

# ELEN90062 High Speed Electronics

## Workshop One

### Cadence Tutorial

Welcome to Workshop 1 for High Speed Electronics. This workshop is simulation-based and is implemented using the Cadence software. The objective of this workshop is to give you an opportunity to familiarise yourself with using Cadence for electronic circuit design and simulations. You will use this knowledge during the assessed Cadence workshops in the coming weeks.

## 1 Instructions

This workshop is not assessed. However, attending the workshop is highly recommended to prepare yourself for the assessed Cadence workshops that start next week.

## 2 Introduction to Cadence Software

Cadence, headquartered in San Jose, California, is one of the world's leading softwares used in electronic design automation (EDA) industry. The software is primarily used to design chips and printed circuit boards. During the High Speed Electronics workshops, we will be using the Cadence Virtuoso Platform. The Virtuoso platform is mainly used in the industry for analog, mixed-signal, RF, and standard-cell designs, memory and FPGA designs.

## 3 Using Cadence Software to Build and Simulate an Inverter Circuit

### Starting Cadence

- (a) In the start menu, select *MobaXterm*
- (b) Select *session* in MobaXterm
- (c) Select *RDP* as your session
- (d) Enter the following and click *OK*:  
Host: *lab-s01.ee.unimelb.edu.au*  
Username: *Your university student username*  
Port: *3389*
- (e) You will be prompted to enter the password. Type in your university student password to login.
- (f) Click *Applications* → *Education* → *Cadence Virtuoso*
- (g) The cadence will open two windows: the CIW (command interface window) and What's New in 5.0 window. Close the latter.

In the Library Manager you can see that things in Cadence are organized into libraries, cells and cellviews. Libraries are used to categorize cells that belong together. You can look through the analogLib and basic libraries which contain all of the basic analog components that we will use: voltage sources (vdc), current sources (idc), grounds (gnd), resistors (res), capacitors(cap) and other basic things. Cells are names of individual parts/circuits that you can place in a schematic. You can place cells within cells to create a hierarchy. Views/cellviews are ways of looking at a cell. We will use the schematic and symbol views. Close the Library Manager for now.

## Creating a Design Library

First we need to create a library for our Cadence work as shown in Figure 1.

- On the Command Interface Window (CIW) click  
*Files → New → Library*
- Choose a library name.
- Click on *Don't need process information.*
- Click *OK*:

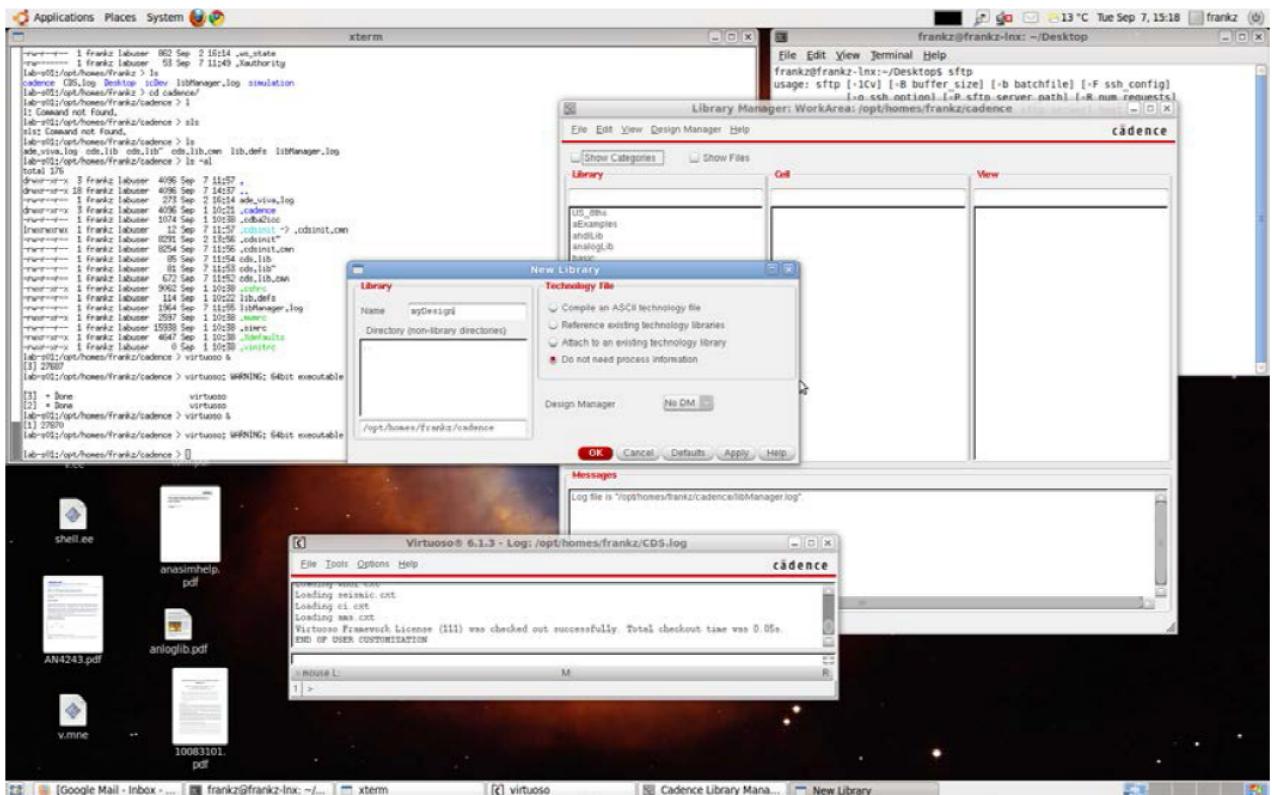


Figure 1: Creating a design library

## Creating testbench schematic in the design library

Now we will open the library manager and create a new schematic.

