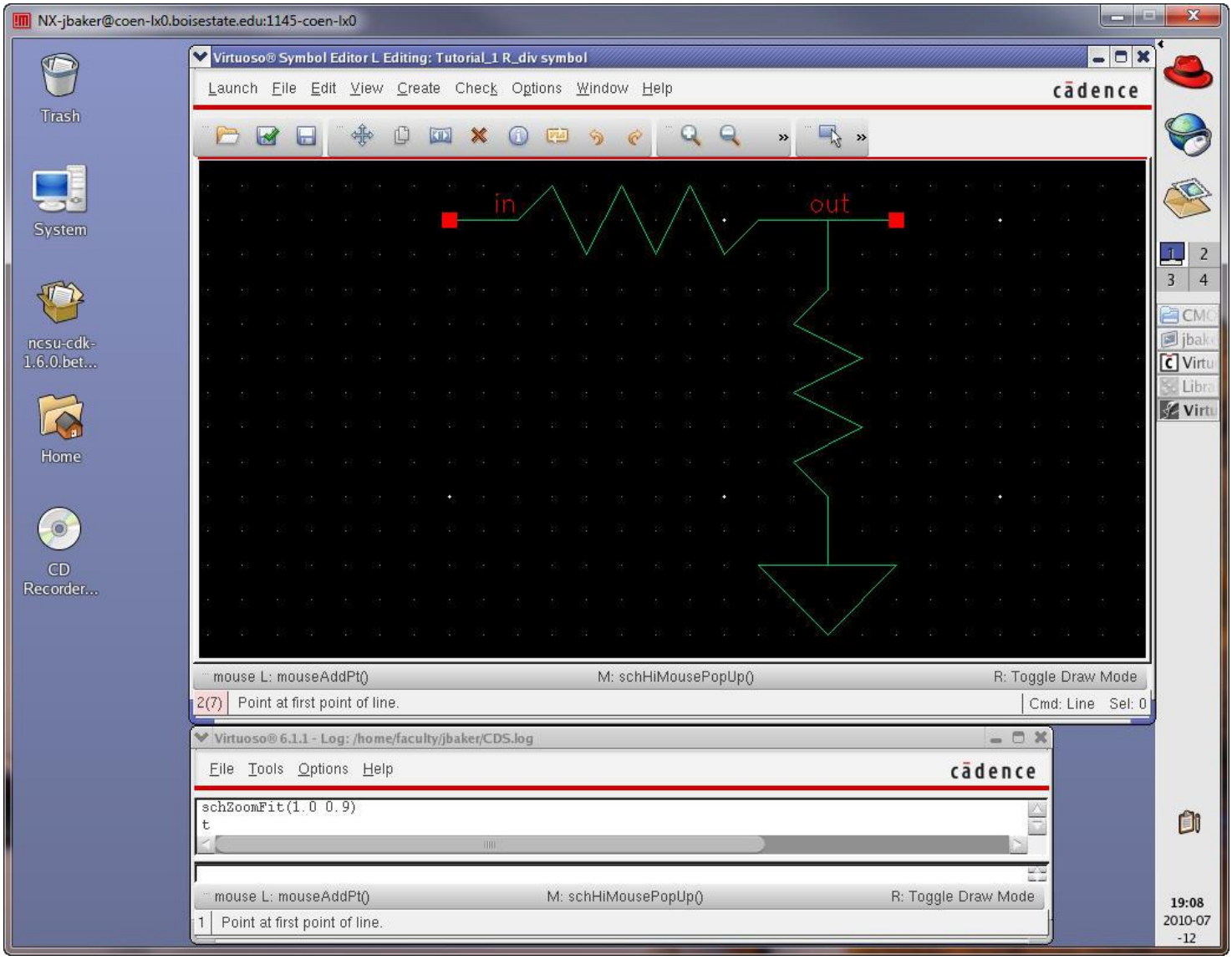
Let's delete everything in the symbol view except for the following. Remember that you can always use the undo command (**u**) if you accidentally delete something.



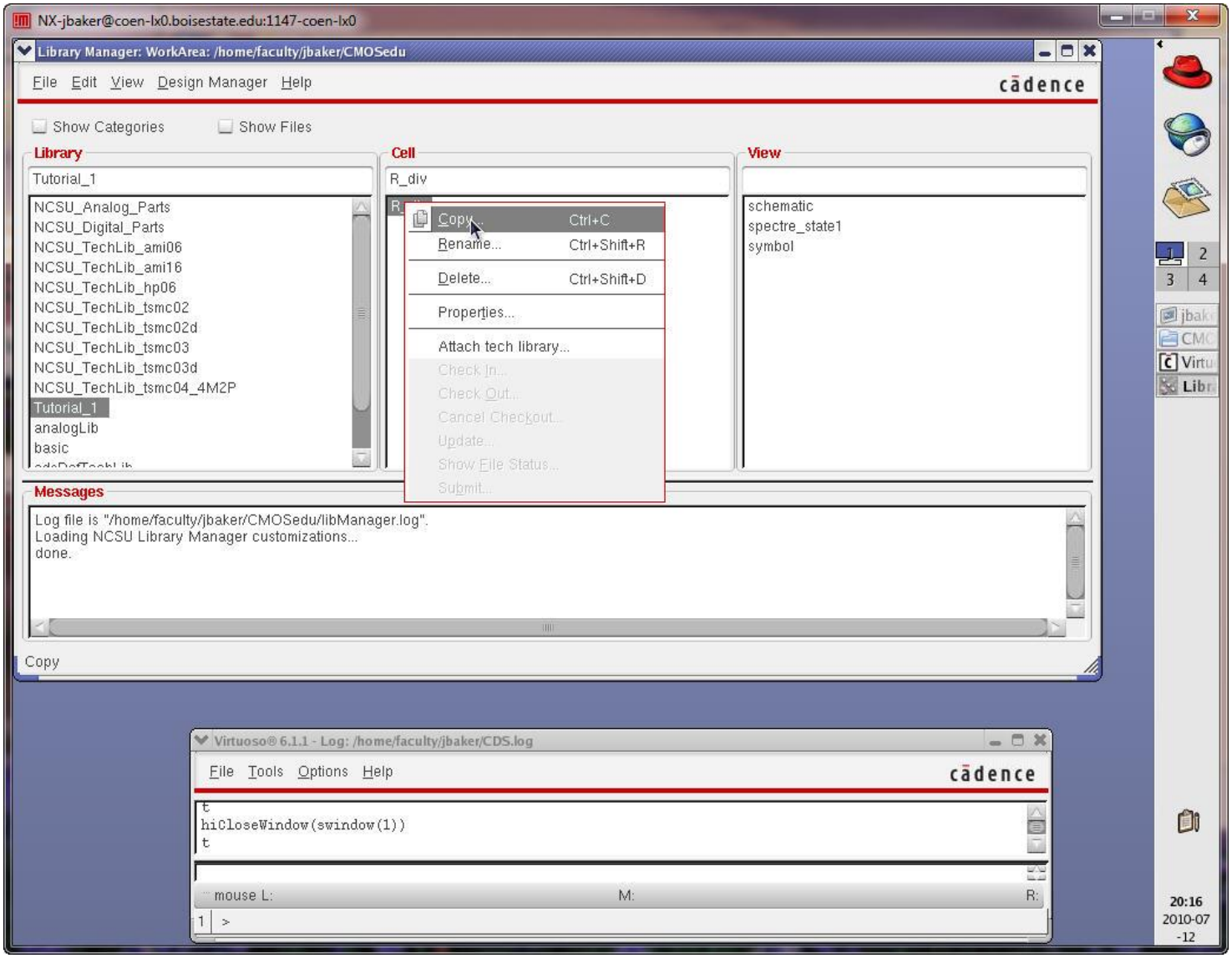
Move the text and then add a line.

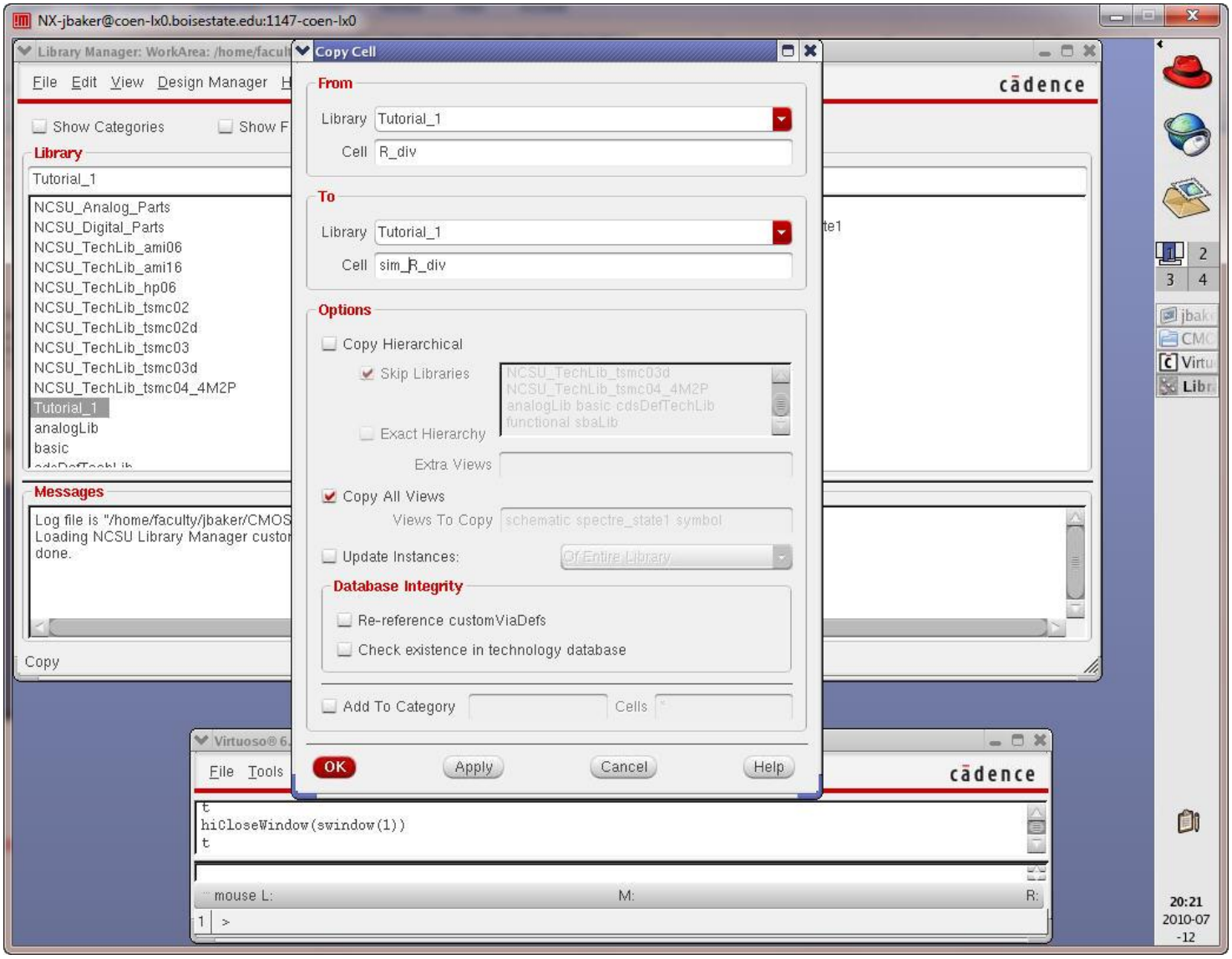


Draw the symbol seen below. Note that **Z** (zoom out by 2) and **f** (fit) can be useful when drawing the symbol.

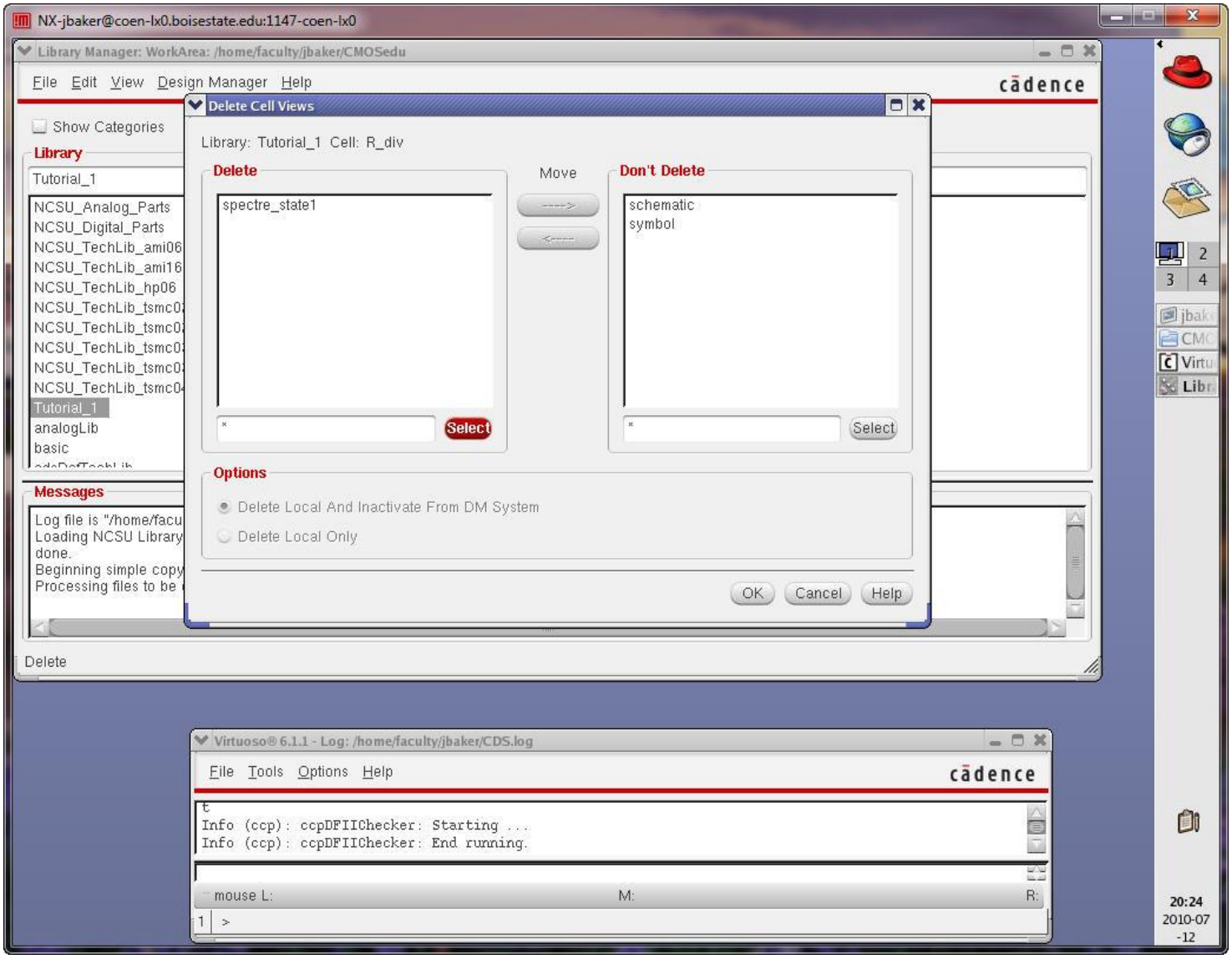


When finished Check and Save the symbol view. Close both the symbol and schematic views of the R_div cell. Before doing the layout for the R_div cell let's
simulate the cell's operation again. To speed things up copy R_div into another cell called sim_R_div (right click on the cell name).

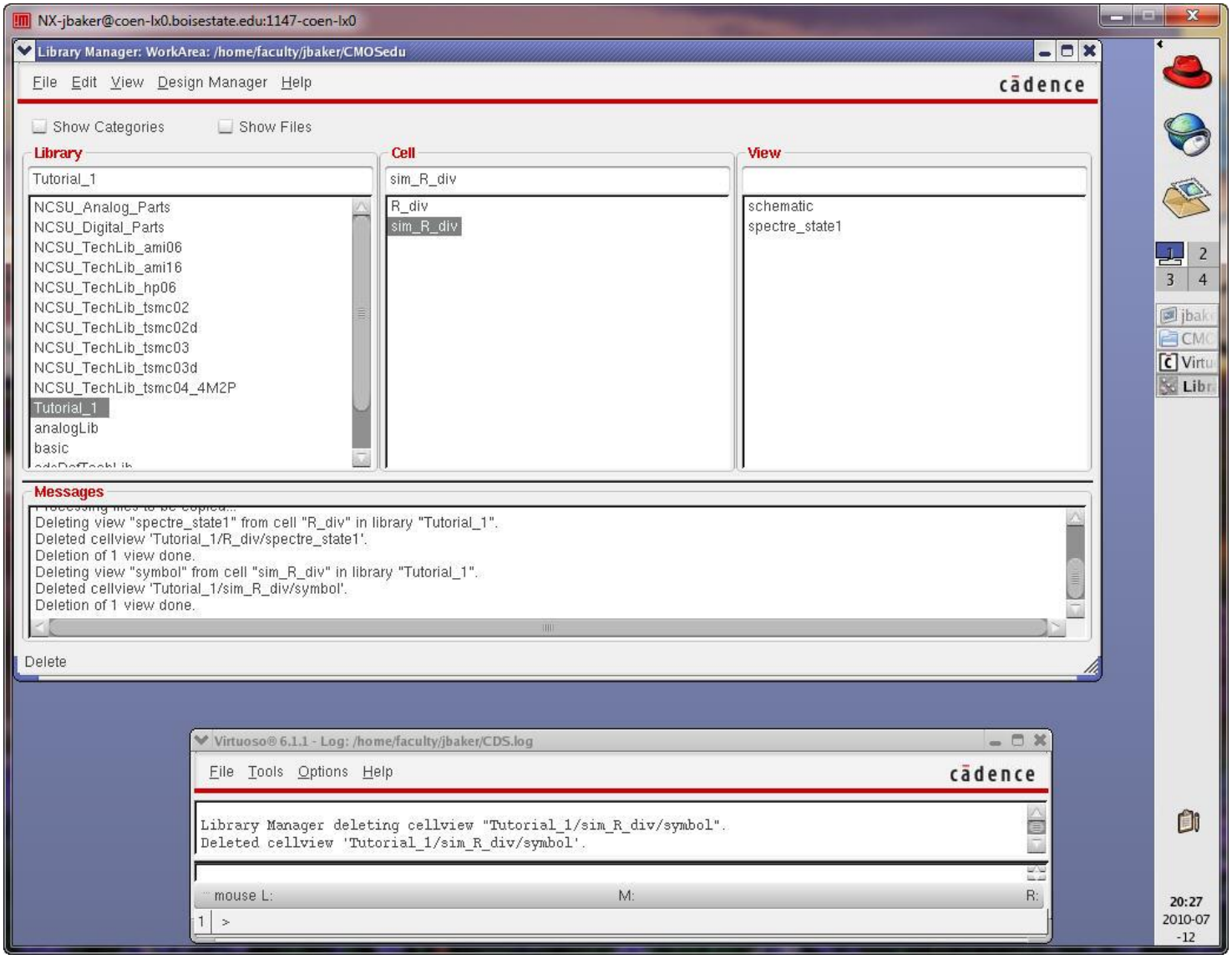

NX-jbaker@coen-lx0.boisestate.edu:1147-coen-lx0
Library Manager: WorkArea: /home/faculty/jbaker/CMOSedu
File Edit View Design Manager Help
cadence
Show Categories
Show Files
Library
Cell
View
Tutorial_1
R_div
NCSU_Analog_Parts
NCSU_Digital_Parts
NCSU_TechLib_ami06
NCSU_TechLib_ami16
NCSU_TechLib_hp06
NCSU_TechLib_tsmc02
NCSU_TechLib_tsmc02d
NCSU_TechLib_tsmc03
NCSU_TechLib_tsmc03d
NCSU_TechLib_tsmc04_4M2P
Tutorial_1
analogLib
basic
Copy... Ctrl+C
Rename... Ctrl+Shift+R
Delete... Ctrl+Shift+D
Properties...
Attach tech library...
Check In...
Check Out...
Cancel Checkout...
Update...
Show File Status...
Submit...
schematic
spectre_state1
symbol
Messages
Log file is "/home/faculty/jbaker/CMOSedu/libManager.log".
Loading NCSU Library Manager customizations...
done.
Copy
Virtuoso® 6.1.1 - Log: /home/faculty/jbaker/CDS.log
File Tools Options Help
cadence
t
hiCloseWindow(swindow(1))
t
mouse L:
M:
R:
1 >
20:16
2010-07
-12



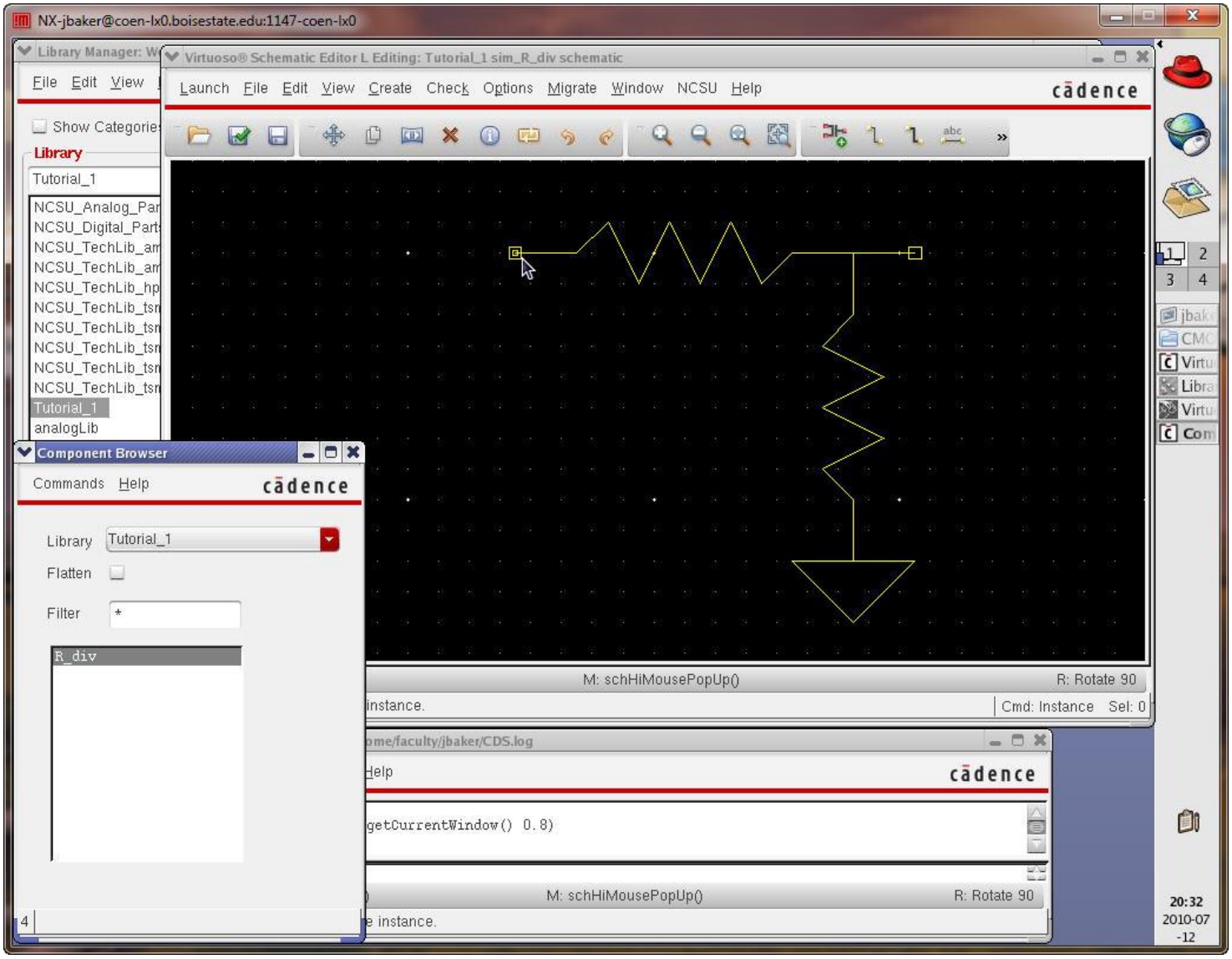
After hitting OK delete the spectre_saved1 in the R_div cell by right clicking on the name in the View category.



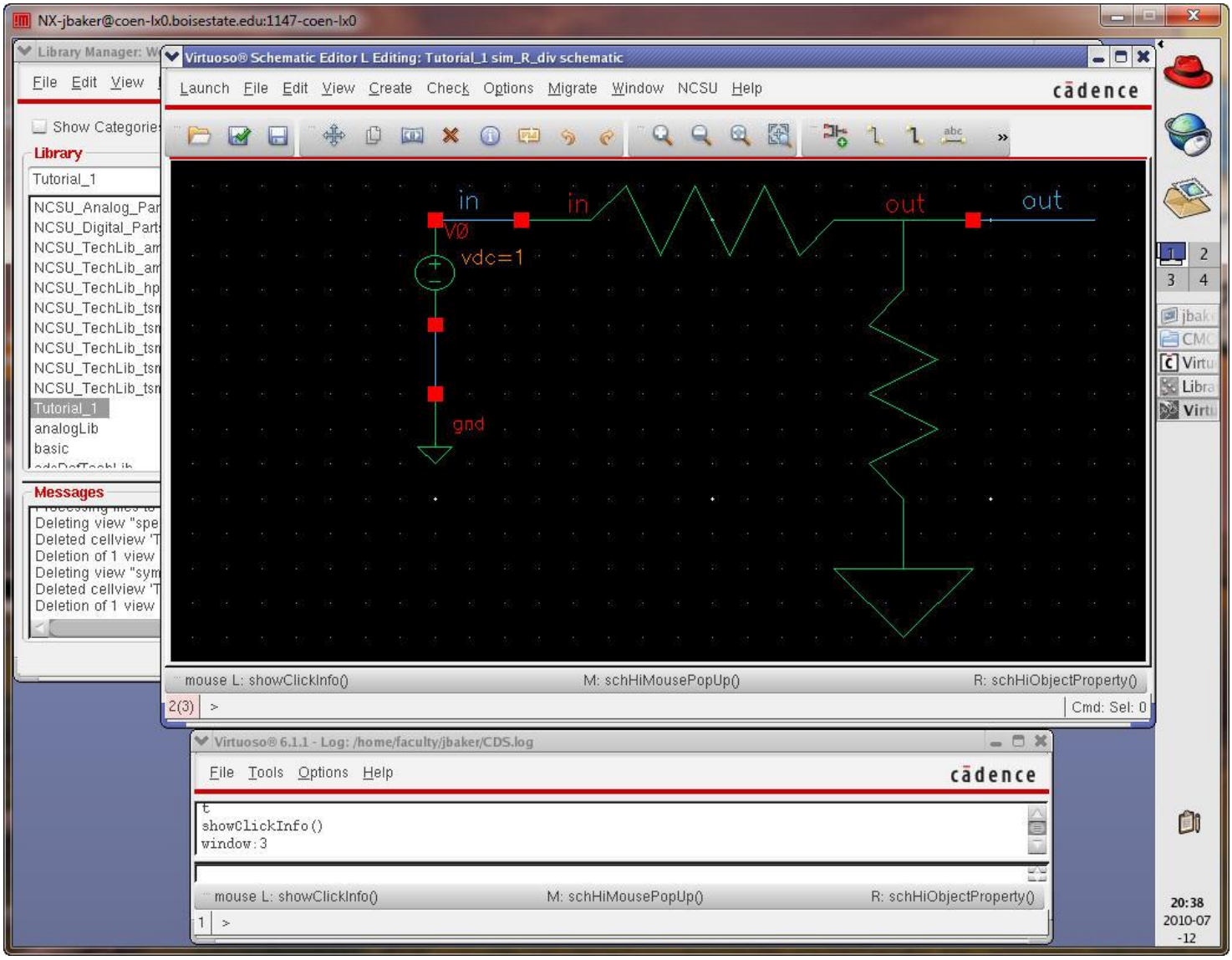
After hitting OK and Yes you are sure delete the symbol view in the sim_R_div cell.



