

## How to use PSPICE

*\*Most of this document was provided as multiple documents by Dr. Brita Olson of Cal Poly Pomona for use by students. Certain changes were made for this workshop. No ownership is claimed by workshop instructor.*

1) Launch Cadence: From the start menu:

**Programs -> Cadence SPB 15.7 -> Design Entry CIS**

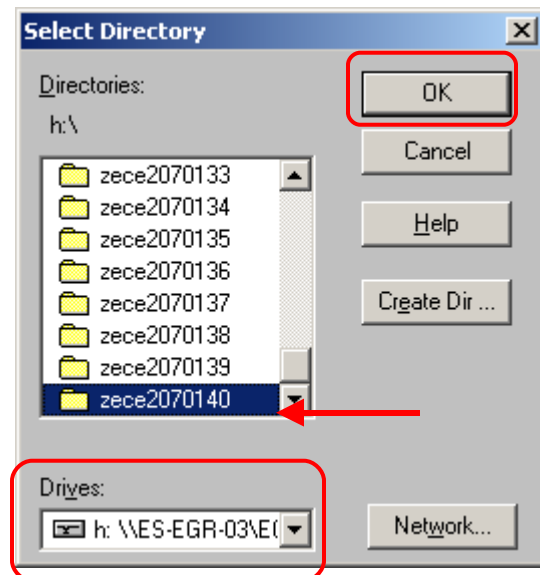
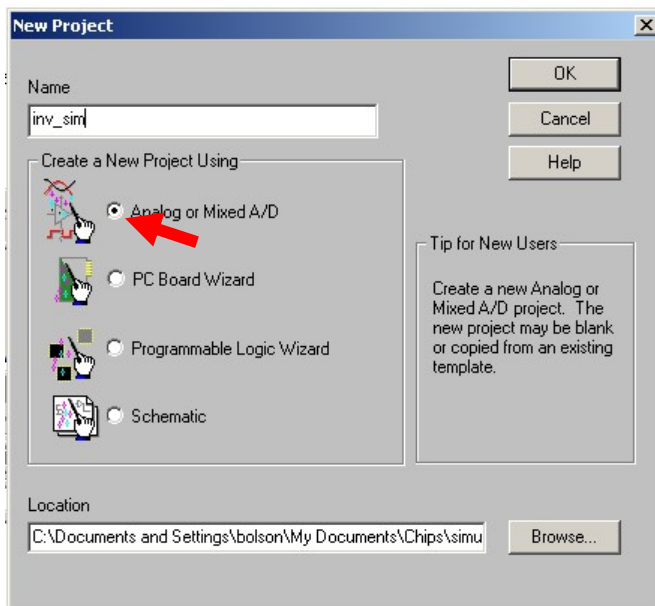
2) From the Pop-up Menu select:

Studio Selection Suite: **Allegro PCB Design CIS L**, hit **<OK>**

3) At this point you will see a large window. From the toolbar at the top select: **File->New->Project**

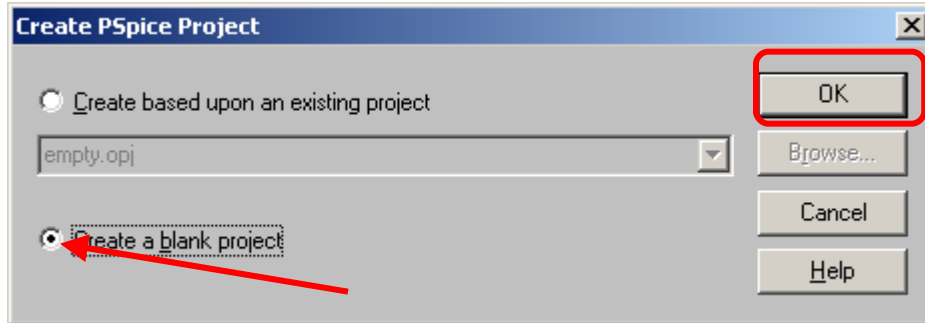
4) You will see the *New Project* pop-up menu shown below on the left.

- Enter a name for your schematic. Here I have created a project called `inv_sim`.
- Make sure that the **Analog or Mixed A/D** radio button is selected (with the school version this is NOT the default). If you do not do this you cannot simulate.
- Hit **<Browse>** to define the path where you want your file to be saved.
- You will then see the *Select Directory* pop-up menu on the right
  - Select the **H-drive** at the bottom
  - Select your class folder. Here I have selected `zece207140`. (you can also create a directory from here to better organize your work)
  - In the *Select Directory* pop-up menu hit **<OK>** to close it
- In the New Project pop-up menu hit **<OK>**



5) You will see the *Create PSpice Project* pop-up menu:

- Select : **Create blank project**
- Hit <**OK**>



**You are now ready to add parts**

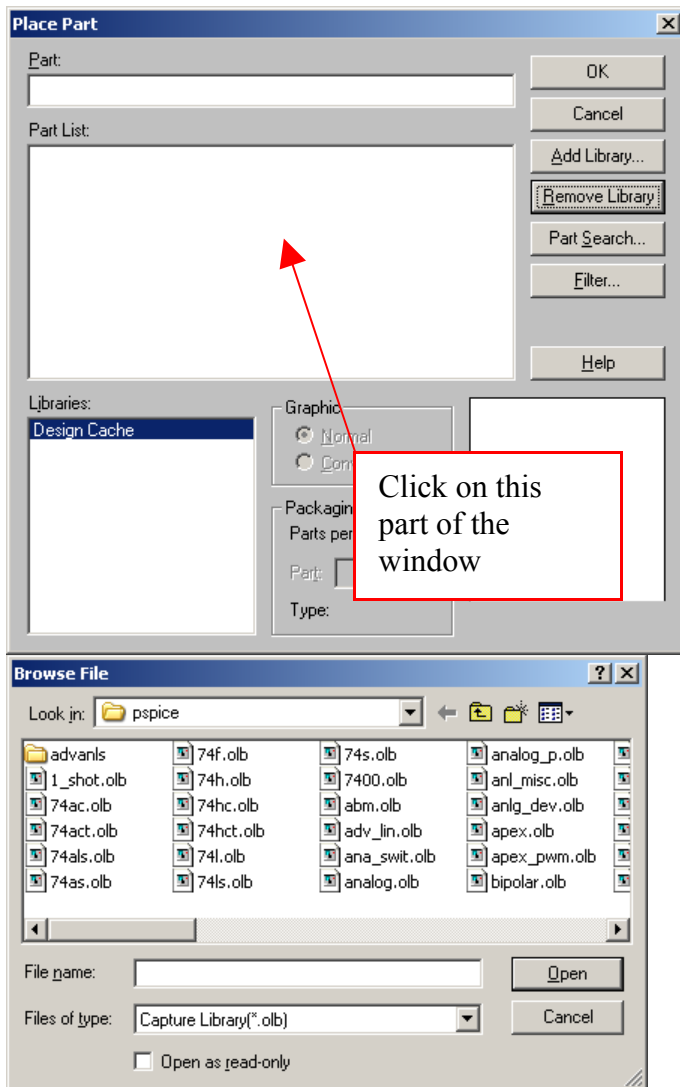
6) First click you mouse on the screen to activate the toolbars

7) To place a part hit hotkey **p** or from the Menu **Place->Part** - *you may have to wait*

When you first launch PSpice (from school) the libraries containing the parts may not be loaded. If this is the case you need to add them. If have libraries loaded skip the Adding Libraries section that follows

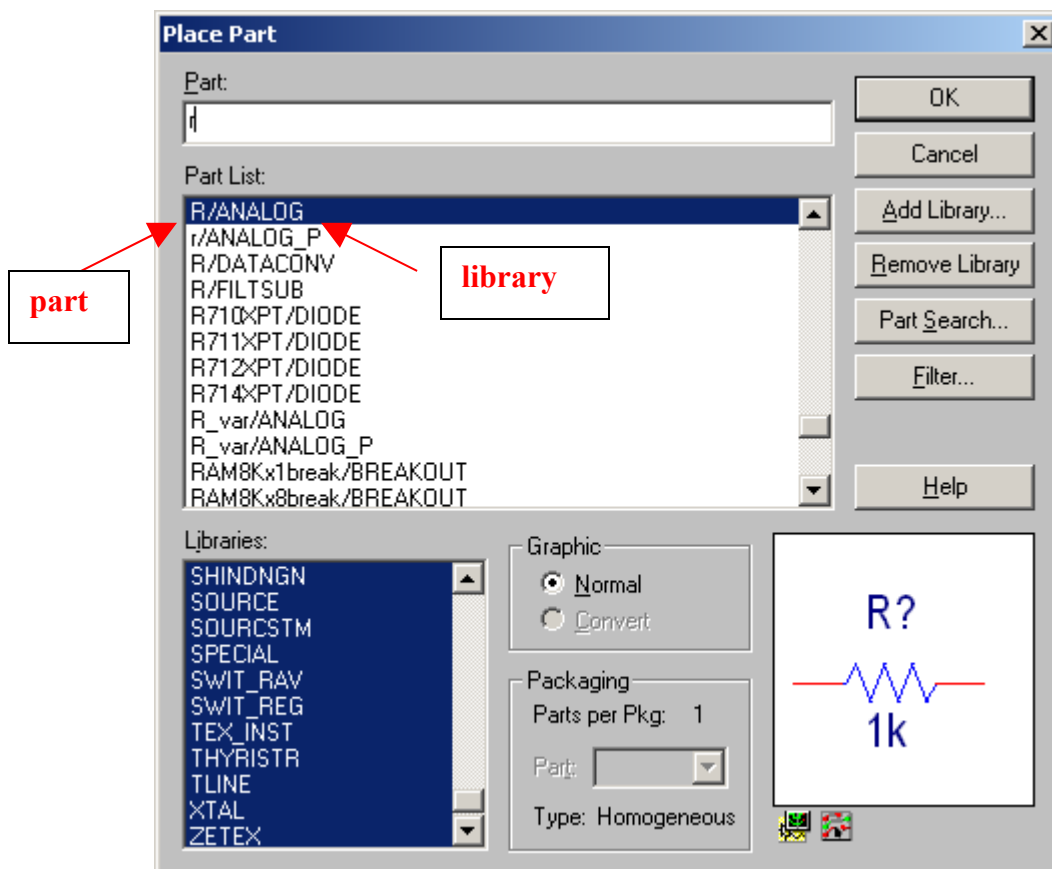
## Adding Libraries

- In the menu below hit **<Add Library>**
- You will see *Browse File* pop-up menu. The path of the library files is:  
**C: Cadence->PSD\_15.0->tools->capture->library->pspice**
  - You can select all the libraries at once in the *Browse File* pop-up menu by: clicking your mouse on the window in the position indicate and then hitting **<ctrl> a**
  - Too add all the selected libraries hit **<Open>** in the *Browse File* window



Your Place Part Menu should appear as shown below with all the libraries loaded. Here all libraries are selected (you can just select a subset of libraries). In the “parts list” the part is listed along with the library that it belongs to. It is important to add the part from the correct library. Otherwise it may not function as you intend. As you can see below there are multiple parts that are called **R**, and each is in a different library.

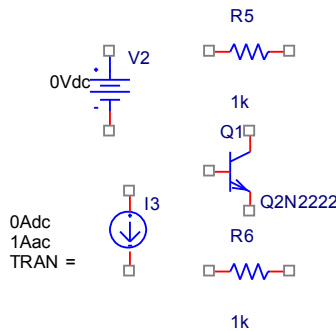
- 8) Enter the name of the part that you wish to add by scrolling or typing it directly.
- 9) Hit **<OK>**
- 10) Sometimes you will see a pop-up menu indicated that the part is not in the design cache. If you see this menu Hit **<yes>**.
- 11) Move your cursor to where you want to place the part. Click your mouse at the location
- 12) You will continue to be able to place parts that are of the type that you selected. Hit **<esc>** to end the place parts command.



