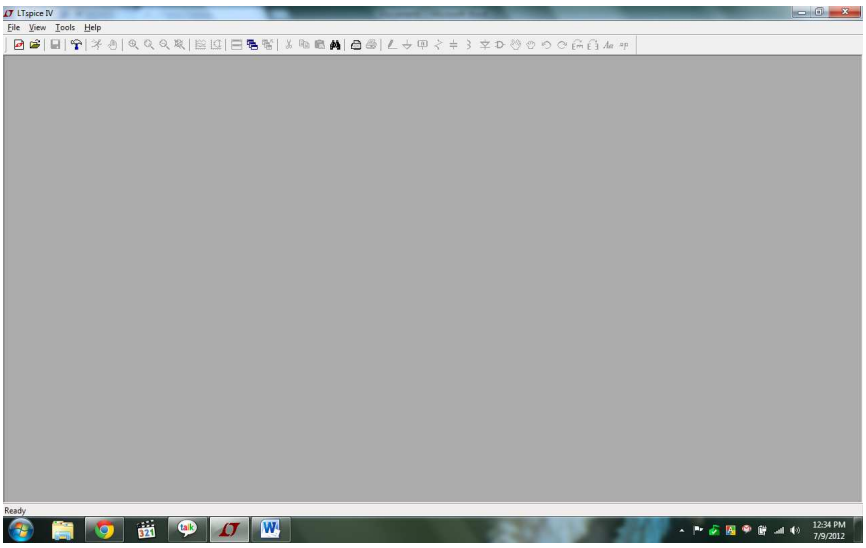
Introduction to LTSpice IV

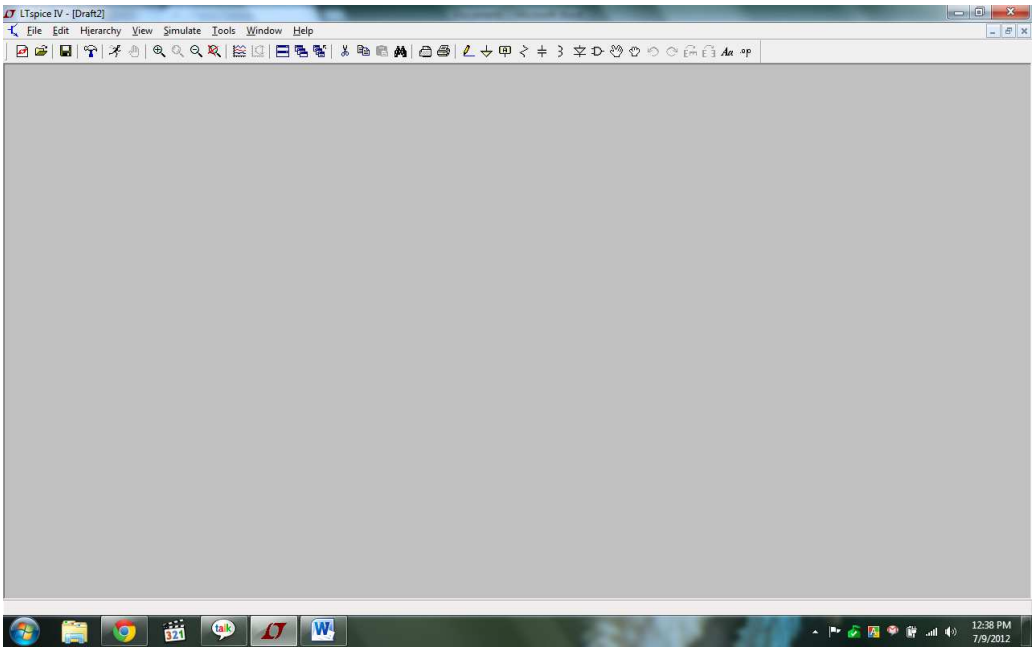
Opening LTSpice IV

1. Download LTSpice IV from Linear Technology Website (www.linear.com).
2. Click on the LTSpice IV shortcut on your desktop. The opening screen will look like this:

