


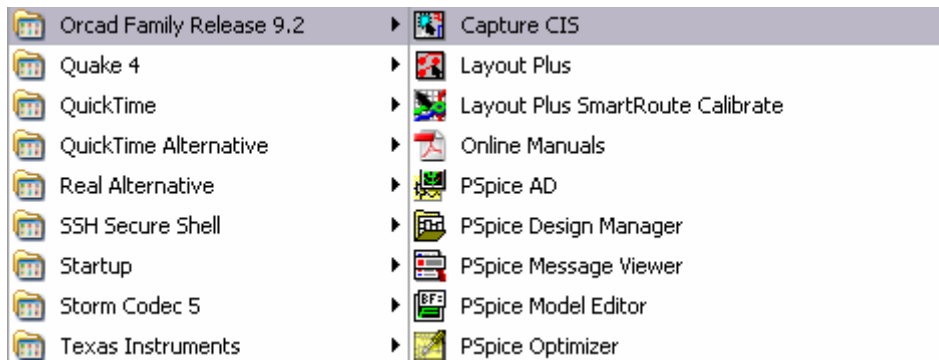
PSPICE Introduction:

Introduction:

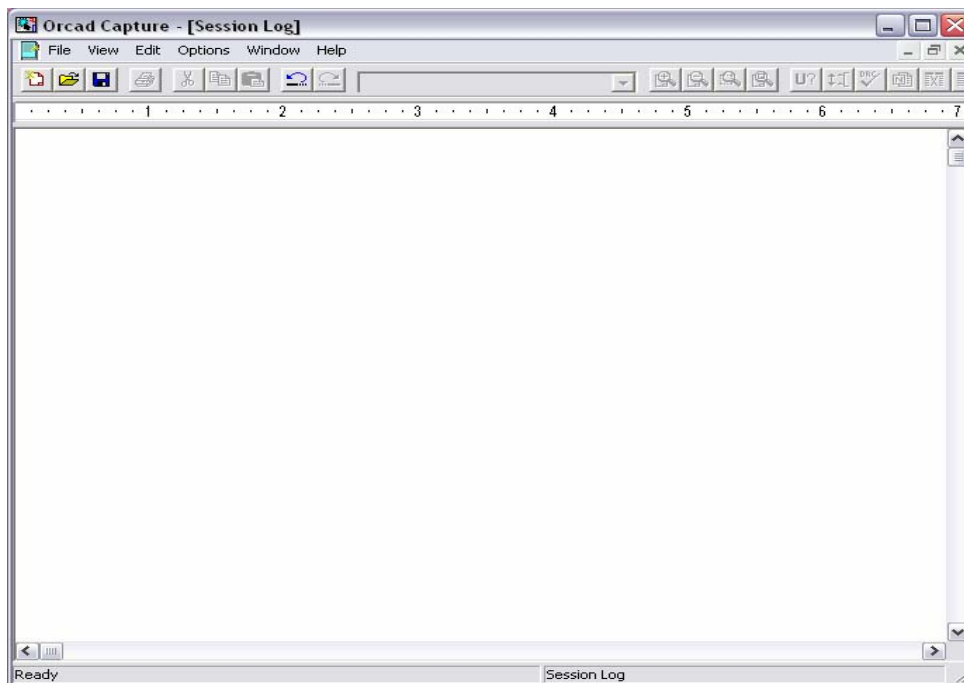
This tutorial is intended to introduce some of the many PSPICE functions you will need to complete the fourth ece35 lab. PSPICE is a computer aided drafting (CAD) tool for various analog and digital circuits, which is used in industry and also in many future classes (ECE 65,102,108). Often in practical engineering, designers must first come up with a circuit model, do appropriate analysis, simulate the circuit, and then finally build the actual circuit. Therefore it is convenient and cost-saving to use a CAD tool to check the circuit for any immediate errors before the actual circuit is built. Hence, it will be beneficial to have some experience with CAD tools for your future classes or maybe even industry work.


Getting Started with PSPICE:

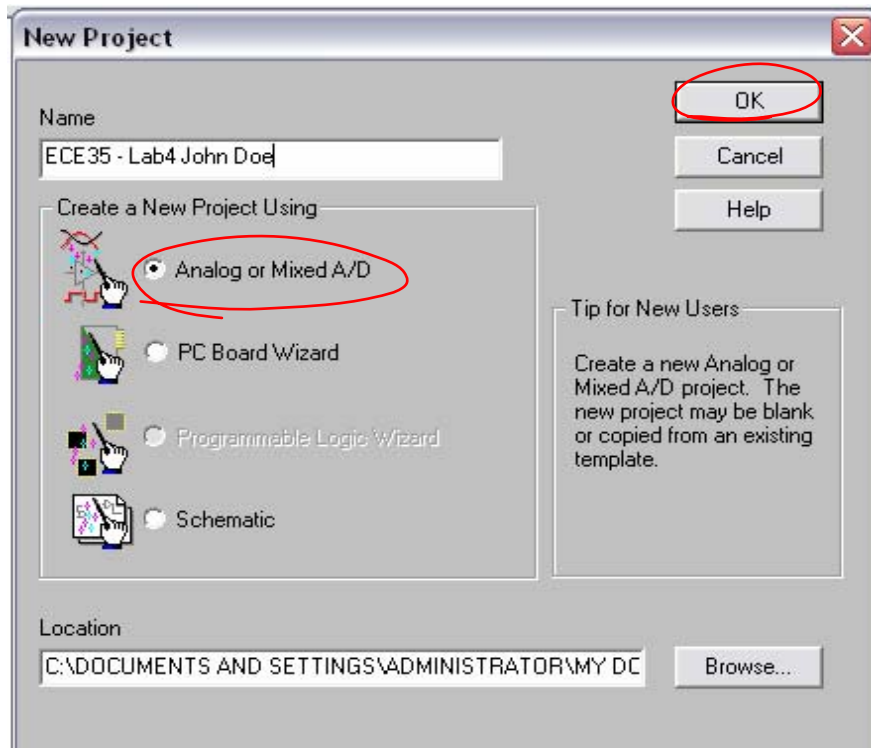
First you must find the program link by first clicking on the  menu, programs, Orcad 9.2, then Orcad Capture Student as shown below.