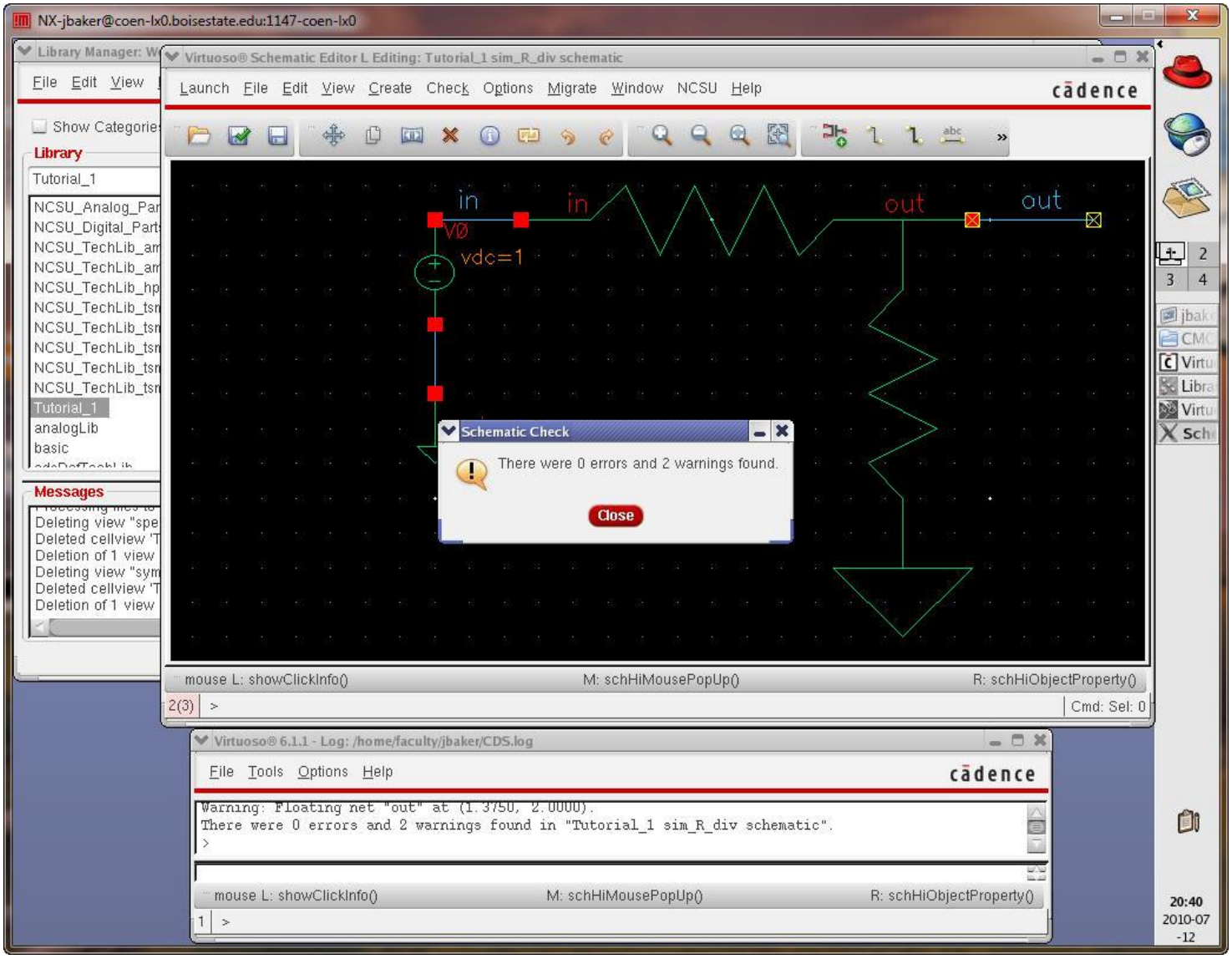
Next open the schematic view of the sim_R_div cell and delete everything in the cell.
Press i to add the symbol for R_div in the Tutorial_1 library.
After you place the cell hit Esc to leave the place instance command mode.



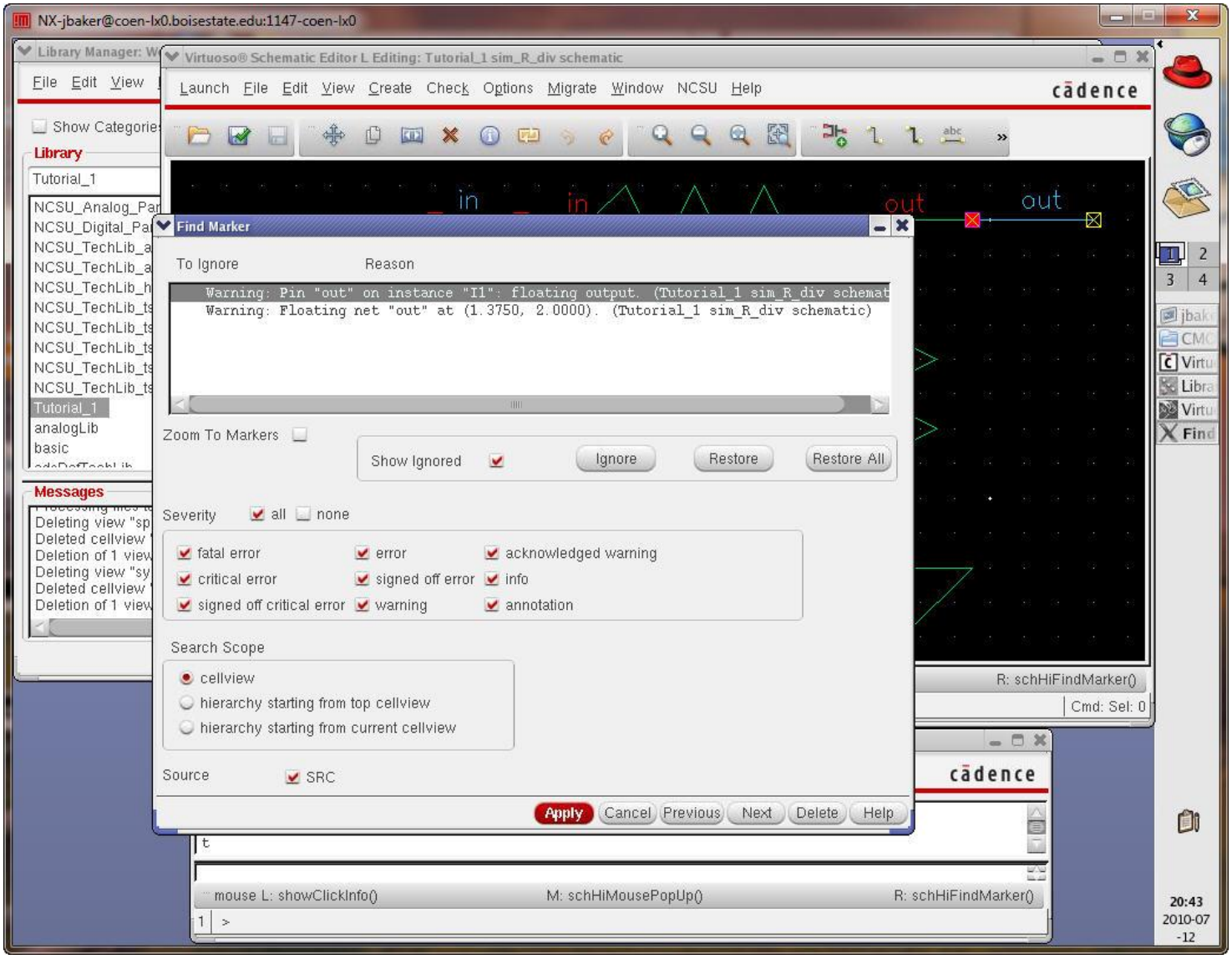
Next add wires, wire names, and the input voltage like we did above.
Set the input voltage to 1 V DC.



When finished Check and Save the schematic.
Note we get two warnings.

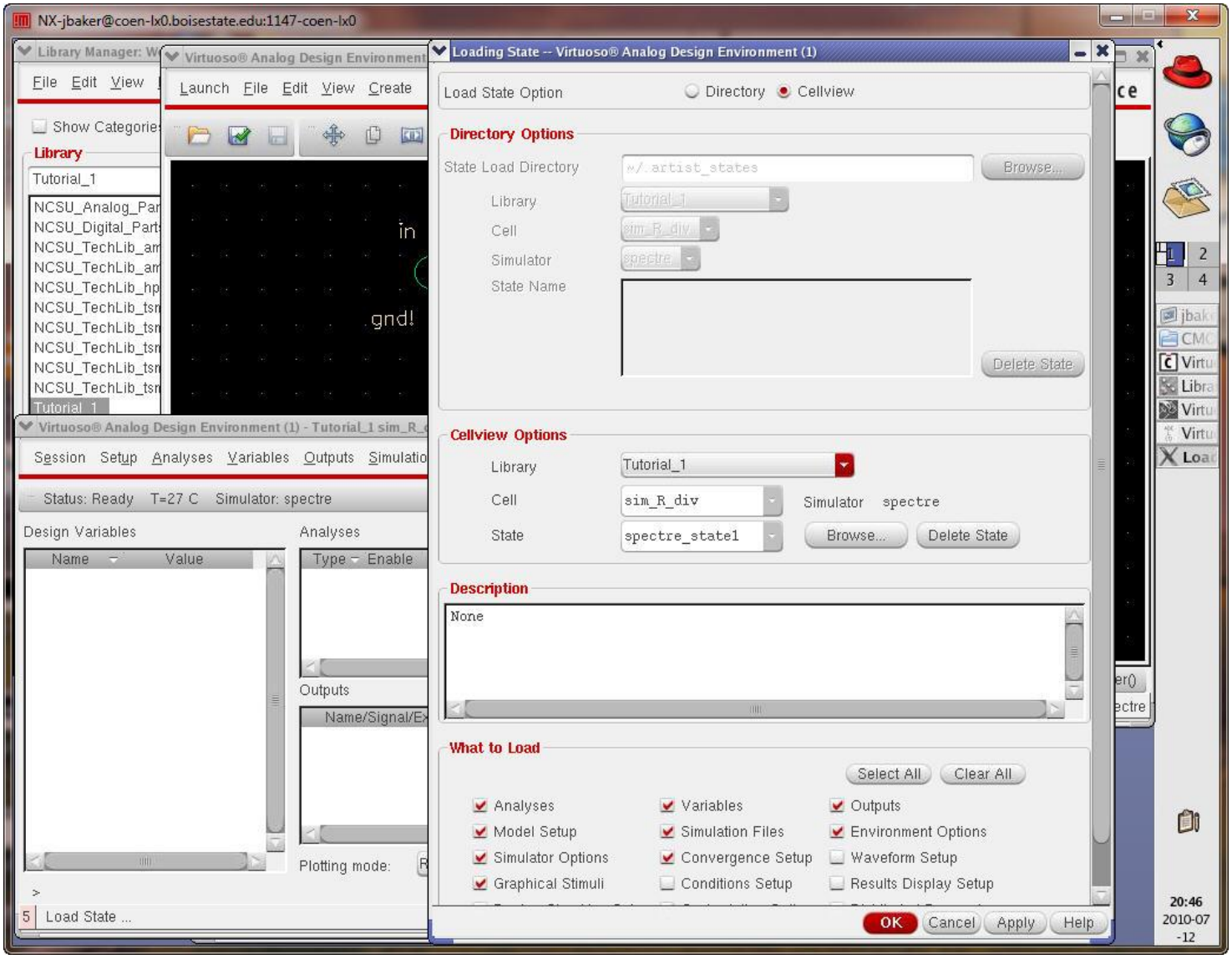


The warnings are from the floating wire named out.
Go to the menu items Check -> Find Marker to see the details.

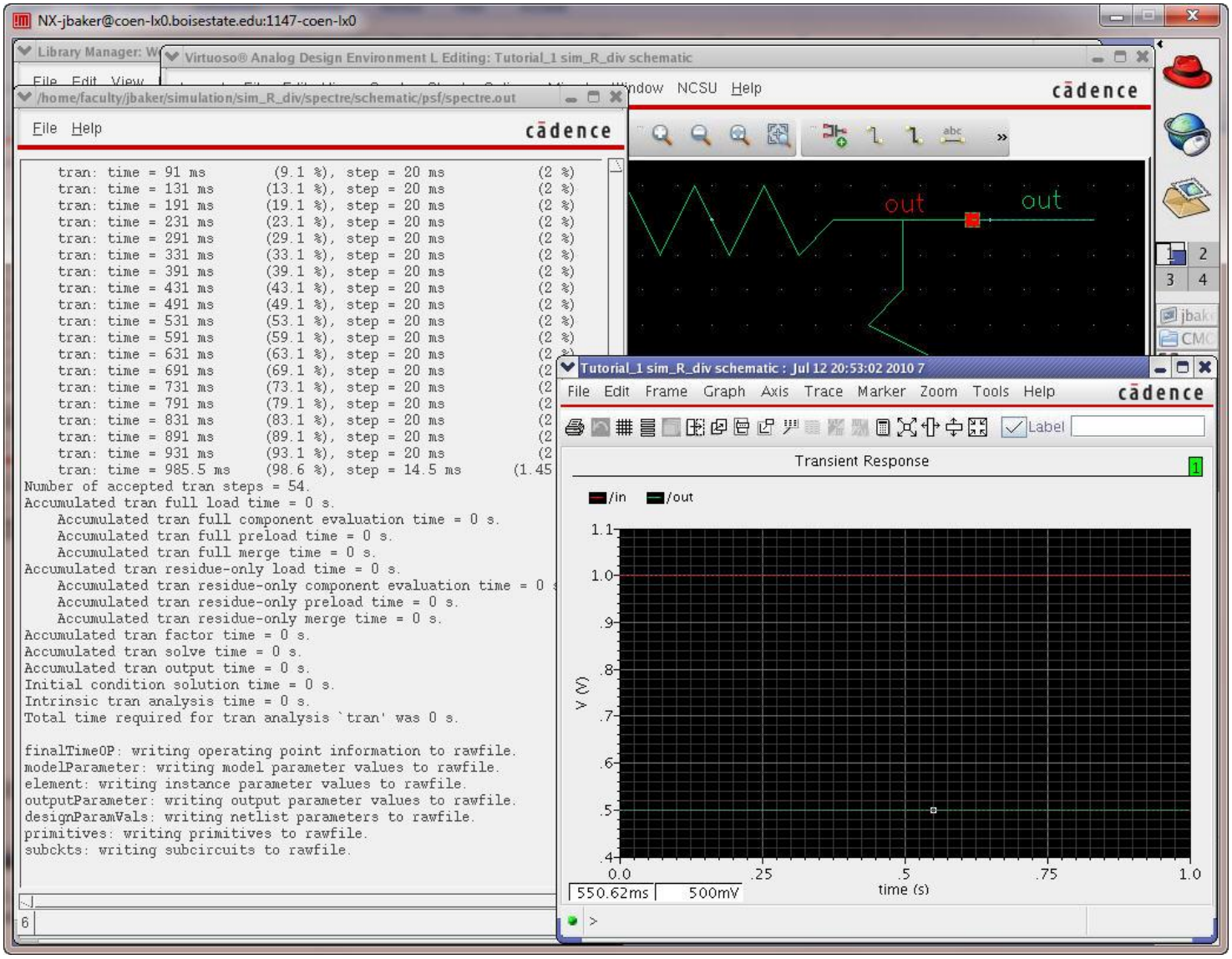


We are okay with this wire floating so press ignore twice and then close the Find Marker window.
Check and Save the schematic again. There should be no warnings or errors.

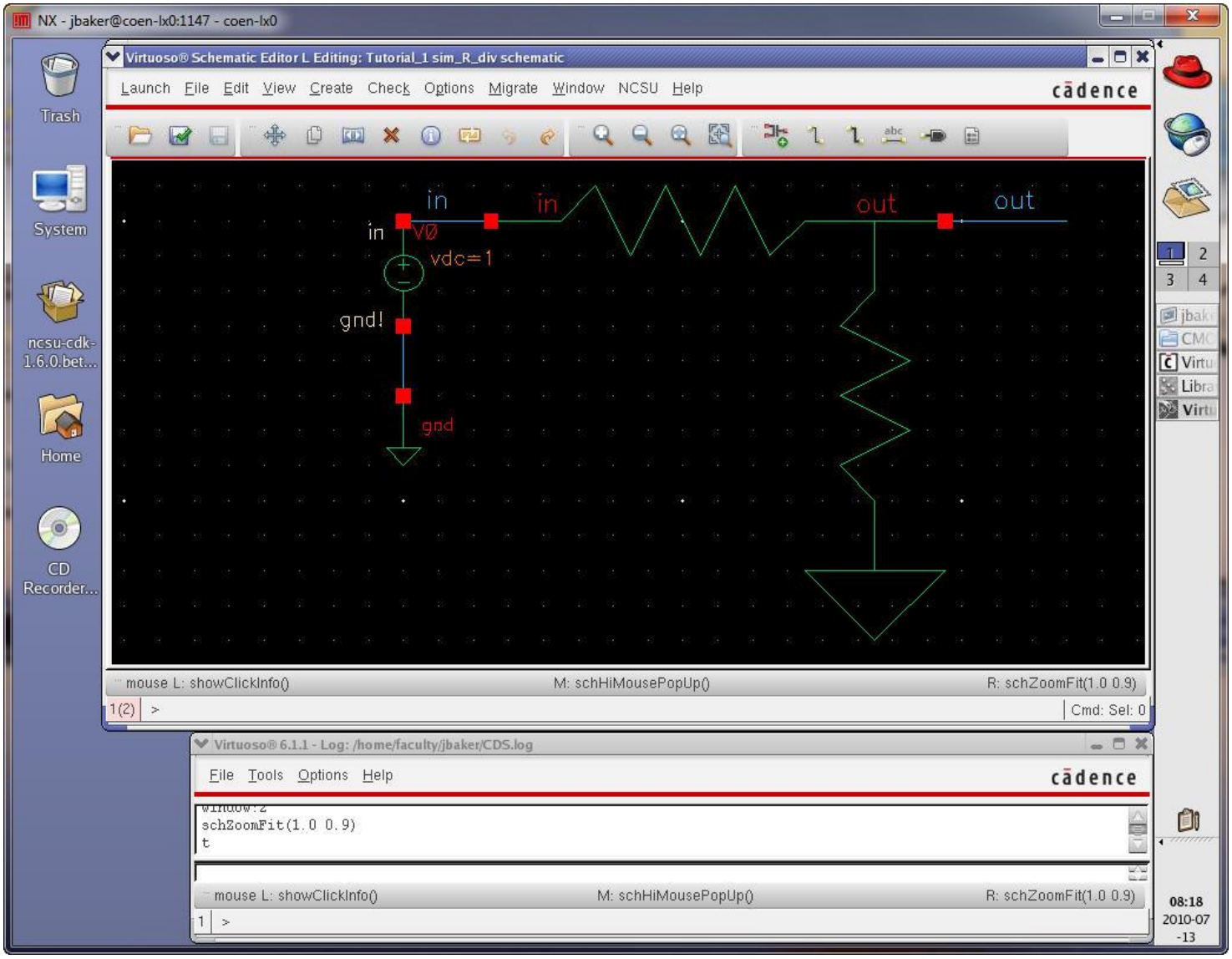
Now let's simulate this circuit. Launch the ADE and then load the state (Cellview…important).



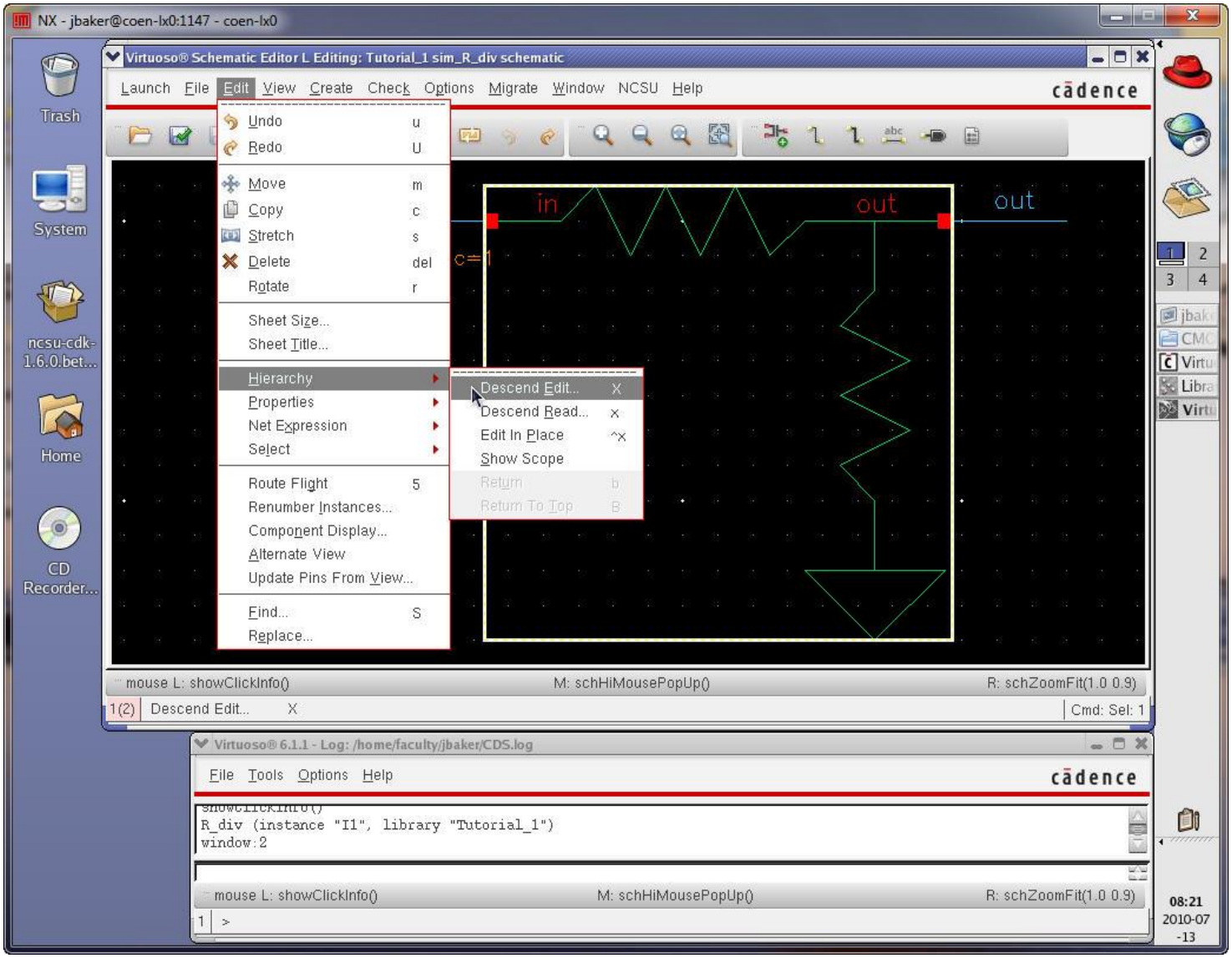
While the type of analyses will be remembered the outputs to be plotted won't be remembered. Use the menu items Outputs -> To Be Plotted -> Select on Schematic
to select the in and out wire nodes. Next use the menu items Session -> Save State to save the state. Hitting the green button to Netlist and Simulate starts
the Spectre simulation.



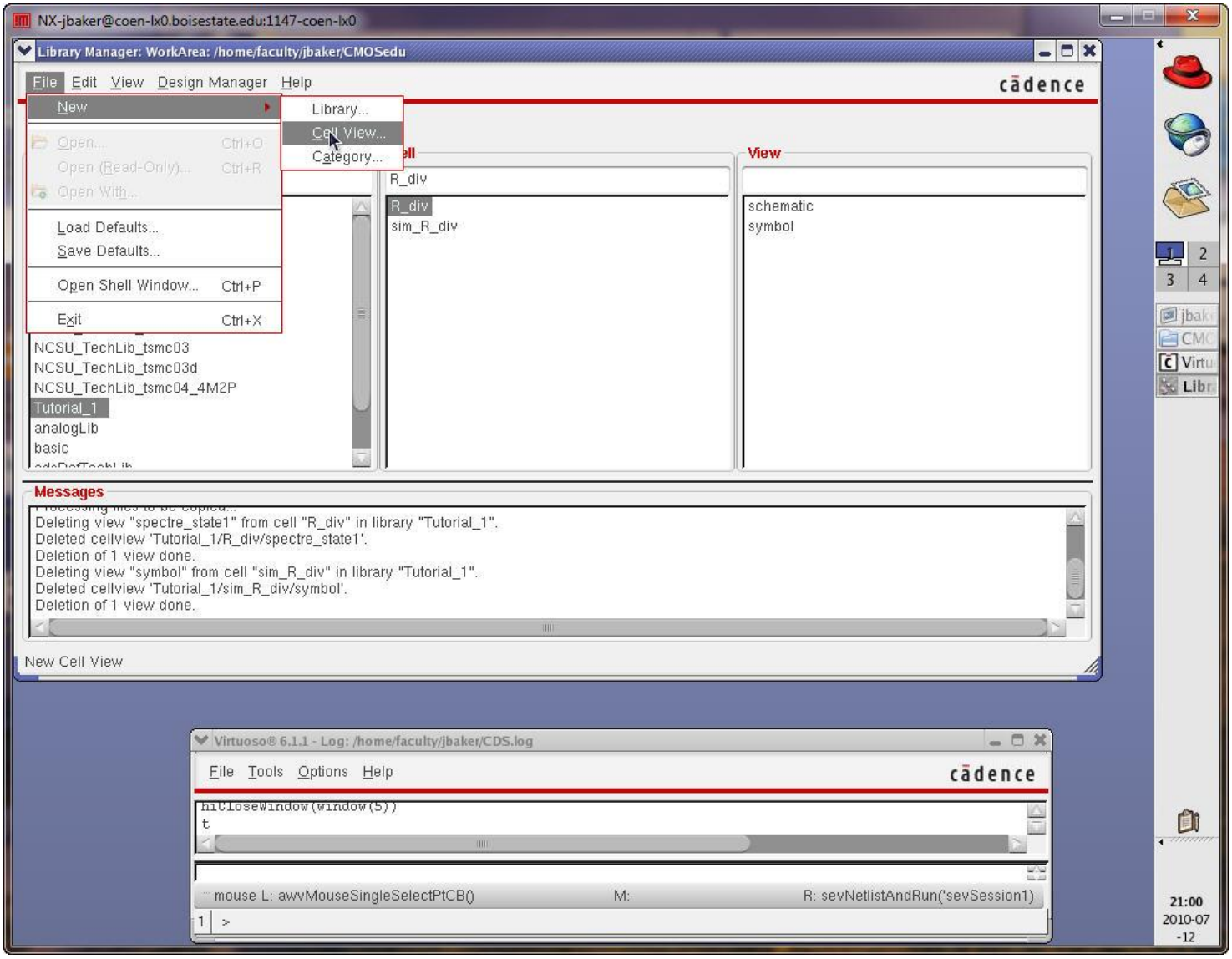
Close the ADE.



Before moving on to the layout lets cover one last item, that is, descending through the hierarchy.
Select the R_div symbol (left+click the mouse button on the symbol) and follow the menu items seen below (or just press **X**).
The contents of the symbol can be edited in the current tab (window) or in a new window. Select current tab and press OK.
Note, also, that you can return back up in the hierarchy by pressing **b** or the menu item seen below (but shaded since we are already at the top).
Descend down into the R_div schematic and then back up to the sim_R_div schematic to get some experience with the commands.