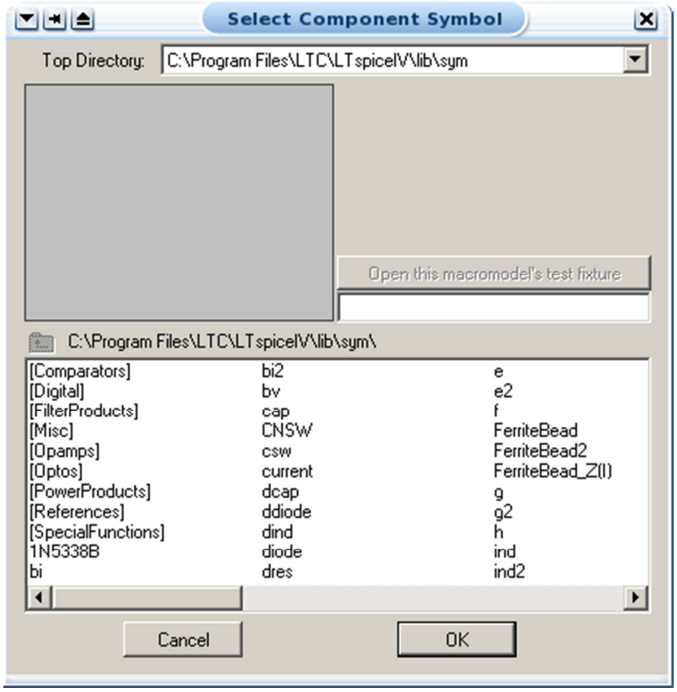


Drawing the Circuit in Schematic Window

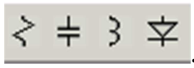
1. Go to *File>New Schematic*, or click on New Schematic icon which looks like this 
   The schematic window will look like this (notice the grey inactive buttons are active now):



2. Get the components you need to draw by clicking the component button  , pressing F2 or going to *Edit>Component*. The component window looks like this:



– To rotate a component, press CTRL+R, click  or go to *Edit>Rotate*. To get the mirror image of a component, press CTRL+E, click  or go to *Edit>Mirror*.

– You can also gate specific components by clicking on their symbols in the menu bar  . This is good for getting common components like resistors, capacitors, inductors and diode.

– To edit the values of the passive elements (inductors, capacitors and resistors), right click on the components and enter the value. Or you can right click on the L/R/C text and just enter the value.

– To edit the voltage and current sources, right click on them. By default, the sources are DC. So if you are dealing with a DC circuit, you can just enter the value of the source. However, to deal with AC sources, after you right click on them, click on the advanced button, and select the source type (sinusoidal, pulse, piecewise linear etc.). Edit the sources as the desired circuit.

3. Click on the wire button  , press F3 or go to *Edit>Draw Wire* to connect the components to draw your circuit. When three or more wires connect at a point, you should see a squire box.

4. Add a ground to your circuit by clicking on  , pressing G key on keyboard, or going to *Edit>Place GND*. This is very important; **you cannot simulate your circuit without a ground**.

5. If you need to move any component, use . If you need to drag a portion of your circuit, use . To delete a component, press Delete key or use . To copy a component, click or press CTRL+C, click on the component you want to copy, and then click on the location where you want the new component. Right click or press the ESC key to end any command.
6. To label any node, click , press F4 or go to *Edit>Label Net*. You can label the nets to identify the nodes of interest easily in the simulation.
7. To edit the properties and values of the components, right click on them.
8. To edit the names of the components, right click on their names.