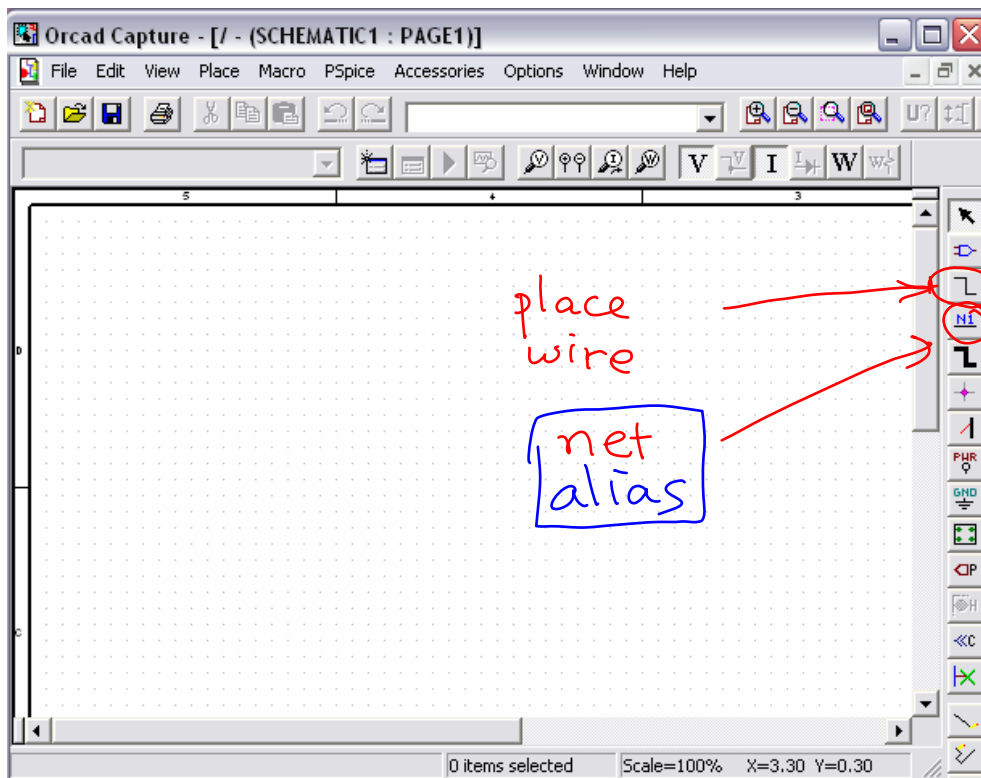
You'll then see a window like this.



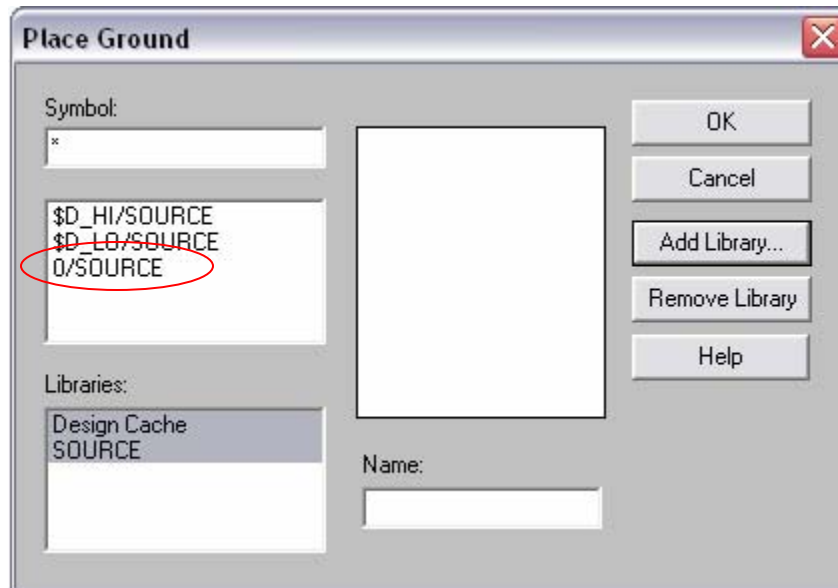
Next, click on this  icon, now you will notice a window pop-up that looks like this:



Type a name for the project (i.e. ECE35 Lab 4 – John Doe... etc). Make sure **“Analog or Mixed A/D”** is checked and **click OK**. Click **“Create a blank project”** and then **click OK**. If you did everything correctly your screen will look something like this. **Keep track of the circled buttons.**



Next, you will need to add several library files. First start by clicking on the “**Place**” button in the menu bar circled from above, and then click on “**Ground...**”.