**Here are some parts that you might want to use:**

- Vdc** DC (constant) voltage source (default value is 0V You might want to change this)
- Vsrc** Voltage source (for DC, AC and Transient simulations)
- Idc** DC (constant) current source (default value is 0A. You might want to change this)
- Isrc** Current source (for DC, AC and Transient simulations)
- R** Resistor (default value is 1K)
- C** Capacitor (default value is 1nF)
- E** Voltage controlled voltage source
- H** Current controlled Voltage Source
- G** Voltage controlled Current source
- F** Current controlled Current Source
- Q2N2222** BJT frequently used for 220 and 320

After placing the parts you should see something like the following: You may wish to rearrange the parts.



**To move parts:**

- Select the part (or parts) you wish to move with your mouse
- *Drag* the part(s) to the new location

**To undo a command:** hit **^z**

**To Copy:**

- Select the part (or parts) you wish to copy with your mouse
- Hit **^c**
- The object is now in the clipboard
- If you are writing a laboratory report this clipboard is available to Microsoft products. So you can use this command to paste entire schematics in your reports (**^v**).

**To paste:**

- Hit **^v**
- Move your cursor to where you want to place the object that is in the clipboard
- Click your cursor at the desired location.

**To rotate**

- Select the desired part Hit hotkey **r** OR from the main menu: **Edit-> rotate**

**To mirror (flip) horizontally or vertically**

- Select the desired part. From the main menu: **Edit-> mirror**

**To delete**

- Select the desired portion of the circuit. Hit **<delete>**

After placing the parts you may wish to connect them using wires also called nets.

### Adding wires:

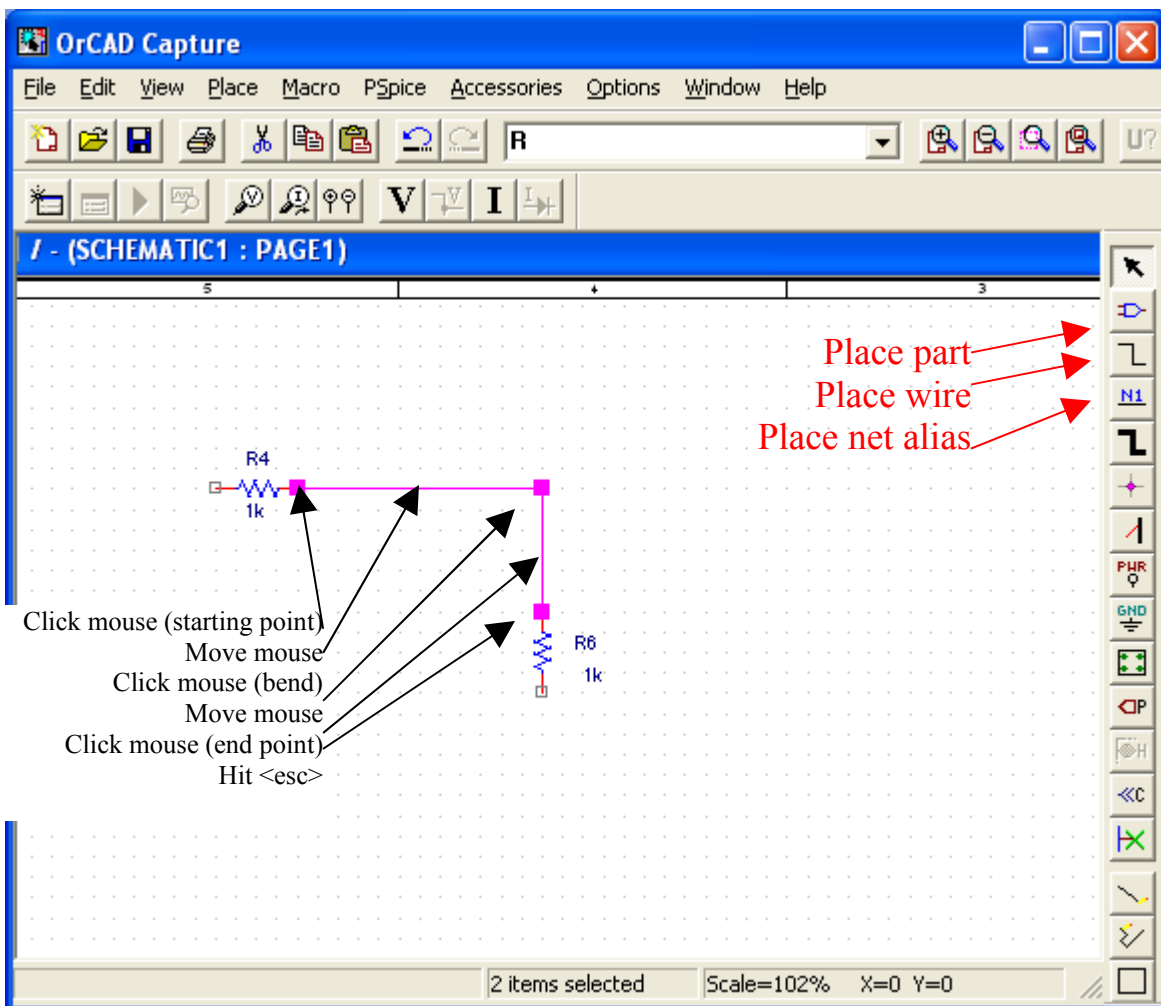
- Select the Place wire icon from the toolbar on the r.h.s. of the screen as shown below. (You can also add parts this way.)

#### One way

- Click your mouse at the starting position of your wire
- Move (do not drag) your mouse to the end location click again.  
(the tool will automatically place the bend in the wire)

#### Alternative (illustrated below):

- Click your mouse at the starting position of your wire
- Move (do not drag) your mouse to the location where you want a bend. Click your mouse. (you may repeat this step for multiple bends)
- Move (do not drag) your mouse to the end location click again. Hit the <esc> key.



