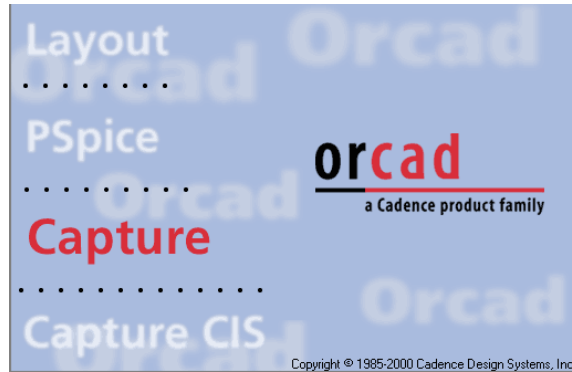# OrCAD PSPICE Tutorial

OrCAD Release 9.2, Capture Lite Edition, PSpice AD Lite Edition

Part 0: Some General PSpice Guidelines

1. All component leads must be connected. Floating leads will result in an error.

2. All circuits must have a zero, "0", reference ground node as shown below.

3. PSpice syntax is case insensitive, thus, PSpice will interpret a resistor component reference "r1" the same as "R1"; "k" (for kilo) the same as "K"; etc.

4. Component values may be designated with the following suffixes:

k, K	=	10^3	"kilo"
m, M	=	10^{-3}	"milli" *[Note: Since PSpice is case insensitive,*
meg, MEG	=	10^6	"mega" ***"M", designates "milli" not "mega"]***
u, U	=	10^{-6}	"micro"
g, G	=	10^9	"giga"
n, N	=	10^{-9}	"nano"
t, T	=	10^{12}	"tera"
p, P	=	10^{-12}	"pico"

5. Do not leave a space between a number and it's suffix, i.e. a resistor value of 1000 Ω can be designated as "1k", not "1 k".

6. Each instance of a component must be assigned a unique reference, i.e. there may only be one resistor labeled "R1" in a circuit.

7. If a PSpice simulation fails due to an error, check the session log or view the output file to determine which node(s) are associated with the error. Then look at the netlist to determine which component(s) are connected to the node(s) where the problem exits.
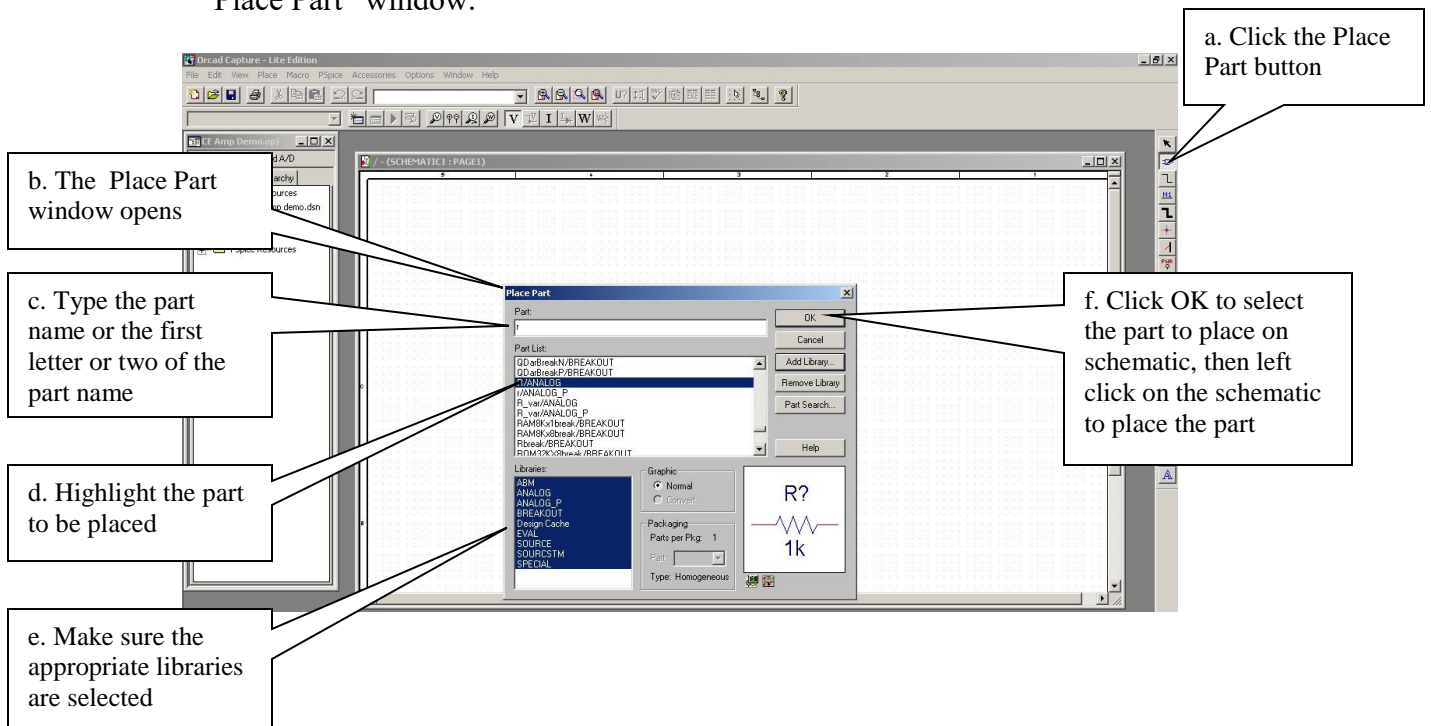
OrCAD PSPICE Tutorial

Simulation Analysis of an
Operational Amplifier Circuit

Part 1: Capturing (Drawing) the Circuit Schematic

1. Start ►Programs ► OrCAD Family Release 9.2 Lite Edition ► Capture Lite Edition. This is the schematic capture program you will use to draw the circuit you wish to simulate, in this case, the operational amplifier circuit shown above. PSpice simulations will generally be run from within Capture.