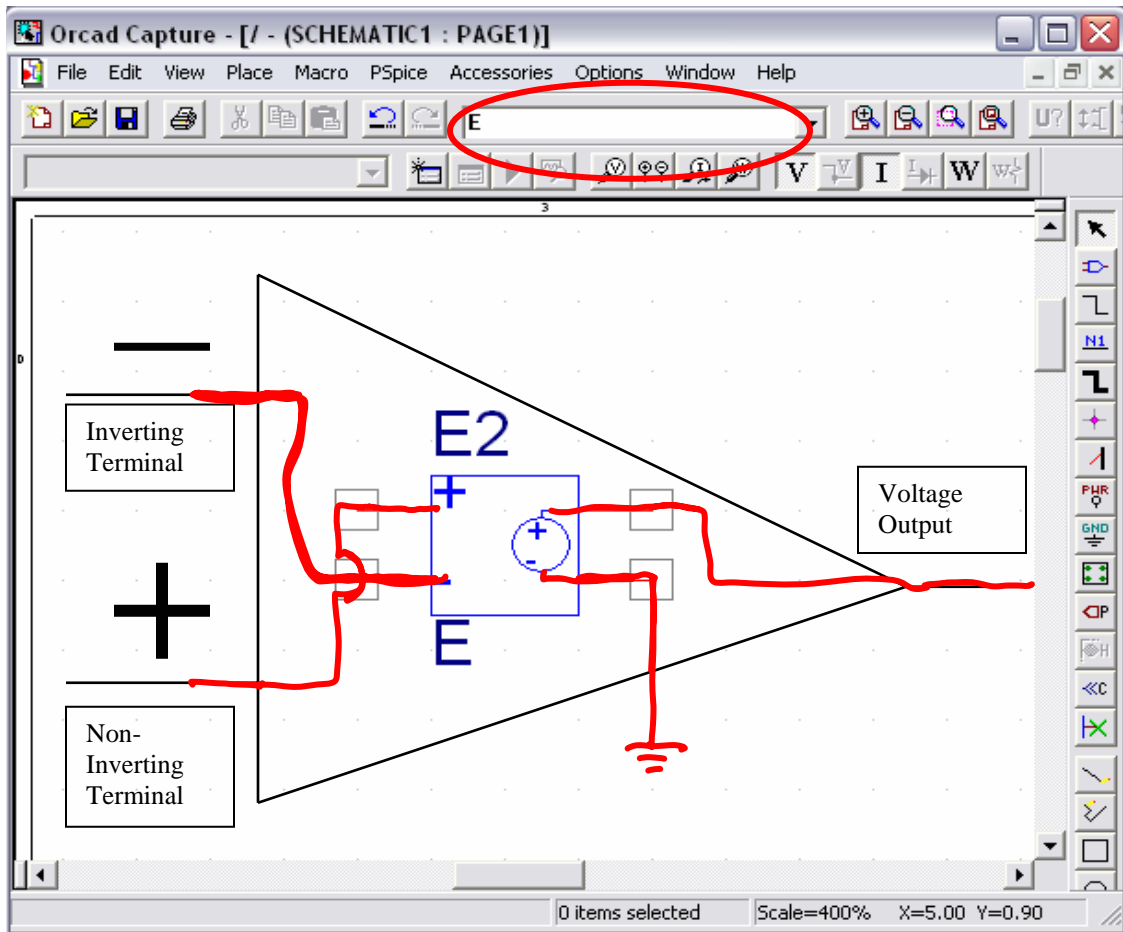


The window might not look exactly the same, but it will be similar. Next, click on “**Add Library**”-- you’ll either start in the directory “**PSPICE**” or “**Library**”. Look for the file “**source.olb**” in the **PSPICE directory** and click open, then press OK to add the source library.

Next you will need to add the appropriate PSPICE parts, to do that, click on “**Place**” on the menu bar. Then, you will have to click “**Part**”. You’ll notice that the window for “**Place Part**” will be similar to the “**Place Ground**” one. Again, click on “**add library**”, but this time, you need to add **ALL the library files**. To do that, press at the same time (**CTRL + A**) you’ll notice that all the files are highlighted. Press **open**, and then press **cancel**. Once you have completed the previous tasks, you can then place circuit elements and starting simulating them.

