University of Technology, Sydney

48551 ADVANCED ELECTRONICS

Faculty of Engineering

OrCAD 10.0 Demo: Computer-Aided Circuit Analysis
1. Basic DC Analysis

Preliminaries

1. If you have not installed the Orcad 10.01 Demo on your computer, you need to install the software first. This software is recorded on a CD available for loan from LDC1. Installation should start automatically after the CD is inserted. In case it does not, run `setup.exe` from the CD. Install Capture CIS Demo (other programs are not needed in this subject). You need Windows 2000/NT (with Service Pack 4 or later) on your computer to run this software.

2. Once the software is installed, you can start it by clicking on **Capture CIS Demo** located in the **Orcad 10.0 Demo** program group. Next use the **Help** -> **Learning Capture CIS** command to bring up the Capture tutorial. Work your way through the lessons and become familiar with the various features of Capture. A comprehensive set of manuals is also included in the directory `...\doc\pspug\`. You'll find the PSpice User Guide (`pspug.pdf`) most useful.

DC Analysis of resistive circuit

1. Before you can draw a diagram of a circuit to be analysed, you must create a PSpice project. First, create a folder `c:\OrcadPSpiceProjects` on your hard drive to hold your circuit files (you may also create a subdirectory for each circuit). Then, start Capture and use the **File** -> **New** -> **Project** command to bring up the window shown in Fig. 1.
   Type the project name: DC_Test01. Then, check the **Analog or Mixed A/D** option, type the name

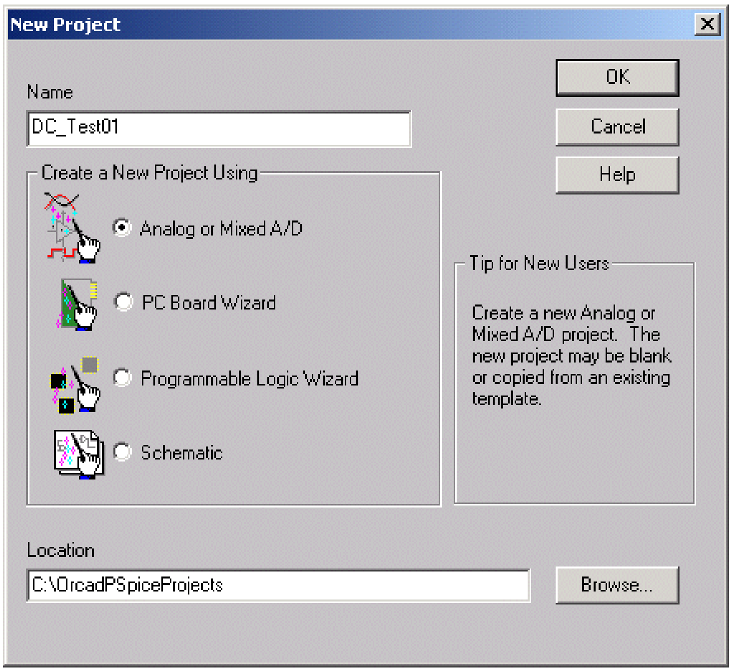

Fig. 1.

of your circuit folder, and left click on **OK**. This brings up the window shown in Fig. 2. Left click on **Create a blank project** and then on **OK**.

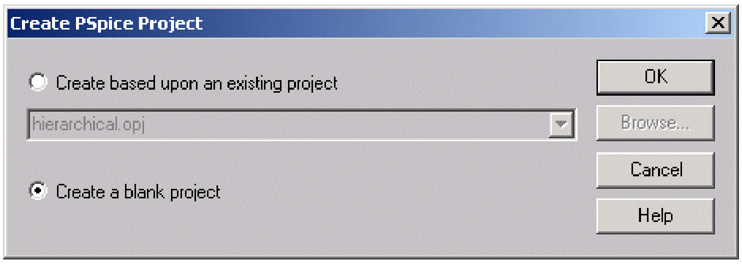

Fig. 2.

This produces the screen shown in Fig. 3, which contains several windows. We will draw the circuit in the window titled SCHEMATIC1: PAGE1.

