York University
Department of Electrical Engineering and Computer Science
EMIL Tutorial Series

Tutorial #1   Basic Analog Simulation in Cadence

In this tutorial we step through how to start Cadence (or at least a very basic version of it), how to define a library linked to an appropriate technology file, how to build a schematic and then how to simulate it with Spectre.

1   Starting Up Cadence

Create a new directory. I called mine `SM_IBM51`. Start-up the Cadence tools

`cad-ibm51`

2   Creating a Design Library

If you only seen on window, it is the icfb window. You now want to create a new library and the easiest way to do that is by creating a library via the **Library Manager**. In the icfb go to **Tools → Library Manager...**.

In the Library Manger window create a new library called SM␣Amp by going to **File → New → Library...**, entering SM␣Amp in the *Name* field and pressing the **OK** button.

In the **Technology File for New Library** window that pops up click the *Attach to an existing techfile* button and hit **OK**.

In the **Attach Design Library to Technology File** window that pops up, select `cmrf8sf` as the *Technology File* and hit **OK**. You'll notice that the SM␣Amp library is created in your Library Manager window.
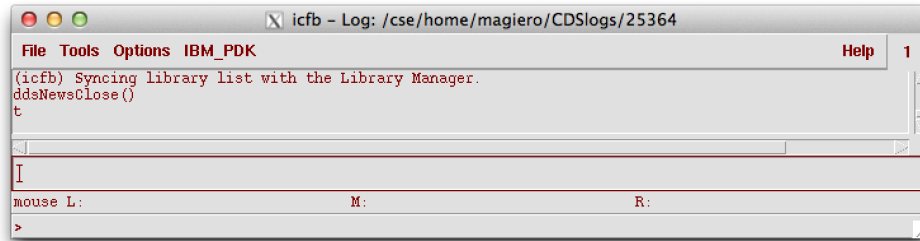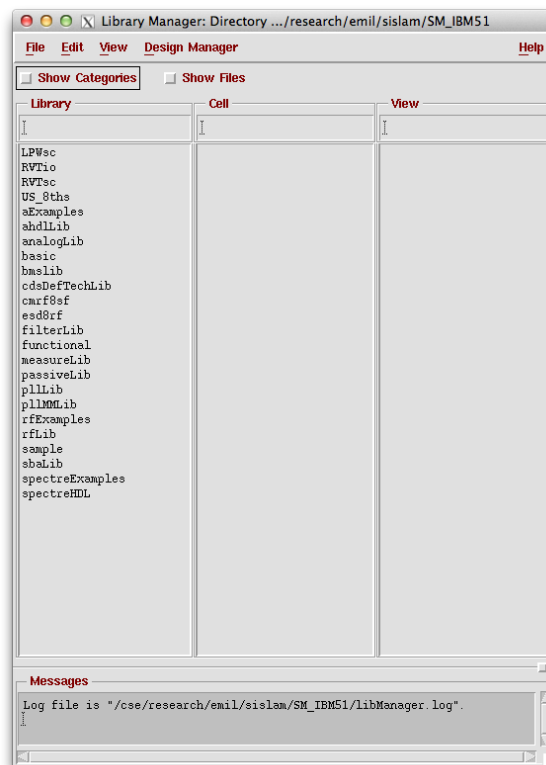
Figure 1: The icfb window
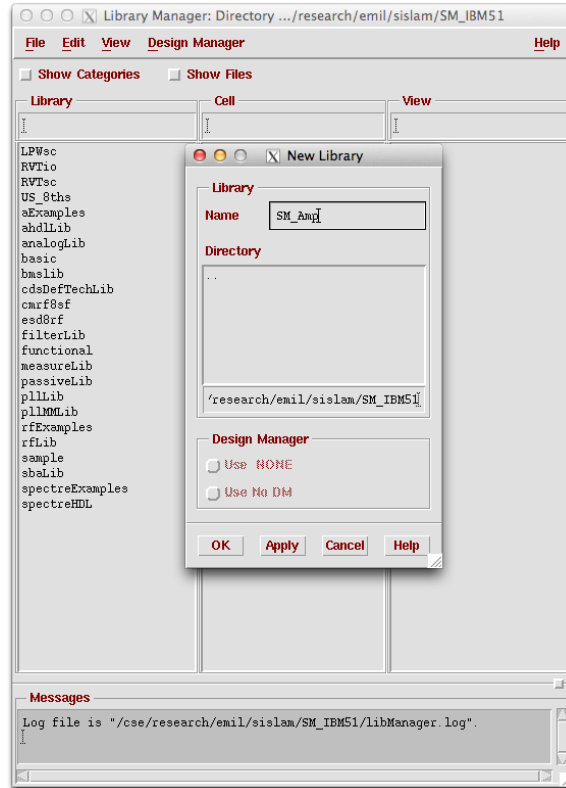

