Creating a new Pspice project
1) Open capture lite
2) Select file ⇒ new ⇒ project
3) In the New Project window, enter project name: example1
4) Under the heading Create a new project using, select analog or mixed A/D
5) Under the Location heading, browse to the OrCAD directory, if it is not already indicated
6) Clicking OK in the New Project window will bring up the Create Pspice Project window
7) In Create Pspice Project window, select Create a blank project
8) Clicking OK in the Create Pspice Project window will bring a blank schematic page

Placing a voltage source
9) Select place ⇒ part, then select the source library and under part list, select vdc
10) Left click to place part
11) Right click and select end mode
12) To change source voltage, double click 0V to display properties, enter 12Vdc

Placing resistors
13) Place part – analog – select R to place resistor
14) Left click to place- then right click, end mode
15) Edit resistor values. 1k, 2k, 3k

Connecting wires
16) Select place ⇒ wire or click on wire button
17) Click to start wire - draw first corner - click for second corner – right click to end wire
18) To get out of the line drawing mode, click the select button

Grounding the circuit
19) Click ground button ⇒ select add library ⇒ select pspice folder ⇒ select source
20) In the place ground window select 0, then click OK to place ground.
21) Move ground to circuit ⇒ right click to end mode
22) Connect a wire from the ground to desired location in the circuit

Running a simulation of DC analysis
23) Select Pspice ⇒ New simulation profile
24) Enter a simulation name: dcanalysis
25) Selecting Create brings up simulation settings
26) Choose bias point for the analysis type, then click OK
27) To run the simulation, click on the run simulation button
28) To view voltages, currents, and powers, use the tool bar.

Warnings:
1. After running a simulation, do not delete displayed voltages, currents or powers. If you do, you will have to re-construct the entire circuit to display the values again. You can safely turn off the display of electrical quantities by using the tool bar.
2. No spaces are allowed in component names.