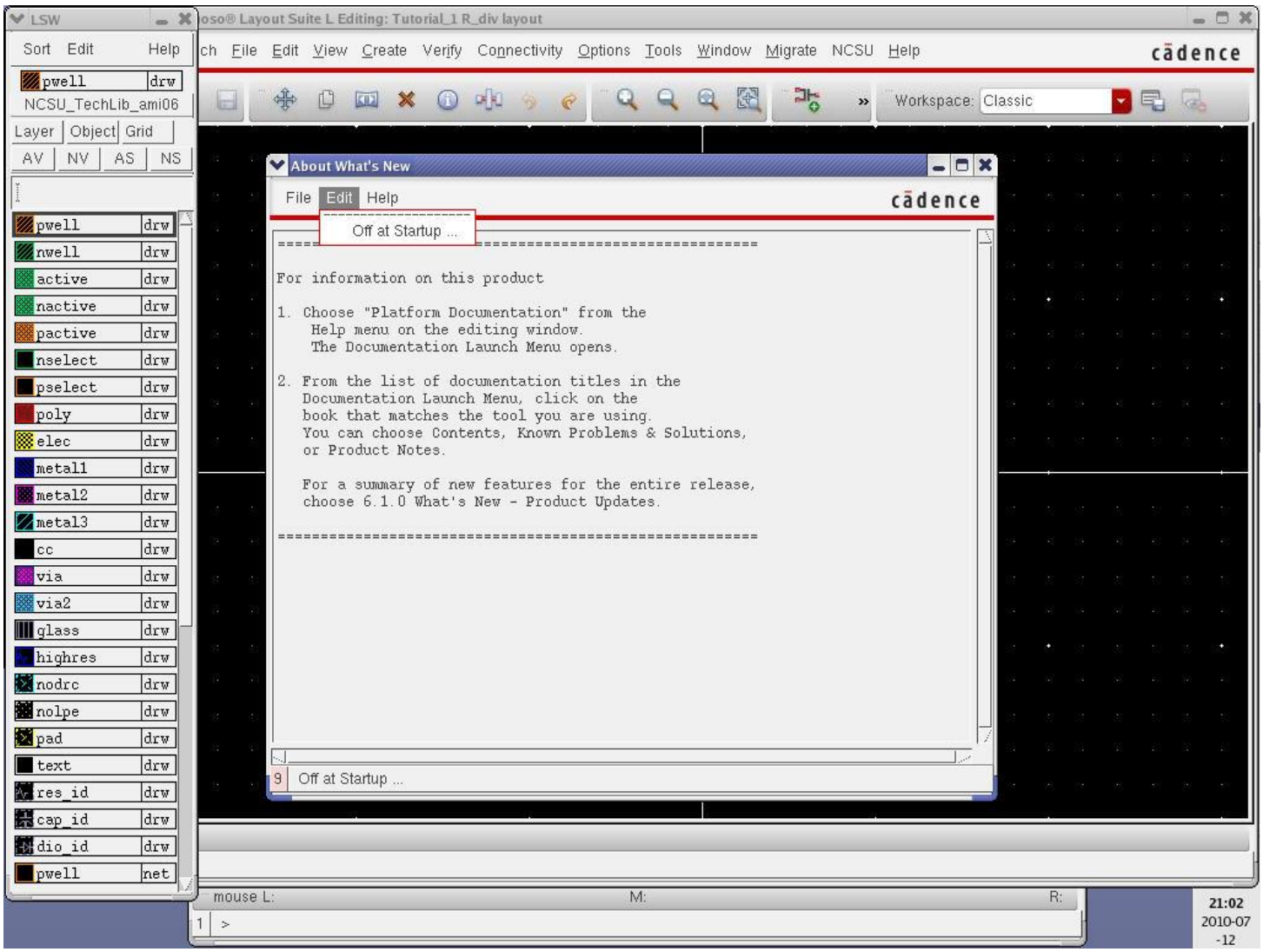
We are now ready to lay out the resistive divider.
To begin make a new Cell View for the layout of R_div.



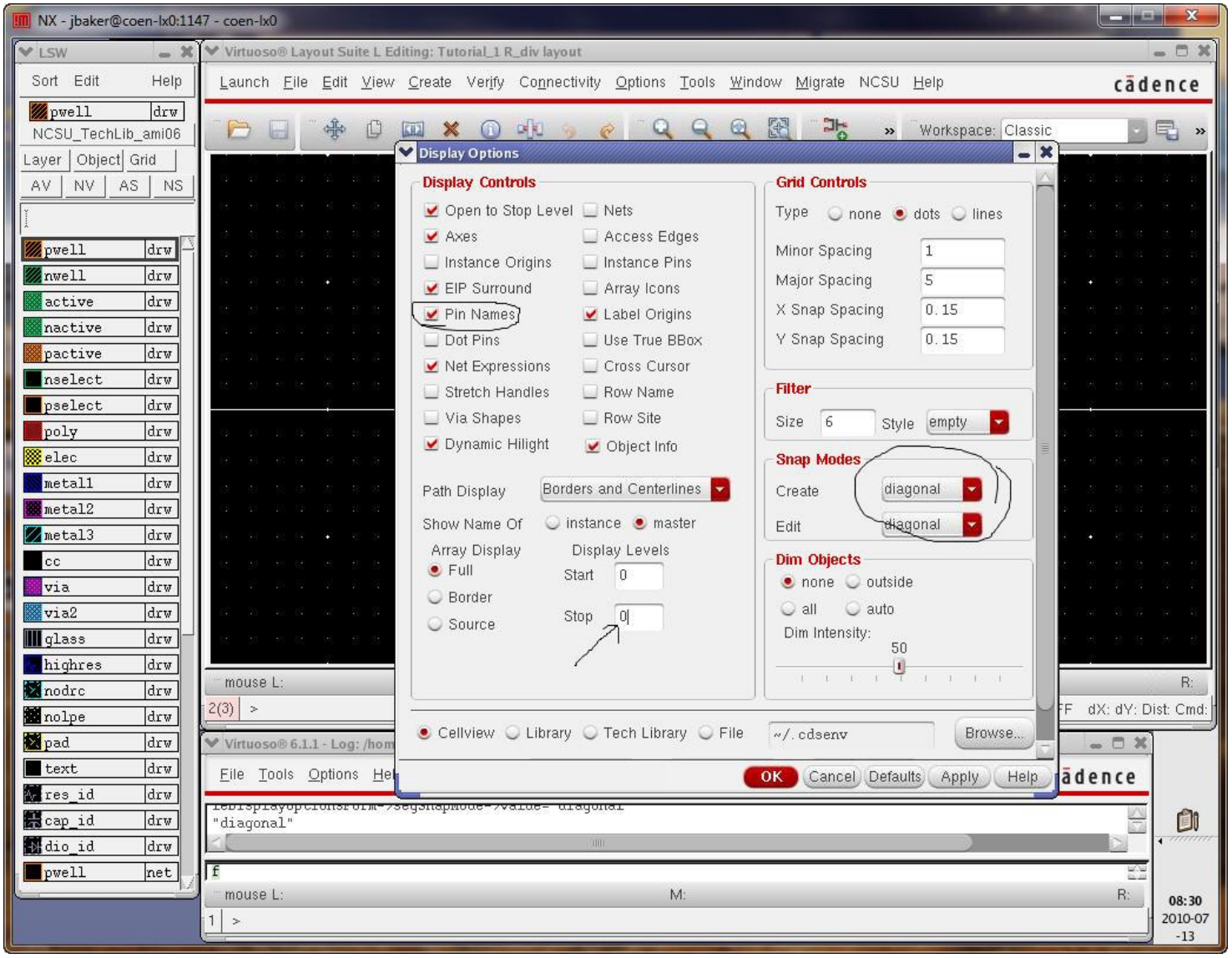
After creating the layout cell view the following will appear.
Select Off at Startup to ensure you don't see this What's New information each time you start-up a layout session.
Note the Layout Selection Window (LSW) that allows you to select specific layers when doing layout.

In the top of the LSW you can select AV (all layers visible) or NV (only the layer selected is visible). After making this selection follow it by re-drawing
the layout window (View -> Redraw) to see the results. AS and NS are used in a similar manner to allow selecting or not selecting specific layers.

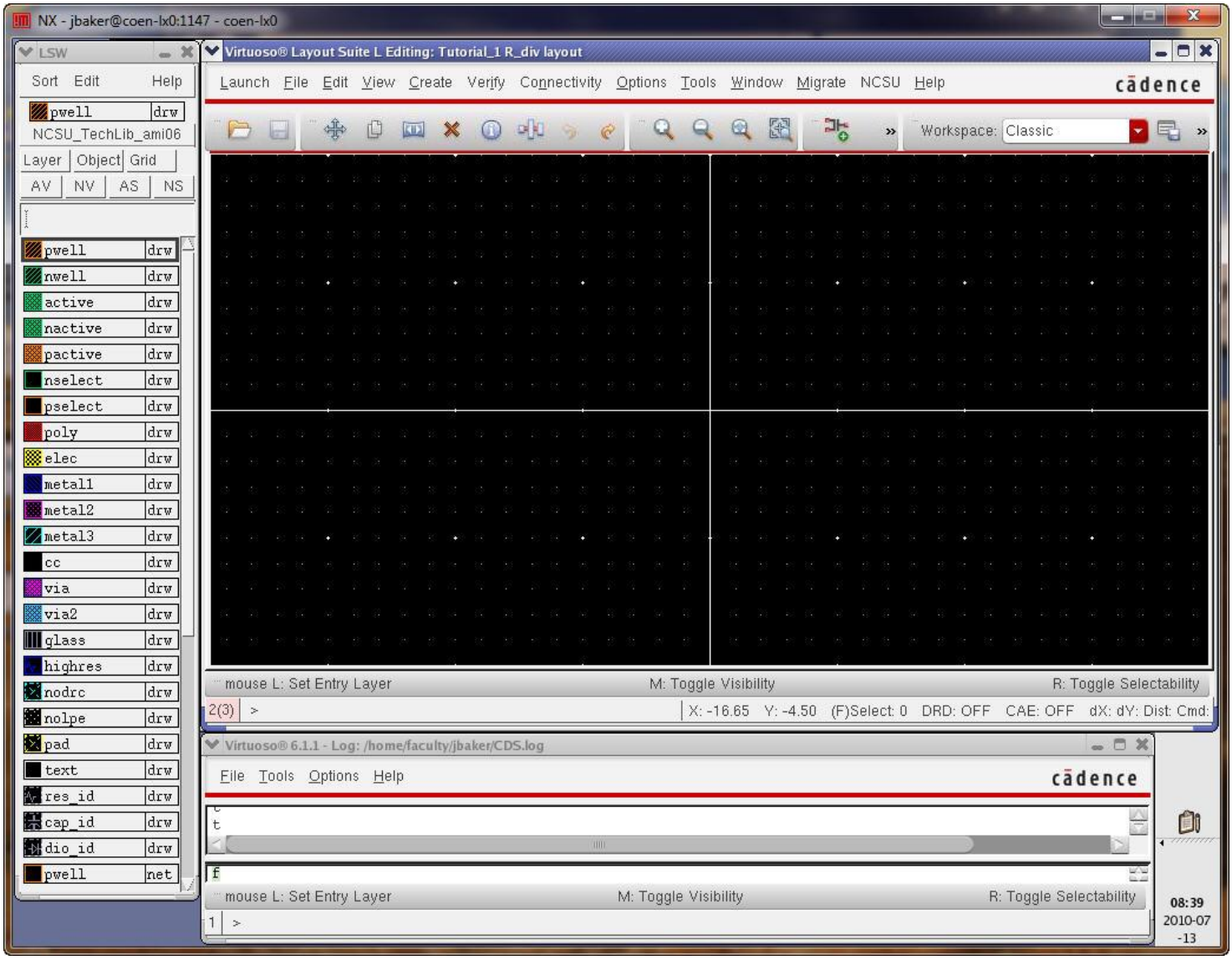
***Note, below, that we use the **drw** layers to draw layouts not the **net** layers, see the top layer is "pwell drw" and the bottom layer is "pwell net." It's
easy to draw shapes, accidentally, using a net layer so if you have an issue with a layout you think should be fine check to see that you haven't done this.***



Next go to the menu items Options -> Display (or just press **e**).
Set the display so that Pin Names are shown.
Change the Snap Modes to diagonal.
Note that the Display Levels has no depth, that is, the layout of a cell placed in another cell will show as an outline. To see the contents we need to increase
the Stop level. We'll do this later just so we can see this window again and what an outline of a cell looks like.



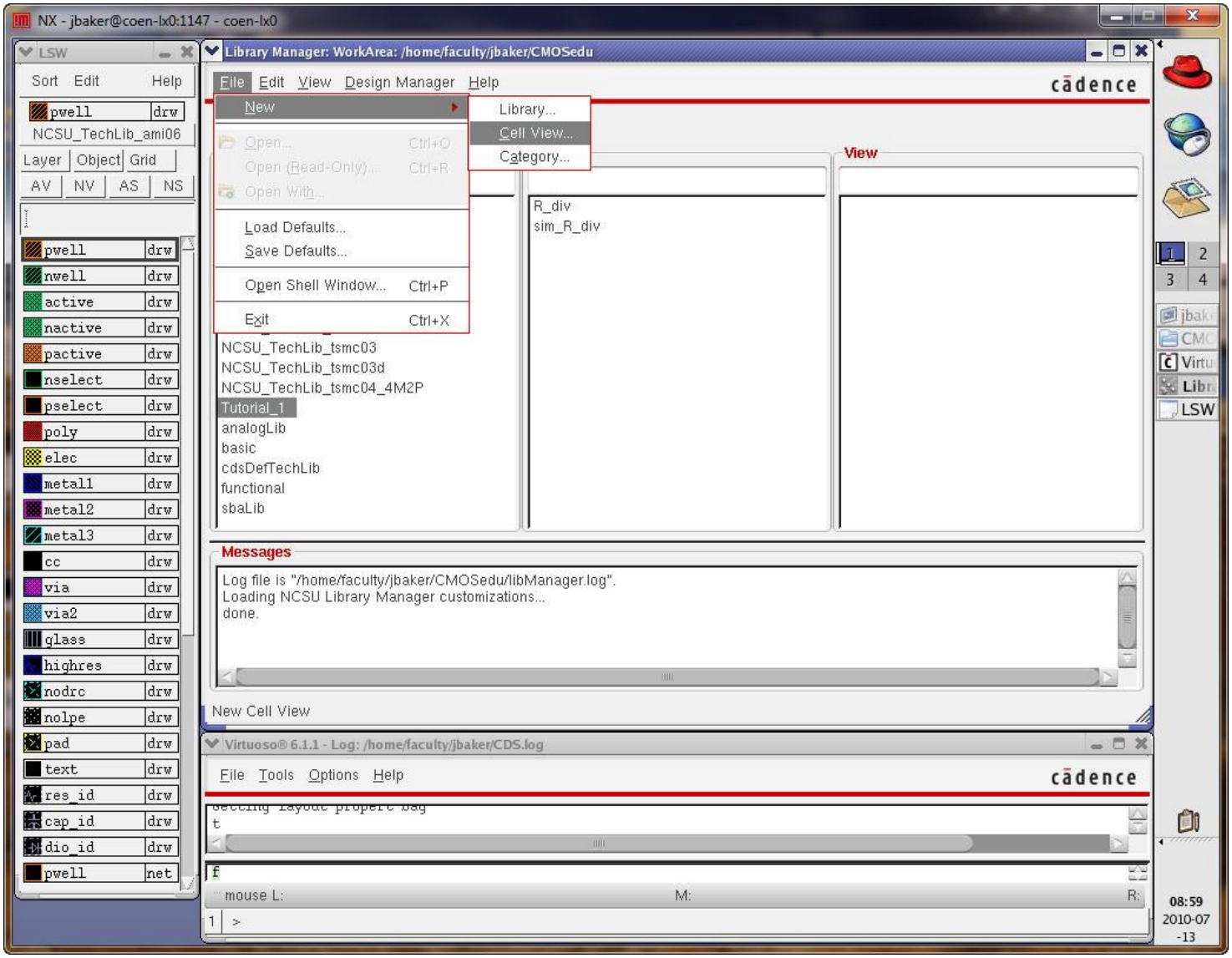
We are now ready to draw some shapes.

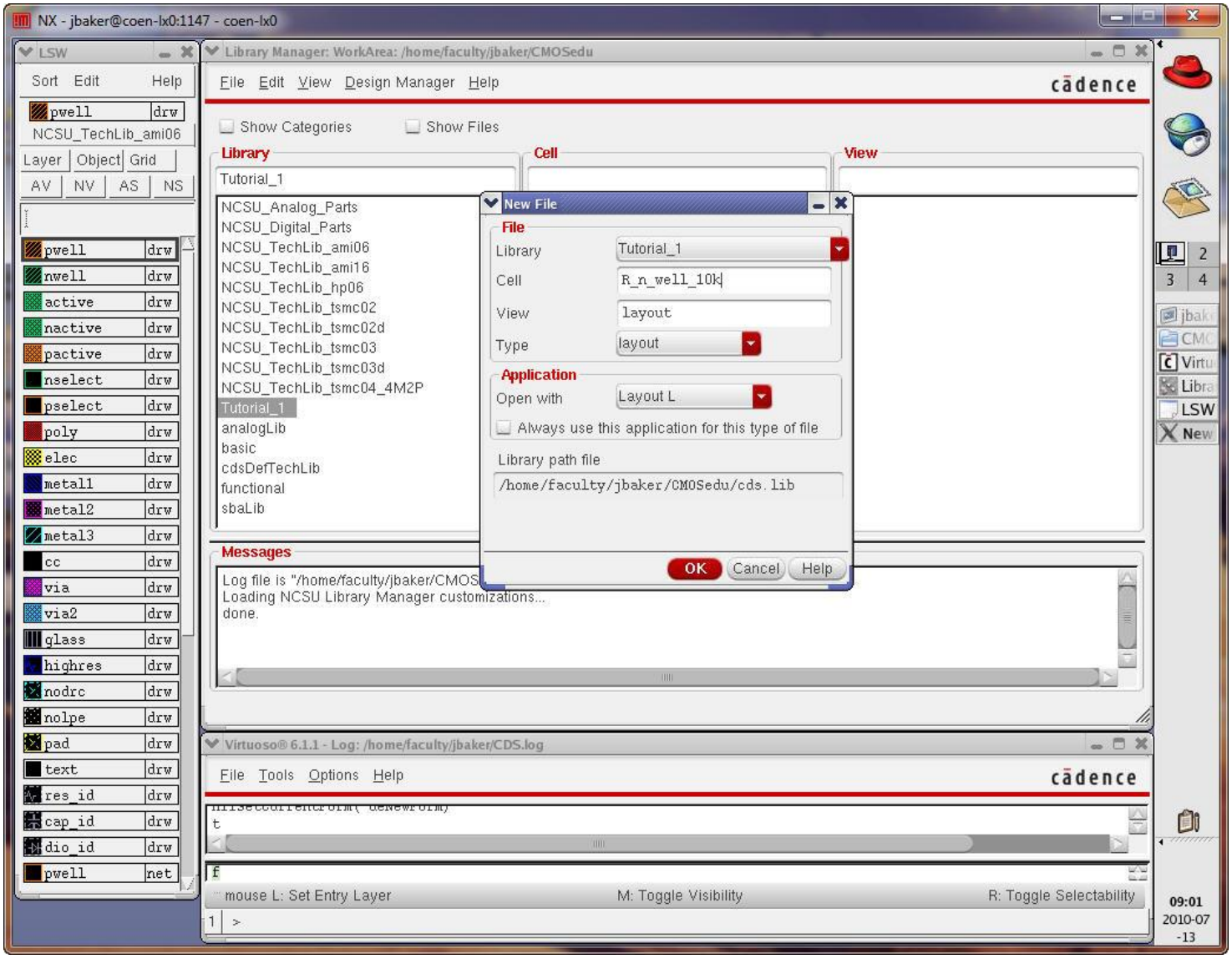


When we started drawing the schematic for the R_div cell the first thing we did was press **i** and place the symbol for a resistor. Here we don't have a layout for
the resistor so we need to create one!

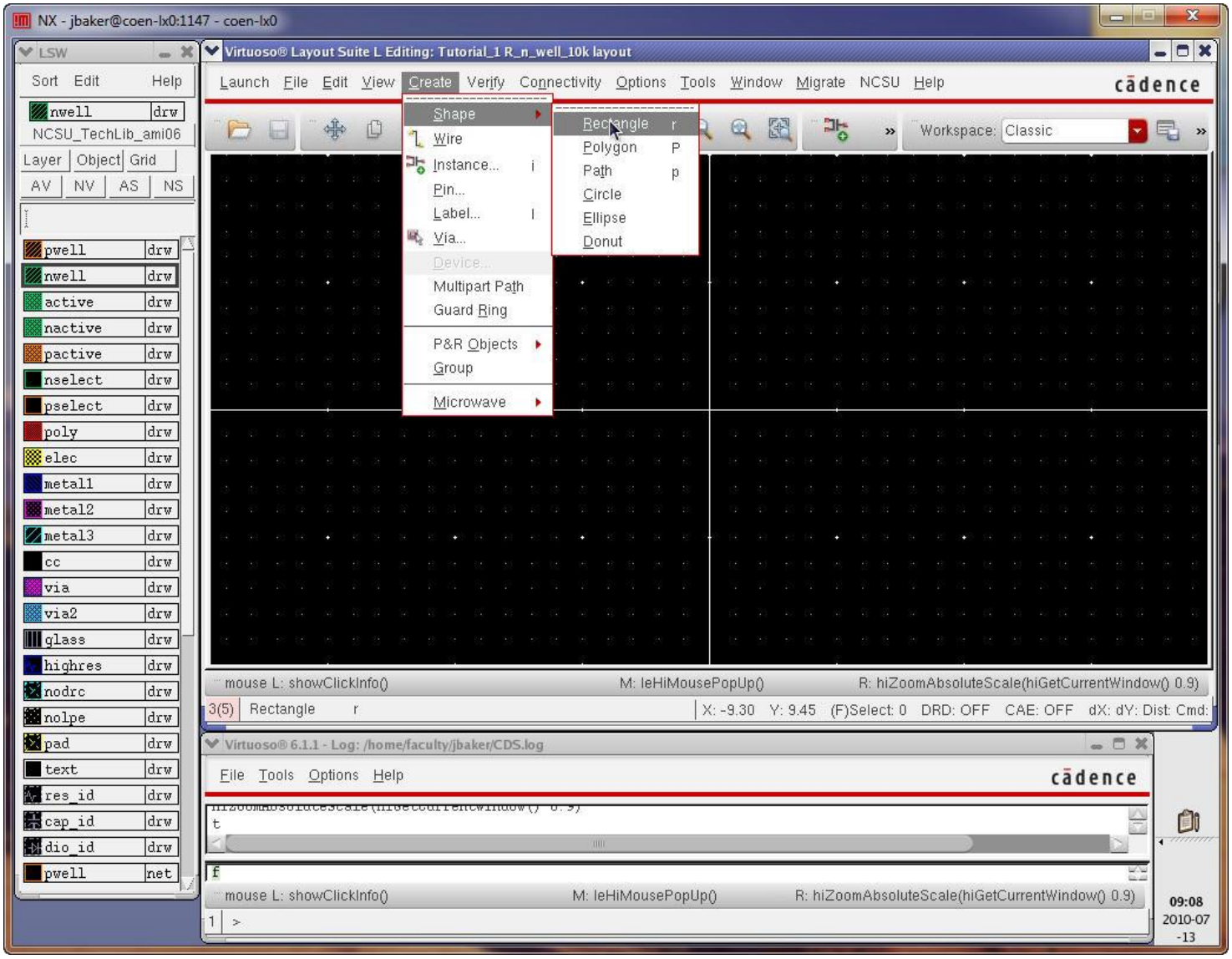
We'll use the n-well layer for the 10k resistor.
The sheet resistance of n-well in the C5 process is roughly 800 ohms.
The minimum width of n-well is 12 lambda (3.6 microns since lambda here is 300 nm) so let's make a 10k resistor using a width of 4.5 um and a length 56 um.

Create a cell (layout view) called R_n_well_10k.