Figure 2: The **Library Manager** window
.

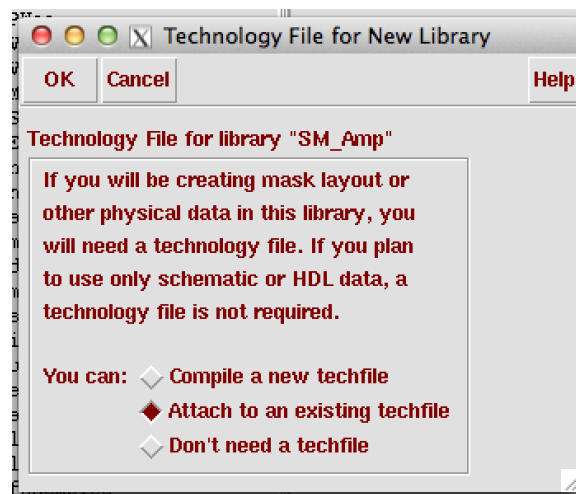Figure 3: Enter SM_Amp as the name of your new library.
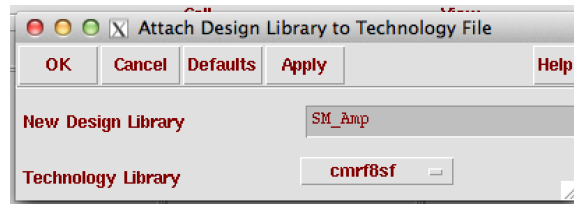.

Figure 4: Attach to an existing techfile.
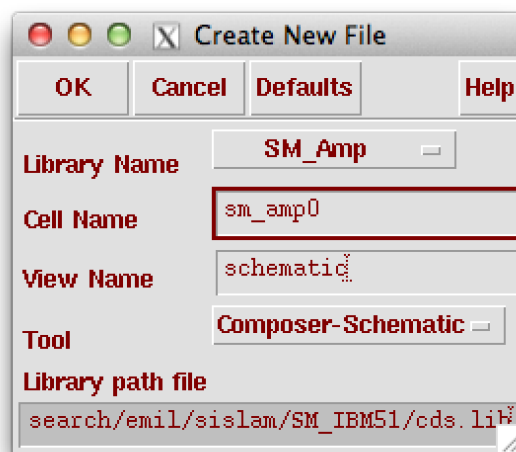.

Figure 5: Attach the right technology file.
.

Figure 6: Enter the file name of the schematic you wish to design.
.

3 Creating a Schematic View

In the Library Manger window select your new `SM_Amp` library and from the window menu follow **File → New → Menu...**. In the **Create New File** window that pops-up, enter the file name of the circuit schematic that you wish to design in the *Cell Name* field. I decided to call it `sm_amp0`. Press **OK**.

This brings up a **Virtuoso Schematic Editing** window in which you can enter your circuit schematic.

Play around with the top menu items at the top of the Virtuoso window. You'll see many hotkey shortcuts to the commands. With time you will probably use these very regularly (get used to it from the very beginning) as they save a tremendous amount of time.