- On the Command Interface Window (CIW), click  
*Tools → Library Manager*  
 You should see the new library that you created under the *Library* section. If you do not see your library, then click on *Library Manager(LM) → View→ Refresh*.
- Click on your library.
- With your library selected, on the Library Manager, click  
*File → New → Cellview*.
- This will open up a window, *New File*, as shown in Figure 2. In the *New File* window, enter the following details.  
*cell - a name of your choice*  
*view - schematic*
- Click OK

This will open a blank Composer schematic capture window. Along the left and the top you will see different buttons and menus that perform different functions. Now is an excellent time to play with all of the buttons and menus, except for the Tools menu. You will notice that many of the command have letters beside them which are their hotkeys, or bindkeys in Cadence parlance.

Before we start here are some useful commands that we will use later.

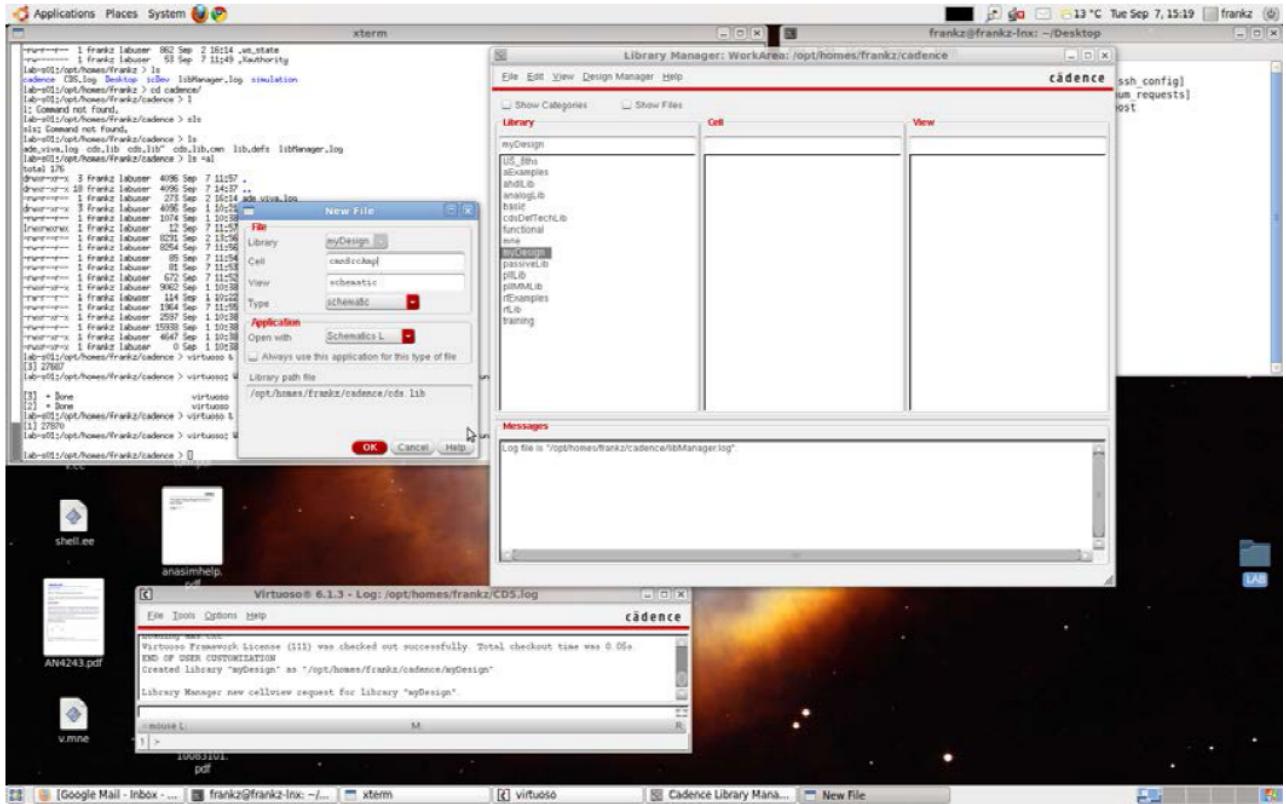


Figure 2: Creating a new schematic

- i - instantiate, creates a new object in the schematic.
- p- creates pins to allow signals to leave the schematic.
- m- moves objects and drags their wiring with them
- M- Moves objects after disconnecting their wires.
- l- allows you to create labels for wires so that they don't get stuck with the default names, like net994875.
- q- opens a dialog box that shows an object properties.
- w- allows you to create wires. After you type w, left click to start a wire, left click to make a corner and double left click to end the wire.

Some commands like copy, move and wire have dialog boxes with more options that can be hidden. (F3) displays these dialog boxes. Also note that Cadence commands use the bottom line of window as an information bar.

## Creating an Inverter Circuit

During this part of the tutorial, you will implement and simulate an inverter circuit as shown in Figure 3. All the components used in this simulation, NMOS, AC and DC sources, and the resistors are from the analogLib library.