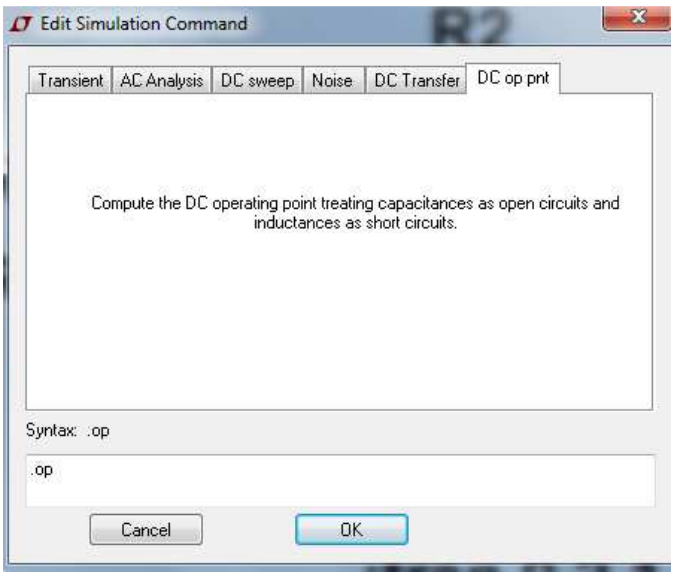
Simulating the Circuit:

Before you start the simulation, make sure that

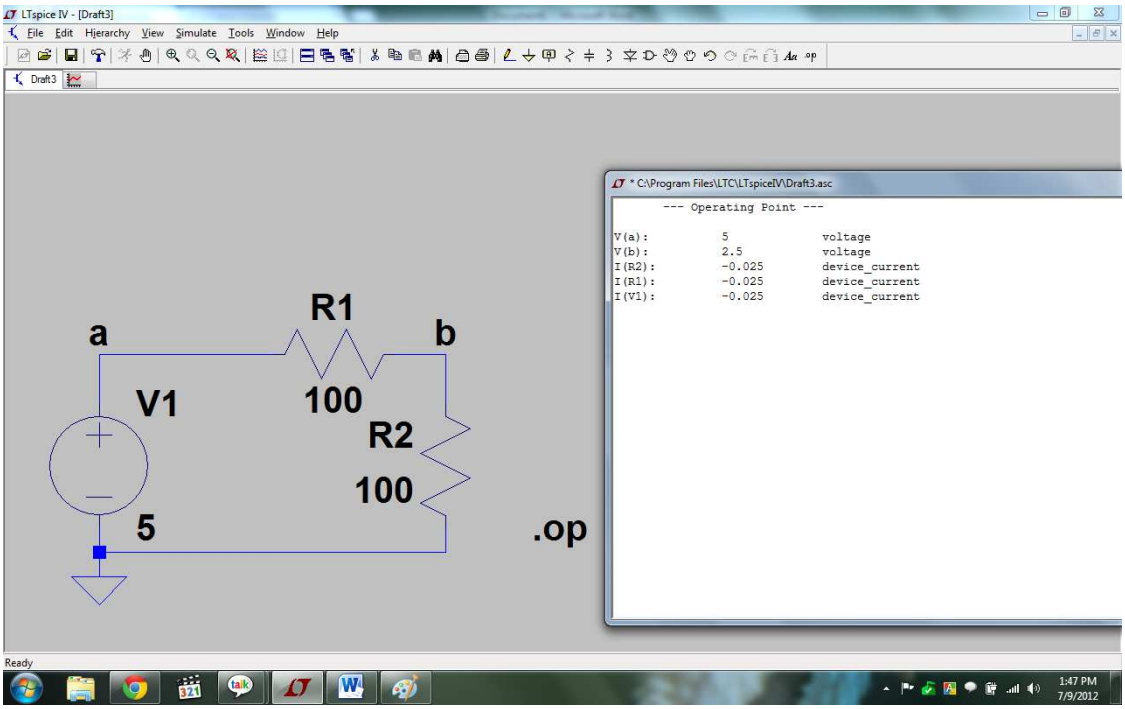
- You have to have your circuit properly drawn and saved.
- There must not be any floating parts on your page (i.e. unattached devices).
- You should make sure that all parts have the values that you want.
- There are no extra wires.
- **It is essential that you have a ground in your circuit.**

DC Simulation

1. Draw the DC circuit by following the steps described above.
2. Go to Edit>Simulation CMD. Choose the type of analysis you want to simulate.
3. To view the DC operating points, write .op in the syntax field of the command window, or select DC op pnt tab as shown below, then click on anywhere in the schematic window to place command:

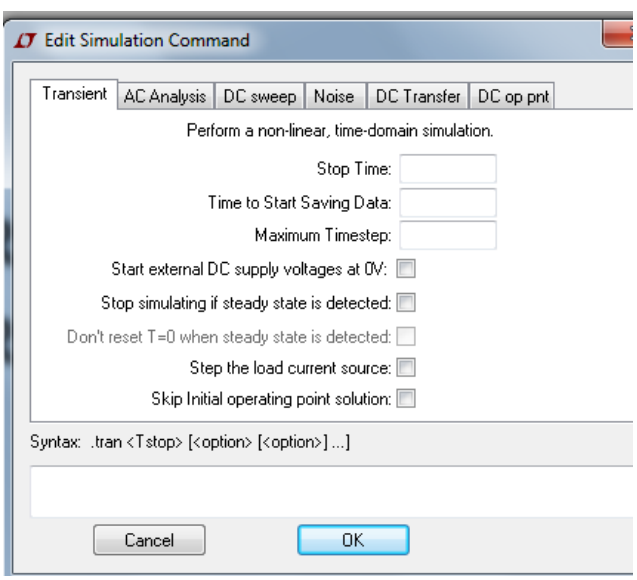


4. Click on the  button or go to *Simulate>Run* to run your simulation.
   A simulation showing only the DC operating points will look like this:

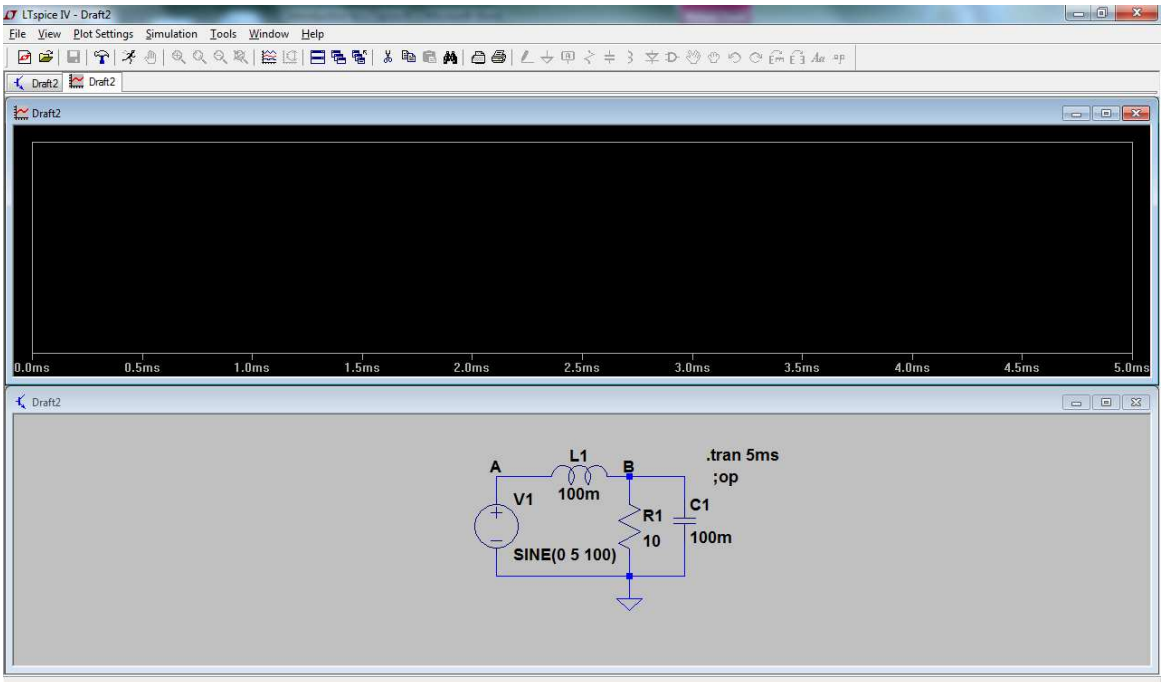


Transient Analysis

1. To do the transient simulation, click on the Transient tab and fill the command window with appropriate values:

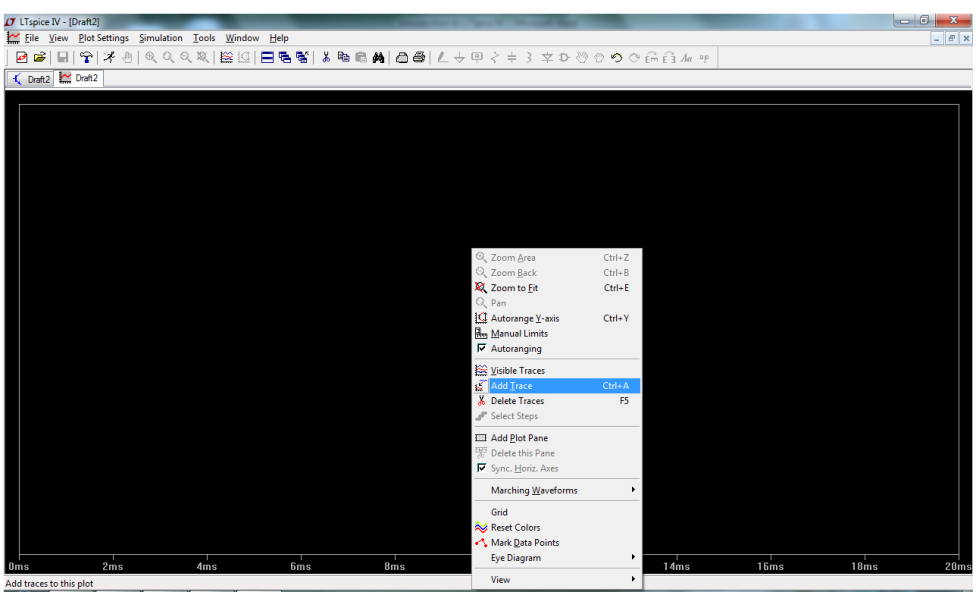


2. As you fill in the command window, the simulation command will be automatically be generated. Click on anywhere in the schematic window to place the simulation command. Then run the simulation as before.
3. A simulation showing the transient analysis will look like this:

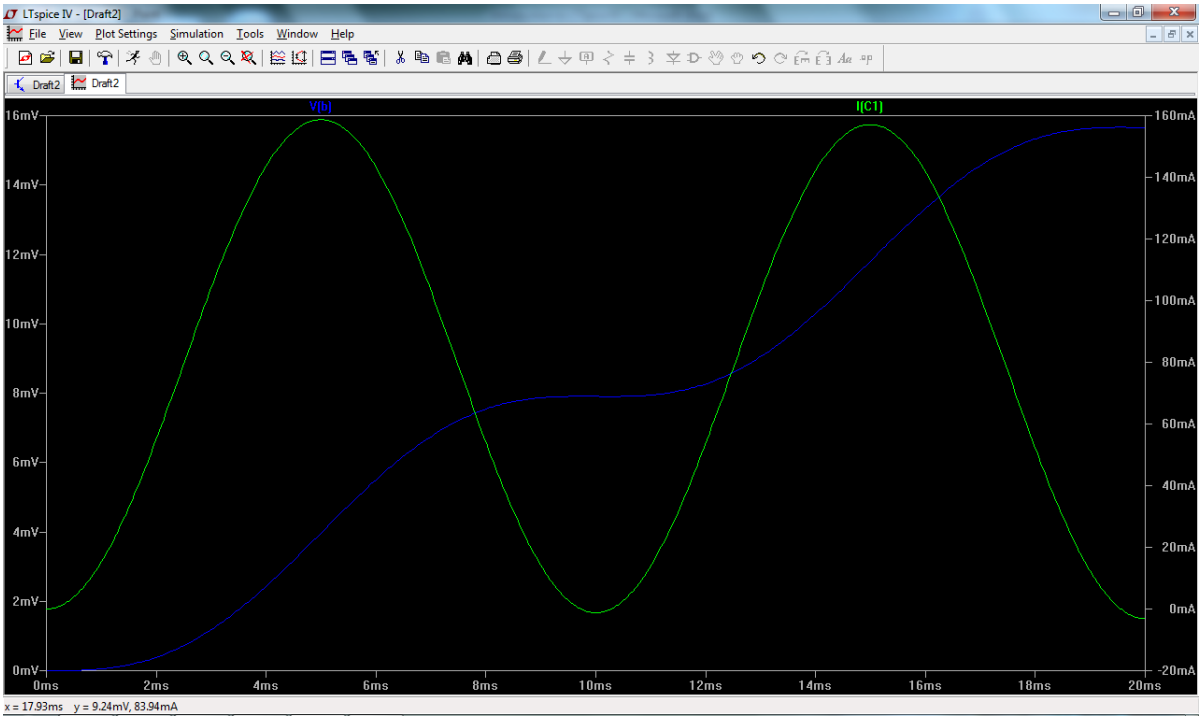


Note that "." denotes the active simulation currently running, and ";" denotes other simulation commands you have saved. You can switch between them.

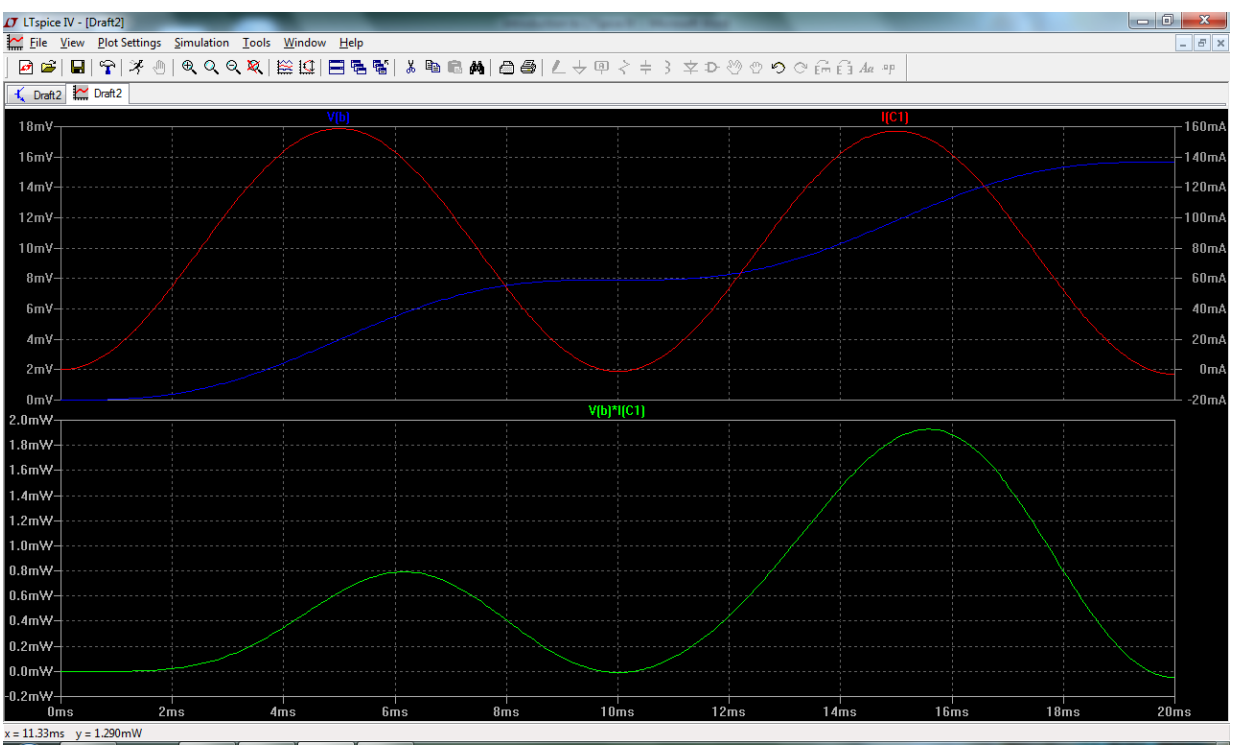
4. In the graph window, you can right click and select Add Traces to view the graphs of interest.