Naming wires: It is much easier to maintain a design if you name the wires (nets) on your diagram. To do this:

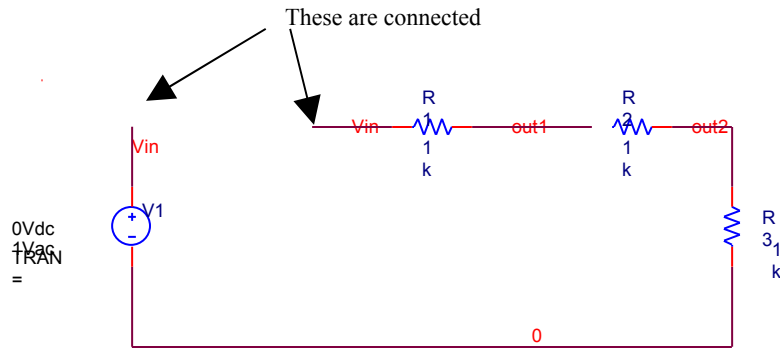
- From the tool bar on the right use the **place net alias** (fourth from the top), OR from the main menu: **Place->net alias**
- Type the name of the net in the menu that follows
- Click your cursor on the net that you wish to name
- Hit <esc> to end the place net alias command

Note wires that have the same name are considered to be connected. You can use this feature to create less cluttered and more readable schematics. Below is an example.

### **Creating Ground:**

To simulate you need to declare a ground. A way that works for both student versions and the version at school is:

***Naming ground:*** name the corresponding net with a net alias of 0 (zero).



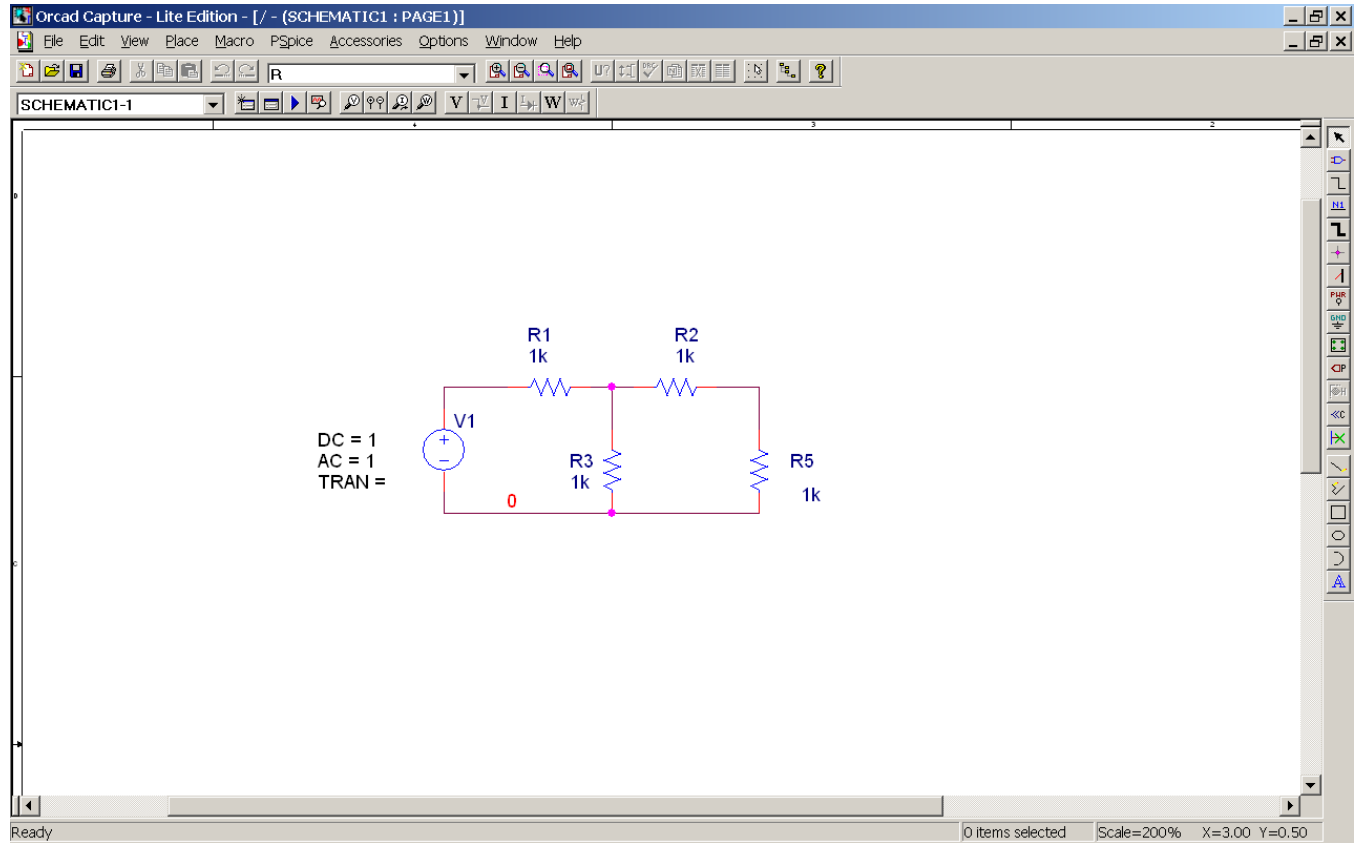
### **Changing the names of parameters of parts:**

Parts are placed with default parameters, for example, resistors default to having a resistance of 1K. To change these parameters. Click on the parameter that you wish to change, for example, for the resistor you would click on 1K. Enter the new value in the resulting menu. Hit **<OK>**. For the voltage source you would change 0Vdc.