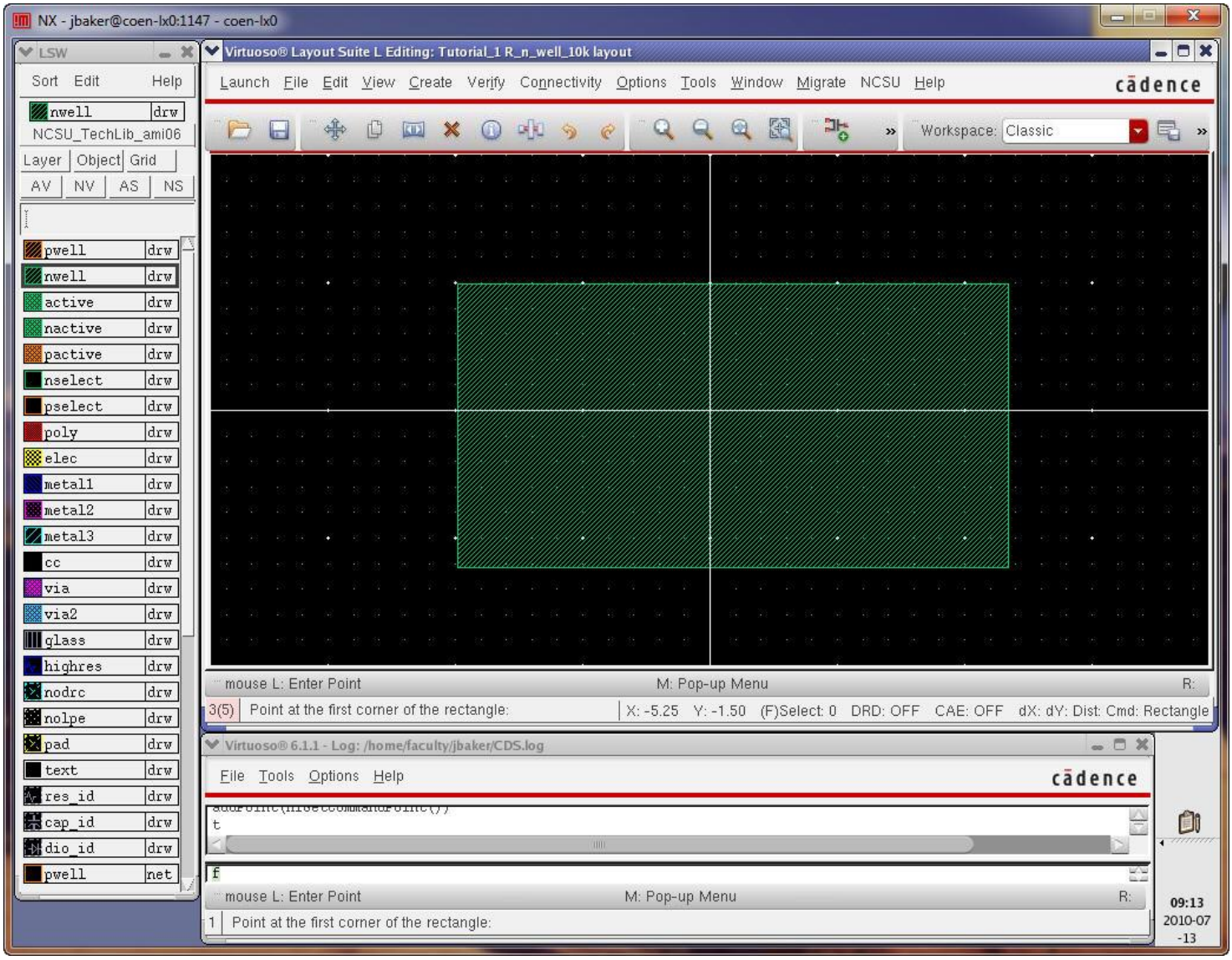

NX - jbaker@coen-lx0:1147 - coen-lx0
LSW
Sort Edit Help
pwell drw
NCSU_TechLib_ami06
Layer Object Grid
AV NV AS NS
pwell drw
nwell drw
active drw
nactive drw
pactive drw
nselect drw
pselect drw
poly drw
elec drw
metal1 drw
metal2 drw
metal3 drw
cc drw
via drw
via2 drw
glass drw
highres drw
nodrc drw
nolpe drw
pad drw
text drw
res_id drw
cap_id drw
dio_id drw
pwell net
Library Manager: WorkArea: /home/faculty/jbaker/CMOSedu
File Edit View Design Manager Help
cadence
New
Open... Ctrl+O
Open (Read-Only)... Ctrl+R
Open With...
Load Defaults...
Save Defaults...
Open Shell Window... Ctrl+P
Exit Ctrl+X
Library...
Cell View...
Category...
View
R_div
sim_R_div
NCSU_TechLib_tsmc03
NCSU_TechLib_tsmc03d
NCSU_TechLib_tsmc04_4M2P
Tutorial_1
analogLib
basic
cdsDefTechLib
functional
sbaLib
Messages
Log file is "/home/faculty/jbaker/CMOSedu/libManager.log".
Loading NCSU Library Manager customizations...
done.
New Cell View
Virtuoso® 6.1.1 - Log: /home/faculty/jbaker/CDS.log
File Tools Options Help
cadence
mouse L: M: R:
08:59
2010-07
-13



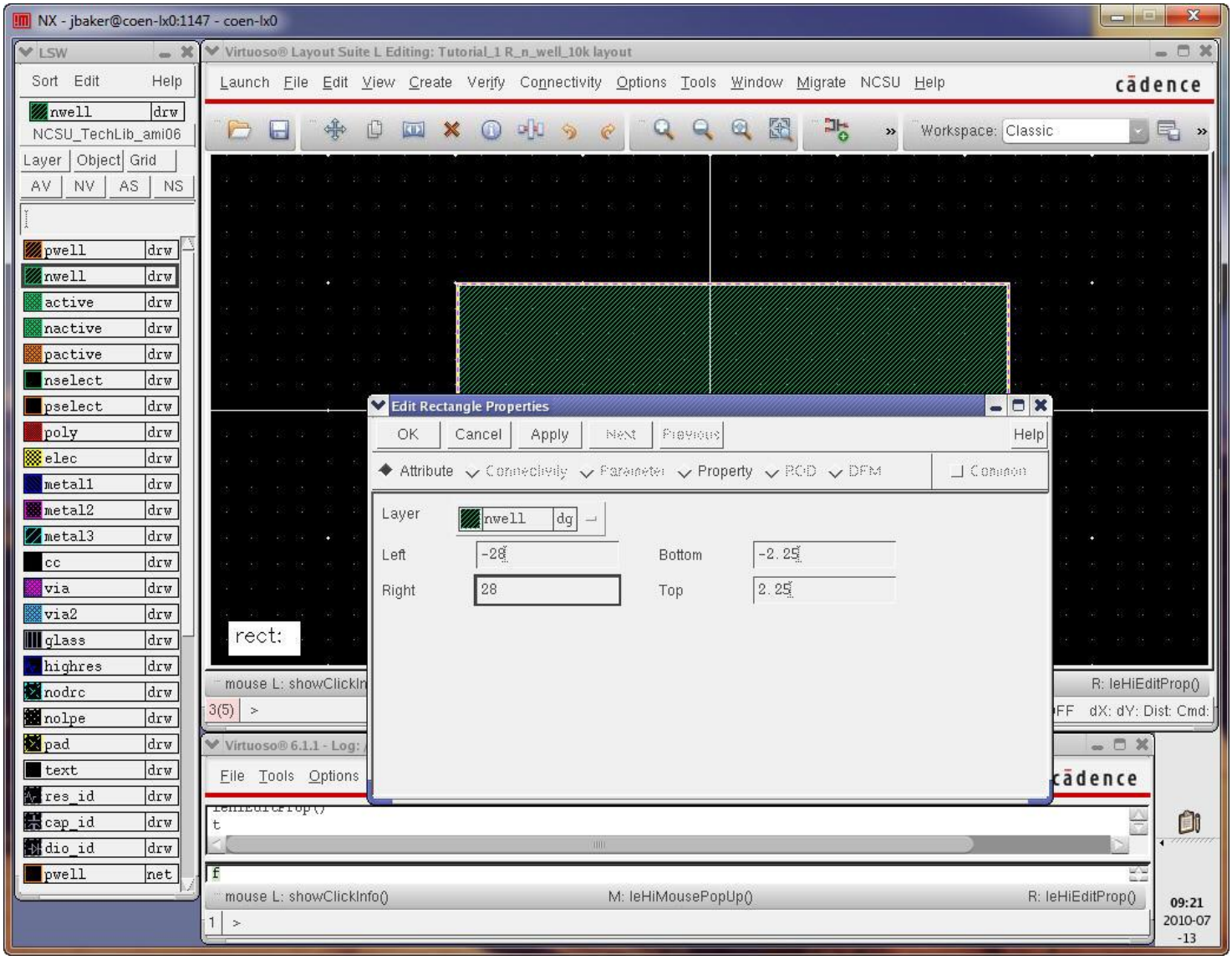
Select n-well in the Layer Selection Window (LSW).
Next create a rectangle (this will be the resistor with a width of 4.5 um and a length of 56 um).



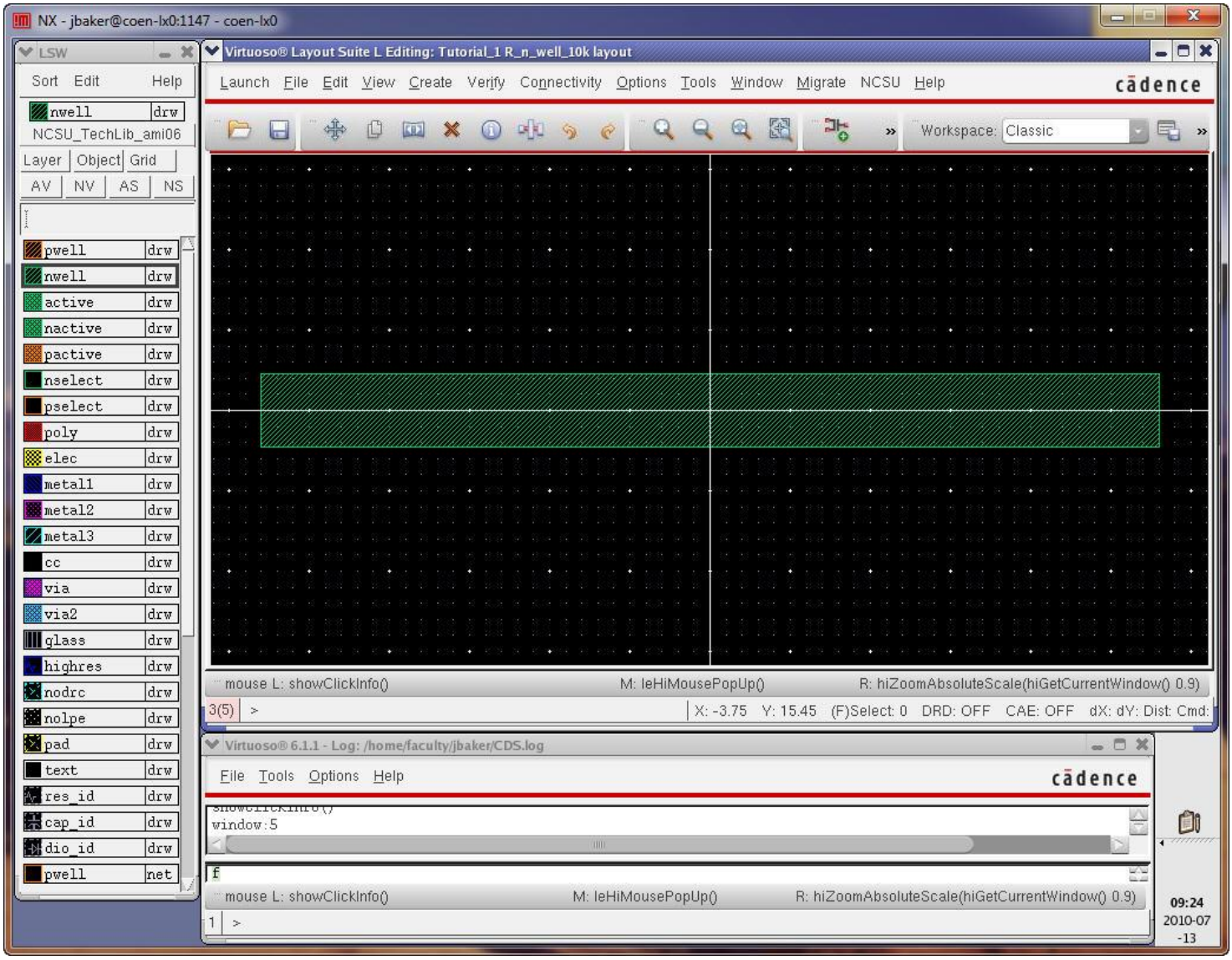
At this point don't worry about the size. Click once to start drawing the rectangle then, after moving the mouse, click again to finish the drawing.



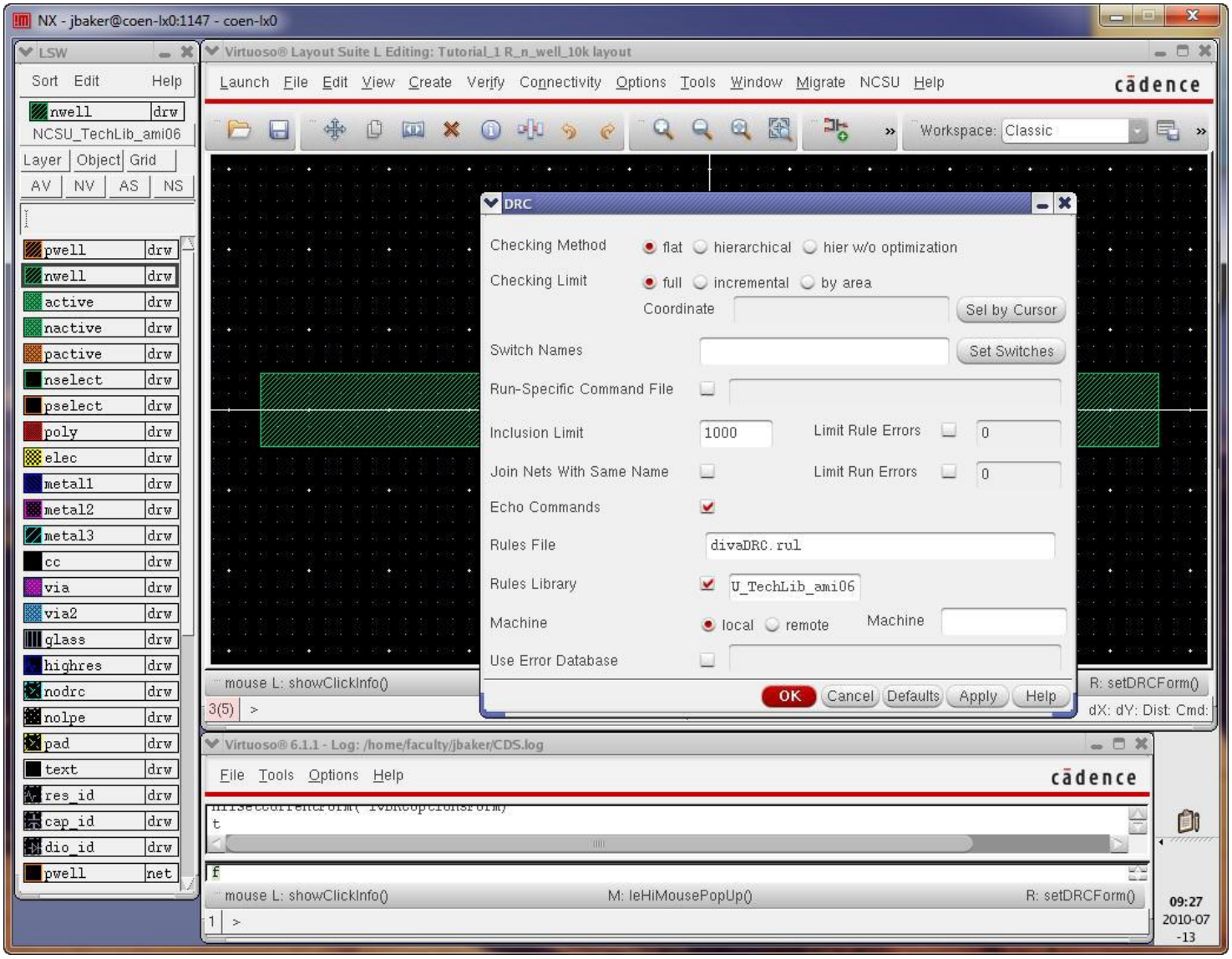
To exit the Create Rectangle mode press Esc (or Virtuoso will continue drawing rectangles).
Next select the rectangle and press **q** (Edit -> Basic -> Properties).
As calculated above we want a resistor that is 56 um long and 4.5 um wide.



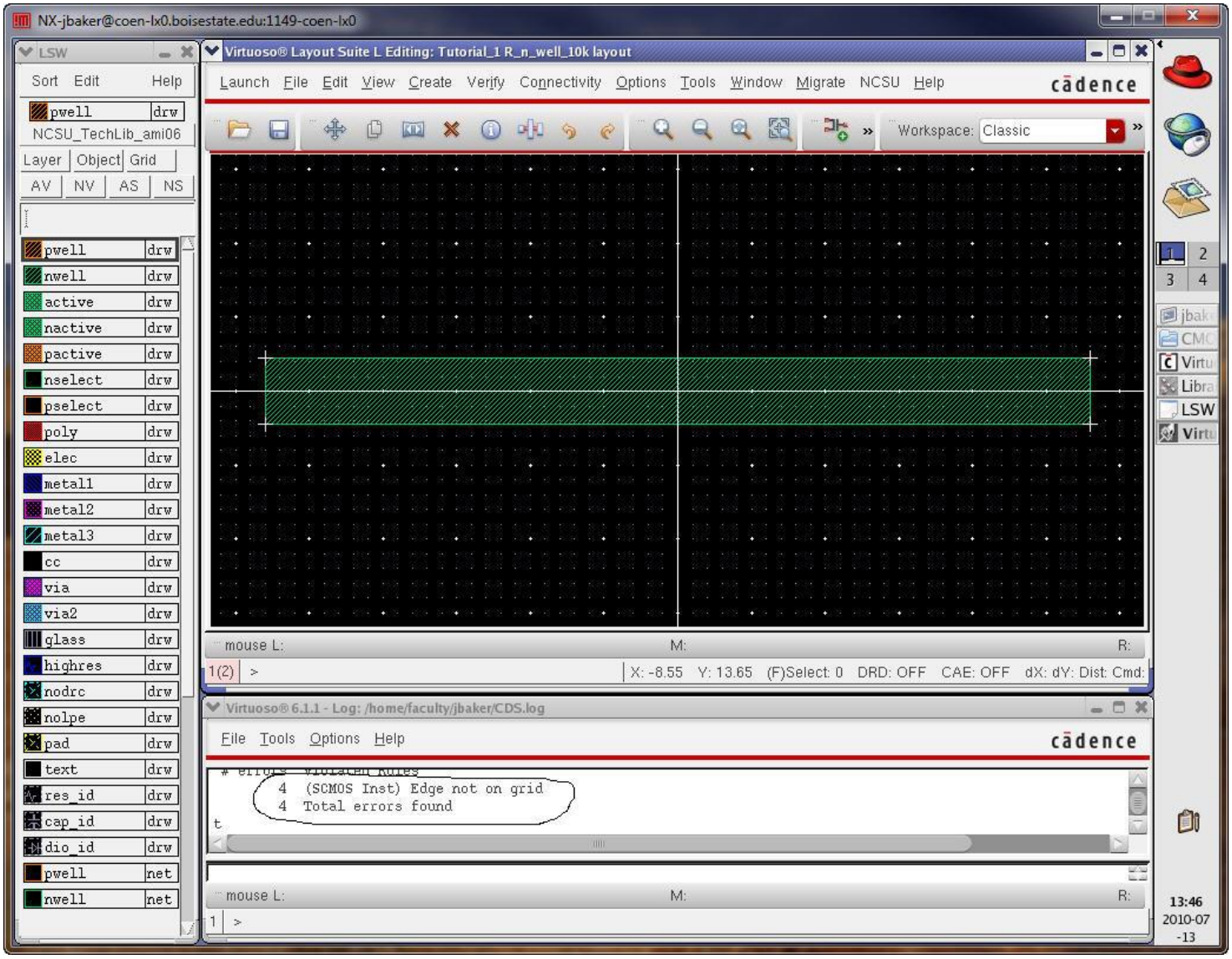
After clicking on OK and fitting the layout we get the following.



Let's design rule check (DRC) this layout before continuing.
Using the menu Verify -> DRC the following window pops up.
The Rules Library field seen below is NCSU_TechLib_ami06

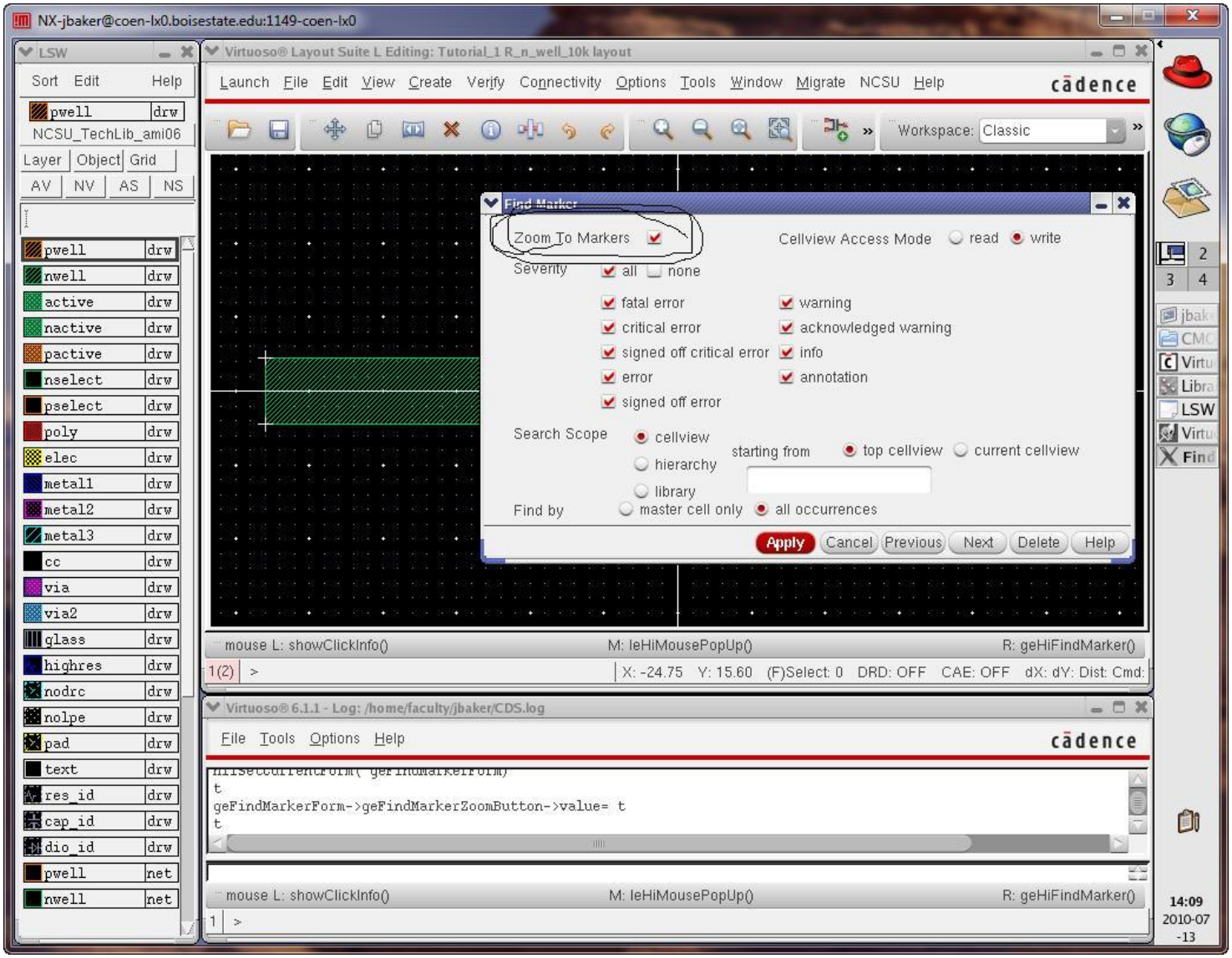


Pressing OK starts the DRC.

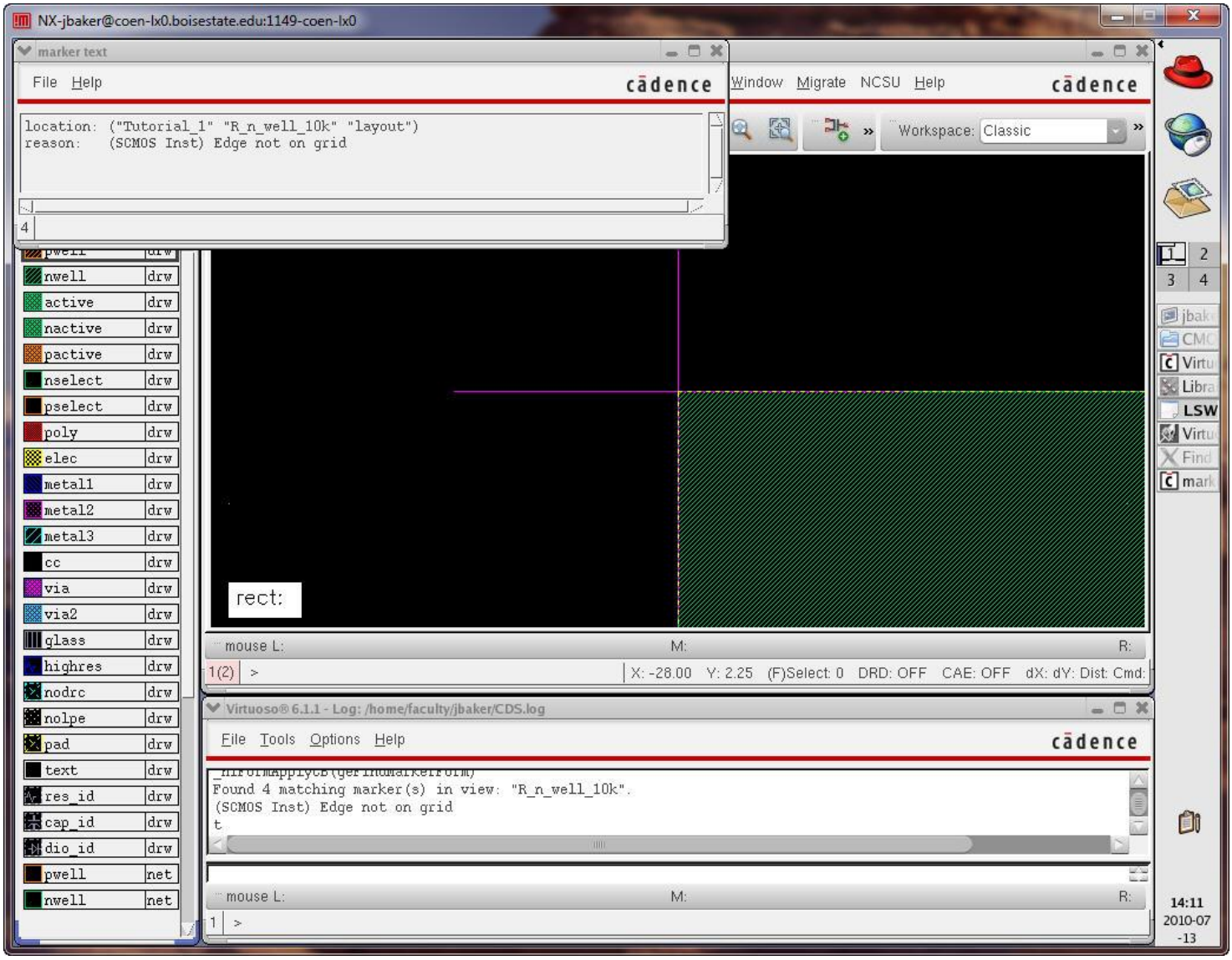


The CIW shows there are 4 errors (the edges are not on grid).
The questions are how do we find out the grid settings and how to view the markers showing the location of the errors (above seeing the markers is easy
since the layout is simple….the crosses in the corners are the markers).