The most important option when building a schematic is the hotkey **i** that opens the **Add Instance** window. Fill in the *Library*, *Cell*, and *View* (and *Names* if you want) of this window to get the component that you want. In the example I have gotten a NMOS transistor
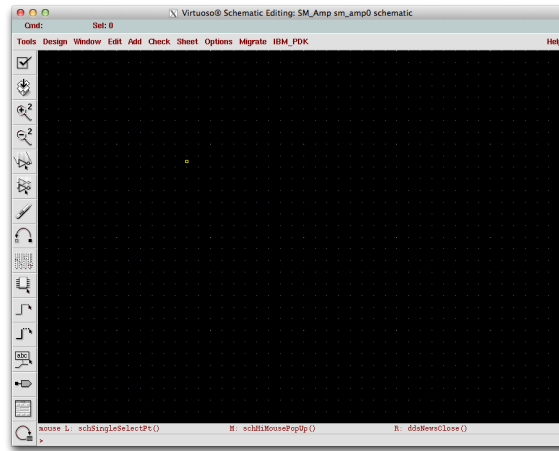
Figure 7: The **Virtuoso Schematic Editing** window wherein you create you circuit schematic.

.

by the name of `nfet` (there are many types of transistors for you to access in modern technologies) from the `cmrf8sf` library. As is usual I have grabbed its `symbol` view and I decided to name it `M0`.

To actually place the component I just click on the Virtuoso window (where I originally clicked **i**) and the component appears in the window. Repeat this process again and again to gather the remaining components needed for your circuit. I have gotten other circuit parts from a library called `analogLib` (you should be able to see it in the Library Manager window). There are many more libraries and many more parts. You have to explore to familiarize yourself with all the options.

After you have place a component in your Virtuoso window you can click on it to highlights and then press **q** to look at and edit its properties.

A view of my schematic with all the parts I'm interested in is shown in Fig. 9. After placing the parts I have altered a couple of them. I have set the DC voltage on the two sources to 0.6 and 2 V respectively and I have set the AC voltage magnitude on the DC voltage connected to the gate of the transistor to 1 V.

Obviously, the parts need to be connected with a wire. You can use the **w** hotkey for that. Hit **w** and then click on the component terminal (the red squares or dots) that you want to connect with a wire. My wired up schematic is shown in Fig. 10.