After you have created your schematic save your work from the main menu: **File->Save**

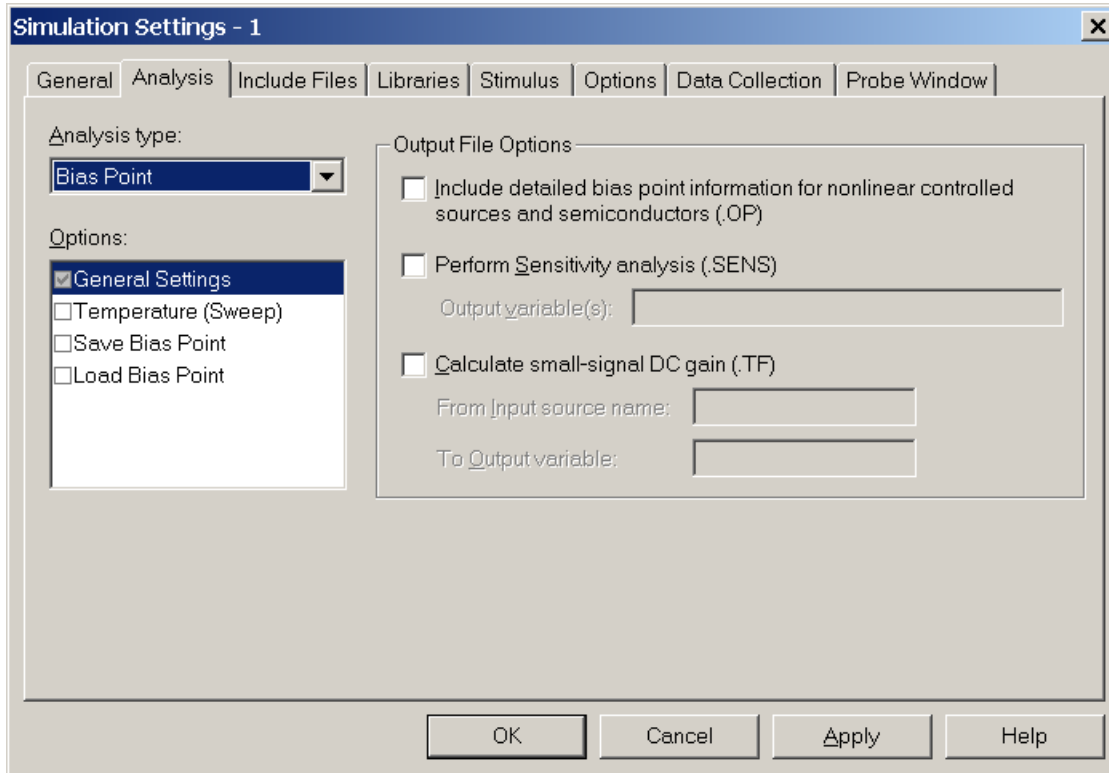
## How to perform Bias Point simulation:

Enter the schematic.



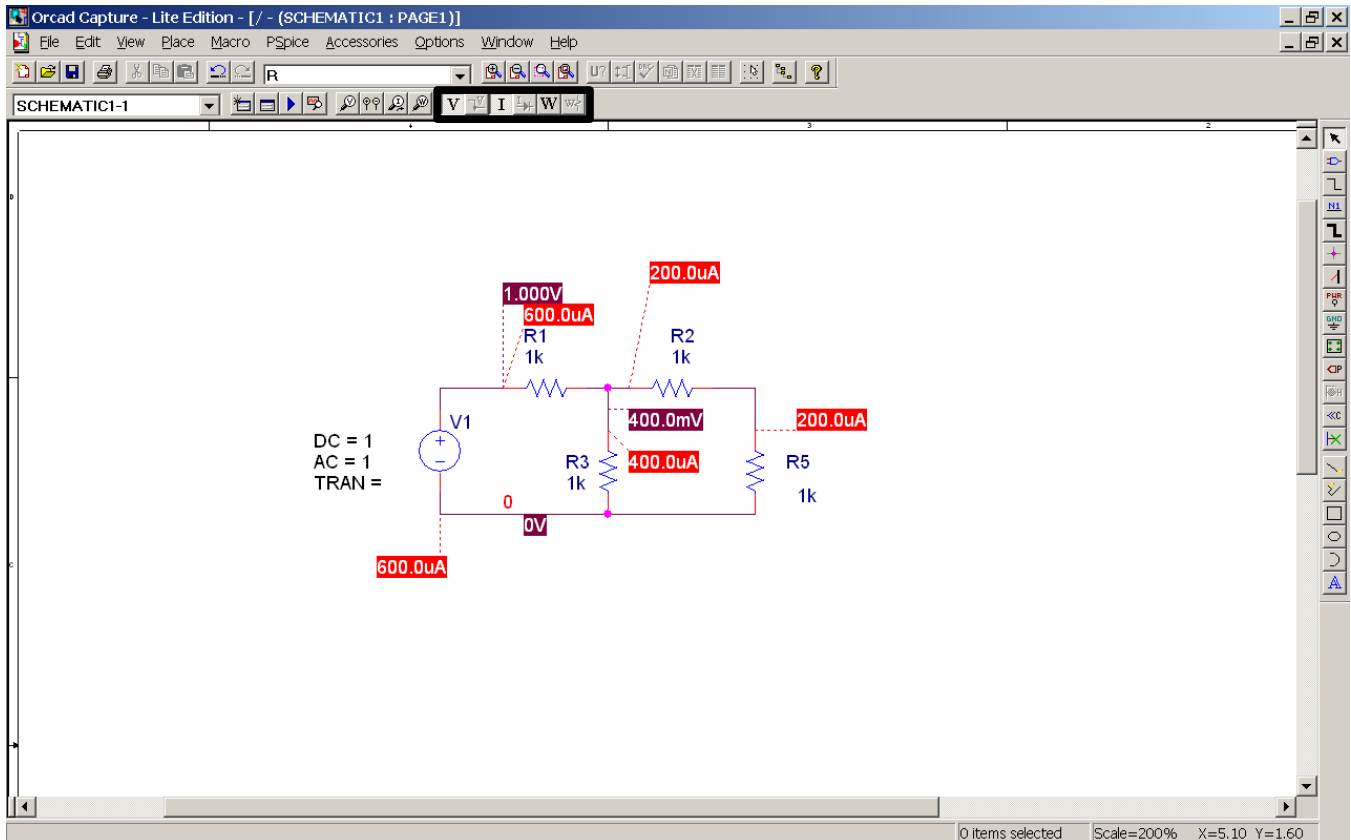
## To run the simulation:

1. **Pspice->New Simulation Profile**(or edit simulation profile) – enter the name of the simulation if it is new one
2. You will see the following menu. Choose **Bias Point** as shown.