2. On the Capture toolbar, click the "Create document" button.

3. The "New Project" dialog box will open. Type in the name of the project, select the "Analog or Mixed A/D" button, specify a location to save the project, then click "OK".

a. Type in the name of the project

b. Select the Analog or Mixed A/D button

d. Click OK

c. Specify a location to save the project
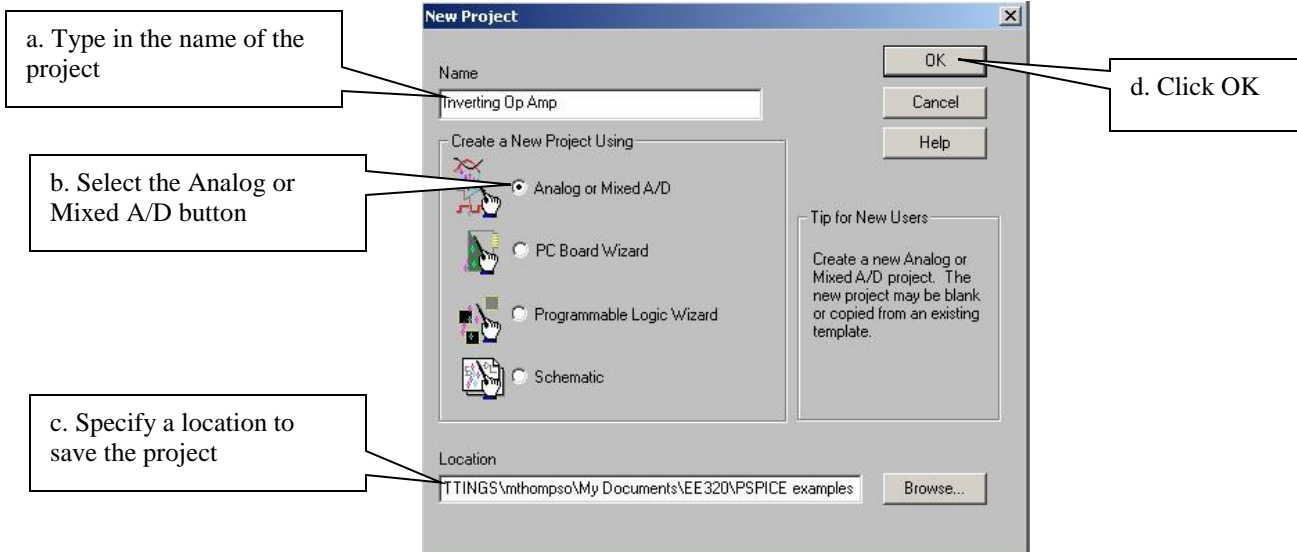
4. The "Create PSpice Project" dialog box will open. Select the "Create a blank project" button, then click "OK".

a. Select the Create a blank project button

b. Click OK
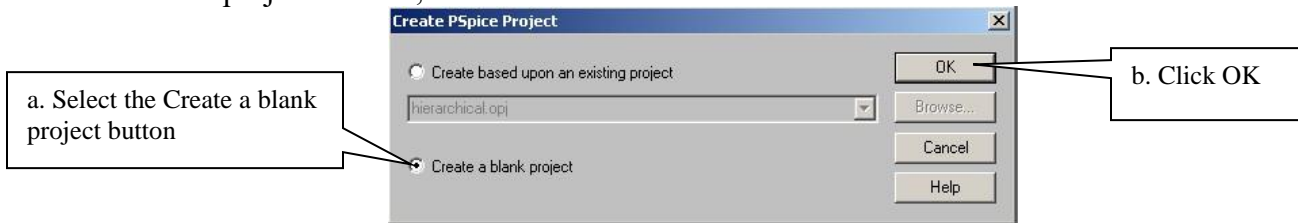
5. The Schematic Editor window will open. Click the "Place Part" button to open the "Place Part" window.

a. Click the Place Part button

b. The Place Part window opens

c. Type the part name or the first letter or two of the part name

f. Click OK to select the part to place on schematic, then left click on the schematic to place the part

d. Highlight the part to be placed

e. Make sure the appropriate libraries are selected

a. Type the name of the part in the "Part" dialog box. If you don't know the complete part name, the first letter or two of a part name will narrow the search, i.e. type "r" and a resistor part such as "R/ANALOG" will automatically be found and highlighted.
b. Make sure all the appropriate libraries are selected. If need be, add libraries by clicking the "Add Library" button.
c. Highlight the part to be placed on the schematic and click "OK", then left click on the schematic to place the part.
d. In this fashion, place all the parts on the schematic for the operational amplifier circuit.
e. Use VDC and VSIN parts for the dc voltages and sinusoidal signal voltage sources, respectively.
f. For convenience, a separate "Place Ground" button for placing grounds (zero reference nodes) is available.