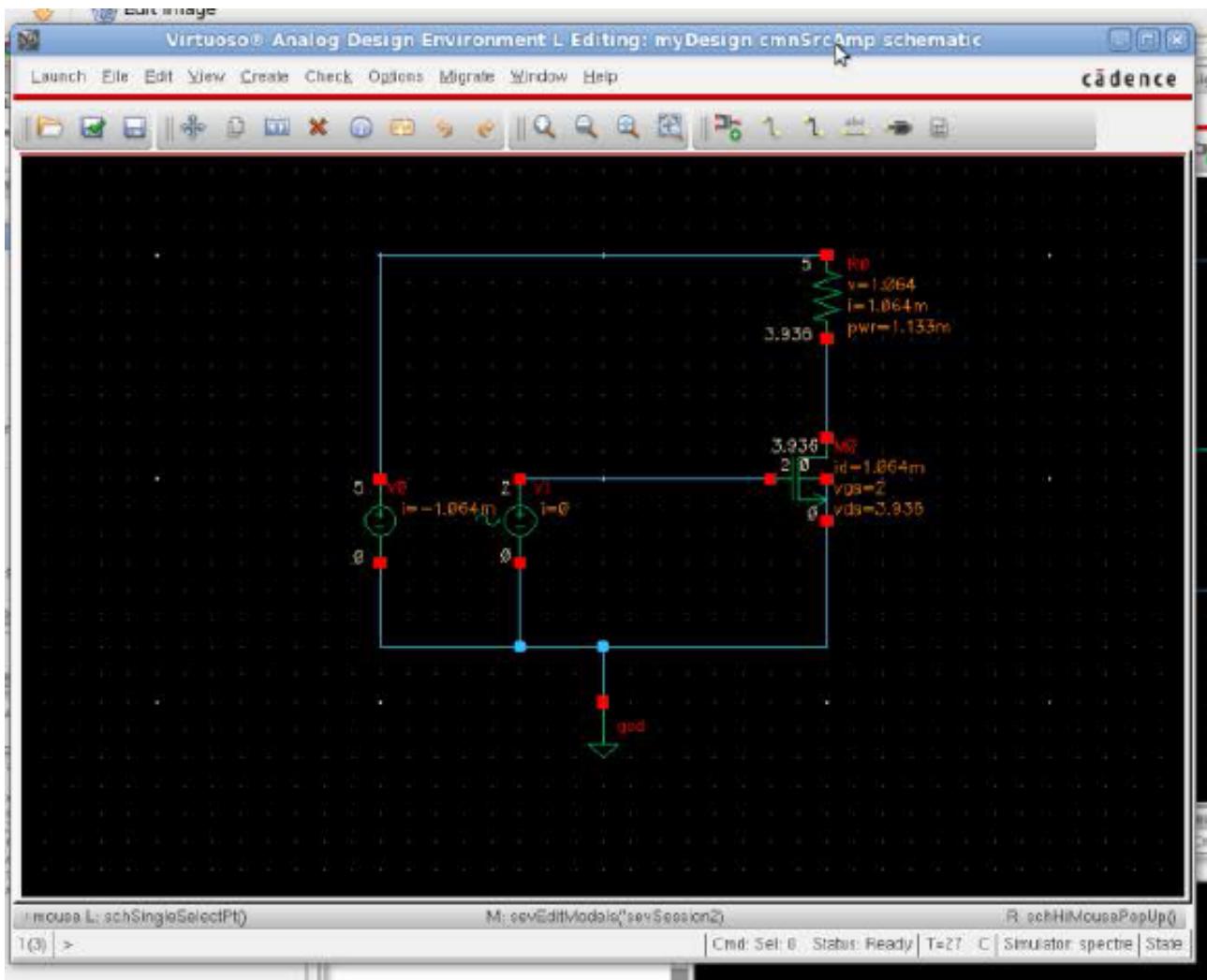


Figure 3: Inverter Circuit Schematic

- Step 1: place an NMOS instance from the analogLib library  
Click *Create → Instance* or just press *i* (the hotkey to place an object in the schematic)
- Step 2: The *Add Instance* window, Figure 4, will open. Click browse and it will take you to the Library Manager window
- Step 3: Select *Library → analogLib, Cell → nmos4, view → symbol* and double click on symbol
- Step 4: Fill out the object parameters, model name, width, and length as shown in Figure 4. Then click on the schematic window to place the component.
- Step 5: Add ground - follow Steps 1-3 to add a "gnd" object from the analogLib library as shown in Figure 5. Click on the schematic window to place the component.
- Step 6: Add a DC voltage source - follow steps 1-3 to add "vdc" object from the analogLib library as shown in Figure 6. Fill out the parameter DC voltage as per Figure 6. Click on the schematic window to place the component.
- Step 7: Add a AC voltage source - follow steps 1-3 to add "vsin" object from the analogLib library as shown in Figure 7. Fill out the parameters, offset voltage, amplitude, and the frequency as per Figure 7. Click on the schematic window to place the component.

- Step 8: Add a resistor - follow steps 1-3 to add "vstin" object from the analogLib library as shown in Figure 8. Fill out the parameter value resistance as per Figure 8. Click on the schematic window to place the component.
- Step 9: Connect the component using wires - use the wire command by pressing *w* to create wires to wire everything together. The wiring command works as follows. Left click with the mouse to attach the wire to a pin, a terminal or another wire. Double click to end wire without attaching it to anything. Hit escape to stop wiring. You have now completed the circuit implementation.