3. Hit **OK**
4. To run the simulation main menu: **Pspice->Run**

There is a toolbar that allows you to display the values of the Voltage, Current, etc. for each component (see figure). In the figure below the values have been moved away from the wires for enhanced clarity. To move each value simply drag the value box of that value. A dotted line will connect to the pin of spot where the value is measured. In the case of current it is the value of the current flowing into that pin.



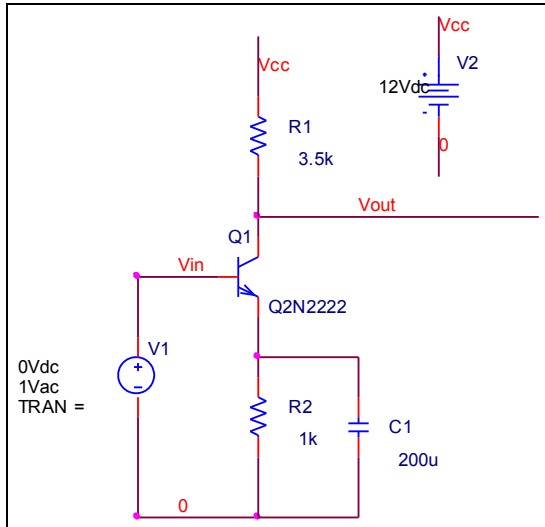
**How to perform DC sweep simulation:**

Below is the schematic of an amplifier with the following parameters:

- DC voltage gain =3.5
- AC Voltage gain of roughly -130 (f = 10KHz)
- input Q-point ( $V_{INQ}$ ) of 1.2Volts.

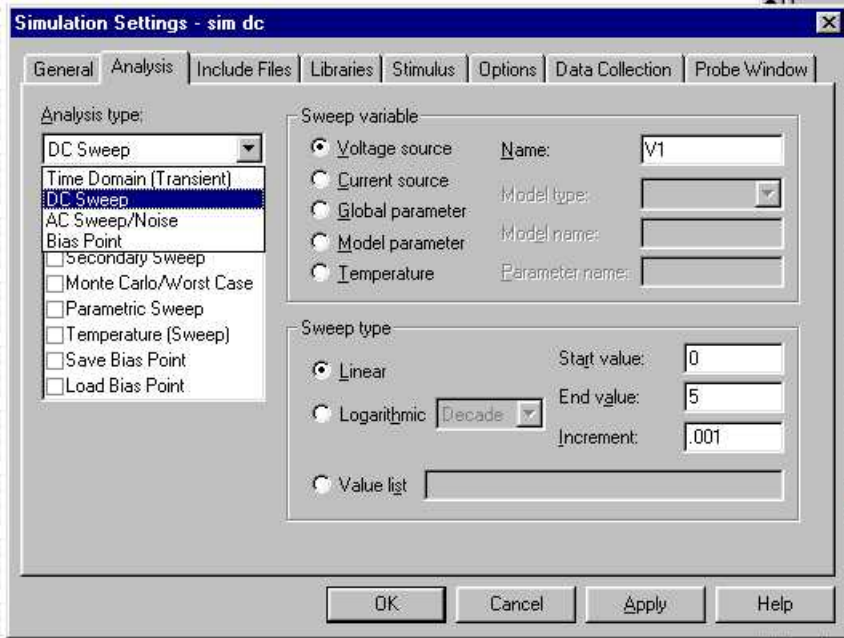
I usually use **VSRC** as the input voltage source for this type of simulation.

Note the name of the voltage source is V1 NOT Vin (you can change this if you wish).



**To run the simulation:**

1. **Pspice->New Simulation Profile**(or edit simulation profile) – enter the name of the simulation if it is new one
2. You will see the following menu. Choose **DC Sweep** as shown. Here SPICE is configured to Sweep the DC voltage **V1** from 0 to 5 Volts in increments of 1mV.



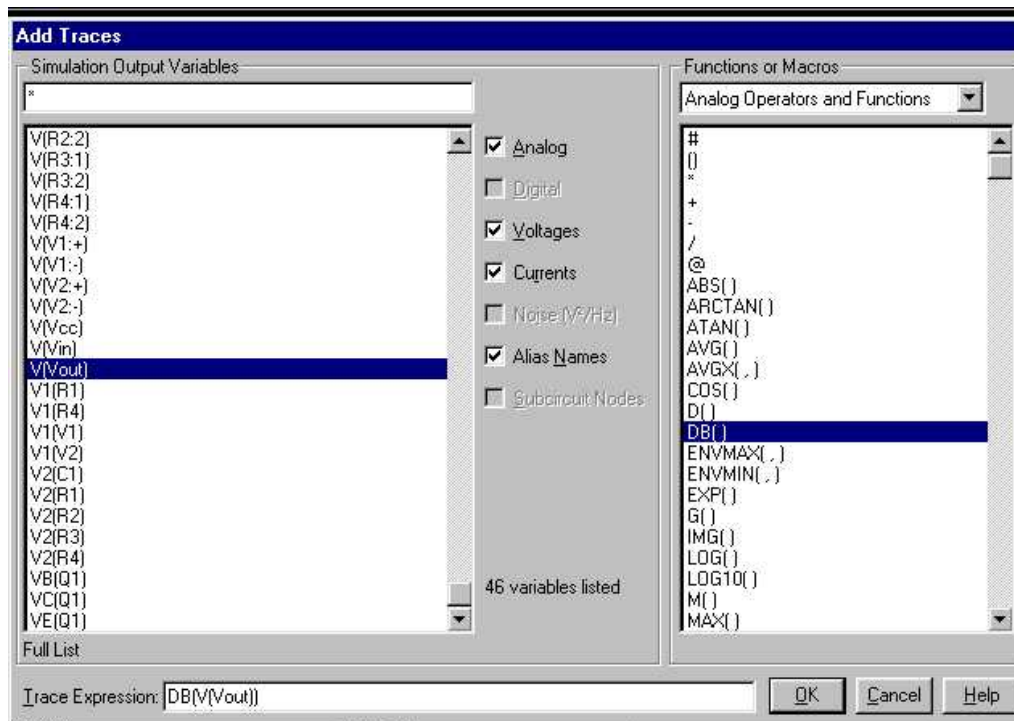
3. Hit **OK**
4. To run the simulation main menu: **Pspice->Run**