Check and Save your work (the box with a checkmark in it in the upper left of your Virtuoso window) to make sure that you connected everything correctly. If Cadence doesn't have any problems with your hook up (e.g. you forgot to attach a wire to a terminal and left it floating) then in the icfb you'll see something like that in Fig. 11
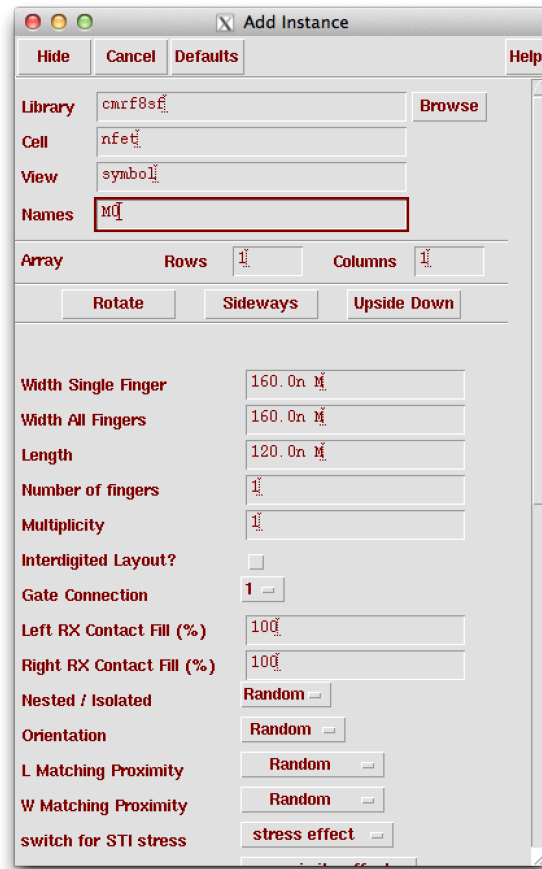
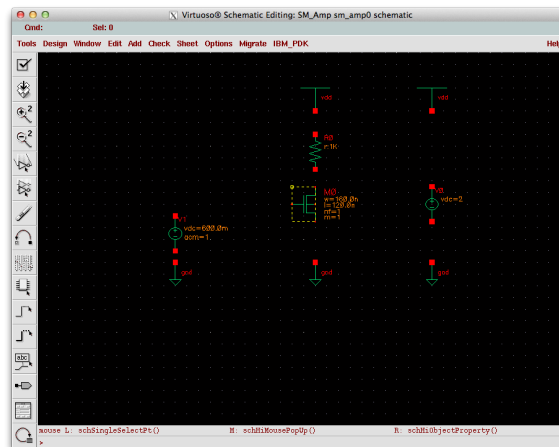Figure 8: An example of a filled out **Add Instance** window.
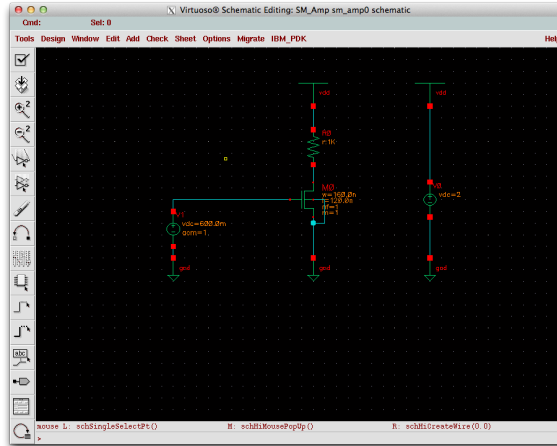
.

Figure 9: My schematic with just the parts imported.
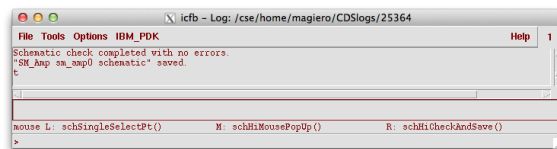
.

Figure 10: My schematic all wired up.

.

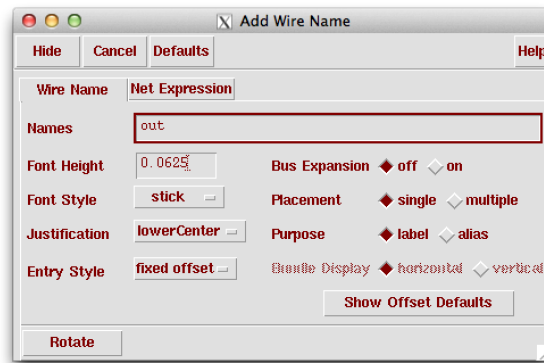Figure 11: icfb say my schematic check and save resulted in no errors.
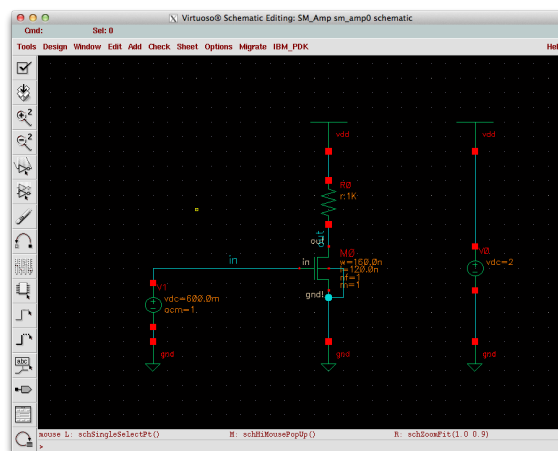
.