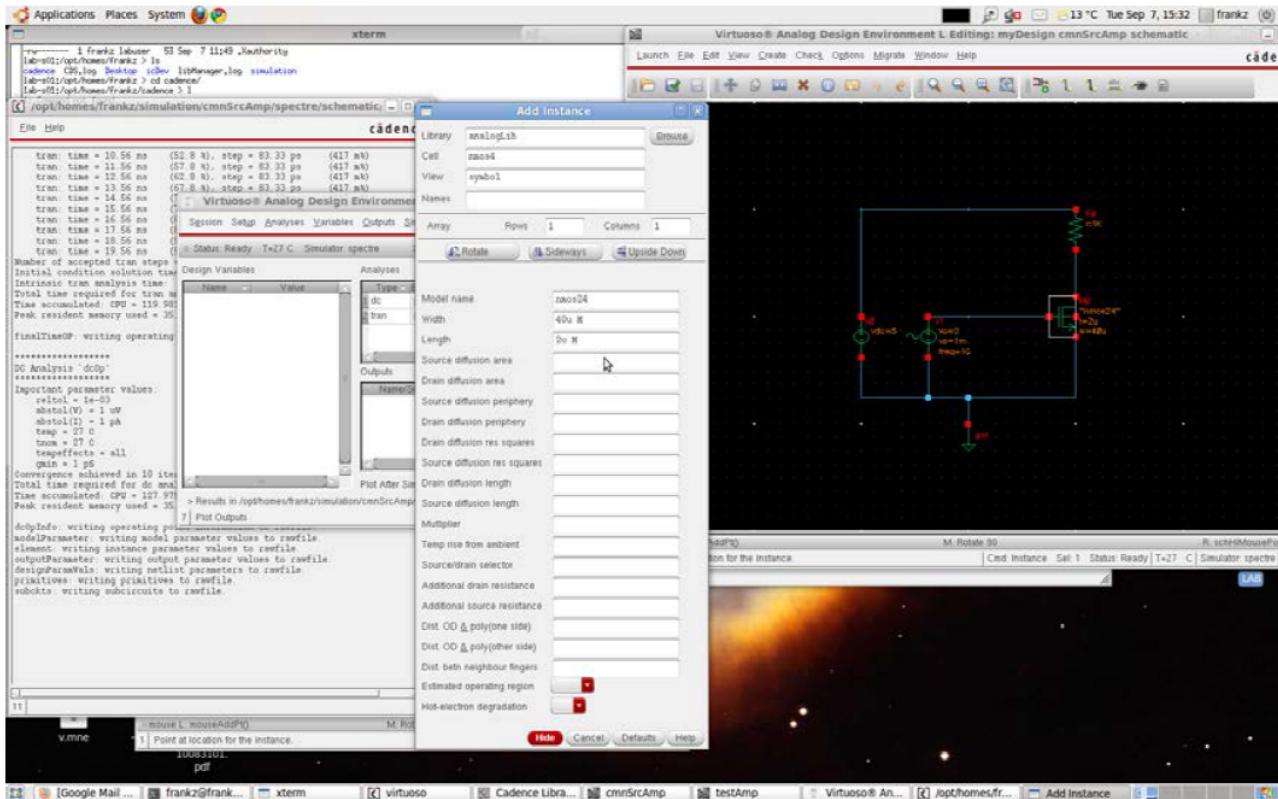


Figure 4: Place NMOS object

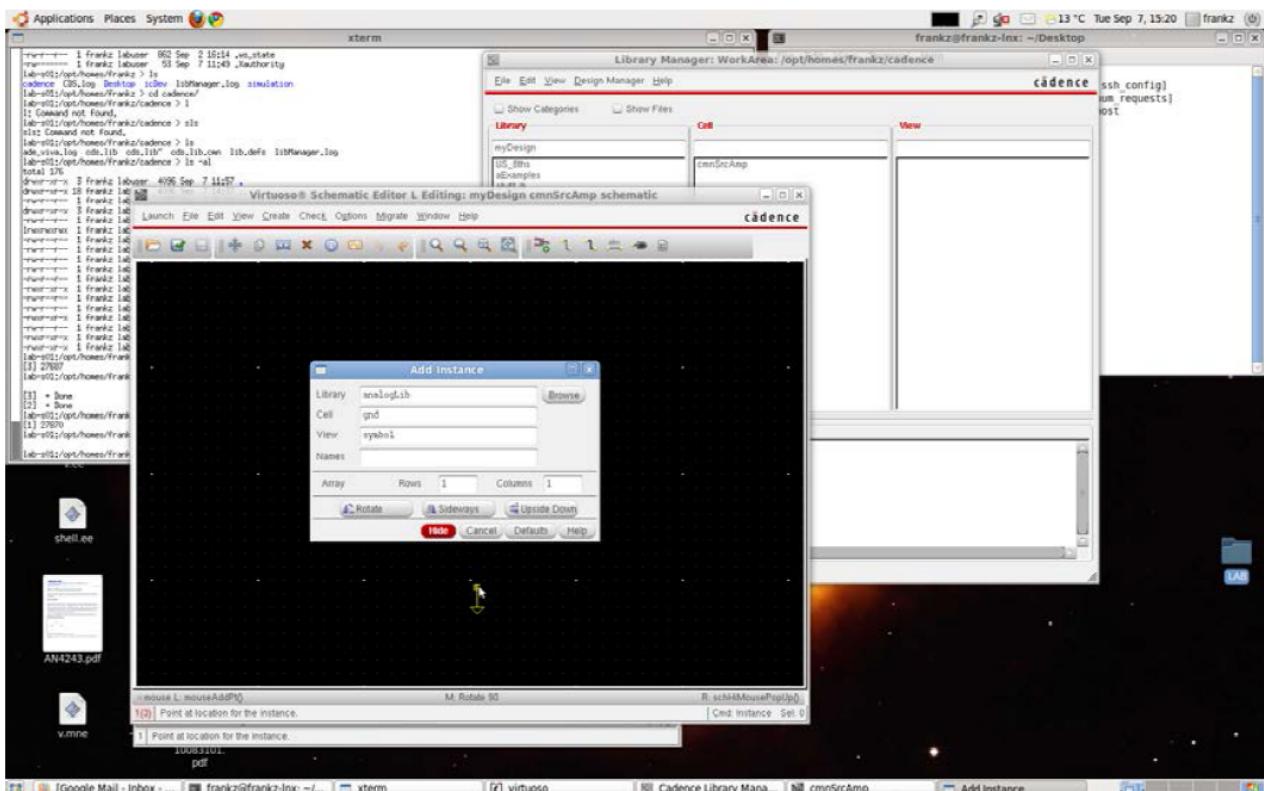


Figure 5: Adding Ground