Place wire button

Place Ground button
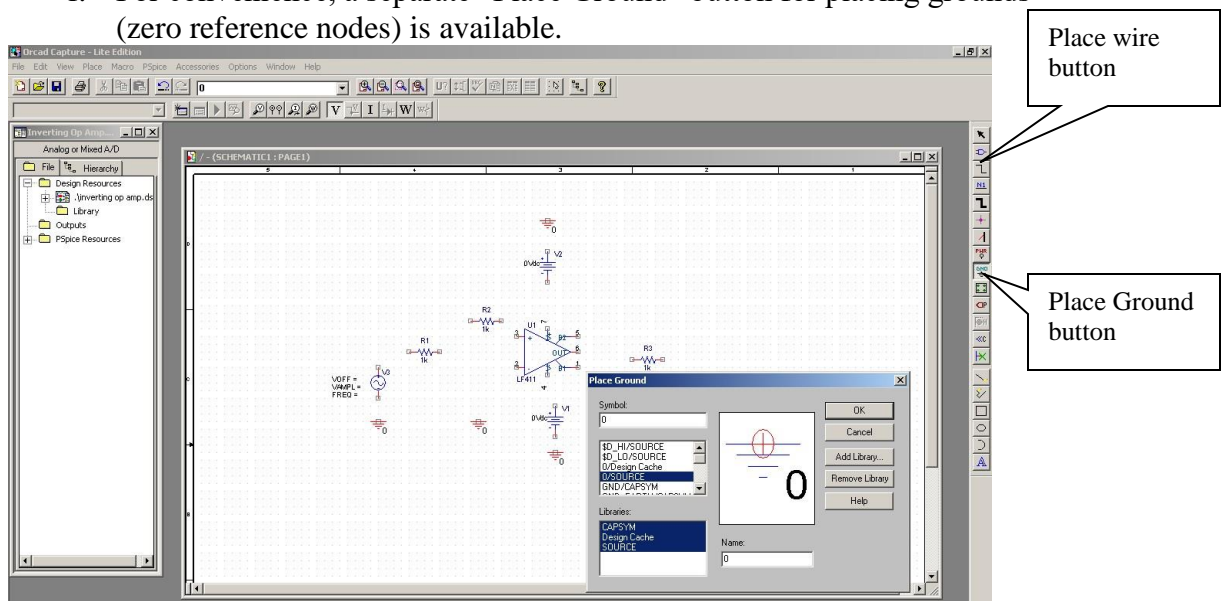
g. After the parts are placed, they can be rotated with "Ctlr-R" or mirrored horizontally or vertically with "H" or "V", respectively and dragged into place.
h. Wiring connections are made by selecting the "Place wire" button. The cursor arrow will change to a cross-hair. Point to a component connection where the wire is to begin and left click the mouse, then point to where the wire is to terminate and left click again. Continue in this fashion until all the wiring connections are made. When finished, right click and select "End Wire" in the pop-up menu to return to normal cursor mode.
i. The default values of component parameters can be changed by double-clicking the parameter on the schematic, i.e. to change the resistance value of R2 from the default value of 1k to 10k, double-click on 1k on the schematic and change the Value to 10k in the "Display Properties" dialog box. Click "OK". Follow the same procedure to change a Part reference description, i.e. change the default R3 reference to RL and it's value to 10k.