Figure 12: The **Add Wire Name** window.

.

Figure 13: The schematic with named wires.

.

Often it is a good idea to label your nodes (wires). Press the **l** hotkey, fill out the name that you want in the *Names* field then, in the Virtuoso schematic, click on the wire that you want to have this name.

My named circuit is shown in Fig. 13.

4   Analog Spectre Simulations

From your Virtuoso schematic window follow the menu options **Tools** → **Analog Environment**. This opens the **Virtuoso Analog Design Environment** (ADE) window. It contains many options by which you can achieve a GUI-driven simulation (not always desirable).
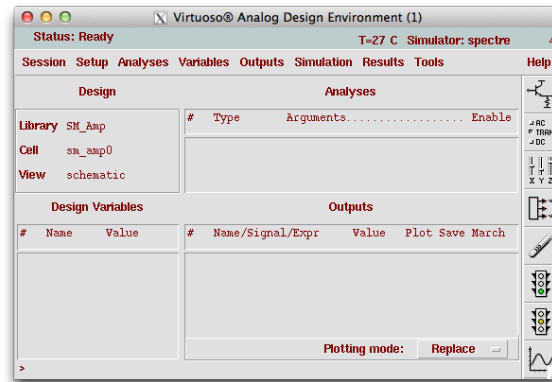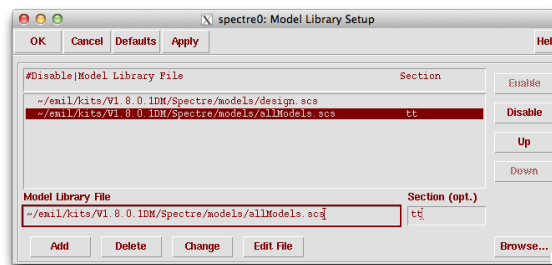
Figure 14: The ADE window.
.

Figure 15: The **Model Library Setup** window.
.

We will only highlight the absolute basics, study the Cadence manuals for more information.

First thing to do is make sure that the simulator has access to the needed transistor models. From the ADE go to **Setup** → **Model Libraries...**. The **Model Library Setup** window pops up wherein you can enter references to the model files. In this setup the basic transistor model libraries are setup for you so you don't have to worry. Otherwise, you have to find this out on your own (somehow). Close the Model Library Setup window.

From the ADE choose **Setup** → **Simulator/Directory/Host...** which launches the **Choosing Simulator** window. Make sure that the *Simulator* field is set to **spectre** as shown in Fig. 16. Click **OK** once you have set this.

Now it is time to define what type of simulations you actually want to run on your circuit. From the ADE window choose **Analyses** → **Choose...**. The **Choosing Analyses** window pops up. As you can see there are over 30 analyses to choose from! Today we'll just work with the simplest a DC analysis and an AC analysis.
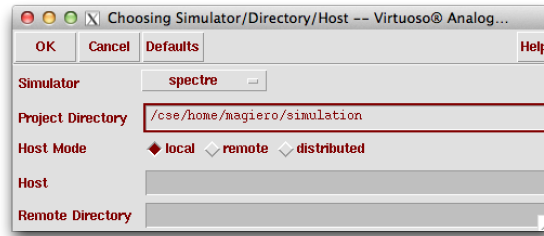
Figure 16: The **Choosing Simulator** window.
.

To choose a DC analysis select the *dc* button in the Choosing Analyses window as well as the *Save DC Operating Point* button. A copy of the window is shown in Fig. 17. Click **OK**.

To add an AC analysis open the Choosing Analyses window and this time select the *ac* button. There are other things to set as well (each simulation has its own characteristic set of options). Note the settings I made in Fig. 18. Hit **OK** after you have finished making your AC settings in the Choosing Analyses window.

The upper left panel in the ADE window should now indicate that you have the AC and DC analyses activated.