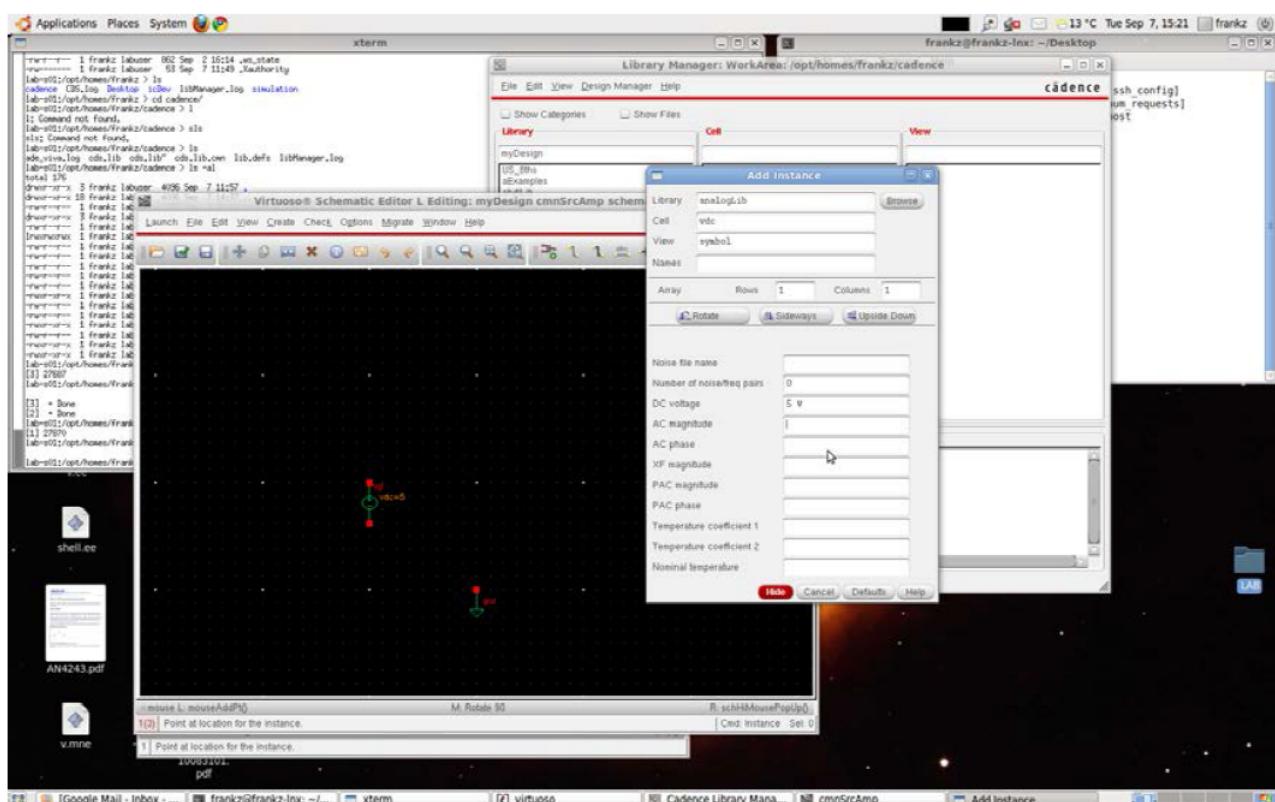


Figure 6: Adding the voltage source "vdc"

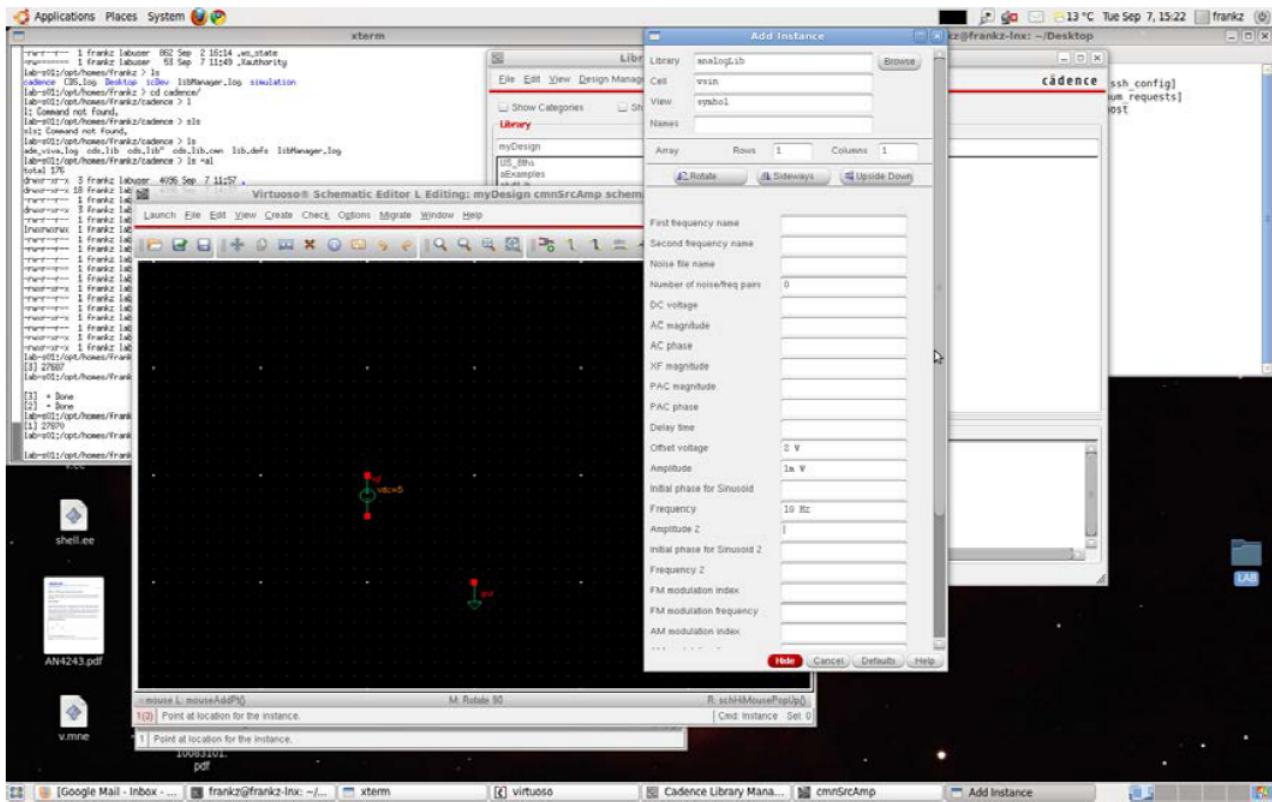


Figure 7: Adding the voltage source "vsin"