a. Change the default value of a resistor component

b. Click OK
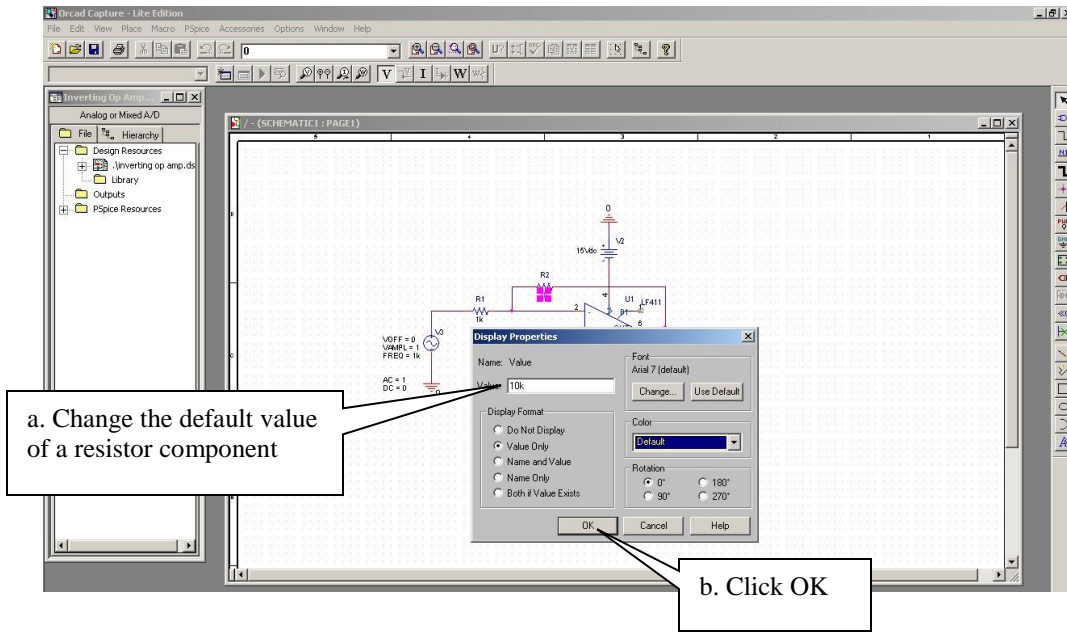
j.   Your completed circuit schematic should look like the one below.
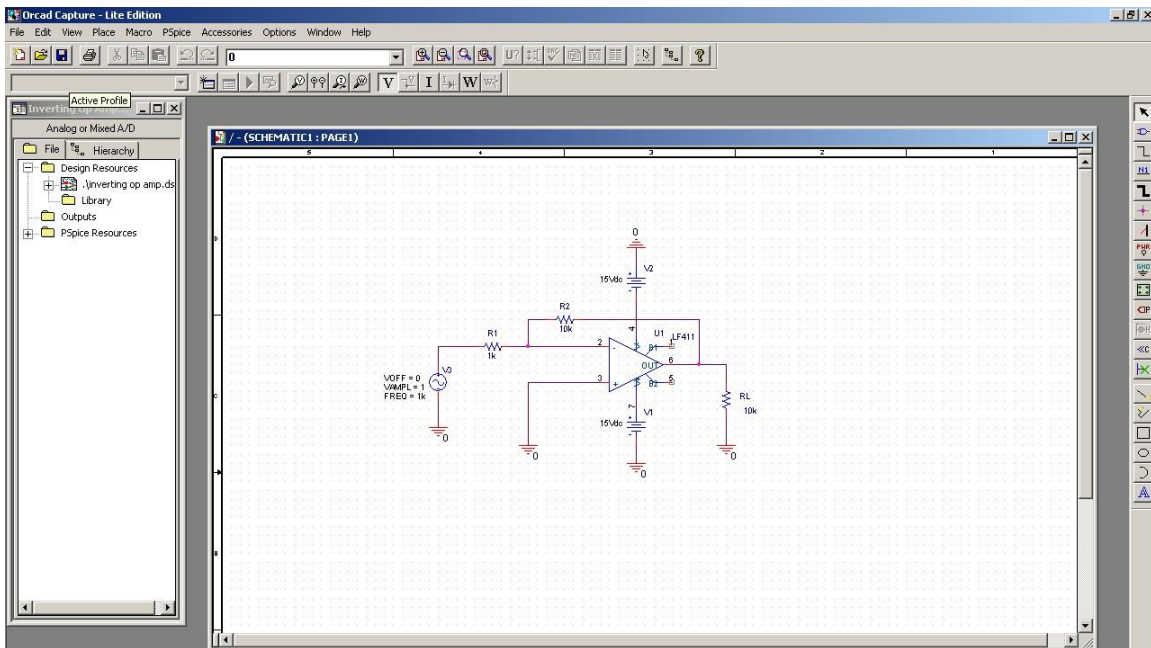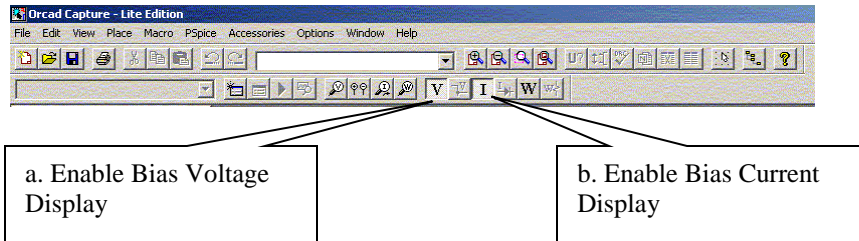
OrCAD PSPICE Tutorial
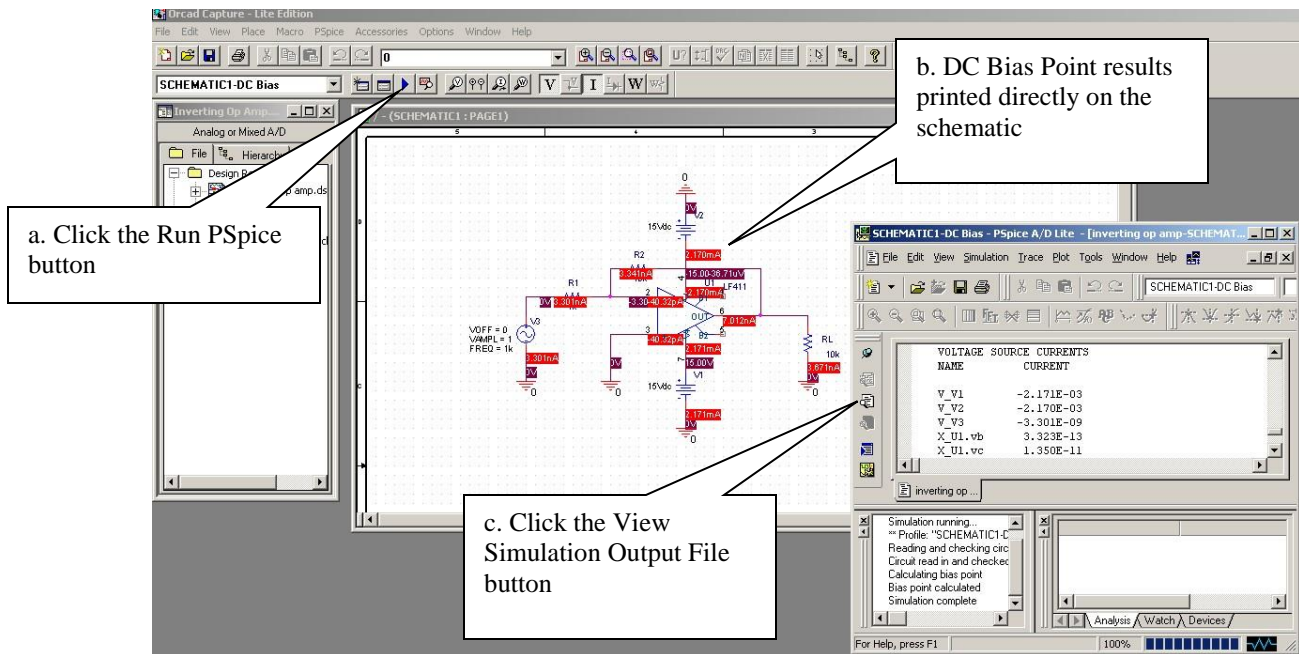## Part 2: DC Bias Point Analysis

1. On the Capture/PSpice toolbar, click the "New Simulation Profile" button. *(You can have different simulation profiles for the same project to allow for different ways of analyzing the circuit, i.e. you can have one simulation profile for dc bias point analysis, one for transient analysis, another one for ac sweep analysis, etc.)* The "New Simulation" dialog box will open. Type in the name of the new simulation, such as "DC Bias Point". Click "Create". The "Simulations Settings" dialog box will open under the "Analysis" tab. In the "Analysis type" drop-down box, select "Bias Point". There are no other settings to change for dc bias point analysis, so click "OK".