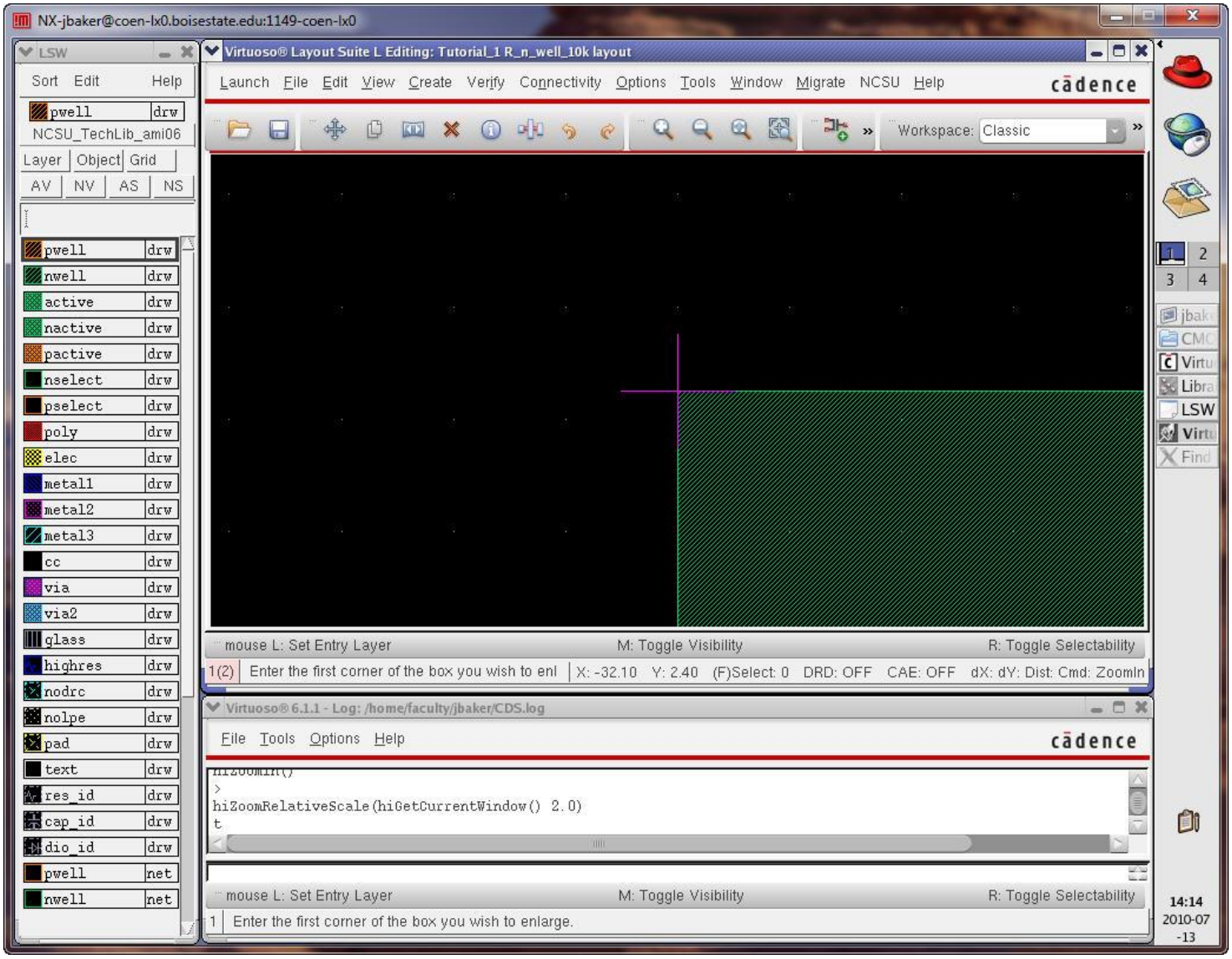
Using the menu Verify -> Markers -> Find Marker (notice you can delete the markers in this menu path too).
Select Zoom To Markers



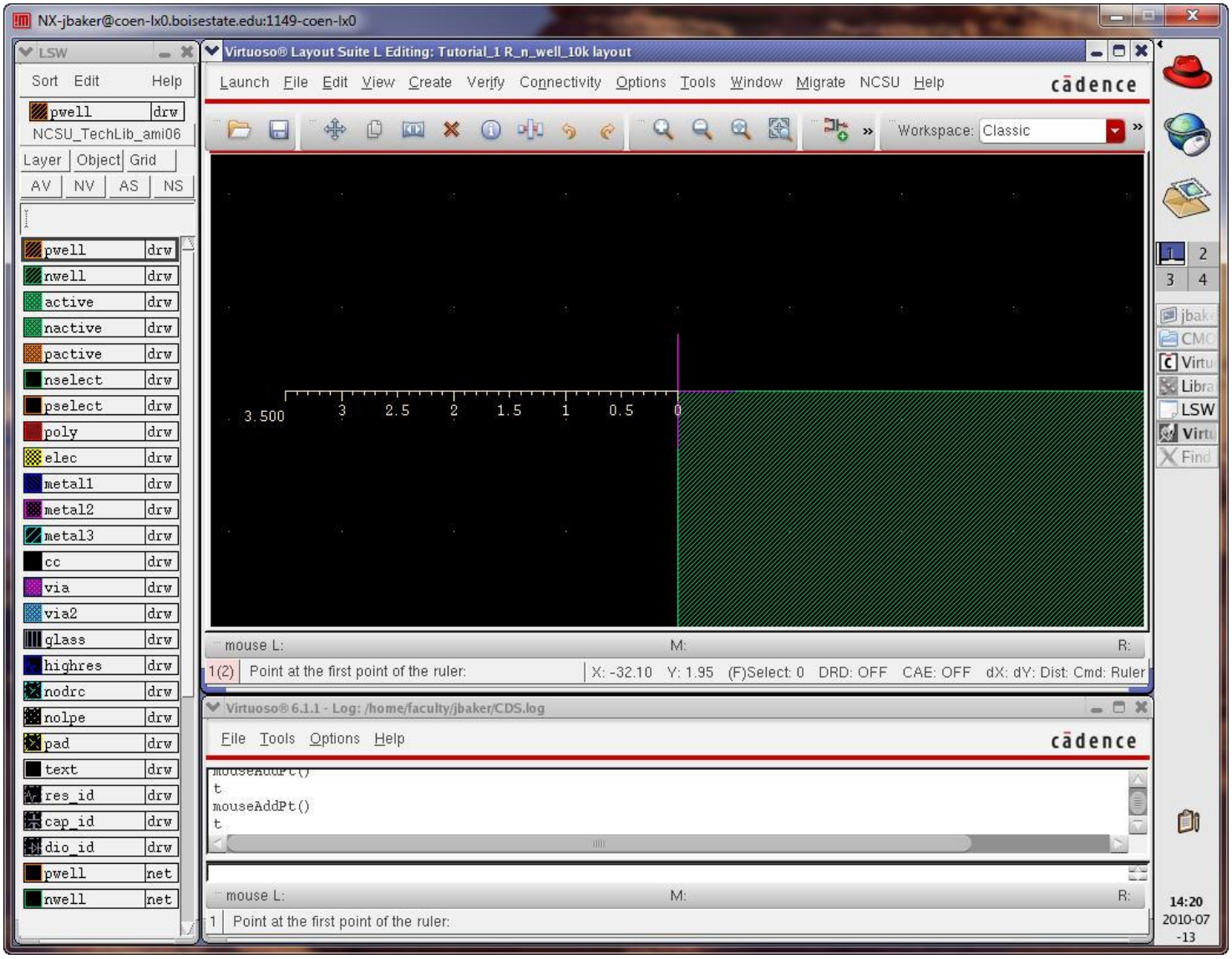
Hitting Apply we get



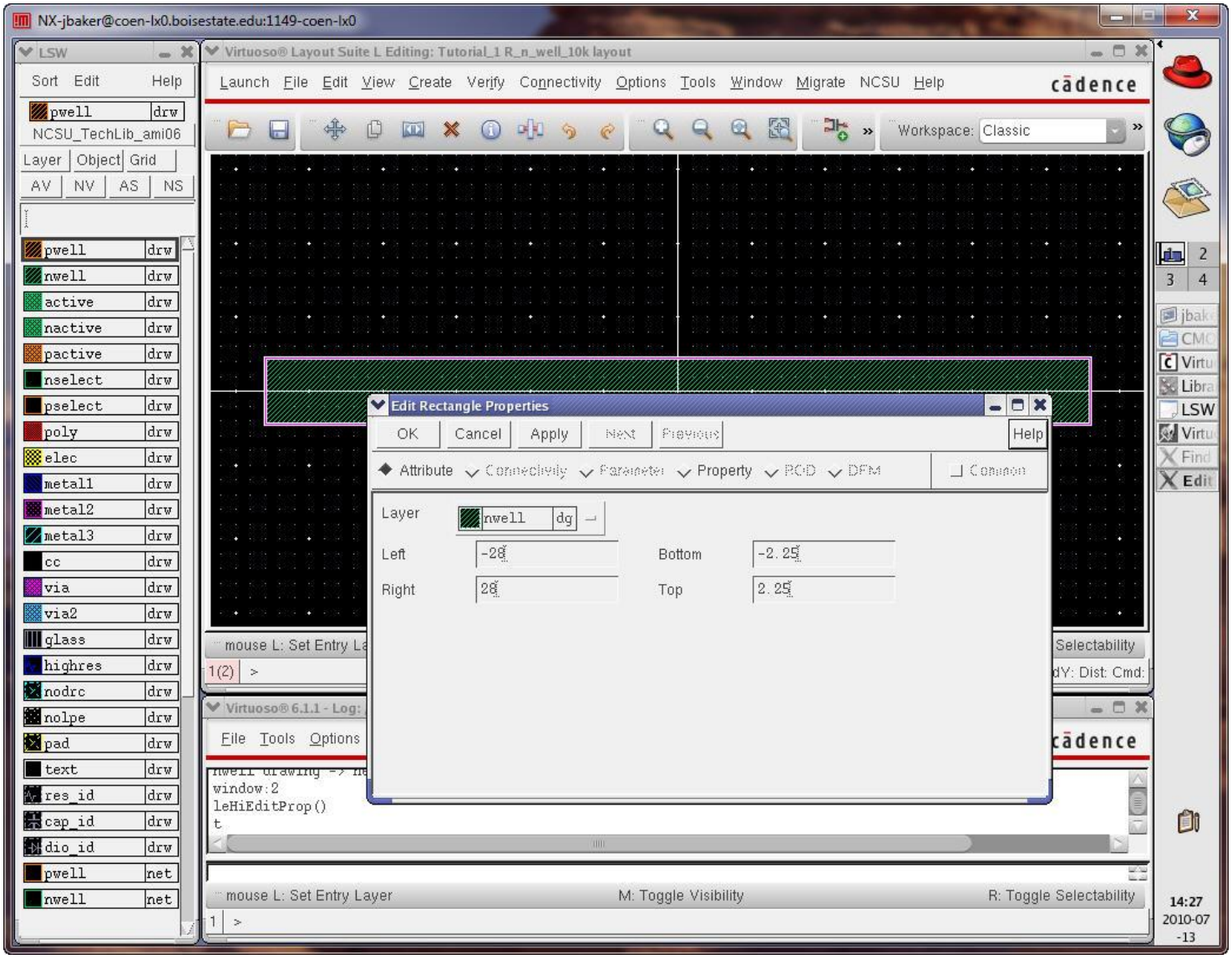
Close the marker text window, select the layout window, and zoom out a couple of times (press **Z**) until you can see the grid.



While it's a little challenging to see in the figure above the corner isn't snapped to the y-axis grid.
Use Tools -> Create Ruler (or the Bindkey **k**) to measure the distance between grid points (or just press e, Options -> Display to see X and Y Snap Spacing is set to 0.15 microns)

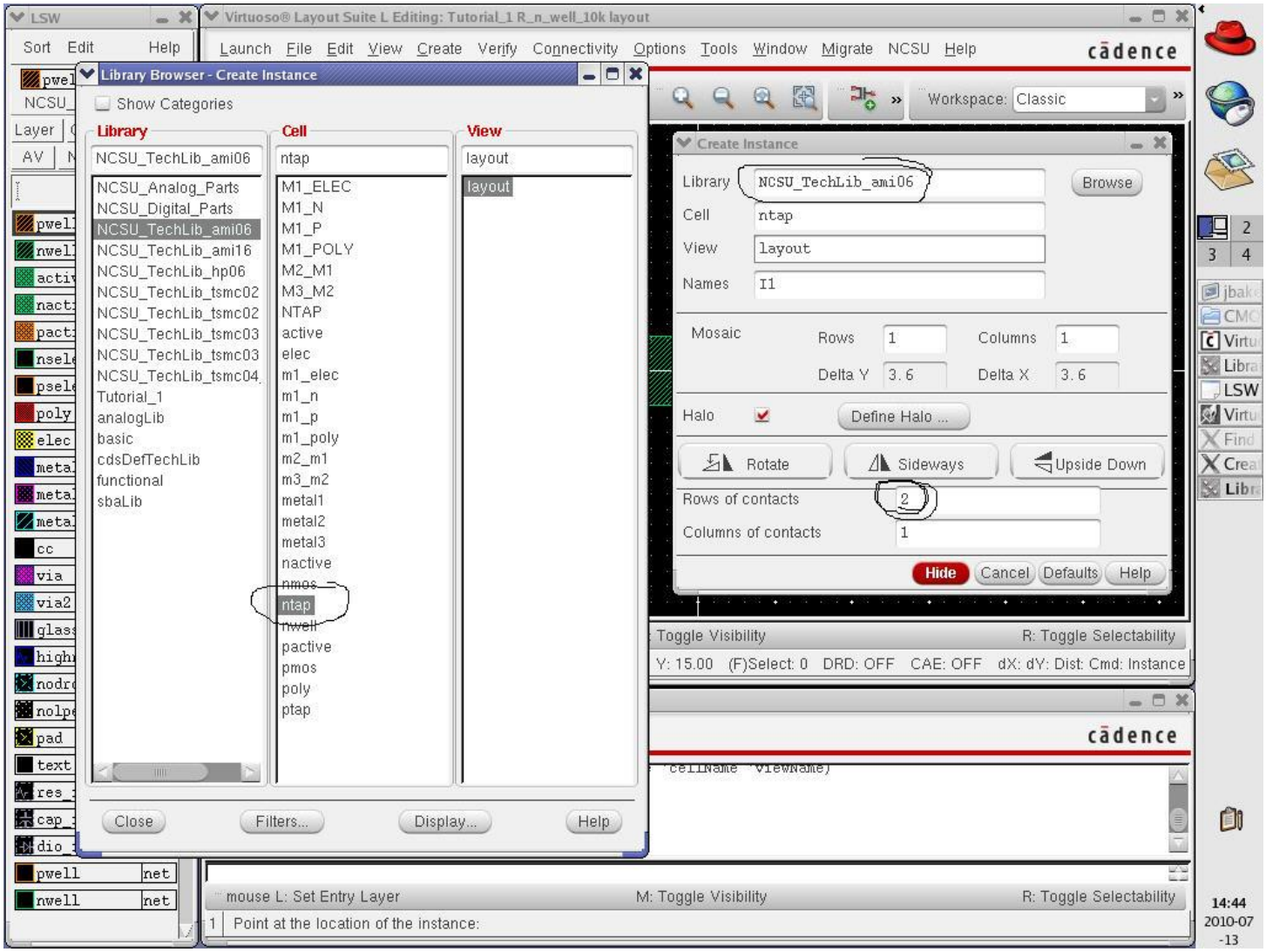


The distance between grid points is 1 micron and, as mentioned, the X and Y snapping is 0.15 microns.
Clear the ruler by using the menu items Tools -> Clear All Rulers (or just press **K**).
Press f to fit the layout in the drawing area.
Use Verify -> Markers -> Delete All followed by OK to delete the markers.
Press Esc a few times to ensure no commands are active.
Next select the layout and press **q** to get the following

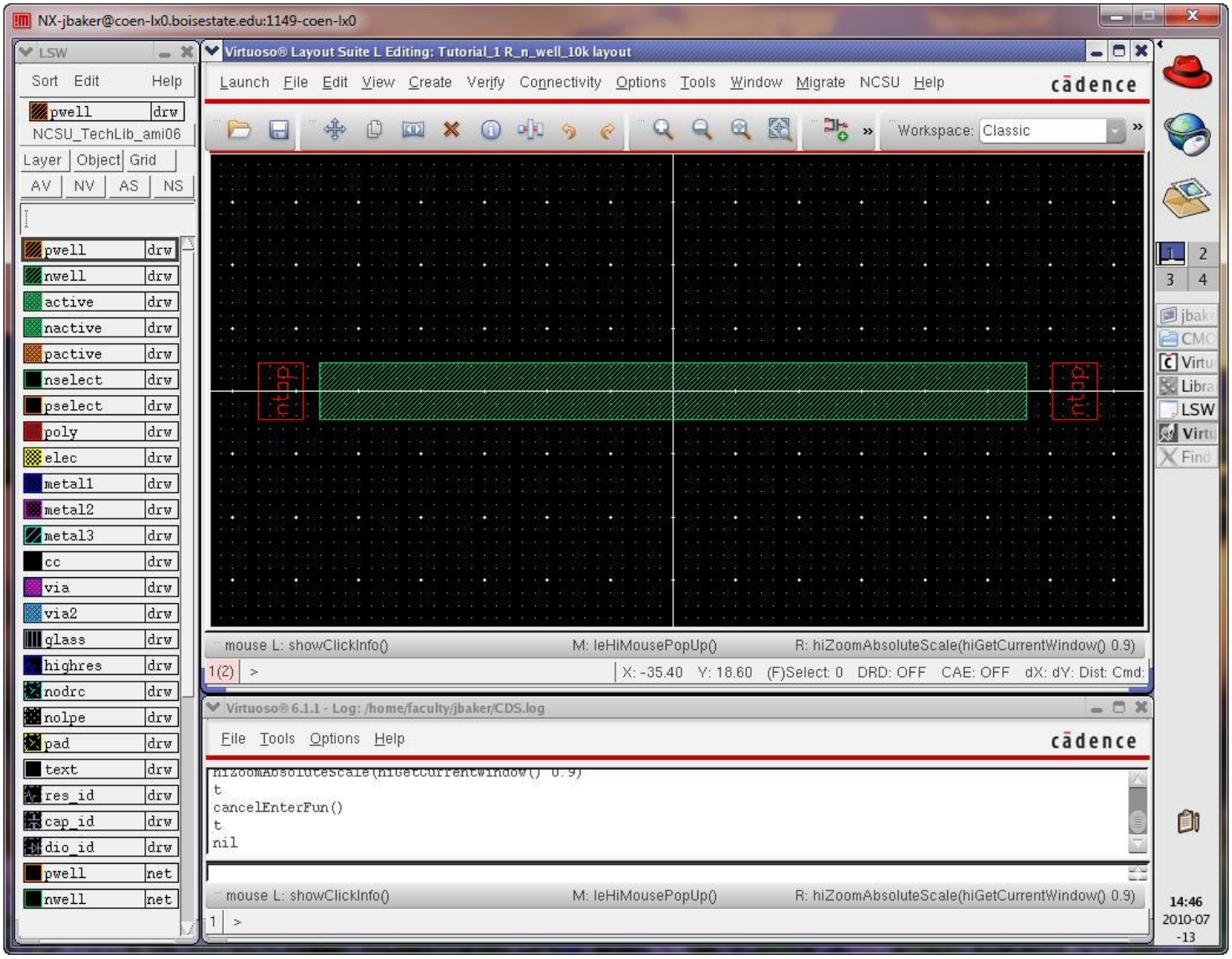


So the length of the resistor is 56 and 56/.15 = 373.3333. To make this a whole number let's increase the length to 56.1 (**so we enter 28.05 in the Left/Right above**).
For the width we used 4.5 and 4.5/.15 = 30 so we are okay there.
Running the DRC shows no errors are found.

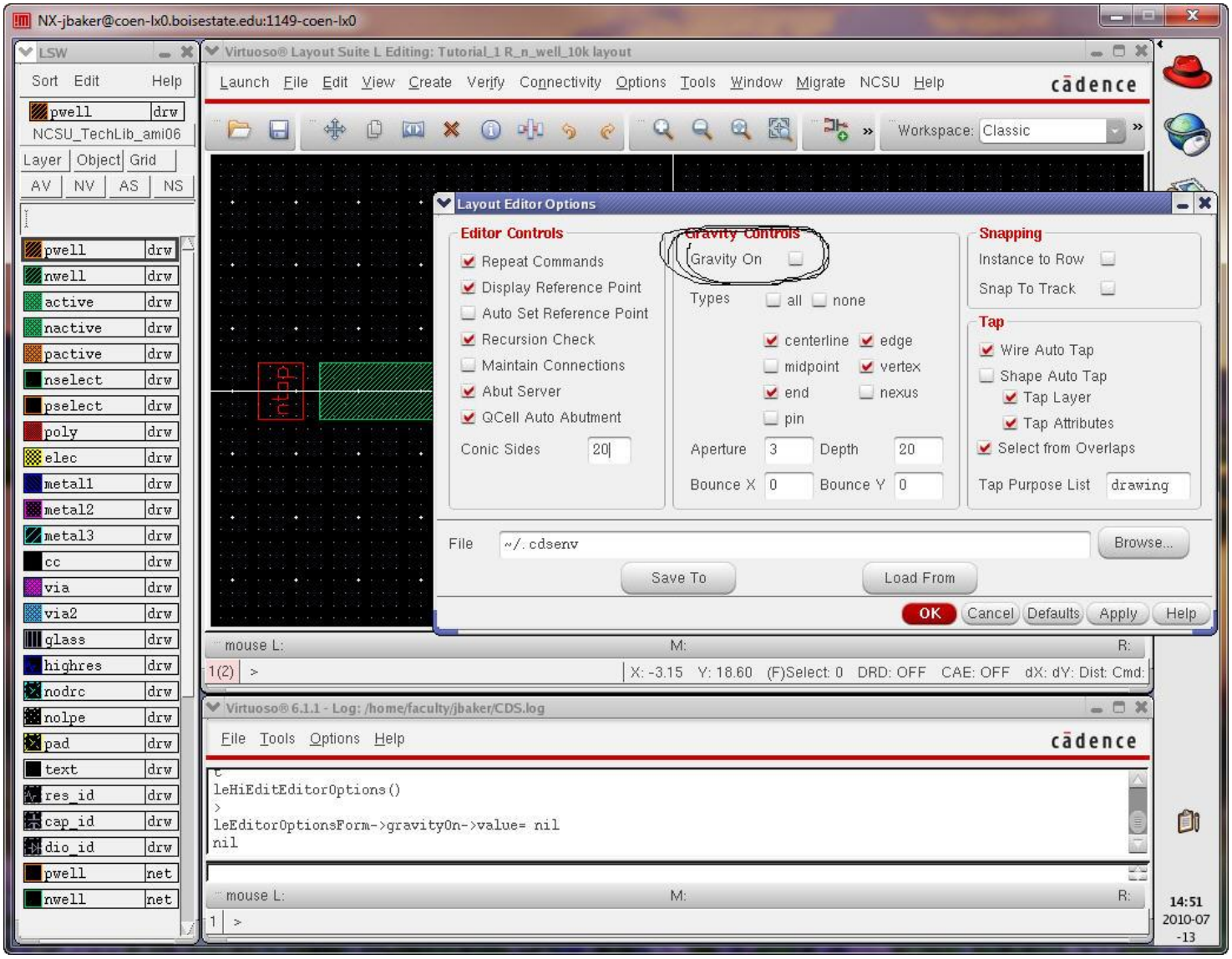
Next let's add the connections to the ends of the resistors.
Press **i** and navigate/select the ntap (metal1 connection to n-well) as seen below.



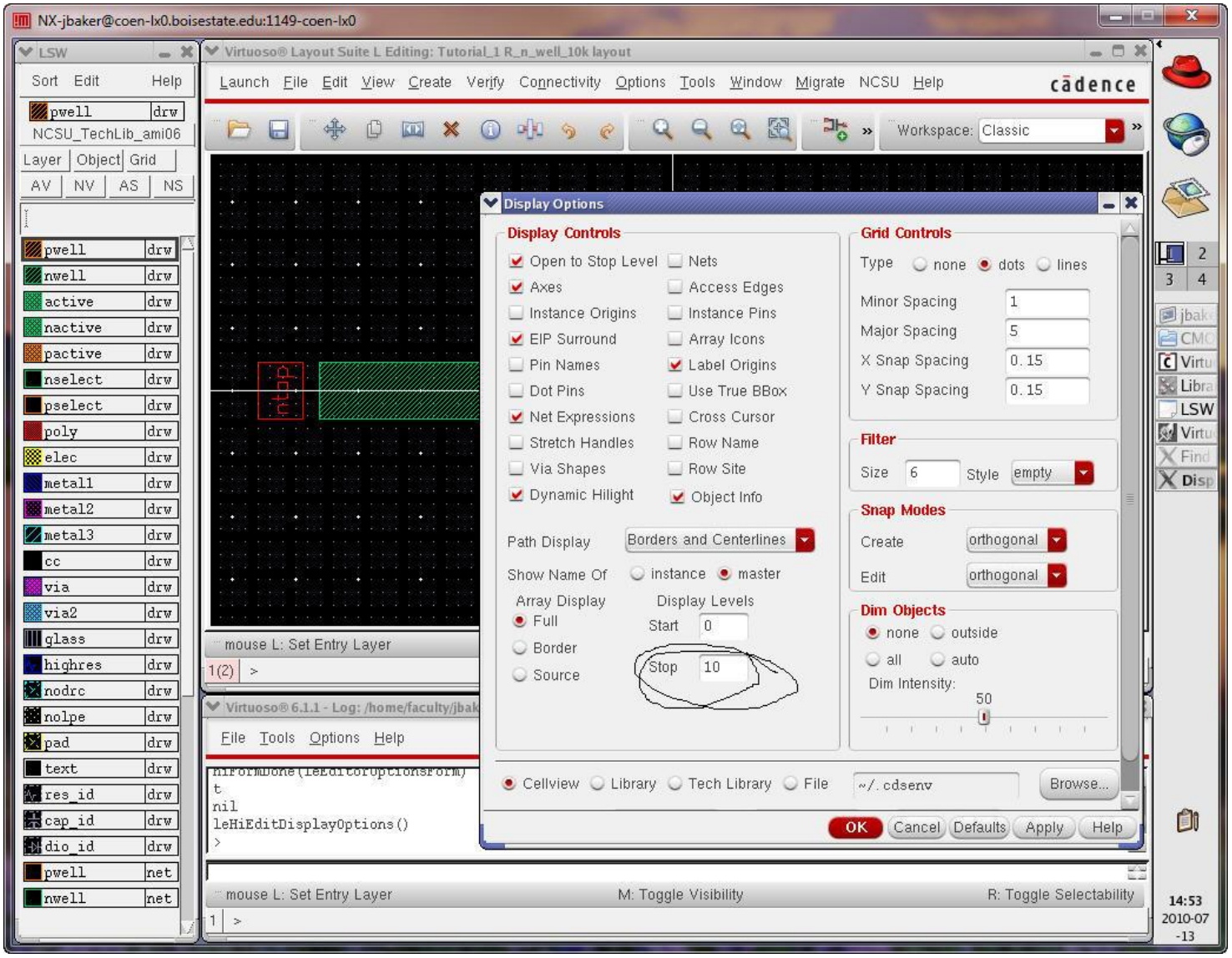
Adding these connections to the ends of the n-well resistor we get the following.



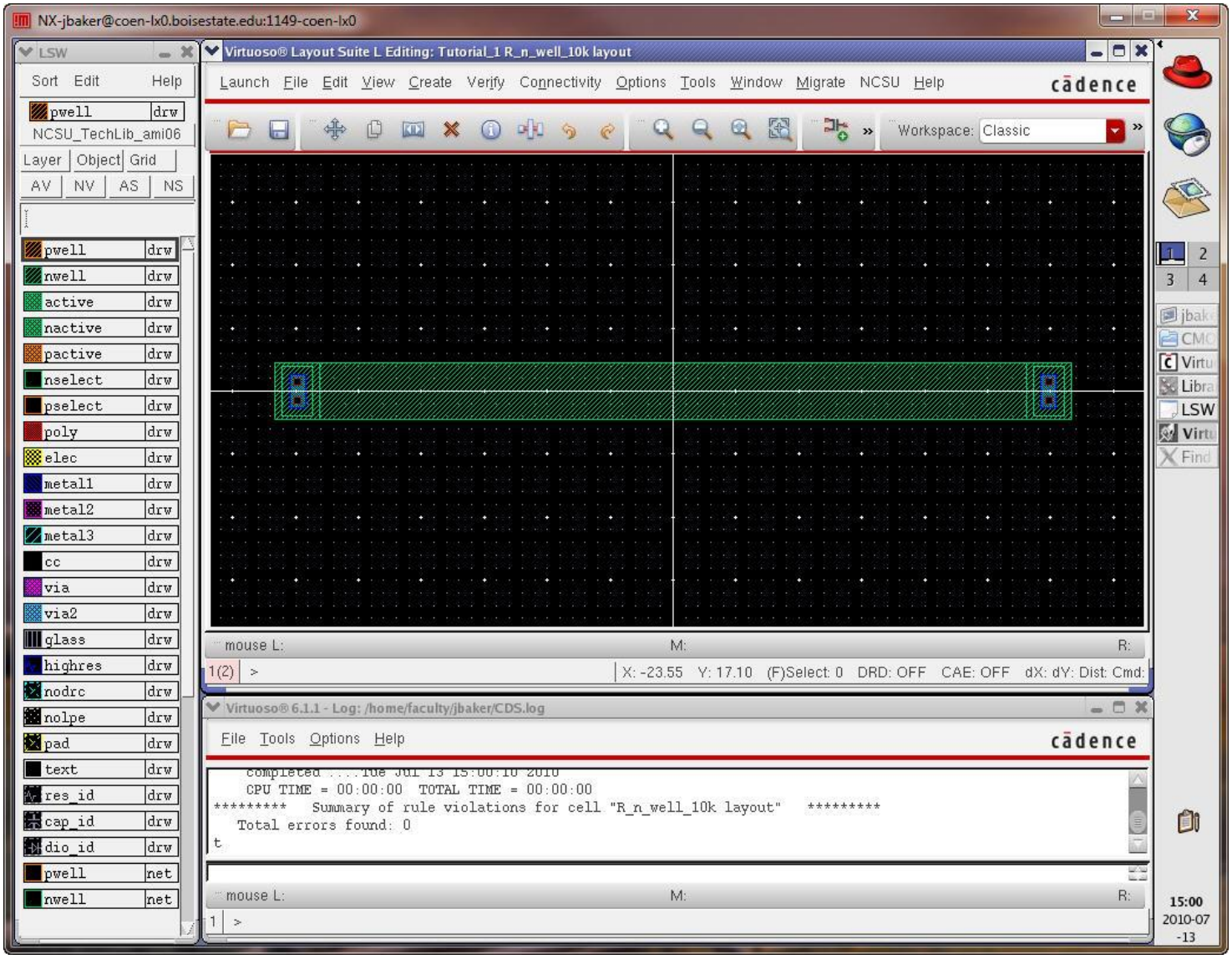
You may have noticed that when placing the nodes they had an affinity to the n-well rectangle. This is called gravity, which can be useful. However,
here it's not useful so let's turn it off. Go to Options -> Editor (or press **shift-E**) and deselect the "Gravity On" check box.