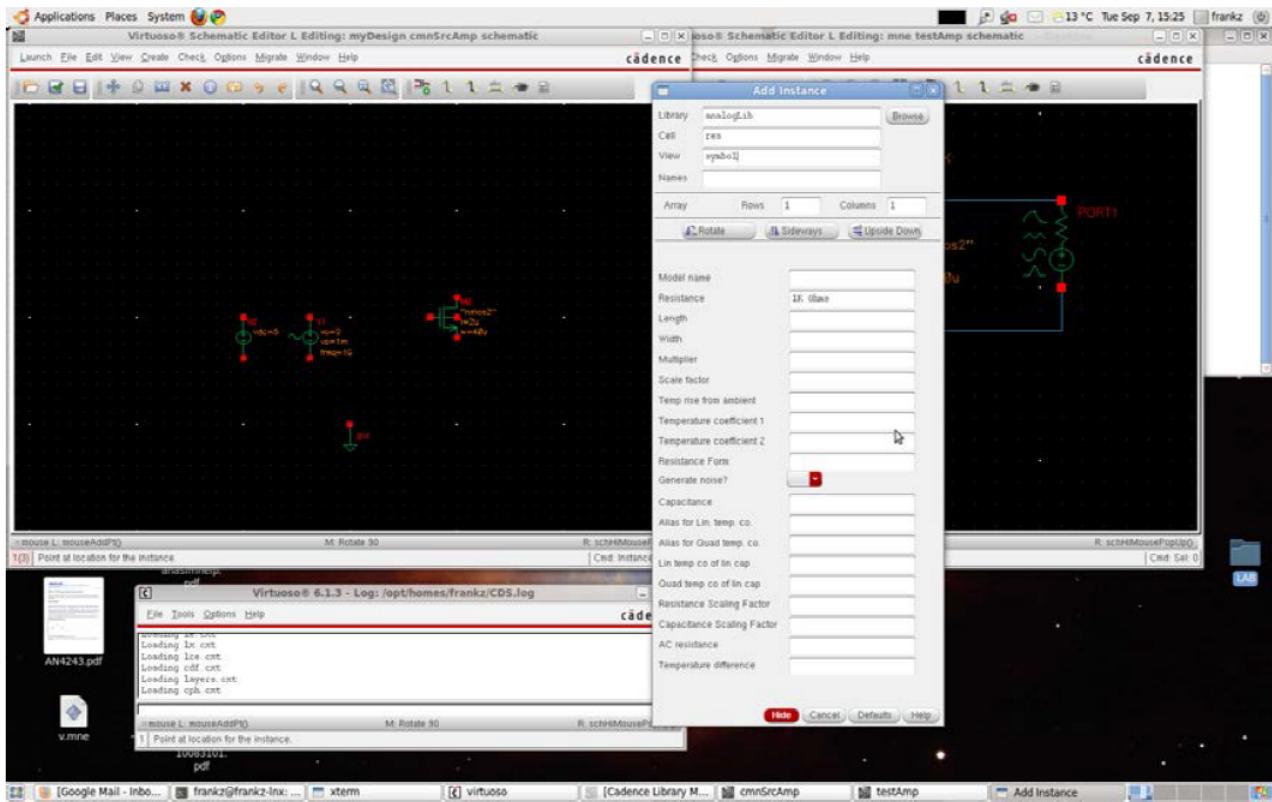


Figure 8: Adding the resistor "res"

## Analog Design Environment (ADE)

- Step 10: Starting ADE - click *Launch*→*ADE L*
- Step 11: Add model libraries - In the ADE window, click *Setup* → *Model Libraries*. This will prompt you to *Model Library Setup* window shown in Figure 9.  
Select *Click here to add model file* and copy and paste the following library path. Then click *OK*  
`/opt/apps/cadence/IC613/tools.lnx86/dfII/samples/artist/models/spectre/basicMos.scs`
- Step 12: Choose DC and transient analysis - In the ADE window, click *Analyses* → *Choose*. This will prompt you to *Choosing Analyses* window shown in Figures 10 and 11. Select the DC/TRansient analyses and fill the simulation parameters as shown in Figures 10 and 11.
- Save and Run the simulation.
- Discuss with your demonstrator on how to plot the graphs shown in Figure 12.