a. Click the New Simulation Profile button

b. Type in the name of the New Simulation

c. Click Create

d. Select Analysis type, Bias Point

e. Click OK
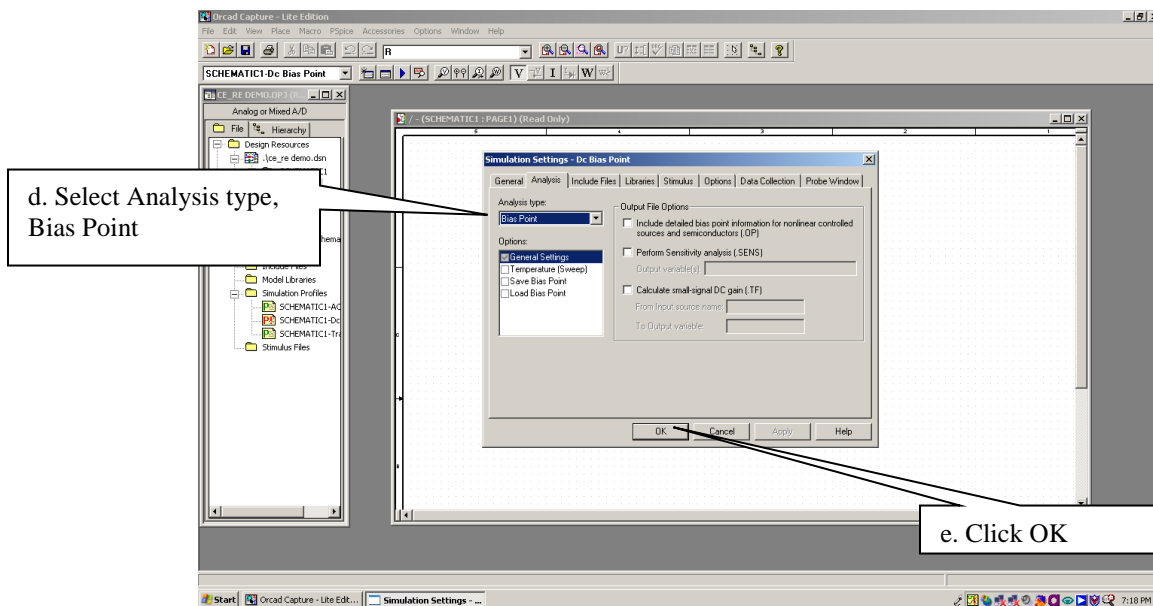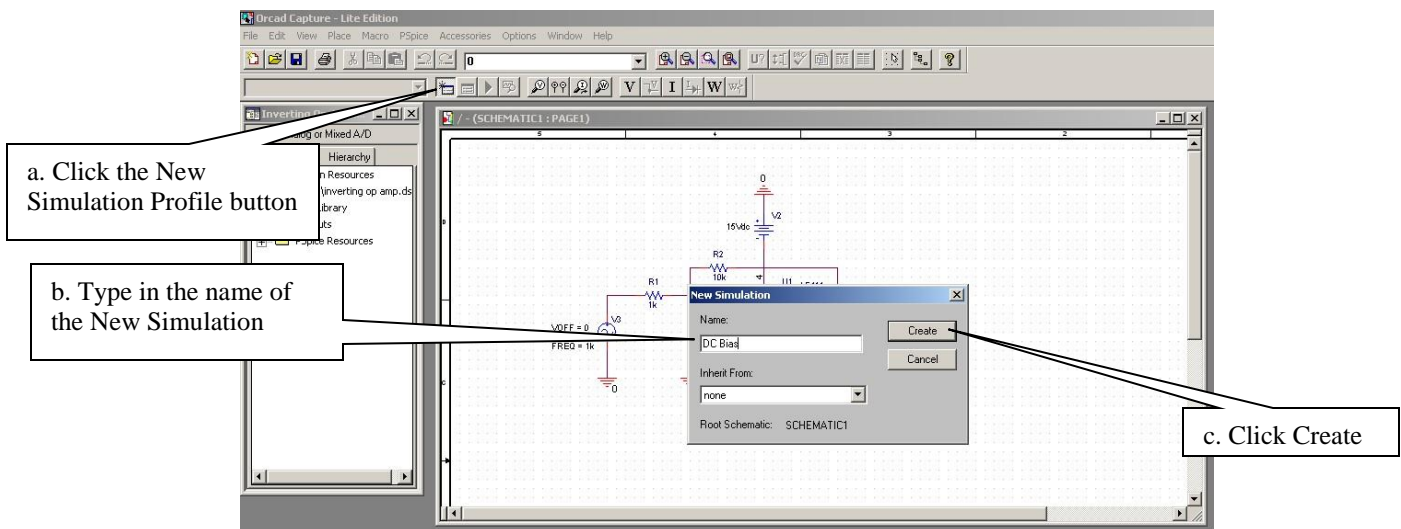
2. On the Capture/PSpice toolbar, click the "Enable Bias Voltage Display", and the "Enable Bias Current Display" buttons to display the results of the dc bias point analysis directly on the schematic.