Placing Parts:

This is what your workspace looks like now. If you notice the circled toolbar thing, that is what you will be using to place your parts (**C** – Capacitor, **E** – Opamp (Voltage Controlled Voltage Source), **R** – Resistor, **L** – Inductor, **0** – Ground (That is a zero), **VDC** – DC Voltage Source, **VAC** – AC Voltage Source). Note that I already typed **E**—when you press enter, you will be able to place the part on your workspace. Place the part on the workspace; it should look like this when you are done:




In order to visualize how this works as an operational amplifier I drew what it would look like as a circuit symbol. Now that we know that an operational amplifier behaves like a voltage controlled voltage source (VCVS), we need to make sure the gain of the VCVS is very high, to do that, double click on the part. You will see a table pop-up with many entries; find the entry "**GAIN**". You'll see that the value for "**GAIN**" is 1. You will need to click on it, delete the 1 and then enter **10e10** after you have done that close the table but make sure **NOT TO CLOSE THE PROGRAM**.

		Color	Designator	GAIN	Graphic	ID	Implementation	Impleme
1	+	SCHEMATIC1 : PAGE1 : E3	Default	1	E.Normal			

Placing Wire

To place wire you can do two things:

- 1.) Click  and when you are done, you can press the **Esc** key on your keyboard
- 2.) Press the **W** key on your keyboard, and once you are finished drawing you can press **Esc**.

Placing Capacitors, Inductors, Resistors, and other things:

When you place a certain component you can adjust its value, whether it is resistance, inductance or capacitance. All you need to do is double click on the default value (for resistors, it is 1k) for the component and type in the desired value. PSPICE will only understand the following values you give it:

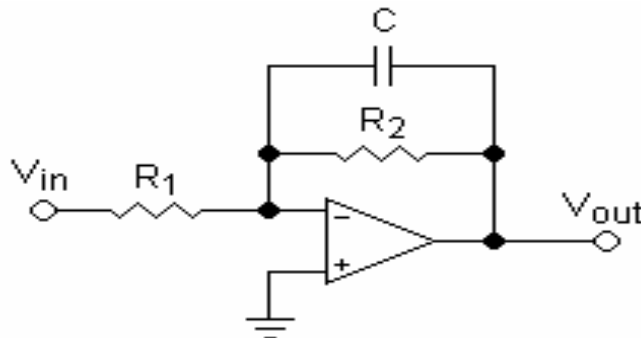
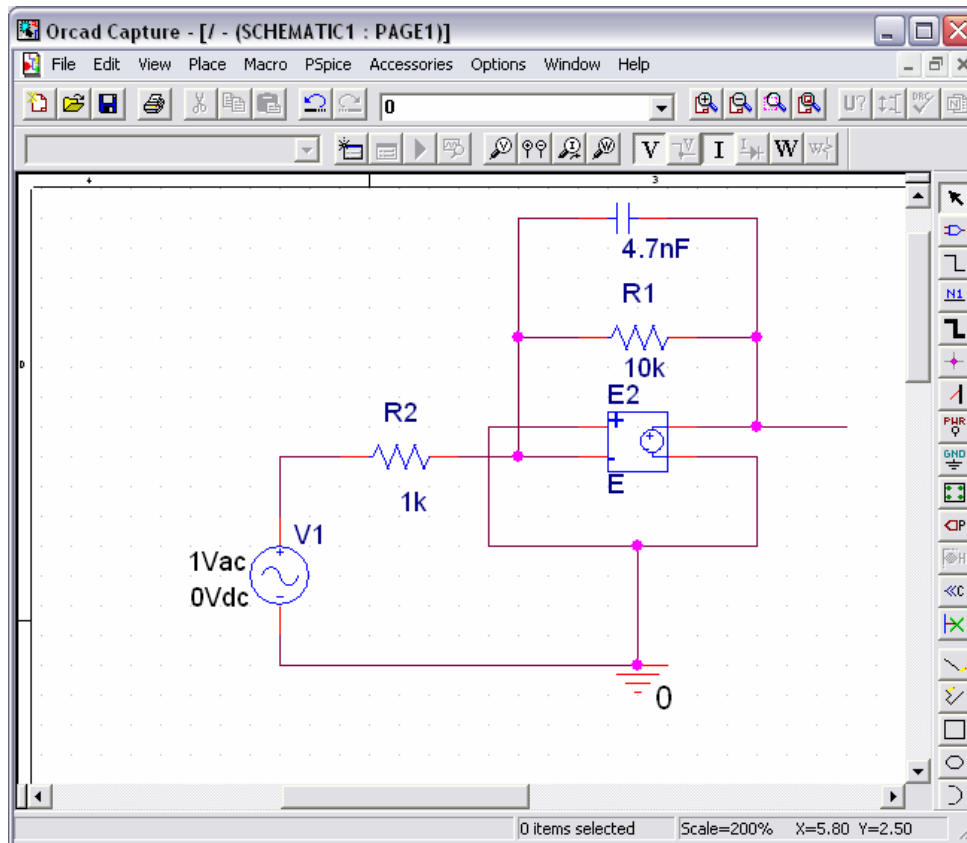
x – Being any floating point number


xp – 10^{-12} **xn** – 10^{-9} **xu** – 10^{-6} **xm** – 10^{-3} **xk** – 10^3 **xMeg** - 10^6 **xG** - 10^9