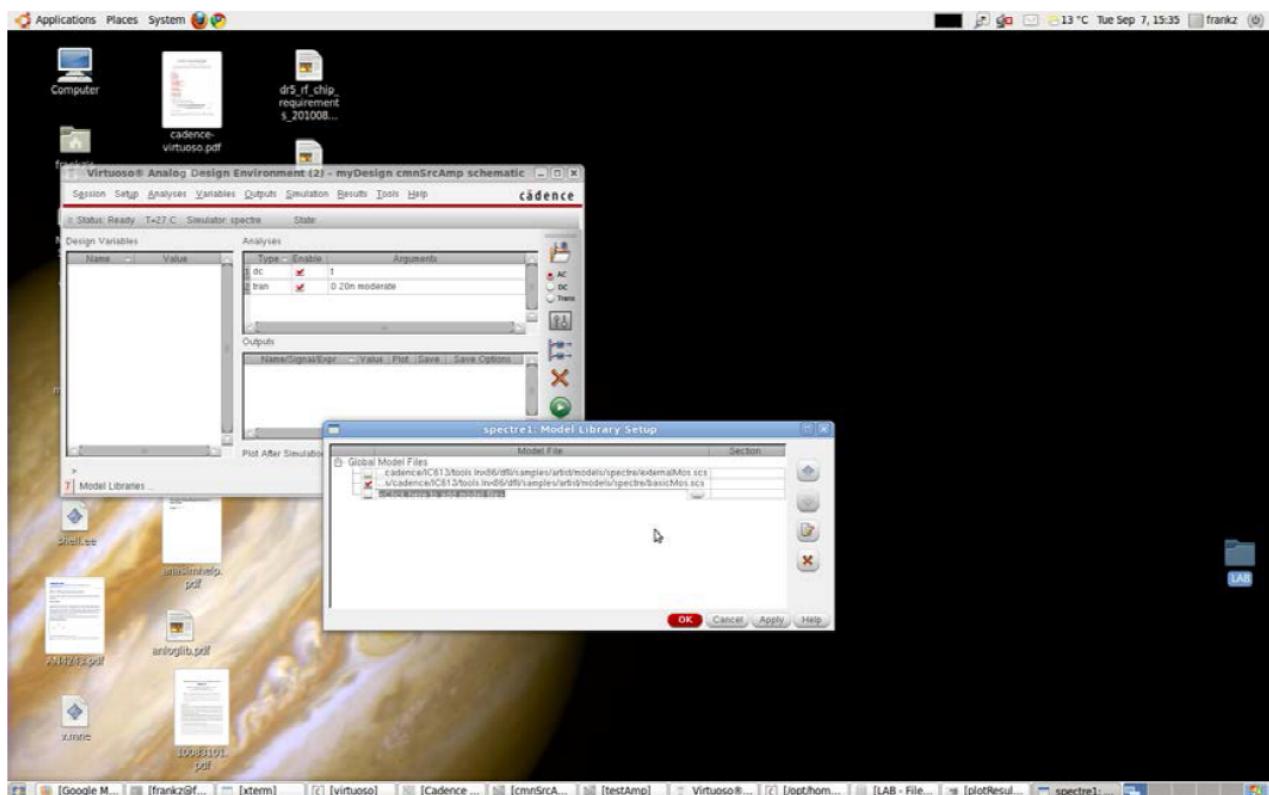


Figure 9: Adding model libraries

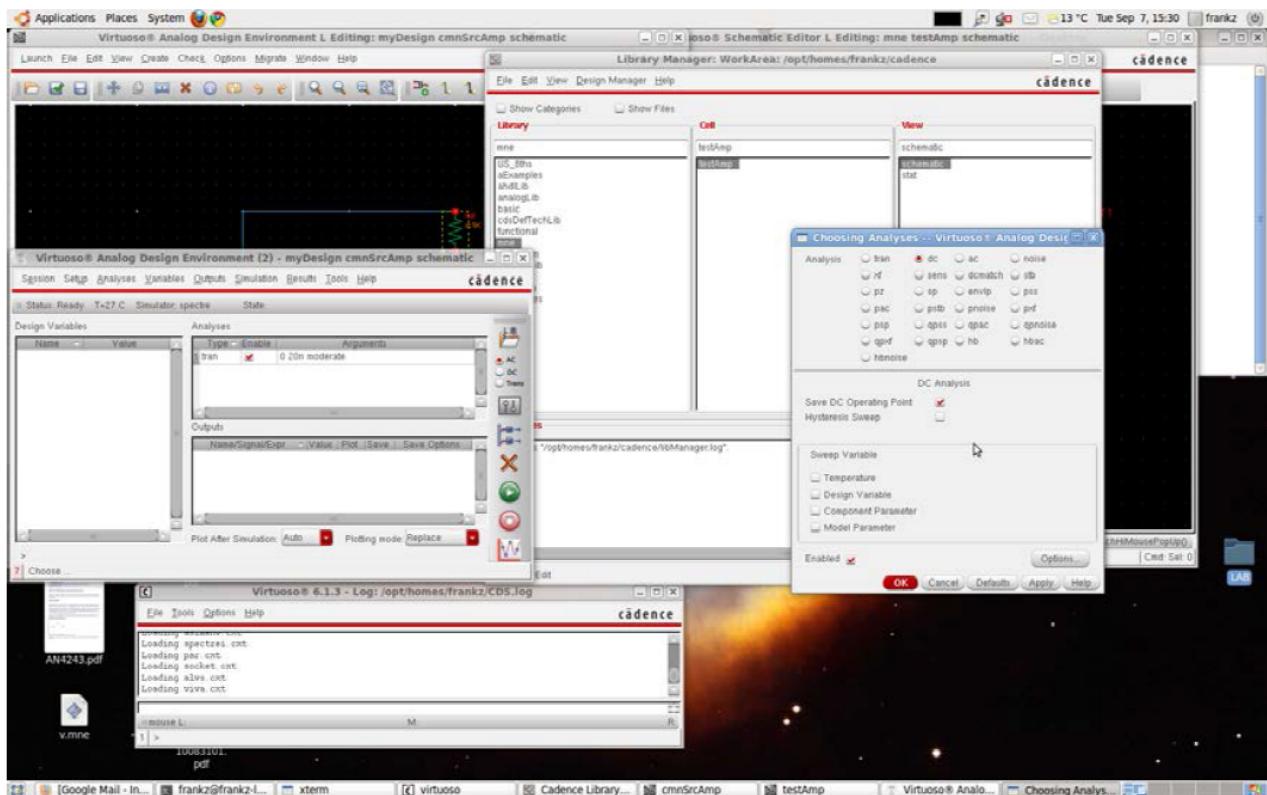


Figure 10: Adding DC analyses