You are ready to run the simulation. Just hit the ADE button with the green traffic light. As usual Cadence gives you a lot of messages during the run. One of them is the status window as shown in Fig. 21 that details some of the simulator's activities. Hopefully your simulation completed without errors.

There are many ways to view your results after a successful simulation. A simple one is to just look at what your voltage settings ended up being in your schematics. In your ADE choose **Results** → **Annotate** → **DC Node Voltages**. This updates your schematic and shows you the DC voltages you simulated.

To see the AC voltage as a function of frequency from your ADE choose **Results** → **Direct Plot** → **Main Form...**. The **Direct Plot Form** comes up from as shown in Fig. 22.

The Direct Plot Form indicates at the bottom that I am to click on the signal of interest in the schematic. When I click on the `out wire on the schematic I get the plot shown in Fig.` **??.**

`Another useful way of getting at your results is by choosing` **Tools** → **Results Browser...** `from the ADE. The` **Results Browser** `window that pops up contains all signals and settings saved by the simulation (very useful when you start writing scripts to run all this stuff automatically instead of having to push through dozens of windows). Note that`
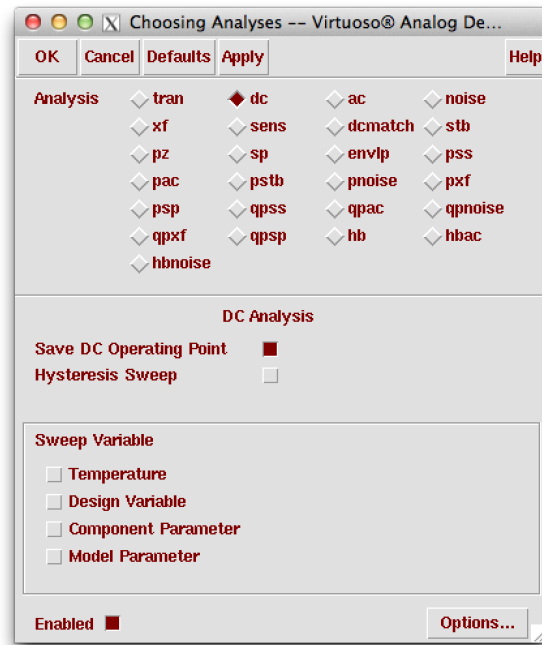
Figure 17: The **Choosing Analyses** window set to choose a DC analysis.
.

in Fig. 24 I have selected the signal out from the ac-ac folder.

Clicking the little calculator icon located in the upper left
of the **Results Browser** (4th from the left) opens up the **Calculator**
window with the out signal already entered in the calculator field
(using Cadence's peculiar syntax for referring to signals).

If you hit the little plot icon (red curve on a little grid) in
the **Calculator** window it will also plot out your signal. With the
calculator though note all the functions you have at your disposal
to start manipulating the signal any way you want (e.g. take
the square, put it in terms of dB, isolate a particular transition,
etc.). Again it is fine to start off this way, but for any serious
signal manipulations you will hopefully learn to drive all of
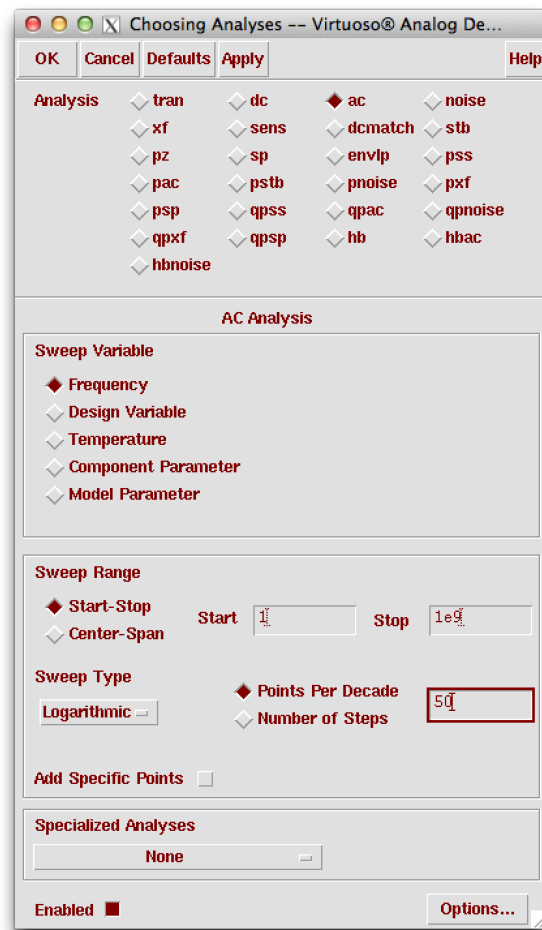this with scripts (the subject of a future Tutorial perhaps).