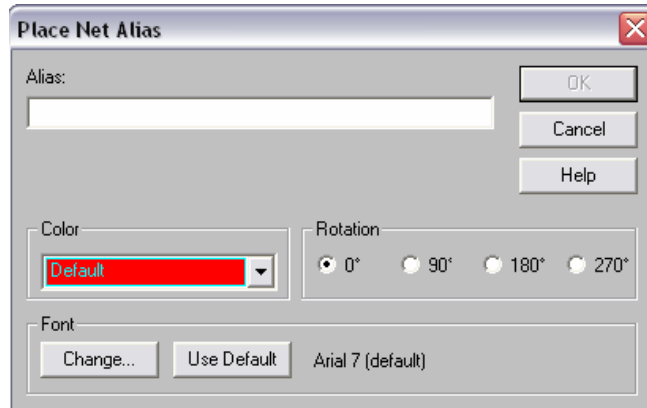
Here are some examples: 4.7p, 12.3n, 5.6m, 10Meg, 30G, 1234k, 3.4k

PSPICE Simulation:

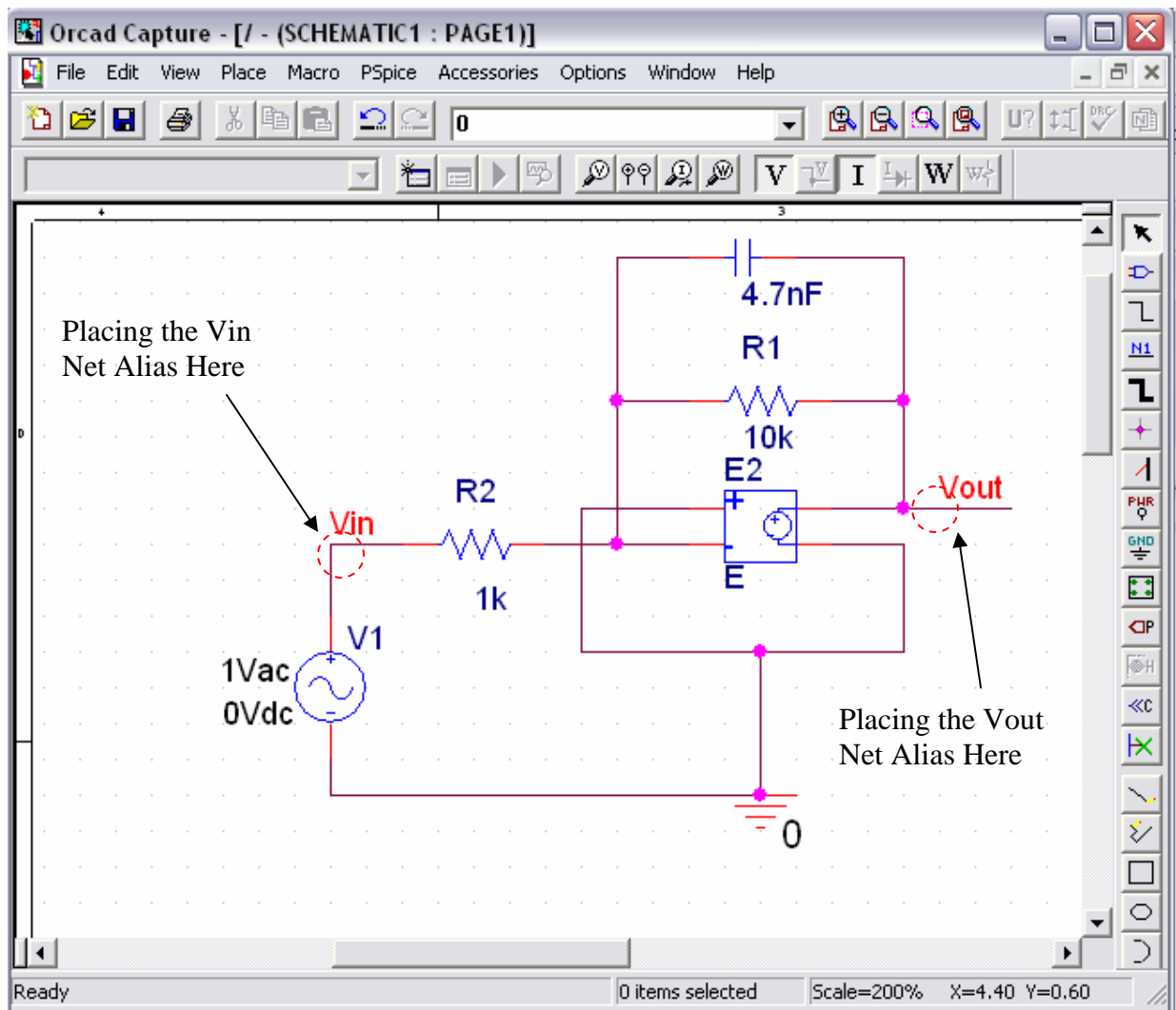
The following is an example low pass filter circuit, using a VCVS as an opamp on PSPICE. Notice how if the wire is actually touching you will see a pink “node” where they are crossing. Hence, all wires with no nodes between them are not touching so be very careful with your wiring.