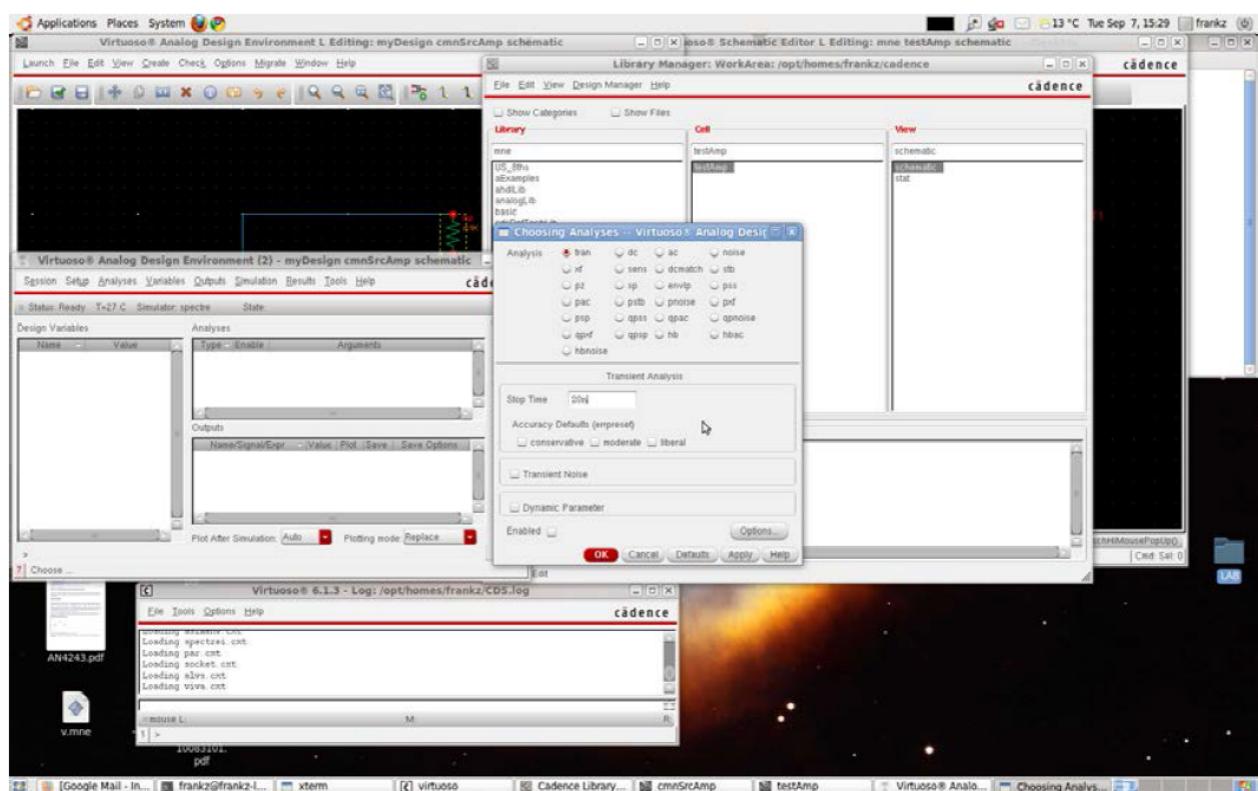


Figure 11: Adding transient analyses

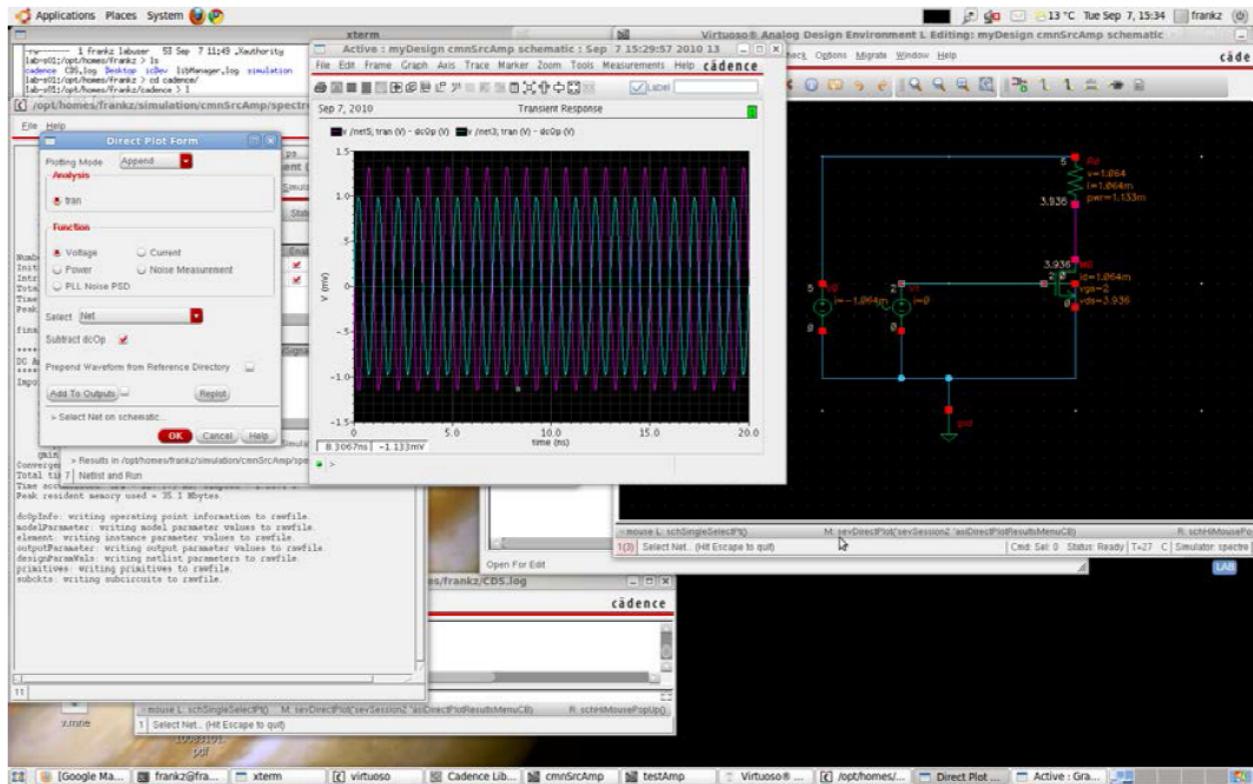


Figure 12: Plotting different network parameters