The voltage and current values of the drawn circuit can be graphically presented as:



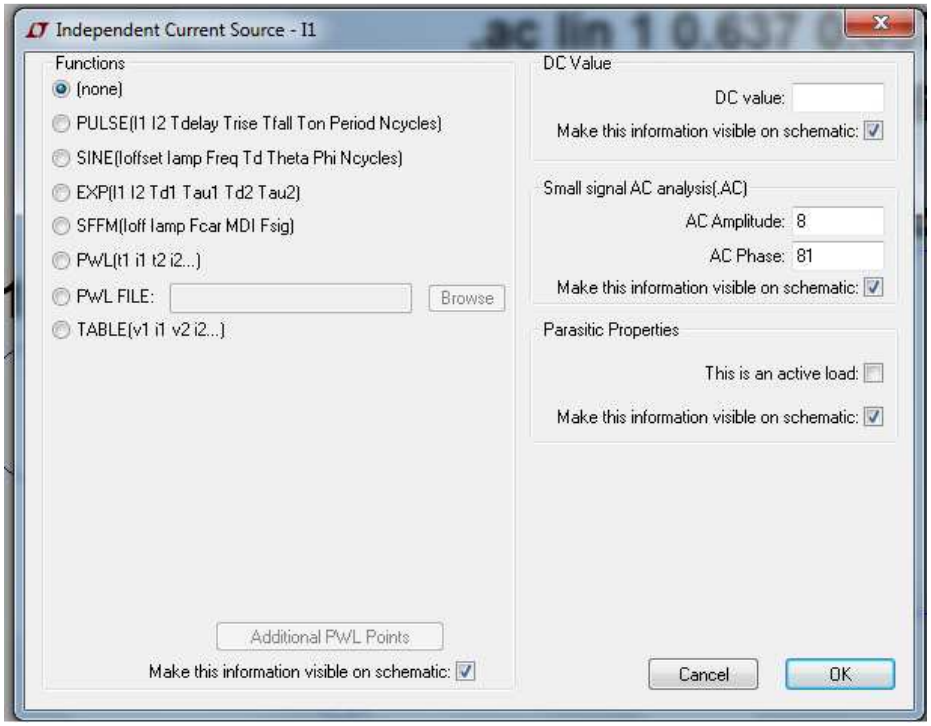
5. To determine power, you can multiply the voltage across a component by the current through that component. For example, in the sample circuit, the power absorbed by the capacitor is given by V(b)*I(C1). While adding trace, you can directly multiply these two quantities. Similarly you can find out differential voltages and currents.



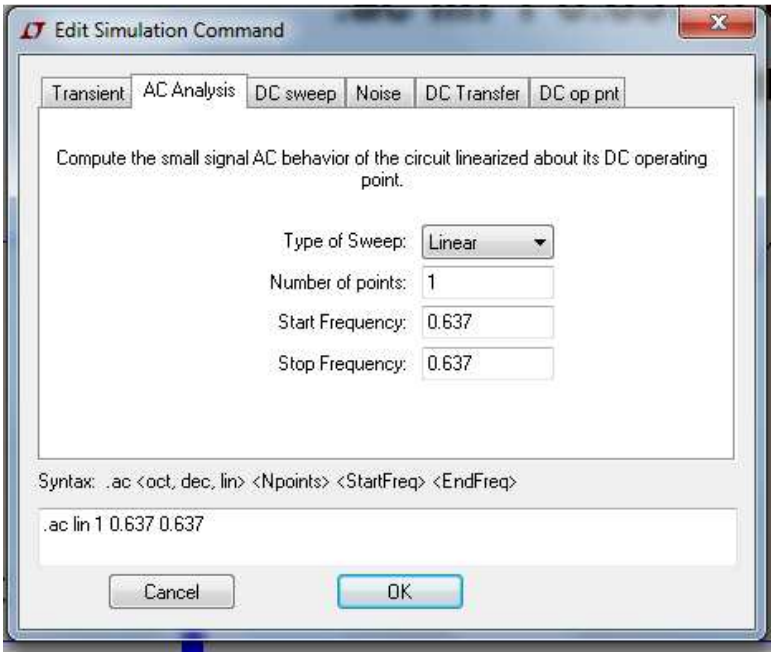
6. You can add multiple plot panes, or turn on the grid for your convenience to interpret the results.