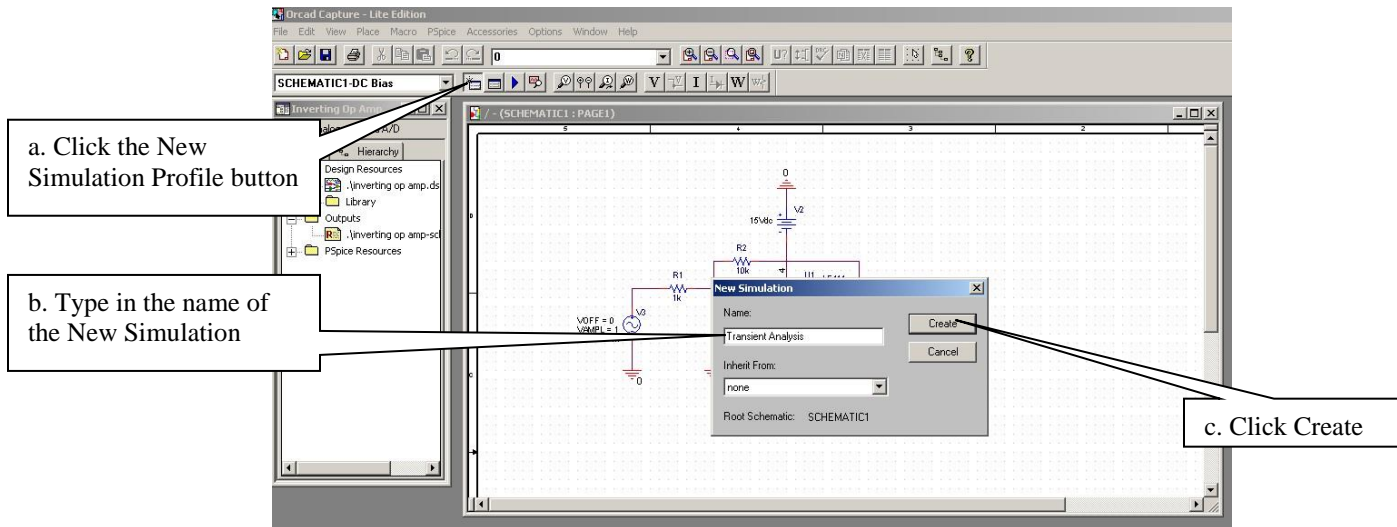a. Enable Bias Voltage Display

b. Enable Bias Current Display

3. On the Capture/PSpice toolbar, click the "Run PSpice" button to perform the dc bias point analysis. The results are printed directly on the schematic and the Probe window opens automatically (though in the case of a basic bias point analysis, there is nothing to plot). If you wish, you can also view the results of the simulation in the Probe window by clicking the "View Simulation Output File" button
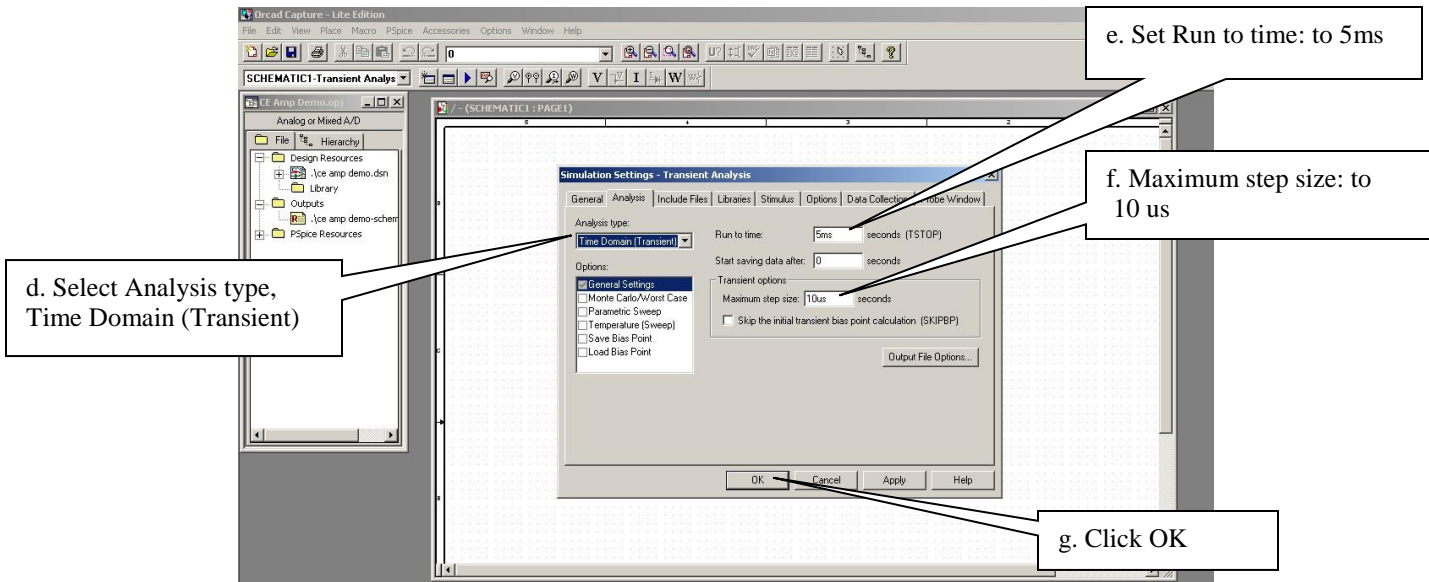

b. DC Bias Point results printed directly on the schematic

a. Click the Run PSpice button

c. Click the View Simulation Output File button

OrCAD PSPICE Tutorial
Part 3: Transient Analysis

1. On the Capture/PSpice toolbar, click the "New Simulation Profile" button. *(You can have different simulation profiles for the same project to allow for different ways of analyzing the circuit, i.e. you can have one simulation profile for dc bias point analysis, one for transient analysis, another one for ac sweep analysis, etc.)* The "New Simulation" dialog box will open. Type in the name of the new simulation, such as "Transient Analysis". Click "Create". The "Simulations Settings" dialog box will open under the "Analysis" tab. In the "Analysis type" drop-down box, select "Time Domain (Transient)". In the "Run to time" dialog box, type 5ms. This specifies how long the simulation will run and is dependent on the stimulation waveform and what you which to observe in the simulation. *(In this case, the stimulation waveform is a sine wave of frequency 1kHz (period equal to 1ms) and thus a simulation run time of 5ms will simulate 5 complete cycles of the sine wave.)* Under "Transient options", type 10us in the "Maximum step size" dialog box. This specifies how often PSpice recalculates the circuit parameters during simulation. To have smooth looking sine waves, you need at least 100 calculations per cycle. *(In this case, a step size of 0.01ms = 10us corresponds to 1/100 of a cycle.)* Click "OK".

a. Click the New Simulation Profile button

b. Type in the name of the New Simulation

c. Click Create

e. Set Run to time: to 5ms

f. Maximum step size: to 10 us

d. Select Analysis type, Time Domain (Transient)

g. Click OK

2. In a time domain (transient) analysis, you often want to view the simulation results as a plot of specific waveforms as a function of time in a Probe window graphics display. To have PSpice do this automatically, place voltage probes at the nodes of interest on the schematic. To do so, click the "Voltage/Level Marker" button on the Capture/PSpice toolbar then left click at a node where you would like a voltage waveform displayed. *(In this case, place two voltage probes, one on the input signal node and one at the output signal (RL) node.)* When finished placing voltage probes, right click and select "End Mode" from the pop-up menu. *(Differential Voltage Marker(s), Current Markers, and Power Dissipation Markers are also available if these waveforms are of interest.)*