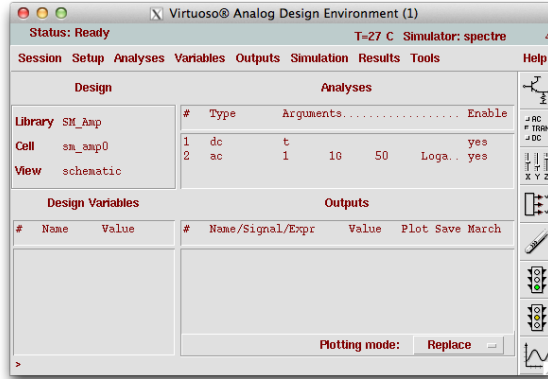Figure 18: The **Choosing Analyses** window set to choose a AC analysis.
.

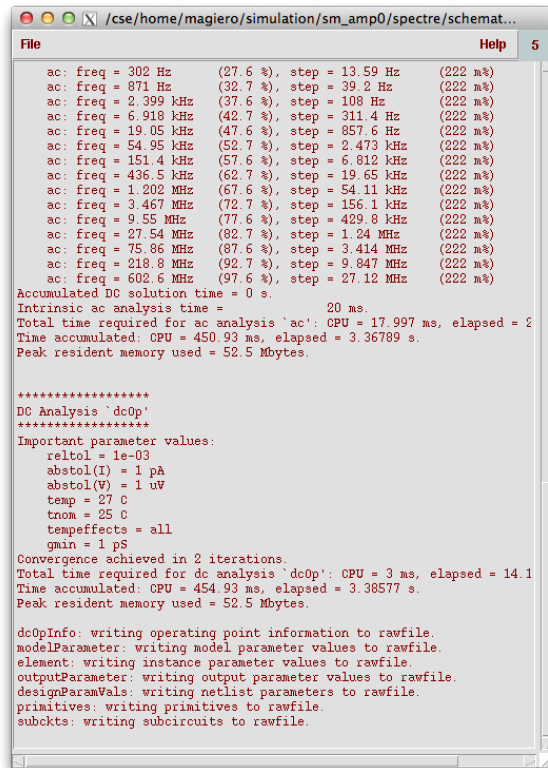Figure 19: The top right panel of the ADE indicates that we have set AC and DC analyses.
.

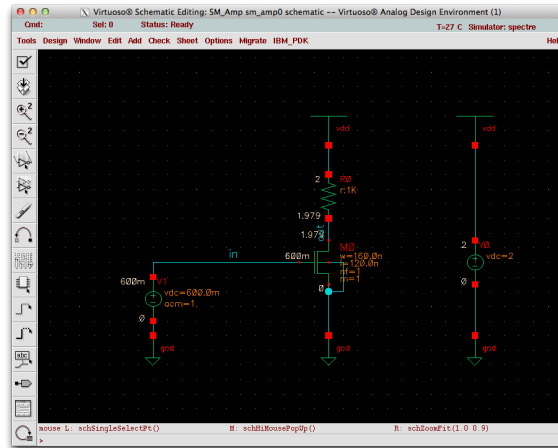Figure 20: The simulation status window after running a spectre simulation.
.