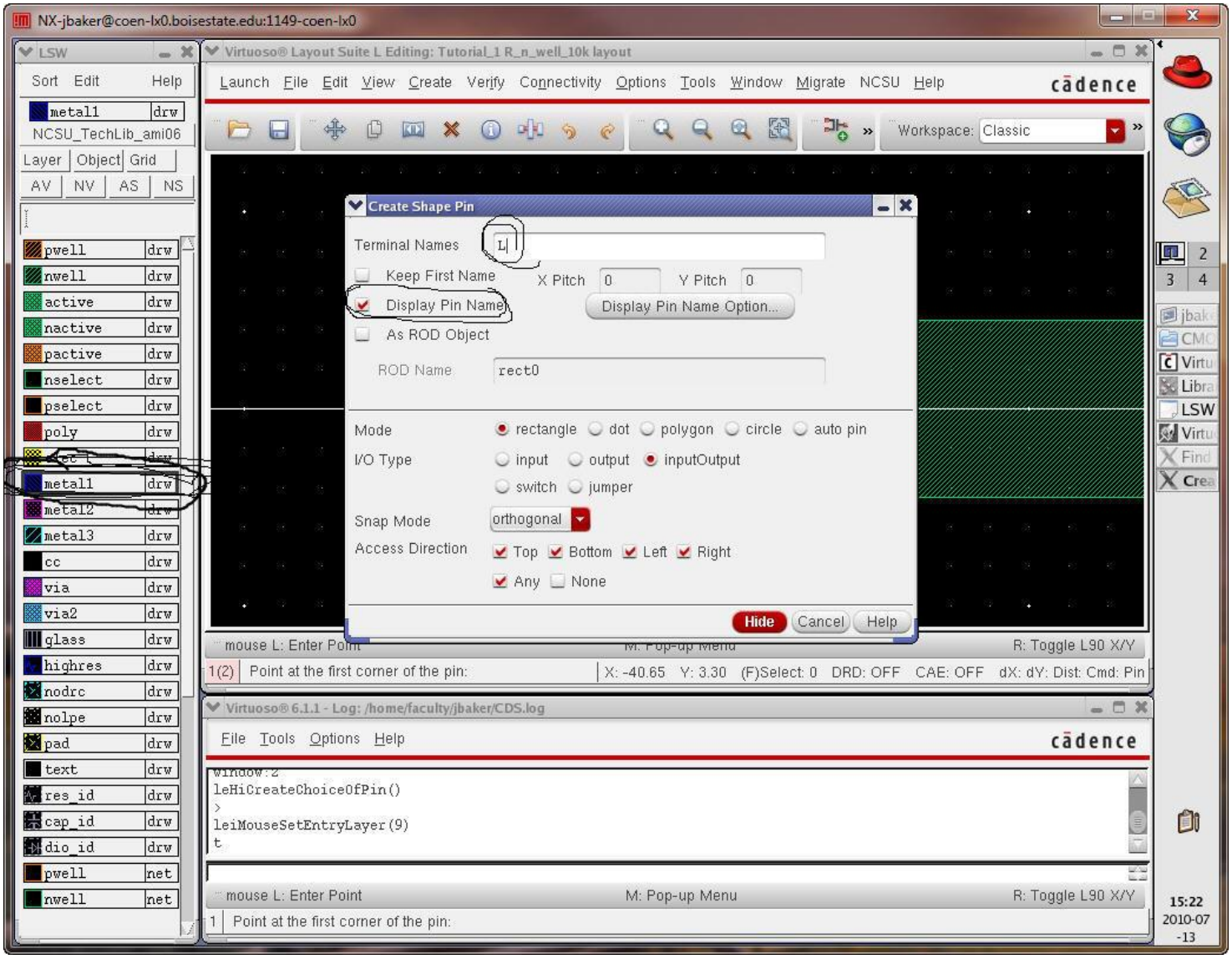
Next notice that the ntap cells are drawn as outlines. Go to Options -> Display (or just press e) and set the depth of display to 10.



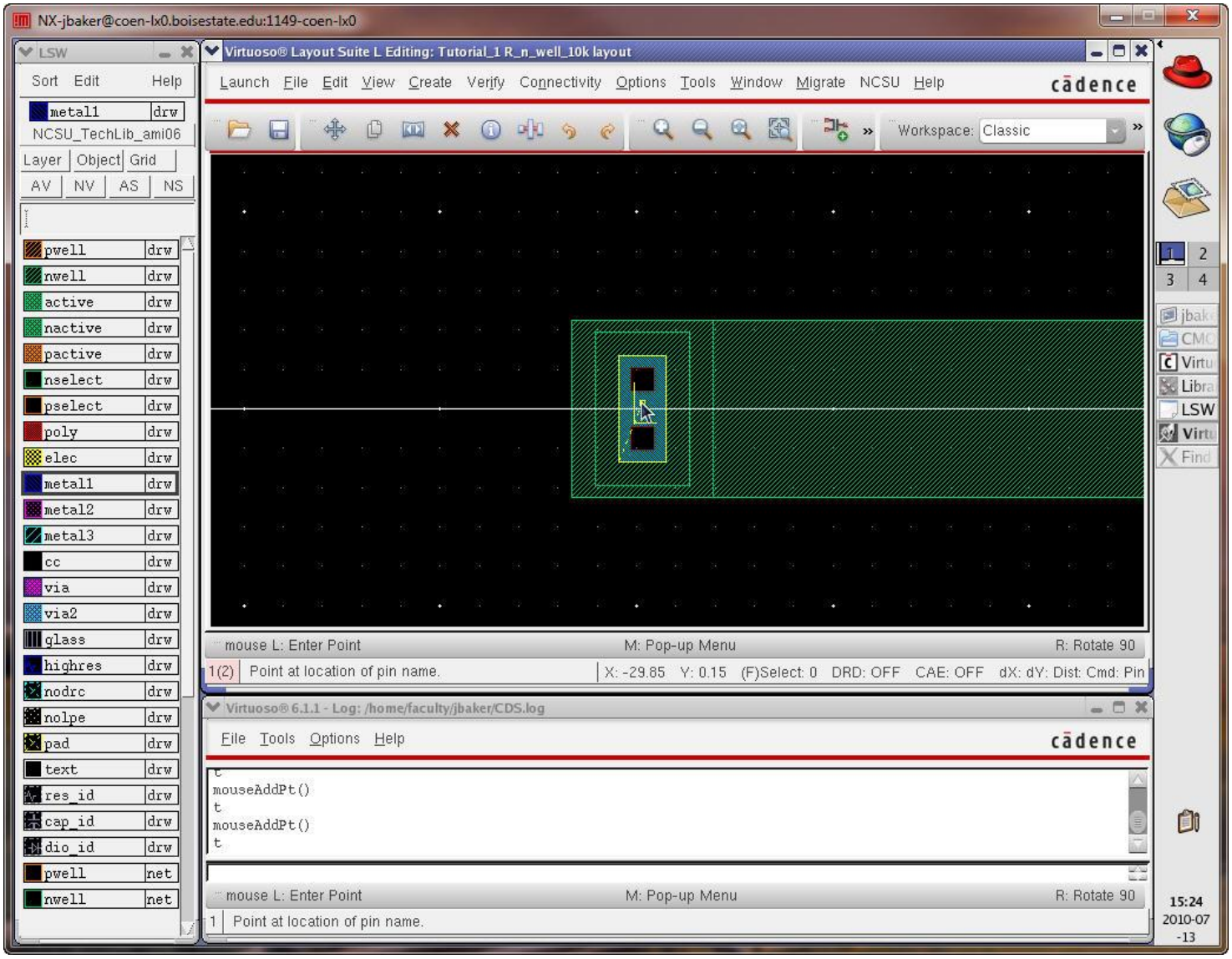
Next select the ntap cells then press **m** to move them (or use the menu Edit -> Move)
Line the cells up as seen below.
Pressing **z** then click, move the mouse, then click again to set the window (you can't click and drag to zoom in).
DRC the layout to ensure no errors.



Next let's add pins to the layout.
Zoom in on the left side of the layout.
Then use the menu commands Create -> Pin
Select "Display Pin Names", a name of L (left), and the metal1 layer as seen below.

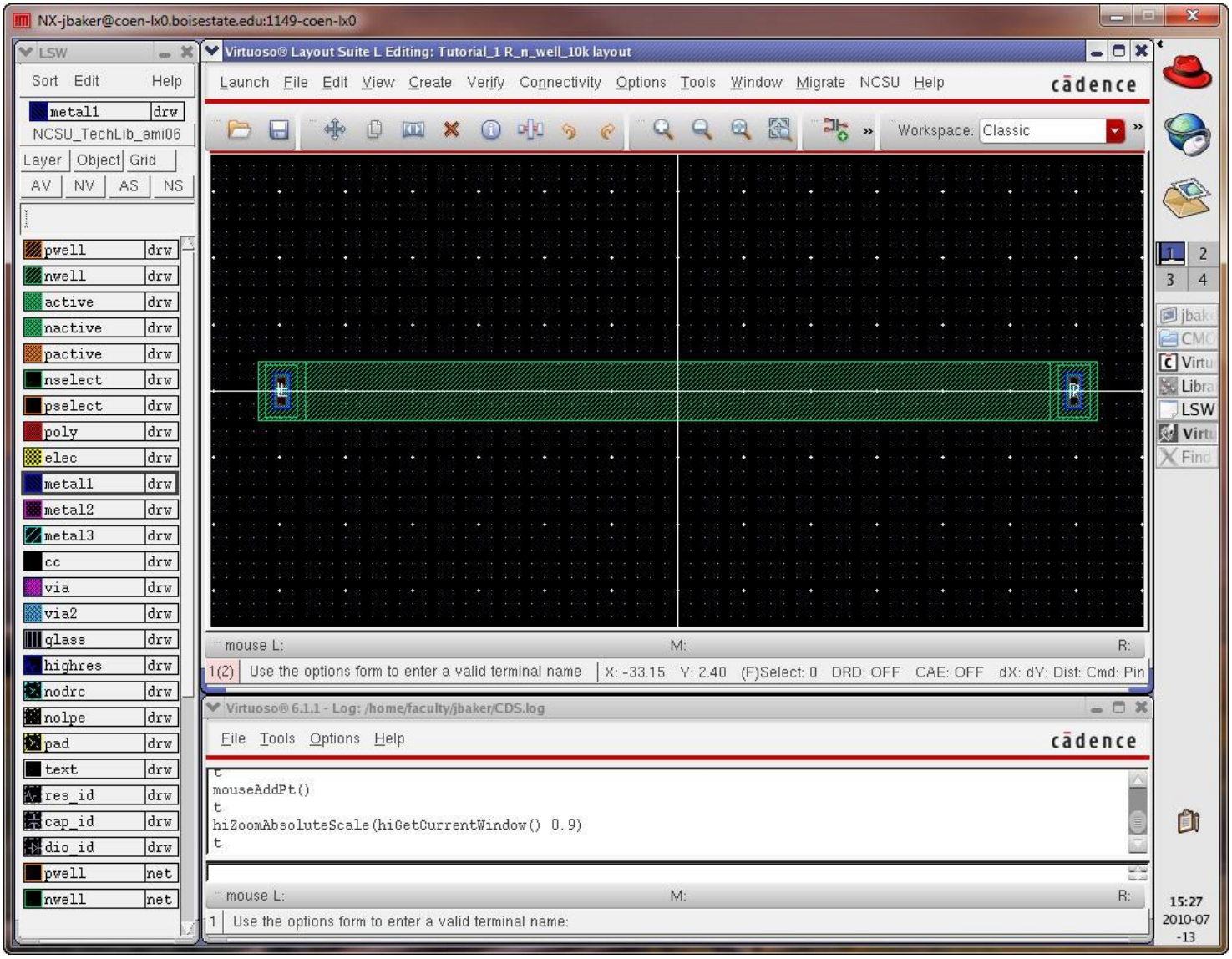


Select Hide and then draw a rectangle around the metal1 on the ntap placing the Pin Name on the center of the metal1 rectangle.



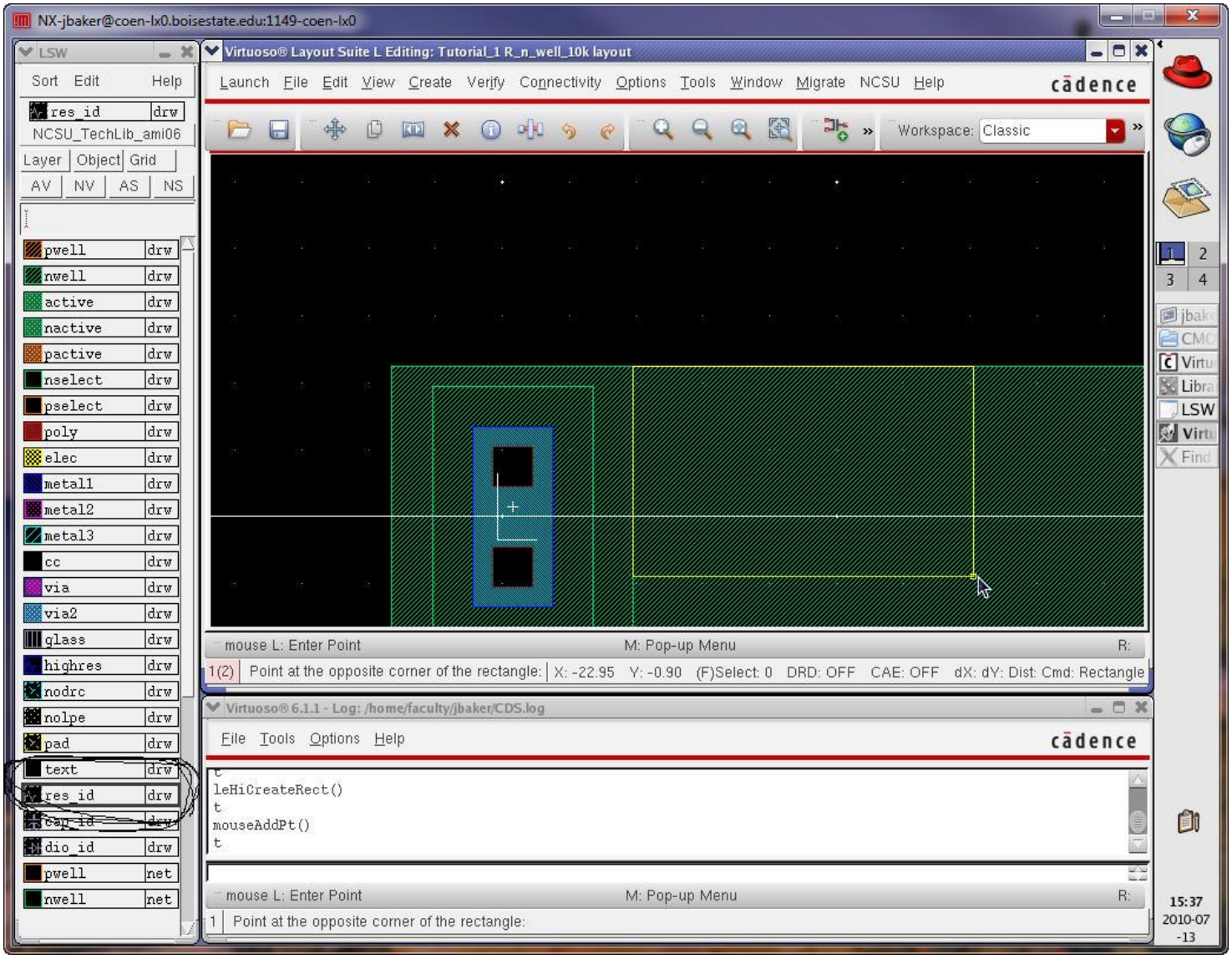
Repeat, but use a Pin Name of R (right), for the other side.

Note that to change the *size* of the text used for the pin's name you select the associated text and edit (**q**) it.
The pin's name, however, can't be changed by selecting the text which can be counterintuitive.
Rather, to change the pin's name, select the shape that the pin is drawn with, here a rectangle on metal1, and then **q** it.
Not understanding this can waste time as you are drawing layouts and trying to change pin names.

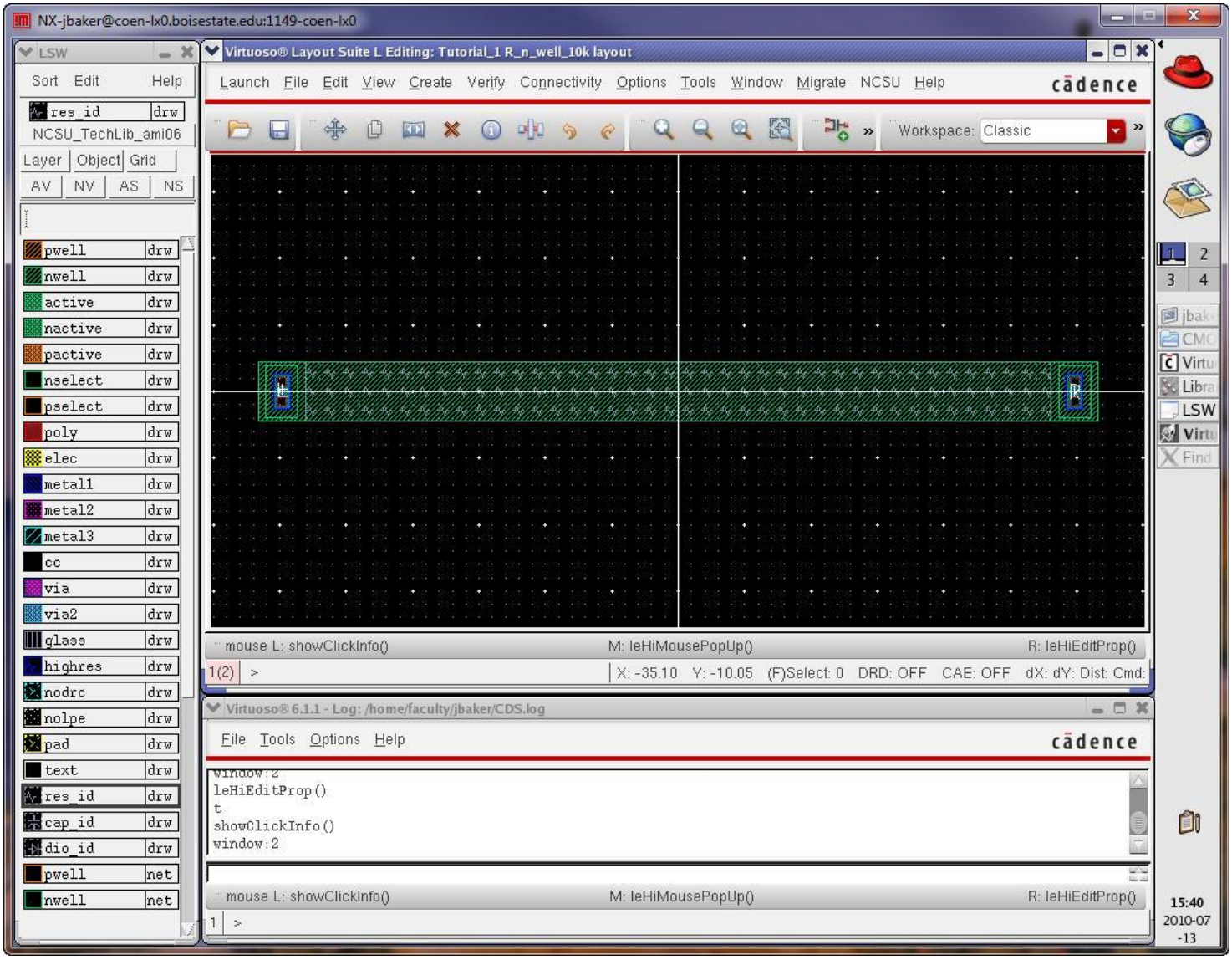


If we were to DRC this layout we would get an error "Label/Pin "R" is on a net already named "L".
The way the layout is now the n-well is treated as a conductor (a wire) not a resistor.
In this situation Cadence thinks that we are trying to label the same wire with two names.
To indicate that we are using the n-well as a resistor, and thus get rid of the error, we use the layer res_id.

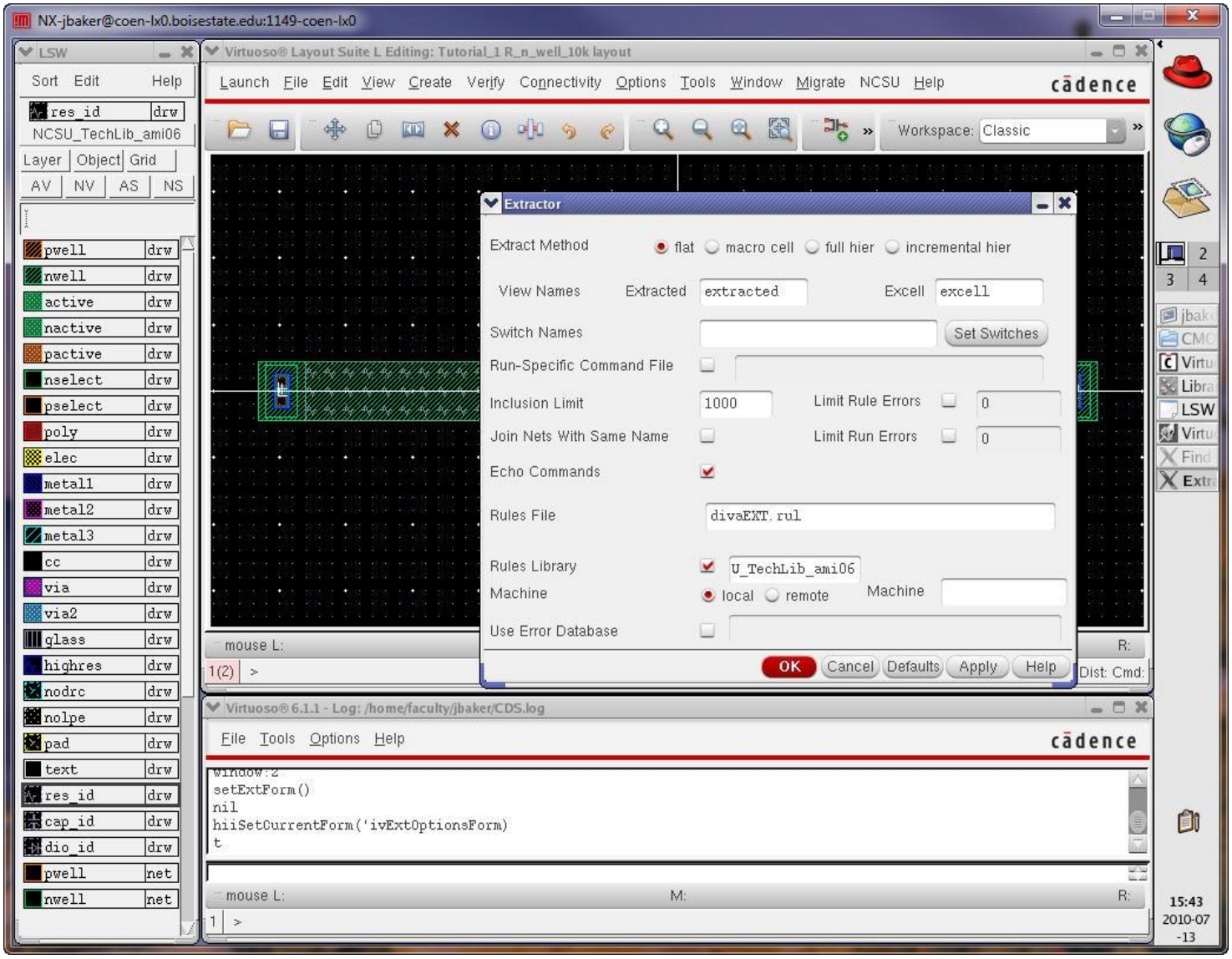
Next select the layer res_id (to identify resistors).
Select r to draw a rectangle.
Zoom in and start drawing a rectangle.