a. Click the Voltage/Level Marker button

c. Place a Voltage probe on the output signal (RL) node

b. Place a Voltage probe on the input signal node
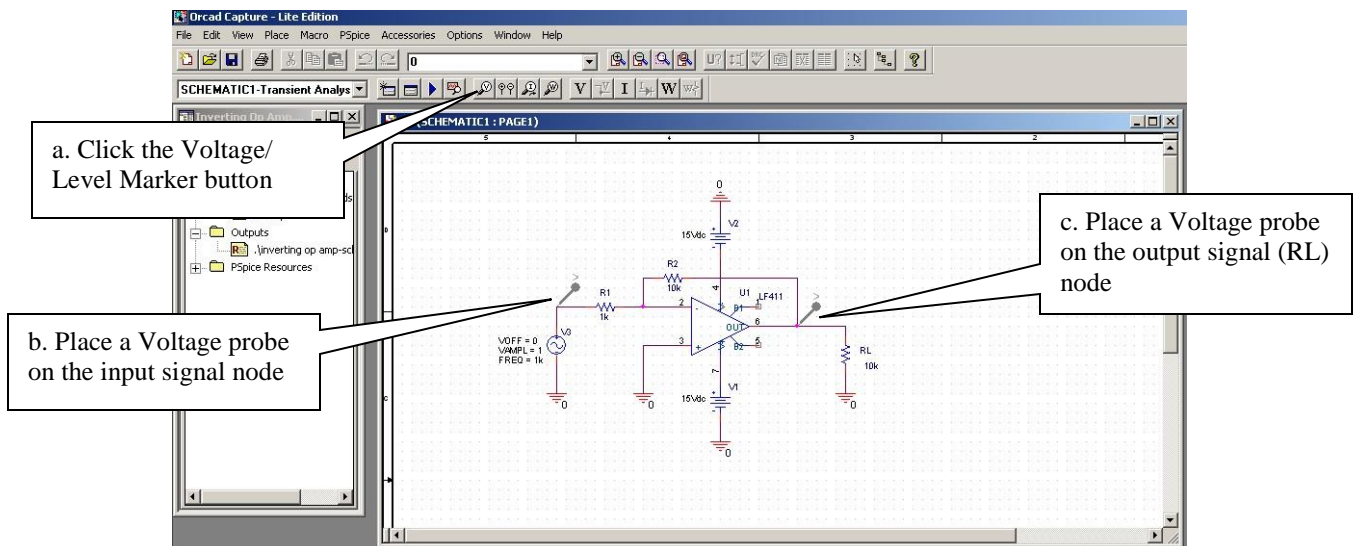
3. On the Capture/PSpice toolbar, click the "Run PSpice" button to perform the transient analysis. The Probe window opens automatically displaying the waveforms that were probed on the schematic *(in this case, the input signal and output signal waveforms of the op amp circuit).* The color of the probe on the schematic corresponds to the color of the trace on the graphics display. *(Note: If you are performing a transient analysis, you do not need to perform a separate dc bias point analysis. The results of the dc bias point analysis can be displayed on the schematic by clicking the "Enable Bias Voltage Display", and the "Enable Bias Current Display" buttons on the Capture/PSpice toolbar, as outlined in the DC Bias Point Analysis section of this tutorial.)*

a. Click the Run PSpice button

b. Green voltage probe displaying the input signal waveform

c. Red voltage probe displaying the output signal waveform

Probe window containing color coded graphics display of probed waveforms

Close-up of graphics display of probed waveforms

4. Experiment on your own with the features of the Probe graphics display window. Utilize some of these features to measure the magnitude and phase shift of the voltage gain of the amplifier at 1kHz. Utilize other features to label the graph with pertinent information. ***Remember: PSpice performs the simulation analysis but you must still interpret the results of the simulation and effectively communicate these results to your audience.***

10

OrCAD PSPICE Tutorial
Part 4: AC Sweep Analysis (Frequency Response)

1. On the Capture/PSpice toolbar, click the "New Simulation Profile" button. *(You can have different simulation profiles for the same project to allow for different ways of analyzing the circuit, i.e. you can have one simulation profile for dc bias point analysis, one for transient analysis, another one for ac sweep analysis, etc.)* The "New Simulation" dialog box will open. Type in the name of the new simulation, such as "AC Frequency Sweep". Click "Create". The "Simulations Settings" dialog box will open under the "Analysis" tab. In the "Analysis type" drop-down box, select "AC Sweep/Noise". Now, specify the simulation parameters as follows: Select the "Logarithmic" sweep button. In the "Start Frequency" dialog box, type 10Hz. In the "End Frequency" dialog box, type 1GHz. In the "Points/Decade" dialog box, type 10. Click "OK".
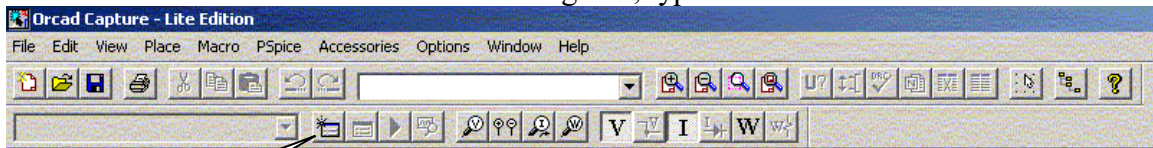
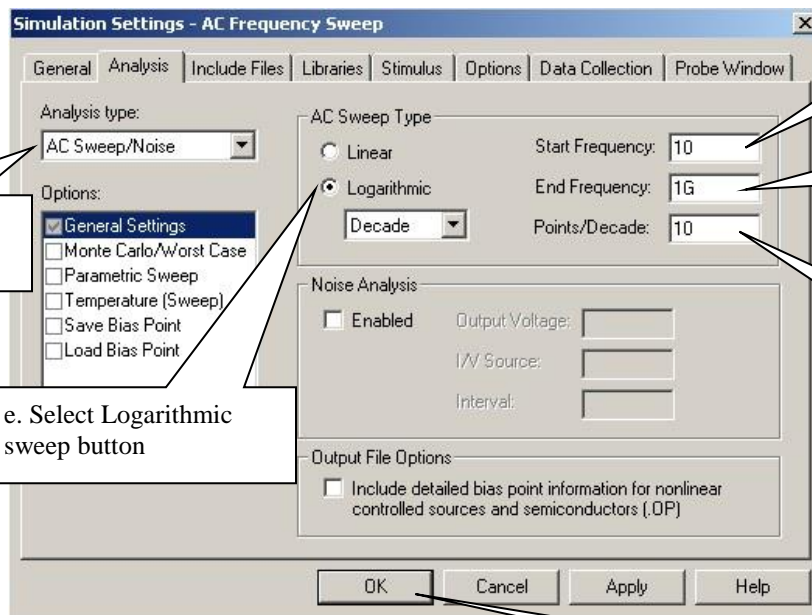a. Click the New Simulation Profile button