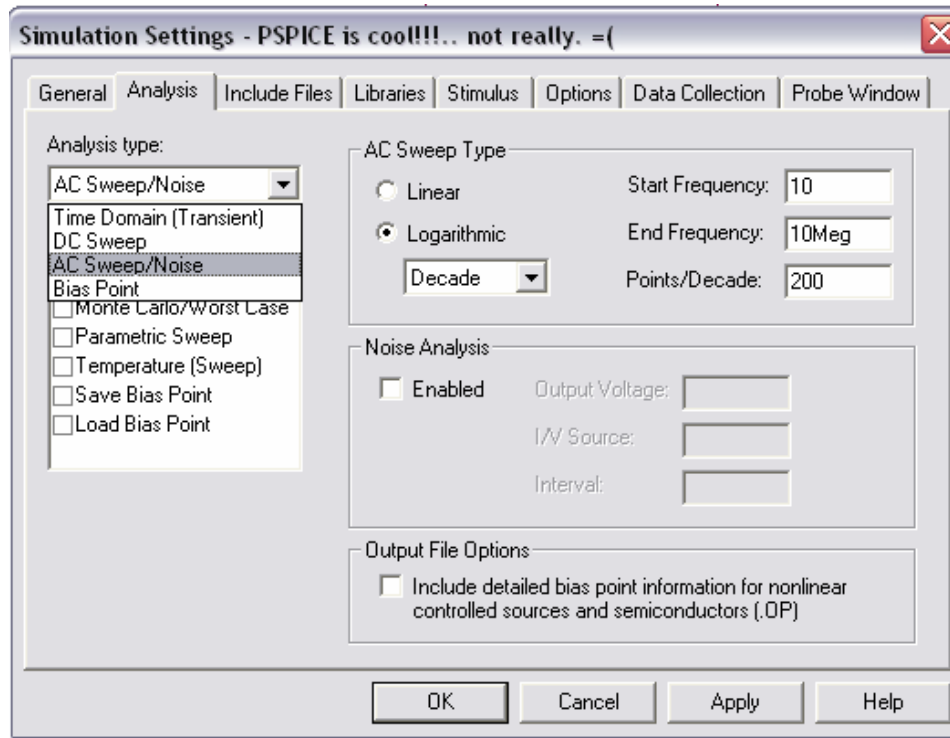
Now that we have our circuit completed, we want to simulate it. An easy way of doing this is to first click on the “**Net Alias**”  button on the right toolbar. You will see a window like this:



Next, type **Vin** in the little bar and press ok. Place this part right before the 1k resistor (where the input voltage signal is coming from). After you have done that, press the **Net Alias** button again and this time, type in **Vout**, and place the part where the output voltage will be measured. If you did this correctly you will see something like this:



Once you have done the previous steps, you need to click on the **New Simulation Profile** () button. Type in anything you want for a name and then press **OK**. You will see the following pop up:



You will notice that there are various **Analysis Types**, but we are mainly interested in **AC Sweep/Noise**. The following is a brief description of what you will use these for. In general, We will use the following values:


Start Frequency: 10
End Frequency: 10Meg
Points/Decade: 200

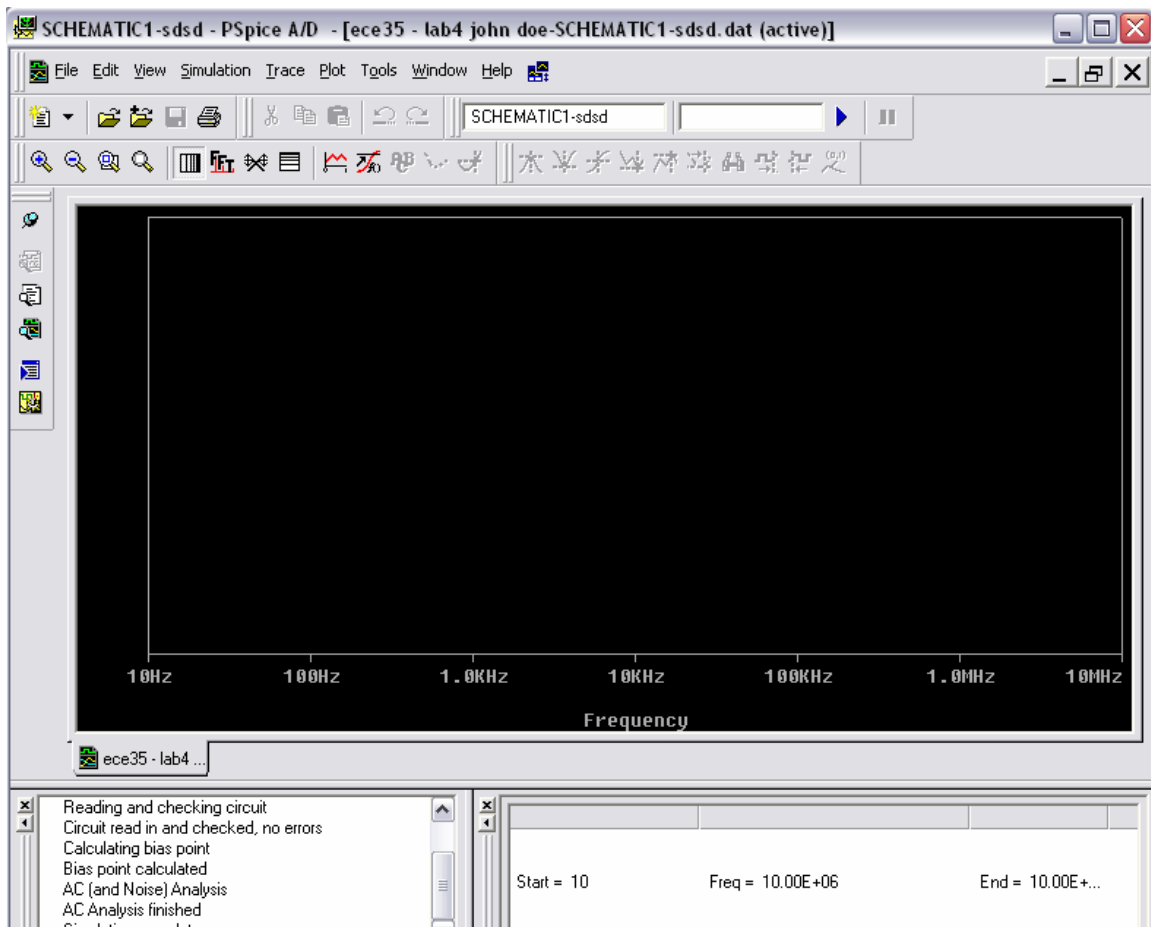
Time Domain (Transient) – Is used to observe the time response of a circuit. For example you can use this option to simulate an RC circuit and find the rise/fall time, and perhaps find the RC time constant.