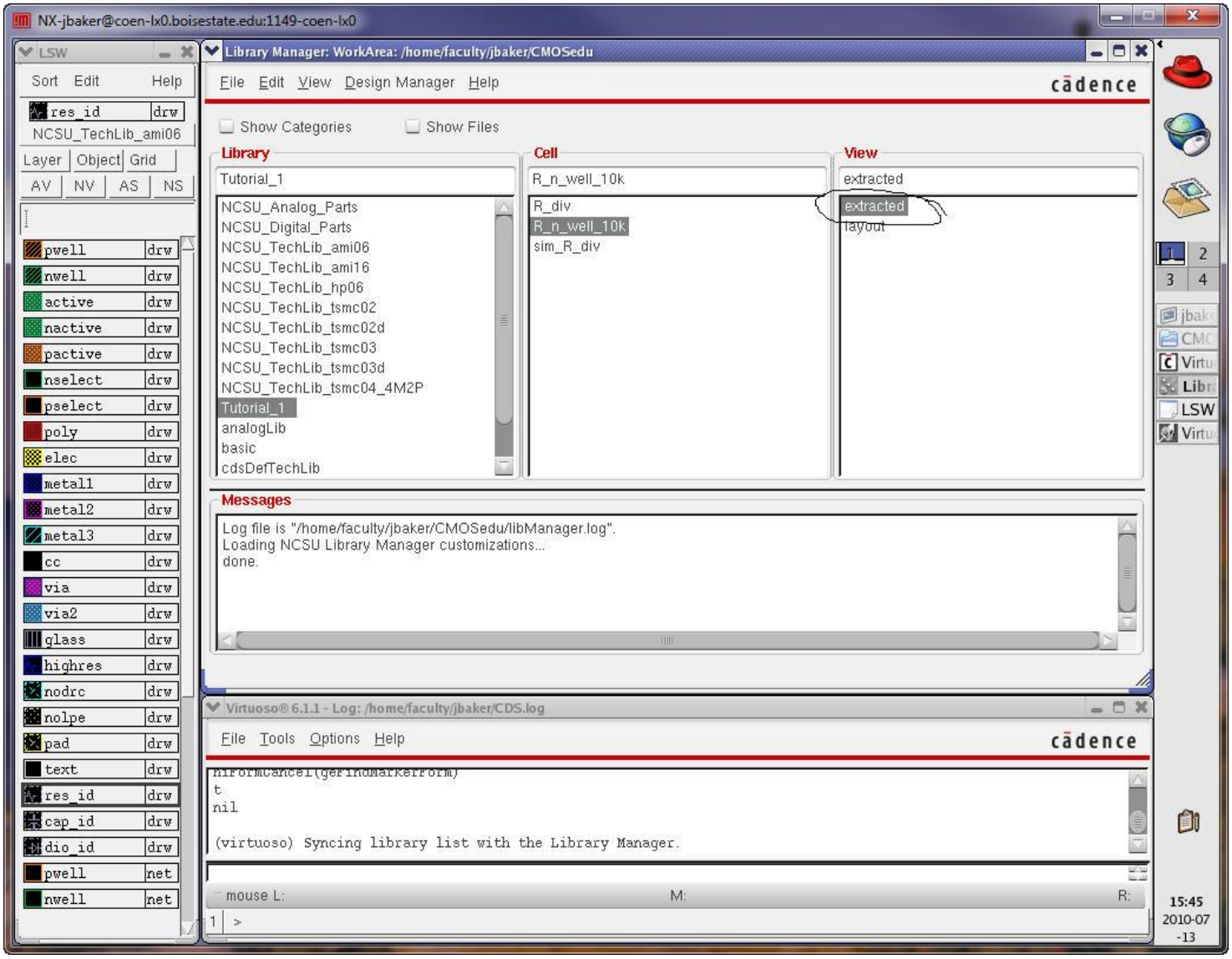
When finished the layout should look like



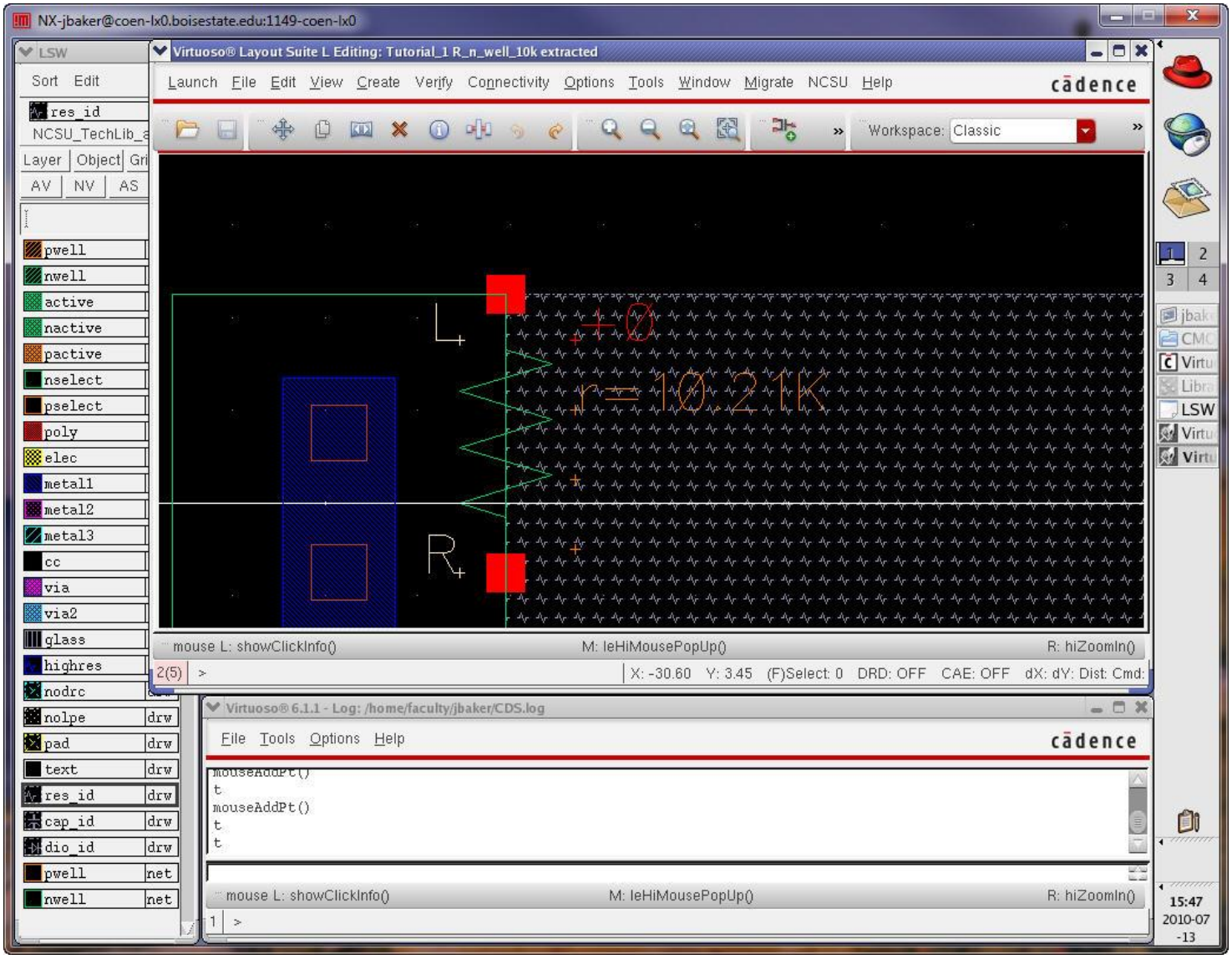
Next let's Extract the layout to determine the resistance's value (and to see if the setups match the 800 ohms n-well sheet resistance we got from MOSIS).
Go to Verify -> Extract



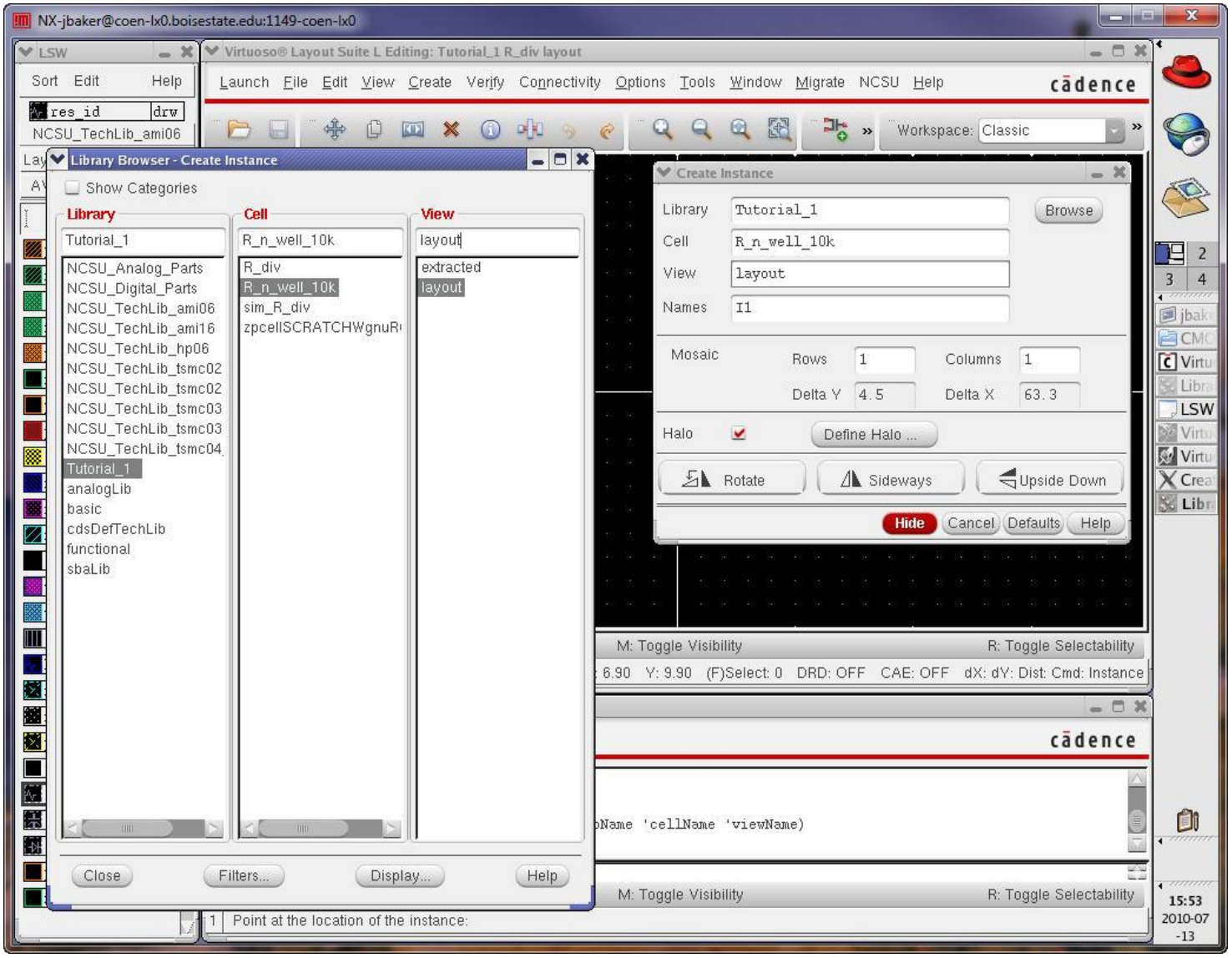
After hitting OK the window closes and an extracted view is created in the R_n_well_10k cell group.

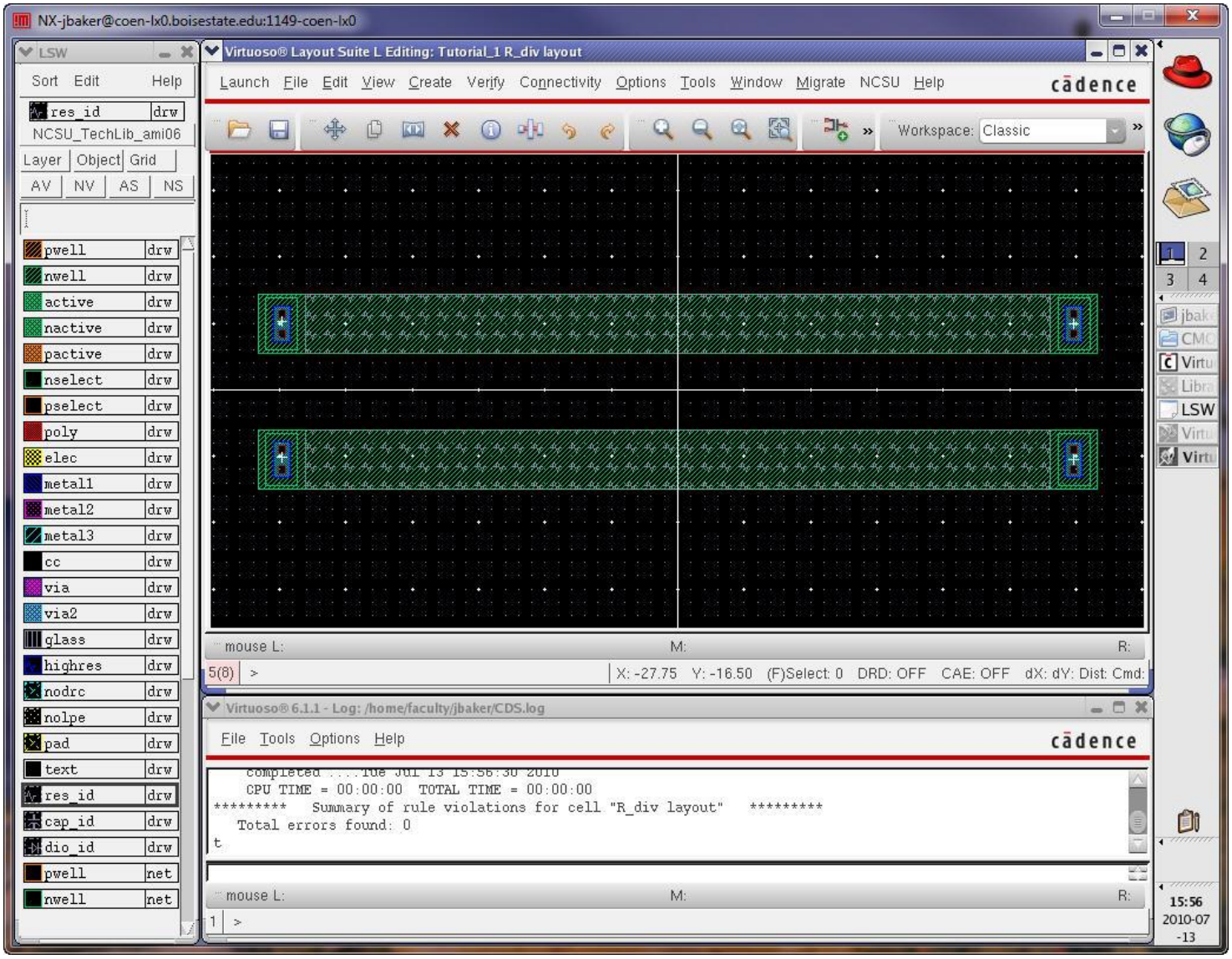


Open this view.
Zoom in to see the resistor's value is 10.21k.



Close the extracted view.
Save and close the layout view of the resistor.
We are now ready to draw the layout of the R_div cell.
Open the schematic view of the R_div cell (so we remember what is in it, like Pin Names).
Open the layout view for the R_div cell (nothing is in this cell).
Instantiate two of the 10k n-well resistor layouts
Remember pressing Esc leaves the instantiate command.
DRC your layout to ensure the resistors are far enough apart.

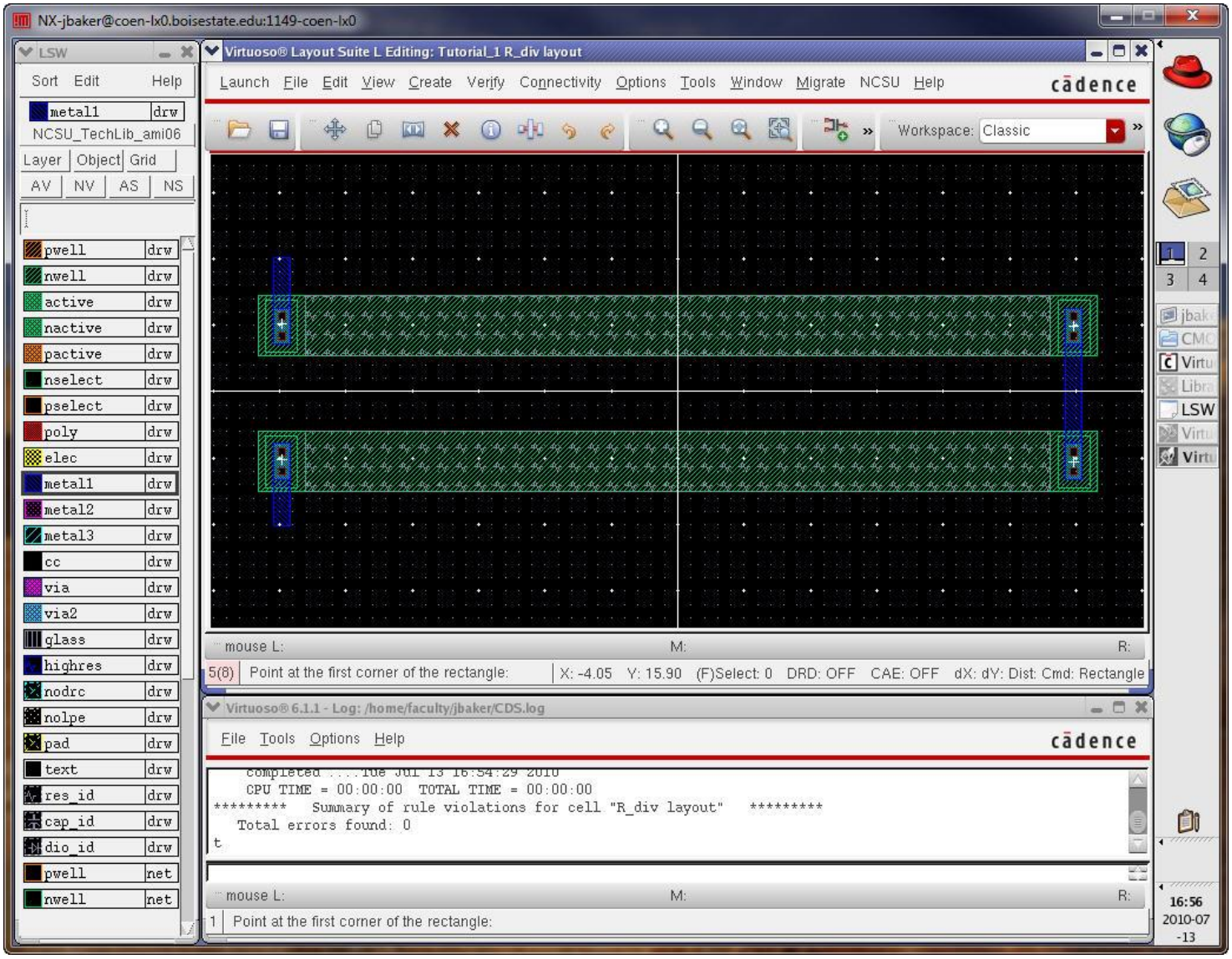

NX-jbaker@coen-lx0.boisestate.edu:1149-coen-lx0
LSW
Sort Edit Help
res_id drw
NCSU_TechLib_ami06
Virtuoso® Layout Suite L Editing: Tutorial_1 R_div layout
Launch File Edit View Create Verify Connectivity Options Tools Window Migrate NCSU Help
cadence
Workspace: Classic
Library Browser - Create Instance
Show Categories
Library
Cell
View
Tutorial_1
R_n_well_10k
layout
NCSU_Analog_Parts
NCSU_Digital_Parts
NCSU_TechLib_ami06
NCSU_TechLib_ami16
NCSU_TechLib_hp06
NCSU_TechLib_tsmc02
NCSU_TechLib_tsmc02
NCSU_TechLib_tsmc03
NCSU_TechLib_tsmc03
NCSU_TechLib_tsmc04
Tutorial_1
analogLib
basic
cdsDefTechLib
functional
sbaLib
R_div
R_n_well_10k
sim_R_div
extracted
layout
Close
Filters...
Display...
Help
Create Instance
Library Tutorial_1
Browse
Cell R_n_well_10k
View layout
Names I1
Mosaic
Rows 1
Columns 1
Delta Y 4.5
Delta X 63.3
Halo
Define Halo ...
Rotate
Sideways
Upside Down
Hide
Cancel
Defaults
Help
M: Toggle Visibility
R: Toggle Selectability
6.90 Y: 9.90 (F)Select: 0 DRD: OFF CAE: OFF dX: dY: Dist: Cmd: Instance
cadence
bName 'cellName 'viewName)
M: Toggle Visibility
R: Toggle Selectability
1 Point at the location of the instance:
15:53
2010-07
-13



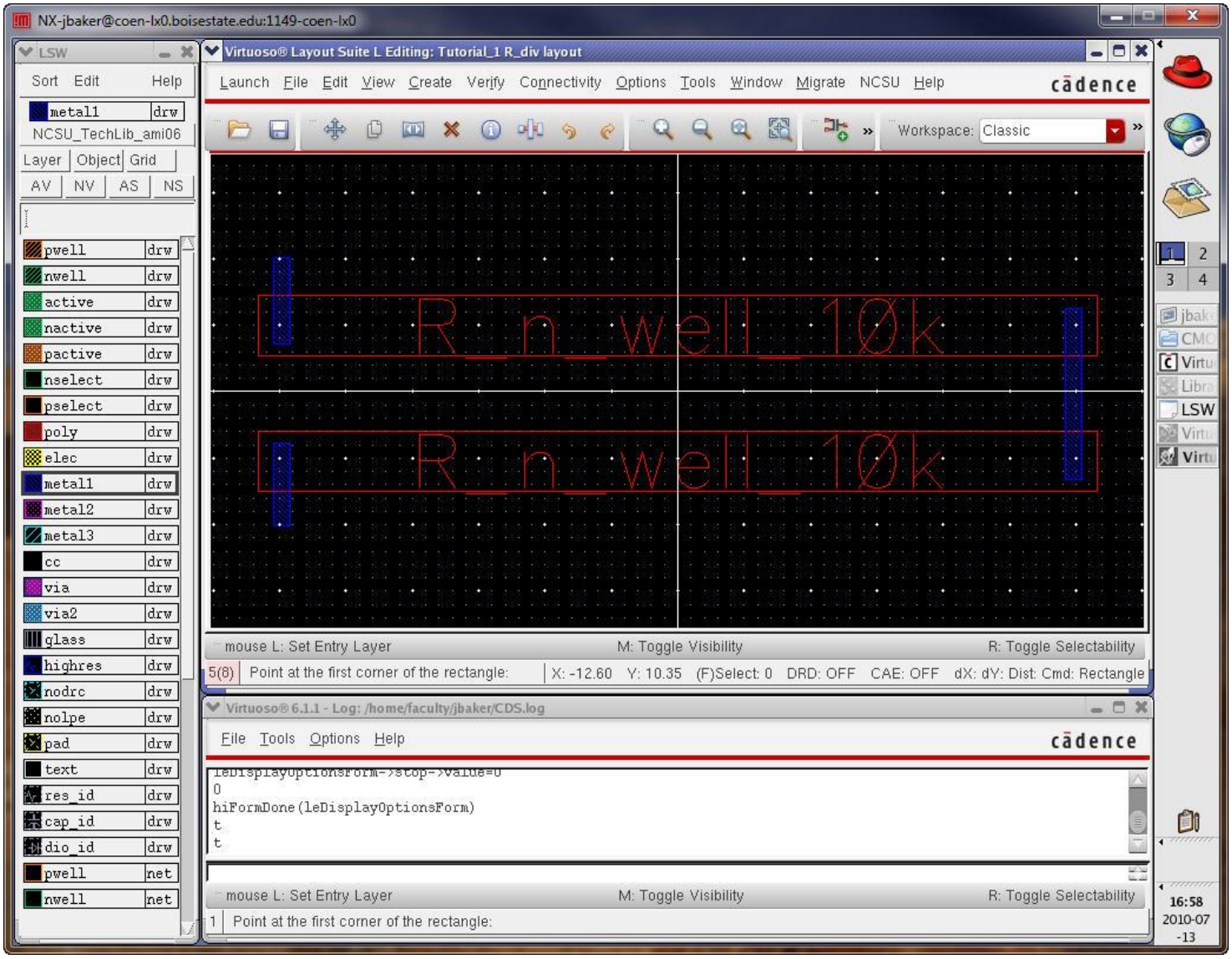
Next select the metal1 layer in the LSW and add rectangles to connect the resistors together and to connect to the Pins of the resistors.

***When drawing layouts always use the "drw" layer and not the "net" layer. Selecting, for example, the metal 1 **net** layer instead of metal 1 **drw**
layer is very easy to do and will waste your time! If a layout looks right to you, and something isn't working, check that you didn't use a net layer. ***

The rectangles don't have to overlap the Pins, just touch (abut) the metal1 Pins on the n-well resistors.
(I like to overlap the Pins with metal1.) One example is seen below.
DRC the design to ensure no errors.

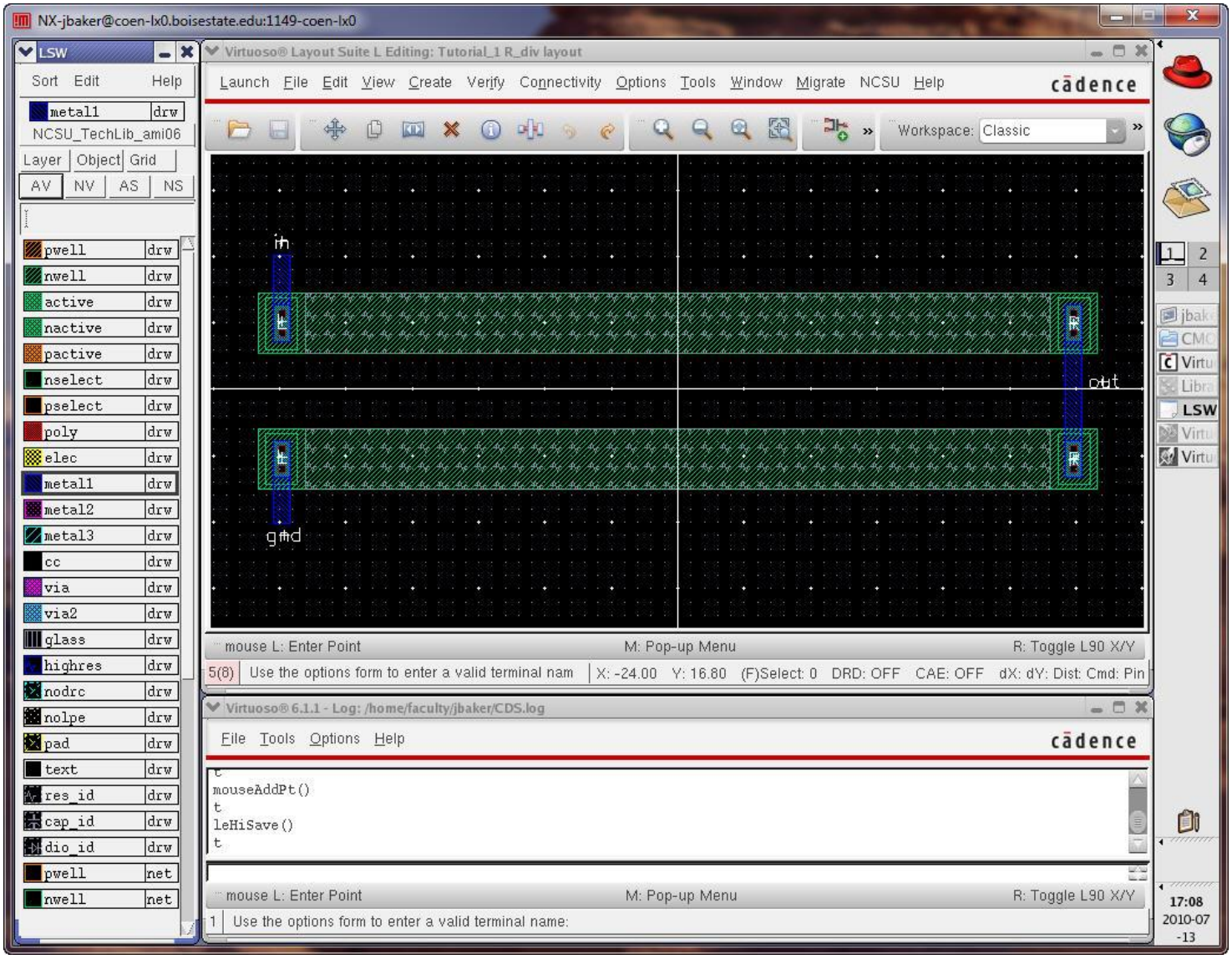


Pressing **e** and set the Stop Depth to zero results in showing the outlines of the cells.

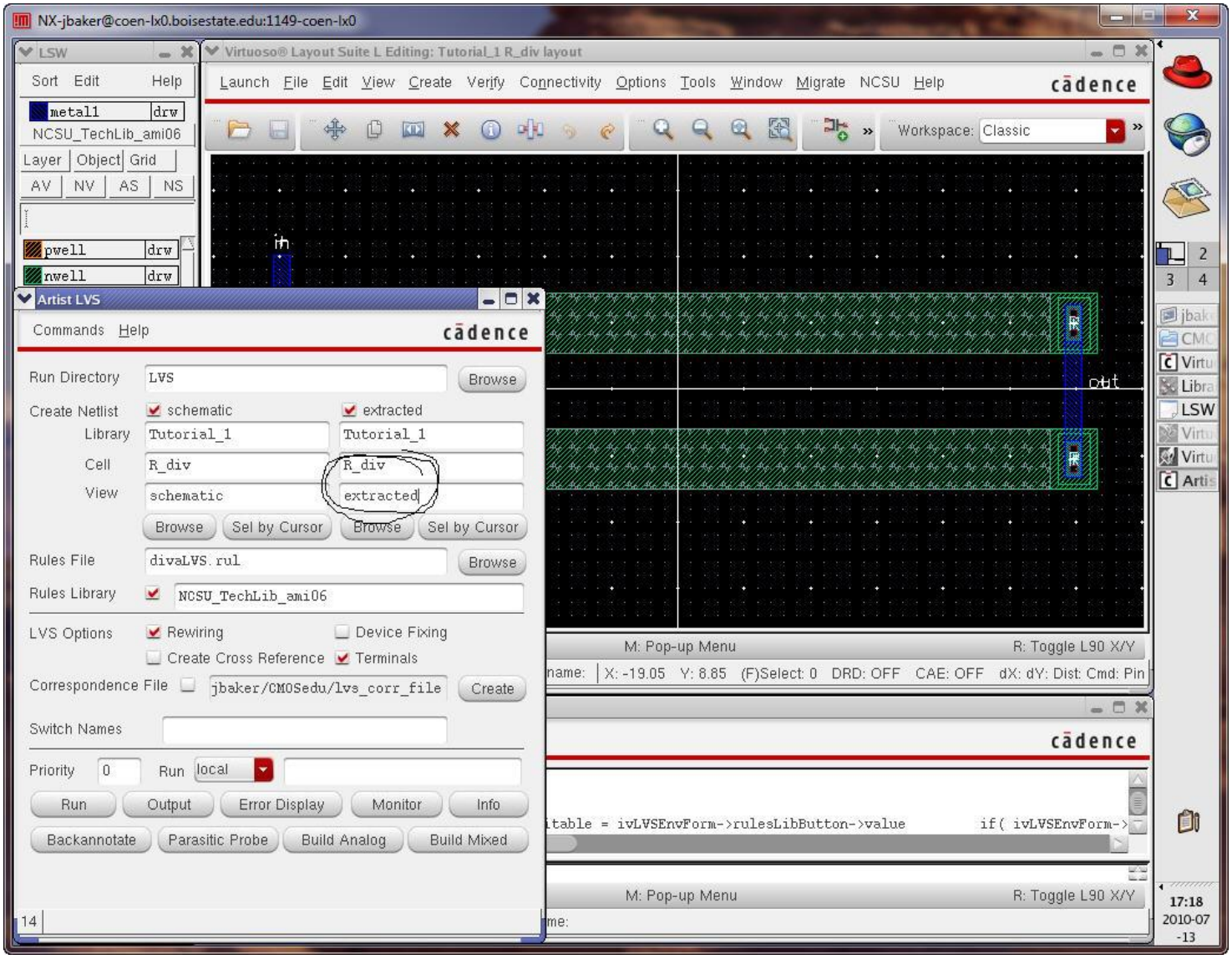


Press e again and set the Stop Depth back to 10. Also ensure, when the Display Options Window is open, that Pin Names is still set to display.