## To Plot the results:

After the simulation is complete you will automatically see a window where you can display the results of your simulation.

- From the menu choose: **Trace->Add Trace**
- You will see the following menu. On the left hand side are all the traces that are possible. On the right hand side are all the functions that you can apply if you wish. Here I will be plotting the output voltage expressed in dB. Note the output voltage is also the collector voltage of the transistor so V(Vout) and VC(Q1), V(Q1:c) are the same thing. You can also plot currents IC(Q1) would be the collector current IB(Q1) the base current.

To see a plot of the phase use function: P()

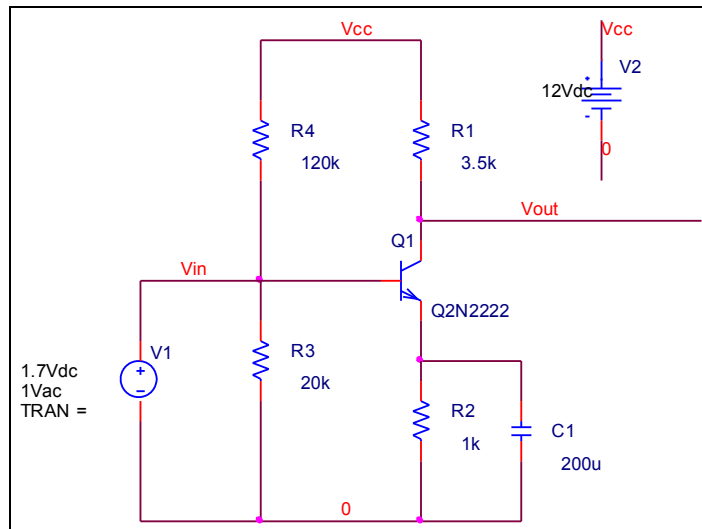


## How to perform an AC simulation:

Below is the schematic of an amplifier with the following parameters:

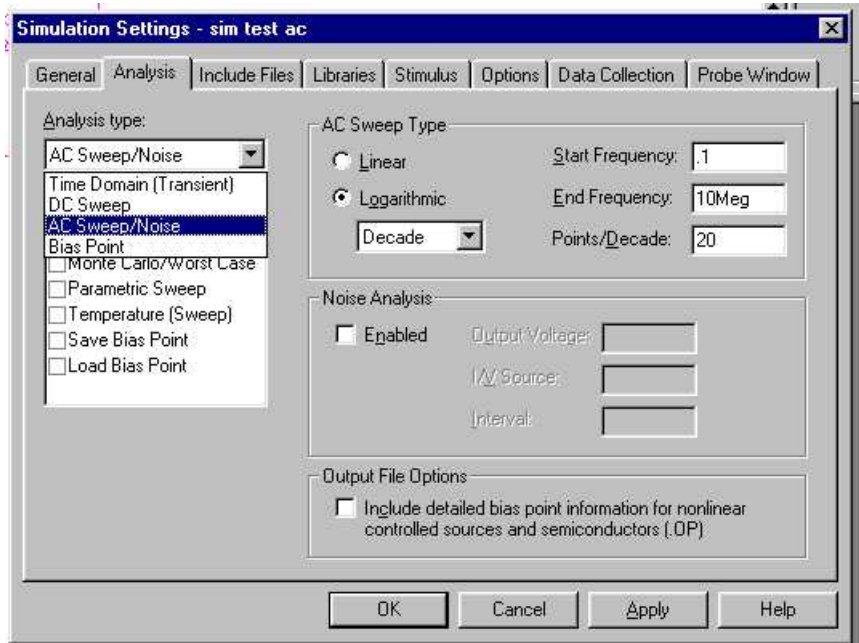
- DC voltage gain =3.5
- AC Voltage gain of roughly -130 (f = 10KHz)
- input Q-point ( $V_{INQ}$ ) of 1.2Volts.

You can use either **VAC** or **VSRC** as the input voltage source for this type of simulation. (I prefer to use **VSRC**). Note the input Q-point,  $V_{INQ}$ , is entered as the DC parameter of the Voltage source. The AC parameter is set to 1V; this is a convenient set up for determining gain parameters. At first glance you might think that this is a problem. The output voltage at 10KHz ( $1V \times -130$ ) should surely exceed the power supply. With this type of simulation, Spice is determining the small signal behavior of the amplifier as a function of frequency. It then takes the computed gain and multiplies it by 1V (the input AC voltage). For this type of simulation SPICE has no concept of what 130 volts is. This is NOT true for DC, bias point and transient analysis.