DC Sweep - Mainly used to find the response of a circuit when you vary or “sweep” a voltage. For example, I can sweep the input voltage from -10V to 10V, and see what the output looks like.

Bias Point - Bias Point will find all the DC currents and node voltages of the circuit.

AC Sweep/Noise - Is used to view the frequency response, or in other words sweep the frequency and view the output voltage or phase of the circuit output is.


Once you have entered all the values you will need to simulate the circuit, by clicking on the simulate button () . Assuming you don't have any errors you will see the following:




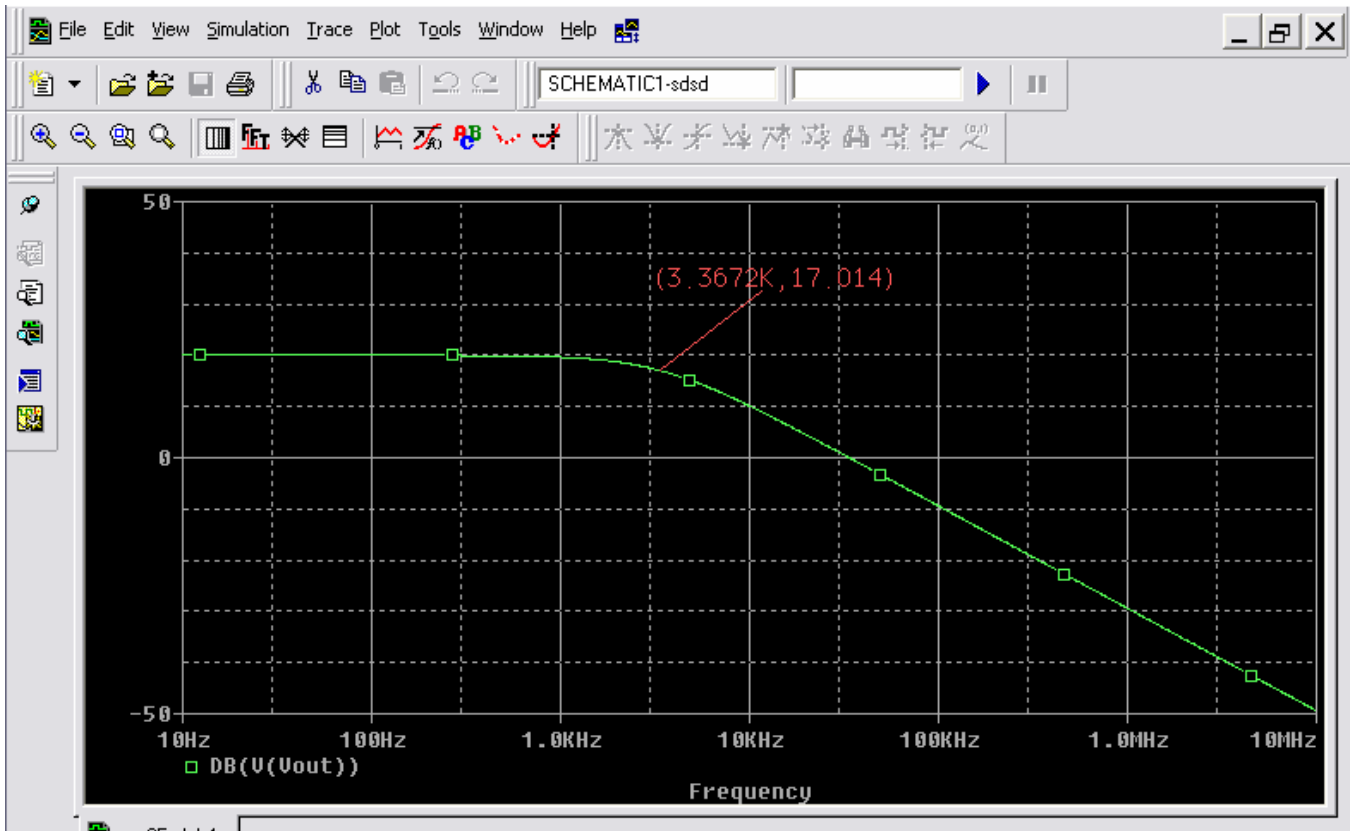
In order to plot the phase or frequency response you will need to click on **trace** and click on **Add Traces**. Under the **Trace Expression** bar you will need to type the following:

DB(V(Vout)) -- This is used to find the Magnitude of the output in Decibels

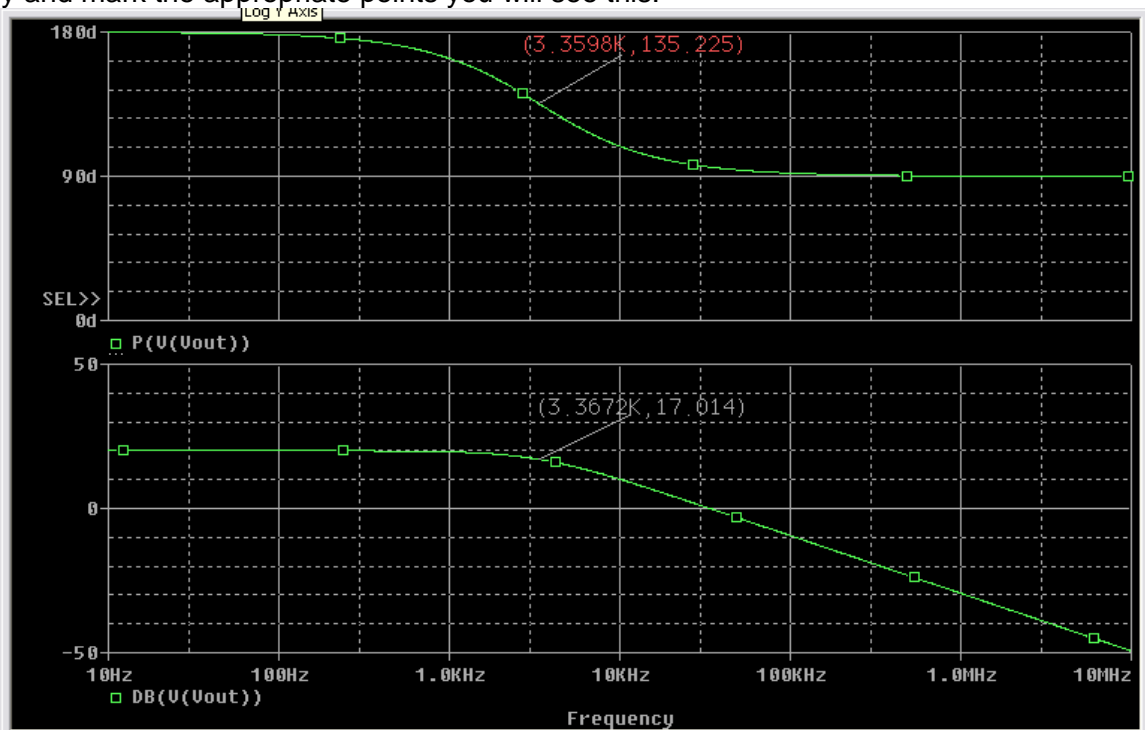
P(V(Vout)) -- This is used to find the phase of the output in degrees.

In order to mark certain points, what you will need to do is press the Toggle Cursor () button. Once you do that, you can use your mouse to drag and click on which data point you are interested in. In

order to mark the data point you will need to press the mark label () button. Once you have done that correctly the graph should look like the following:



You will notice that I marked the 3dB half power bandwidth frequency. The data point is in the form of an ordered pair (x, y). The value x will correspond to the x axis which is the frequency in Hz and the value y will correspond to the y axis which is the magnitude in decibels. What you can do afterwards if you want to read the phase response of the circuit, you can click **Plot**, and then **Add Plot to Window**. Next you can repeat the steps for plotting the magnitude but instead plot the phase. Once you do this correctly and mark the appropriate points you will see this:



Be aware that in order to modify one of the plots you need to click on the green diamond
(`□ DB(V(Uout))`) if you want to work with that plot. and make sure the (`SEL>>`) button is pointing to the correct plot.

Concluding Remarks:

This is a very brief tutorial, but it will give you enough information to be able to finish the PSPICE portion of the fourth lab for this class. In the next class, ECE 65, you guys will be using PSPICE more frequently, so hopefully after this class you guys will have had some exposure to the program.

If you find any errors, have any suggestions or comments please e-mail me at pling@ucsd.edu.