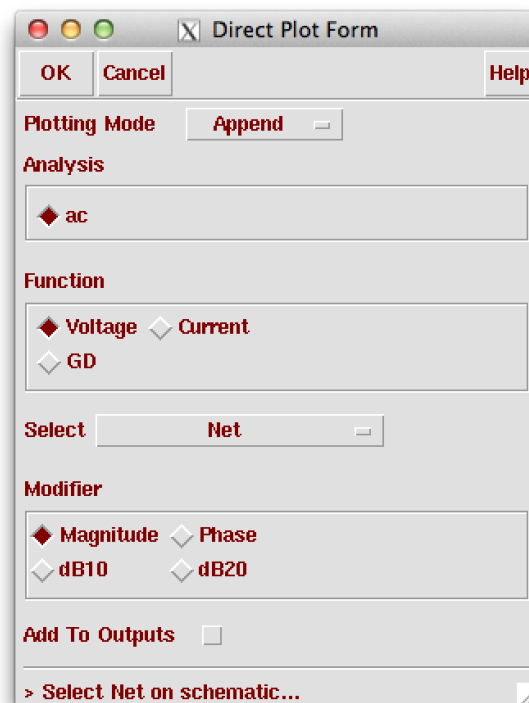Figure 21: The schematic window after choosing **DC Node Voltages**.
.

Figure 22: The **Direct Plot Form** allows you to choose the signal you want to plot right from the schematic.
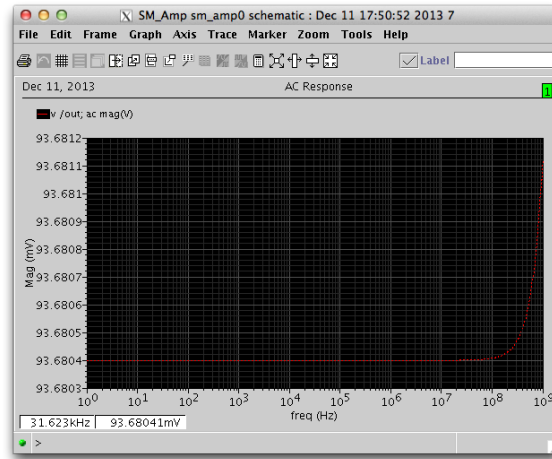.

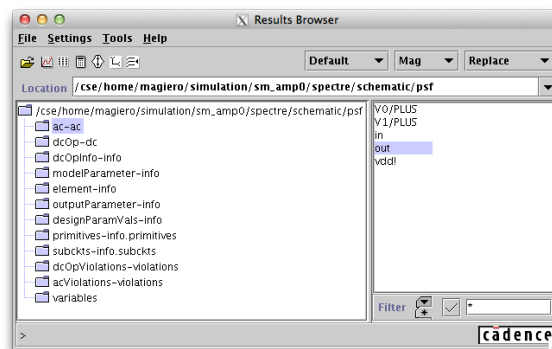Figure 23: A plot of the AC response at the `out node`.
.

Figure 24: The **Results Browser** a very useful summary of all the values saved by the simulation.
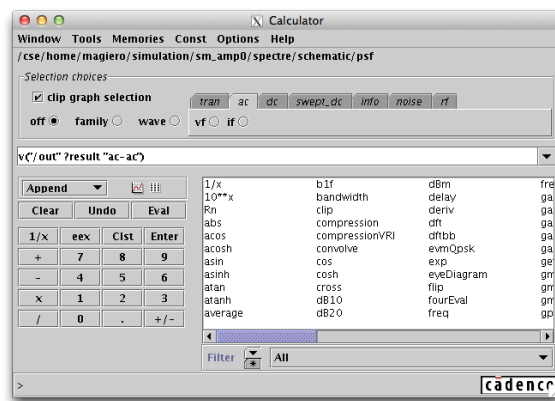.

Figure 25: The calculator with the signal of interest defined in its signal field.

.