AC Analysis

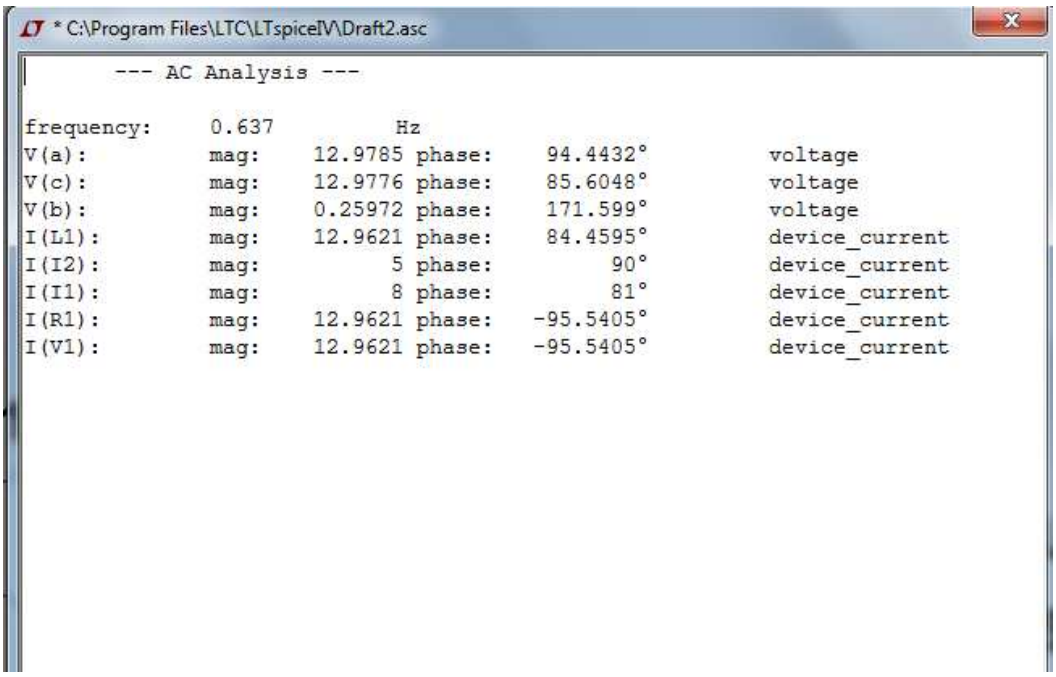
1. To perform the AC analysis, draw the desired circuit. Instead of using the sine wave as your sources, use the small signal AC analysis to input the source amplitude and phase.



2. Click on the AC analysis tab in the simulation command window. Select the type of sweep to be linear. Since at this moment, we are interested in only one frequency, input number of points to be 1 and as select the desired frequency for both start and stop frequency.



3. Click to run the simulation, a new window will open with the AC analysis summary.



```
* C:\Program Files\LTC\LTspiceIV\Draft2.asc

       --- AC Analysis ---

frequency:     0.637          Hz
V(a):          mag:    12.9785 phase:    94.4432°         voltage
V(c):          mag:    12.9776 phase:    85.6048°         voltage
V(b):          mag:    0.25972 phase:    171.599°         voltage
I(L1):         mag:    12.9621 phase:    84.4595°         device_current
I(I2):         mag:          5 phase:         90°         device_current
I(I1):         mag:          8 phase:         81°         device_current
I(R1):         mag:    12.9621 phase:   -95.5405°         device_current
I(V1):         mag:    12.9621 phase:   -95.5405°         device_current
```