**To run the simulation:**

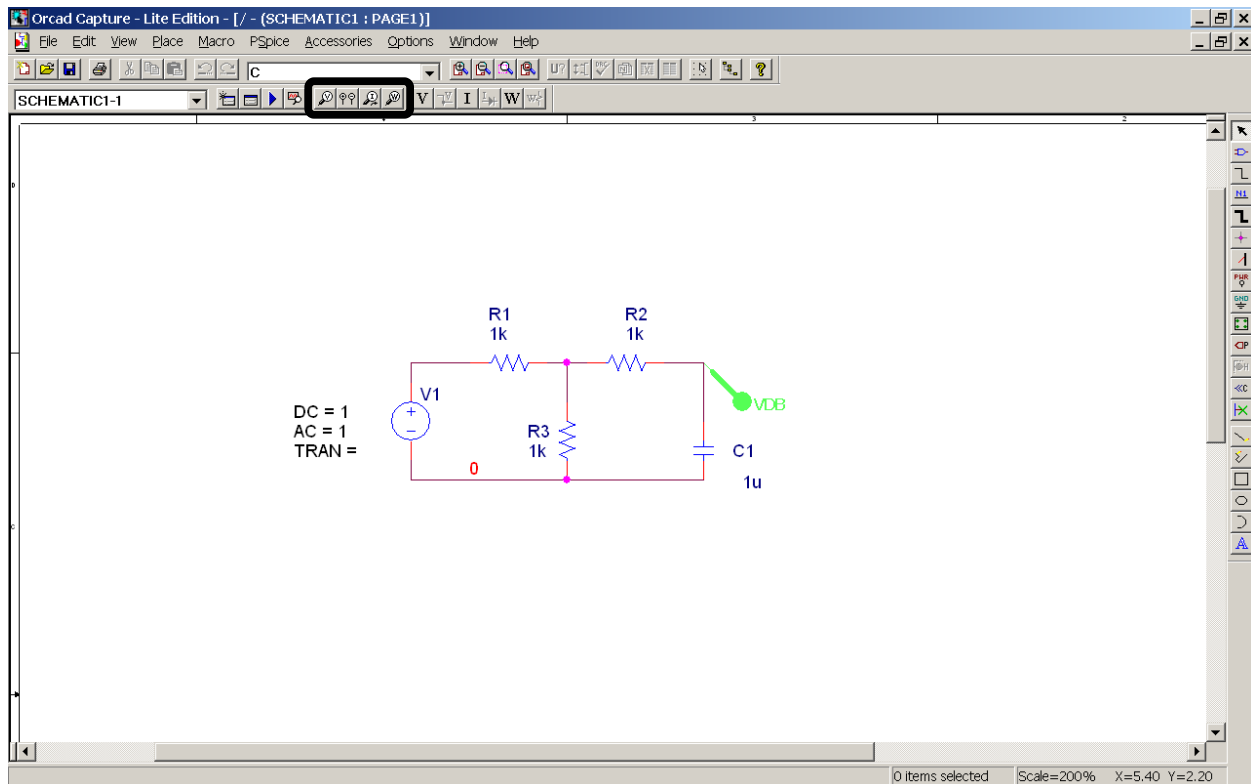
1. **Pspice->New Simulation Profile**(or edit simulation profile) – enter the name of the simulation if it is new one
2. You will see the following menu. Choose **AC Sweep/Noise** as shown. Here SPICE is configured to perform an AC sweep from 0.1 to 10M Hz. Do NOT make the starting frequency 0. MHz is indicated as 10Meg NOT 10M. The later would be 10mHz ( $10 \times 10^{-3}$ Hz).



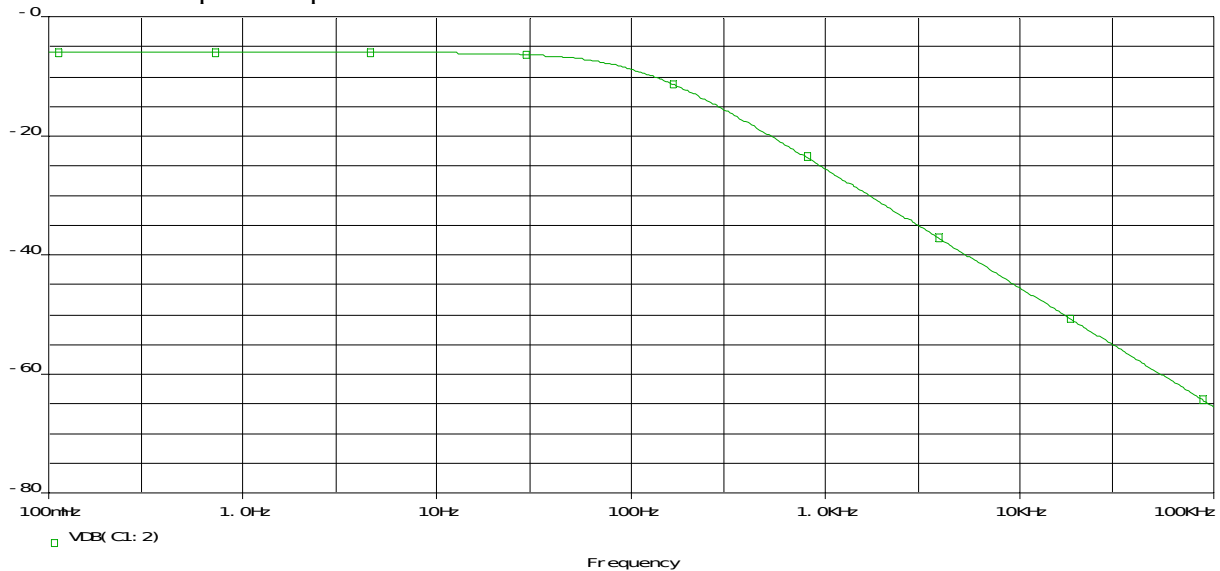
3. Hit **OK**
4. To run the simulation main menu: **Pspice->Run**

## Markers:

You can also use the GUI to place a marker. To place a normal Voltage, Current, Power, or Voltage Differential marker use the toolbar (see figure below). For other markers go to **PSPICE -> Markers -> Advanced**. Here you will find markers such as Voltage measured in dB (as shown on the schematic below).



This marker will produce plots like this one:



**\*Remember:** certain advanced markers will not be selectable from the menu unless the correct simulation type has been selected. For example, dB of Voltage will not be selectable in Bias Point analysis, so make sure you have set the correct simulation type first.