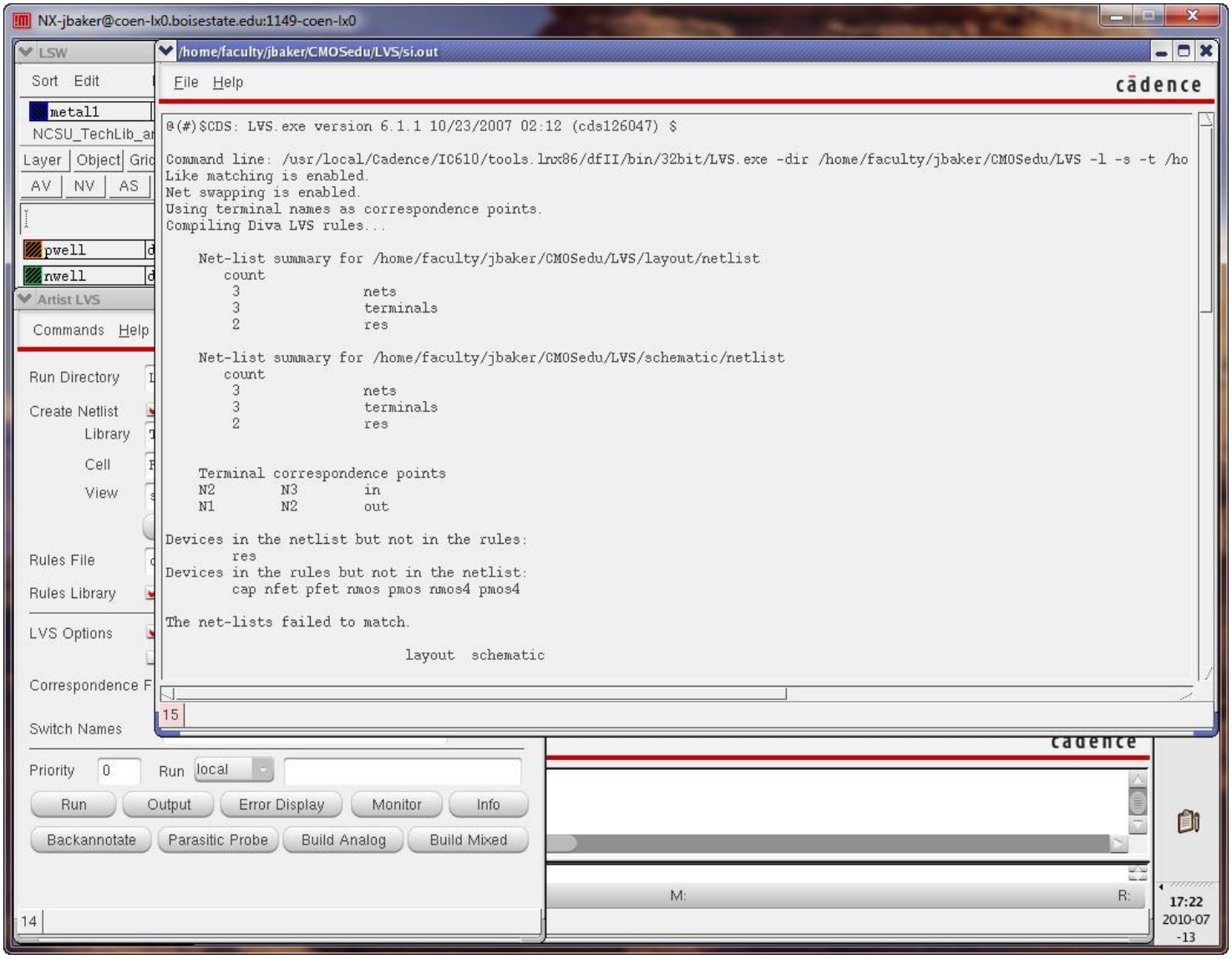
Next add Pins on the metal1 layer named in, out, and gnd. Set the rectangle size of the Pin to the same size as the metal1 seen above.



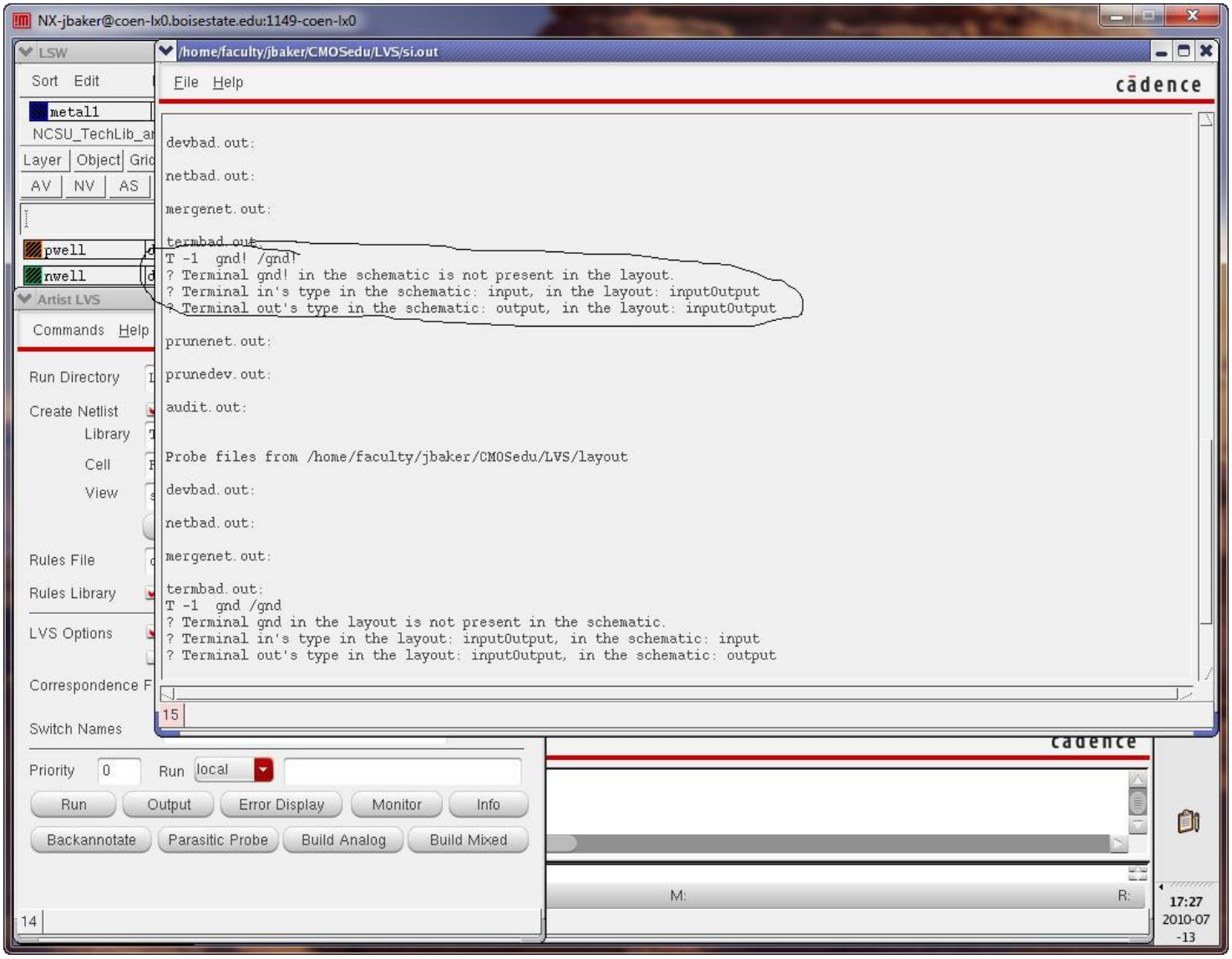
DRC your layout.
Extract your layout (we need to do this before we do a layout versus schematic, LVS)
Finally, select Verify -> LVS and set the extracted view's cell name (here R_div) and that its view is extracted as seen below.
Hit Run and OK to "Save Cellviews" (if asked)
When the LVS is done it will, hopefully, tell you it has succeeded (hit OK).



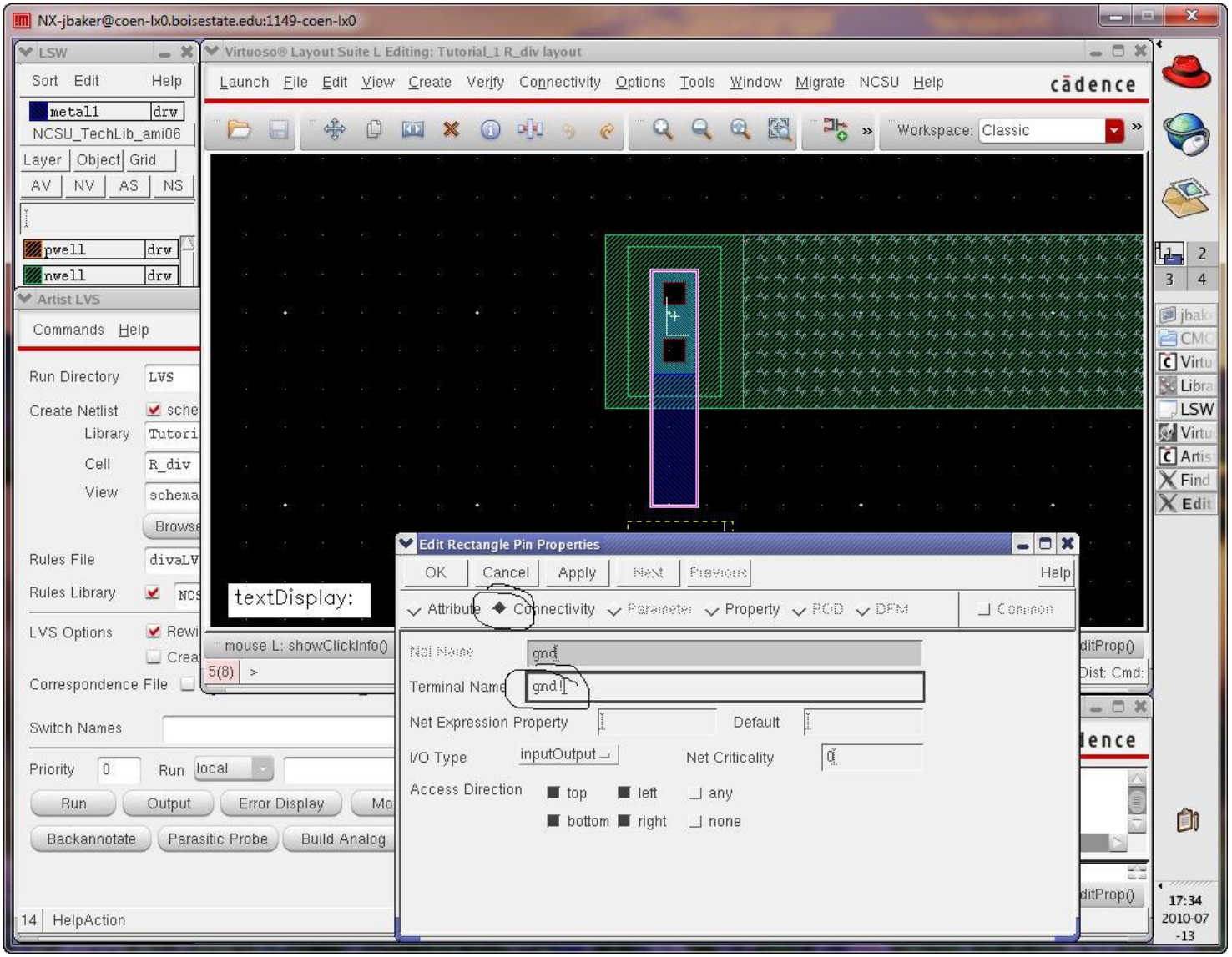
While the LVS succeeded to run this does not tell us if the layout and schematic match!
After pressing Output above we get the following.

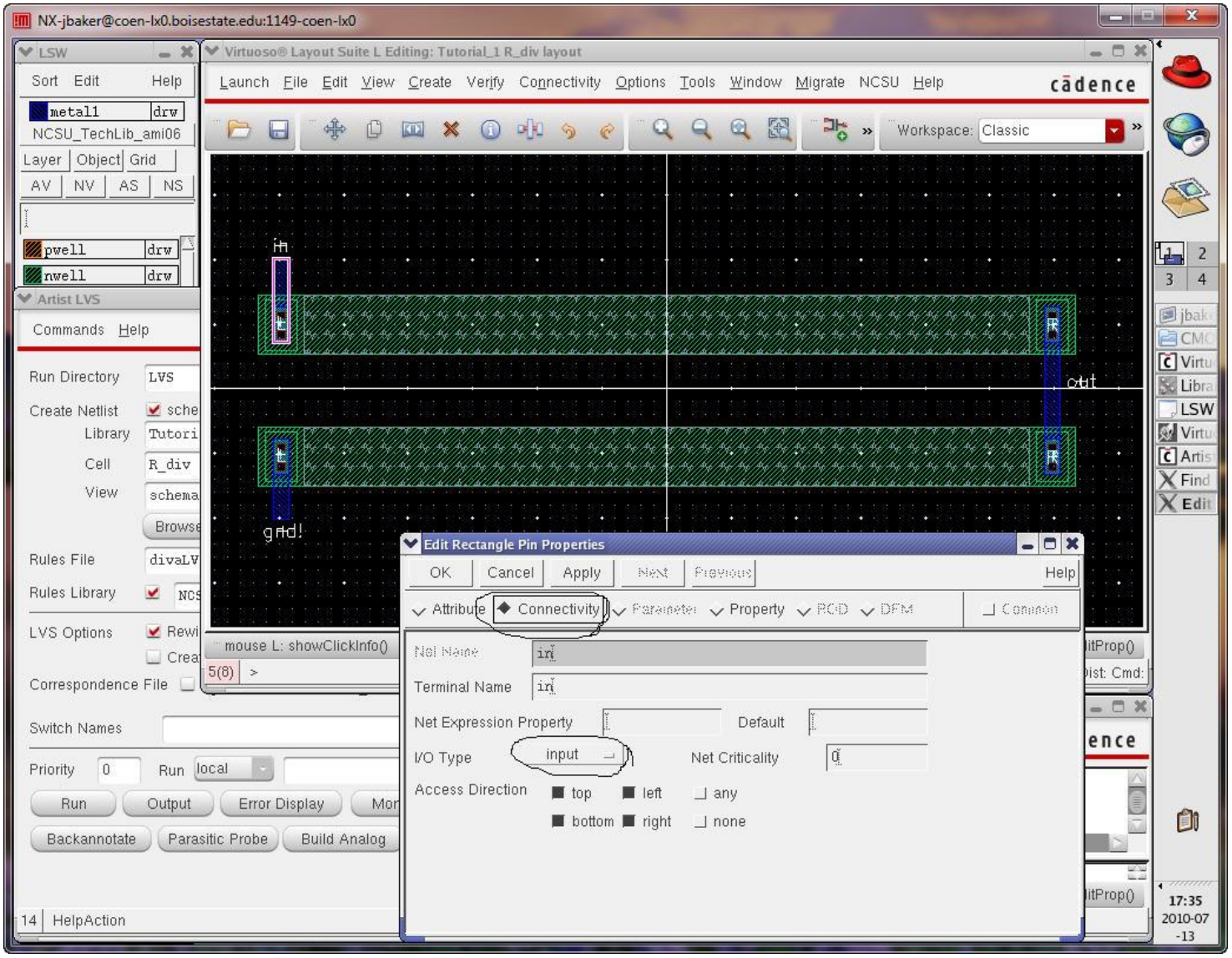


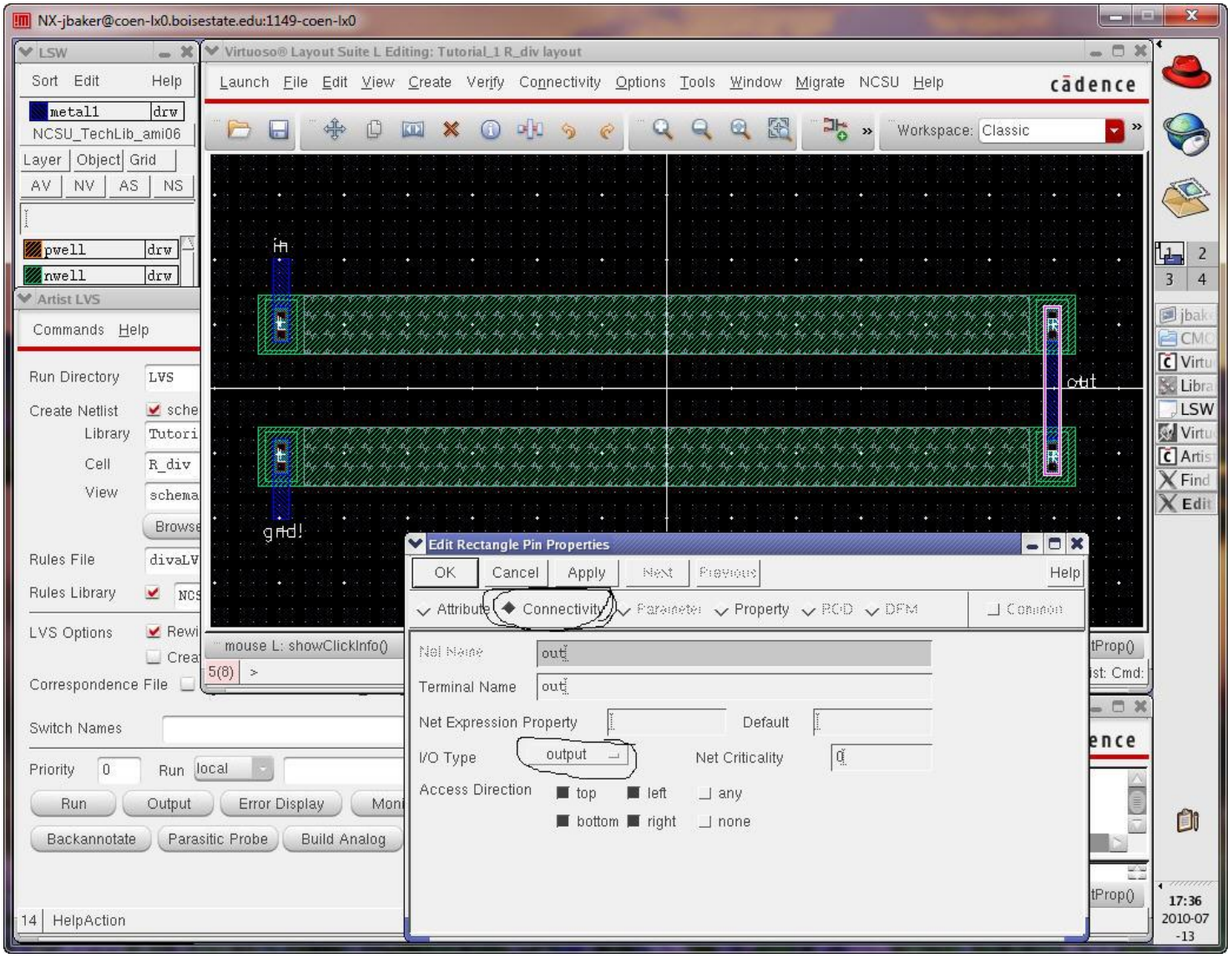
Scrolling down in the si.out file shows the problems.



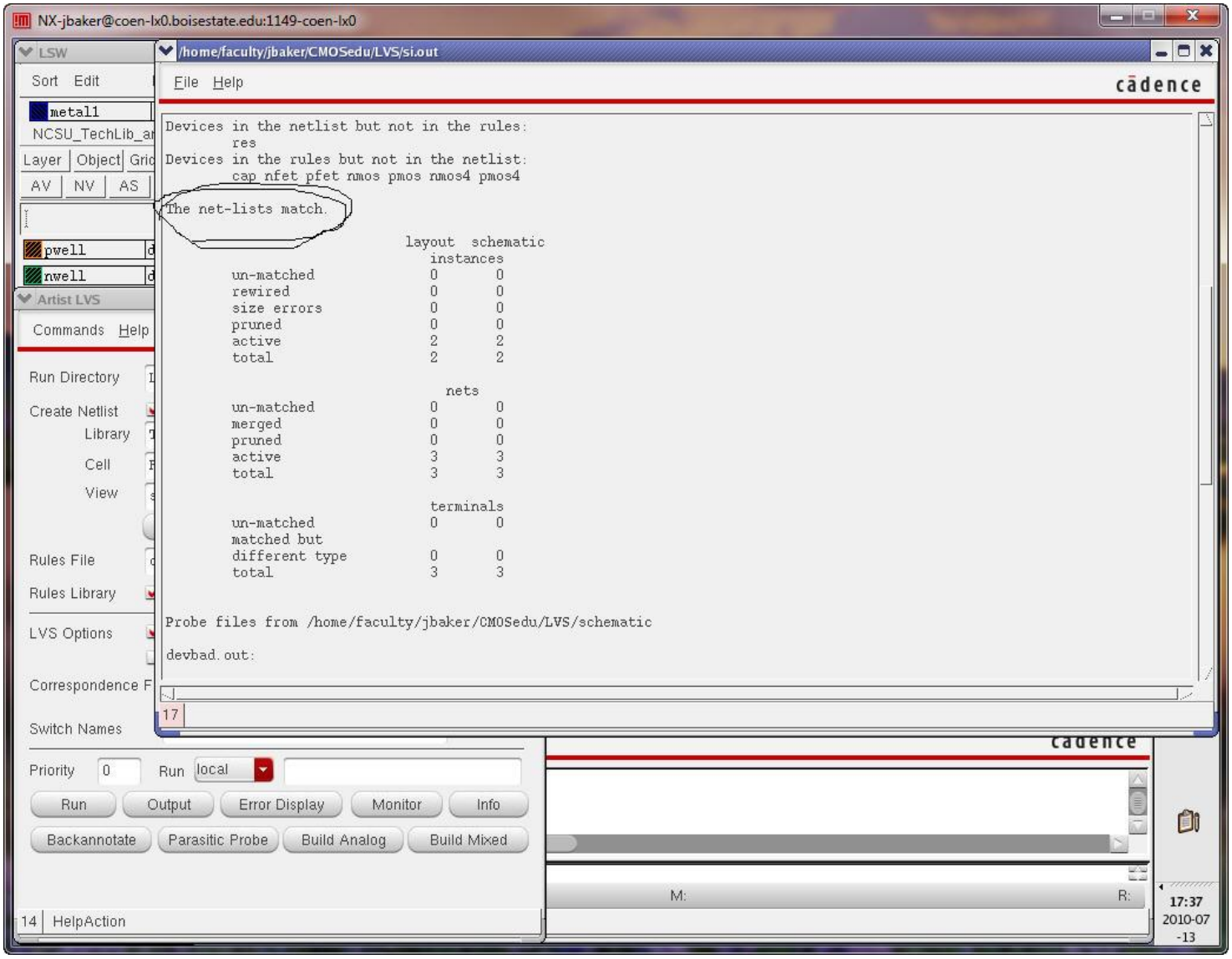
We should have labeled gnd in the layout gnd! (the exclamation point indicates a global value), and our in pin should have characteristics of input (not input/output)
and the out pin should be an output (not input/output).
Note that we could also select the Error Display button above and view the errors in the extracted view (often much easier than viewing text).


NX-jbaker@coen-lx0.boisestate.edu:1149-coen-lx0
LSW
Sort Edit Help
metal1 drw
NCSU_TechLib_ami06
Layer Object Grid
AV NV AS NS
pwell drw
nwell drw
Artist LVS
Commands Help
Run Directory LVS
Create Netlist sche
Library Tutori
Cell R_div
View schema
Browse
Rules File divaLV
Rules Library NCS
LVS Options Rewi
Crea
Correspondence File
Switch Names
Priority 0
Run local
Run
Output
Error Display
Backannotate
Parasitic Probe
Build Analog
14 HelpAction
Virtuoso® Layout Suite L Editing: Tutorial_1 R_div layout
Launch File Edit View Create Verify Connectivity Options Tools Window Migrate NCSU Help
cadence
Workspace: Classic
textDisplay:
mouse L: showClickInfo()
5(8) >
Edit Rectangle Pin Properties
OK Cancel Apply Next Previous Help
Attribute Connectivity Parameter Property PCD DFM Common
Net Name gnd
Terminal Name gnd!
Net Expression Property
Default
I/O Type inputOutput
Net Criticality 0
Access Direction top left any bottom right none
editProp()
Dist: Cmd:
17:34
2010-07-13


NX-jbaker@coen-lx0.boisestate.edu:1149-coen-lx0
LSW
Sort Edit Help
metal1 drw
NCSU_TechLib_ami06
Layer Object Grid
AV NV AS NS
pwell drw
nwell drw
Artist LVS
Commands Help
Run Directory LVS
Create Netlist
Library Tutori
Cell R_div
View schema
Rules File divaLV
Rules Library
LVS Options
Correspondence File
Switch Names
Priority 0
Run local
Run Output Error Display
Backannotate Parasitic Probe Build Analog
14 HelpAction
Virtuoso® Layout Suite L Editing: Tutorial_1 R_div layout
Launch File Edit View Create Verify Connectivity Options Tools Window Migrate NCSU Help
cadence
Workspace: Classic
in
out
gnd!
mouse L: showClickInfo()
5(8)
Edit Rectangle Pin Properties
OK Cancel Apply Next Previous Help
Attribute Connectivity Parameter Property RCD DFM Common
Net Name in
Terminal Name in
Net Expression Property
Default
I/O Type input
Net Criticality 0
Access Direction top left any
bottom right none
17:35
2010-07
-13



After saving the layout, **extracting the layout again**, and then running the LVS again we get the following.



This ends our first tutorial.
In this tutorial we've covered the fundamental operation of Cadence.
Mastering the topics in this tutorial is important before moving on to the other Tutorials.

For your reference the Tutorial_1 directory is available in Tutorial_1.zip.
Again, to use this design directory put the Tutorial_1 folder (unzipped from above) in the CMOSedu directory and then add, to the cds.lib

DEFINE Tutorial_1 $HOME/CMOSedu/Tutorial_1

Return