b. Type in the name of the New Simulation
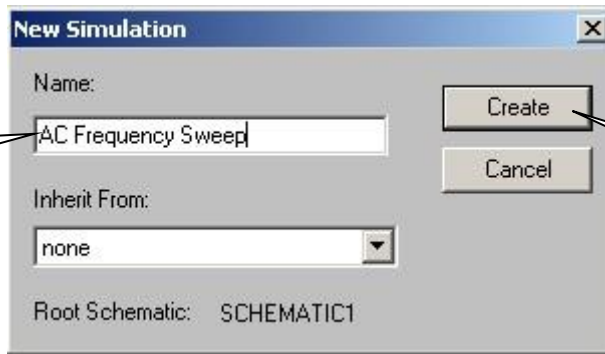
c. Click Create

d. Select Analysis type, AC Sweep/Noise

e. Select Logarithmic sweep button

f. Type in the Start Frequency

g. Type in the End Frequency

h. Type in the Points/Decade

i. Click OK

2. In a frequency response (ac sweep) analysis, you often want to view the simulation results as a Bode plot of the gain function in a Probe window graphics display. To have PSpice do this automatically, we will set the AC magnitude of the input voltage equal to 1 (a useful analysis trick that makes the output voltage magnitude equal to the voltage gain magnitude) and place a single voltage probe at the output node on the schematic. Most time-varying sources also have an AC magnitude parameter, though it is not usually set to be displayed by default. To set the small-signal AC magnitude of the time varying source, VSIN, equal to 1 and display it on the schematic, double click on the circuit symbol of V3 on the schematic to open the "Property Editor" window.

a. Double click on the circuit symbol on the schematic to open the Property Editor window

In the "Property Editor" window, highlight the "AC" parameter box and type in "1" to set the AC magnitude equal to 1, then select the "Display" button to open the "Display Properties" window. Under "Display Format" in the "Display Properties" window, select the "Name and Value" button, then click "OK".

c. Select the Display button.

b. Highlight the AC parameter box and type in "1".

d. Select the Name and Value button.

e. Click OK

In a similar fashion, set the DC parameter equal to "0" and display it on the schematic. Next, place a single voltage probe at the output signal (RL) node on the schematic as you did in the Transient Analysis section of this tutorial.

3. On the Capture/PSpice toolbar, click the "Run PSpice" button to perform the ac frequency sweep analysis. The Probe window opens automatically displaying the output voltage magnitude (representing the voltage gain magnitude) as a function of frequency. The color of the probe on the schematic corresponds to the color of the trace on the graphics display. *(Note: If you are performing an ac sweep analysis, you do not need to perform a separate dc bias point analysis. The results of the dc bias point analysis can be displayed on the schematic by clicking the "Enable Bias Voltage Display", and the "Enable Bias Current Display" buttons on the Capture/PSpice toolbar, as outlined in the DC Bias Point Analysis section of this tutorial.)* To display the voltage gain magnitude in decibels, open the "Modify Trace" window by double clicking on the output voltage reference in the legend of the Probe window display. Edit the "Trace Expression" dialog box to read "dB(V(XX:YY))", where XX and YY reference the output voltage. This is equivalent to "20*log(V(XX:YY))". Click "OK" to display the Bode plot of voltage gain magnitude in dB as a function of frequency.

a. Click the Run PSpice button

b. Probe window displaying the output voltage magnitude (representing the voltage gain magnitude) as a function of frequency.

c. Open the Modify Trace window by double clicking on the output voltage reference in the legend of the Probe window

Modify Trace

Simulation Output Variables

`*`

```
Frequency
I(R1)
I(R2)
I(RL)
I(U1:+)
I(U1:-)
I(U1:OUT)
I(U1:V+)
I(U1:V-)
I(V1)
I(V2)
I(V3)
I(X_U1.c1)
I(X_U1.c2)
I(X_U1.css)
I(X_U1.dc)
I(X_U1.de)
I(X_U1.dln)
```

☑ Analog
☐ Digital
☑ Voltages
☑ Currents
☑ Power
☐ Noise (V²/Hz)
☑ Alias Names
☑ Subcircuit Nodes

189 variables listed

Full List

Functions or Macros

Analog Operators and Functions ▼

```
#
()
*
+
-
/
@
ABS( )
ARCTAN( )
ATAN( )
AVG( )
AVGX( , )
COS( )
D( )
DB( )
ENVMAX( , )
ENVMIN( , )
EXP( )
G( )
IMG( )
LOG( )
LOG10( )
M( )
MAX( )
```

d. Edit the Trace Expression dialog box to read dB(V(RL:2))

Trace Expression: dB(V(RL:2))

OK   Cancel   Help

e. Click OK

□ dB(V(RL:2))

Frequency

f. Display of Bode plot of voltage gain magnitude in dB as a function of frequency.

4. Experiment on your own with the features of the Probe graphics display window. Utilize some of these features to measure the 3 dB bandwidth of the op amp circuit. Utilize other features to label the Bode plot with pertinent information. *Remember: PSpice performs the simulation analysis but you must still interpret the results of the simulation and effectively communicate these results to your audience.*

14