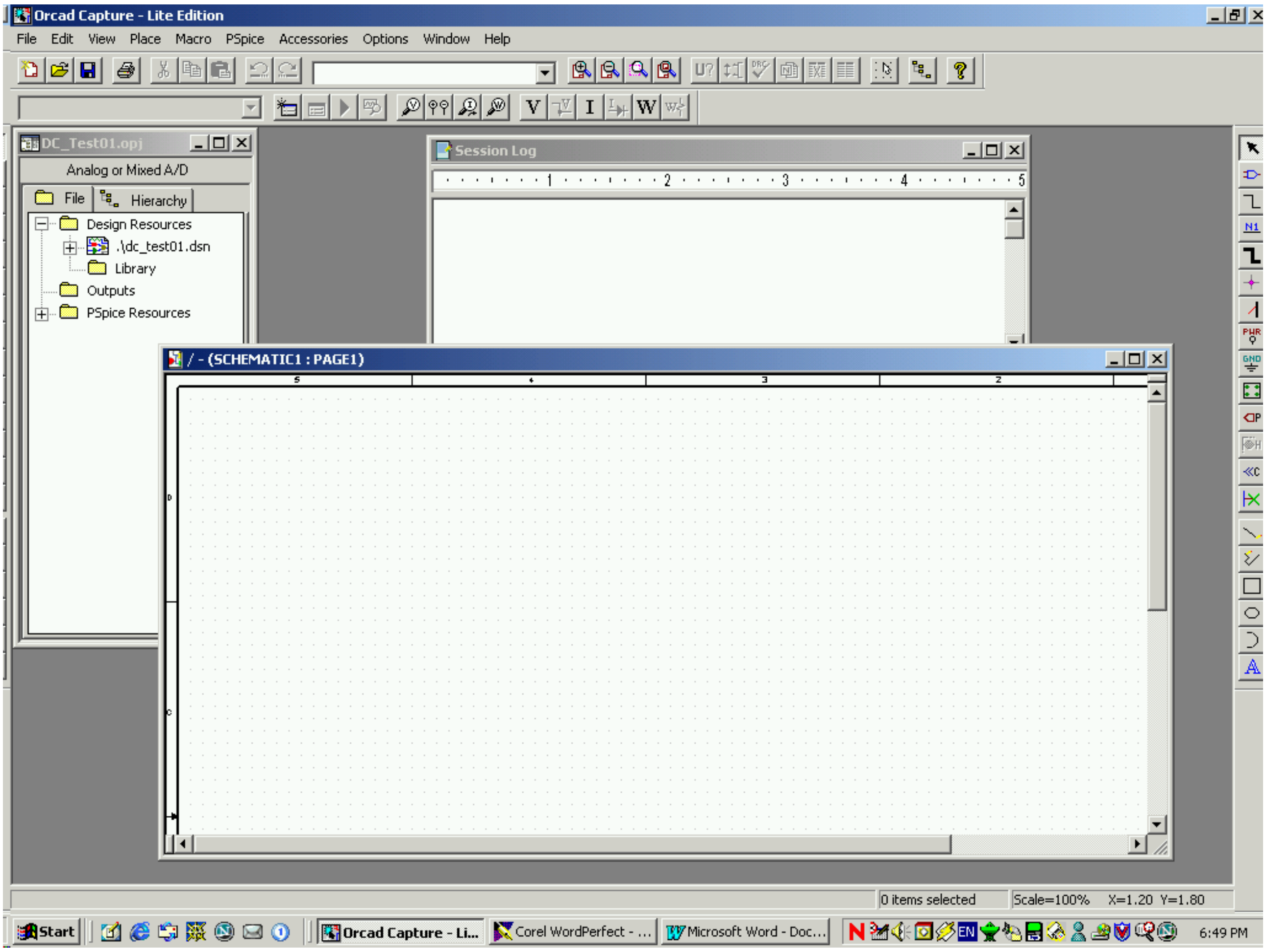

Fig. 3.

2. Suppose we want to find voltages, currents and powers dissipated in circuit elements for the resistive circuit in Fig. 4. To draw the diagram of this circuit you must place all parts, connect them by wires, assign values to components and designate one circuit node as the reference node.

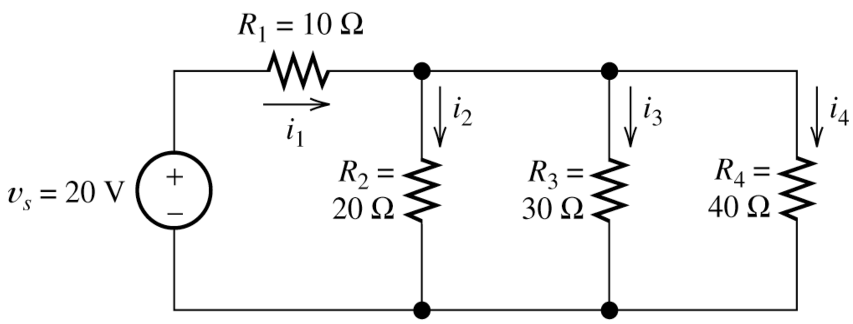

Fig. 4.

3. To draw the parts we use the **Place** -> **Part** command to bring up the window shown in Fig. 5. From the Part List: box choose a resistor (R/ANALOG). If you cannot find a resistor on the part list, click on the **Add Library...** command button and choose `analog.olb` file from the Libraries: window. (The libraries are normally installed in the folder: `...\OrCAD_10.0_Demo\tools\capture\library\pspice` .) At this point, you can move the cursor around on the *SCHEMATIC* window and place a resistor each time you click the left mouse button. The orientation of the resistor symbol can be rotated by simultaneously pressing the "Ctrl" and "r" keys <Ctrl+r>. Place four resistors to resemble the pattern in Fig. 3. Then, click the right mouse button and select **End Mode** from the pop-up menu (or press the "Esc" key). Next, place the voltage source (VDC) by again using the **Place** -> **Part** command. Your *SCHEMATIC* window should look like the one shown in Fig. 6.

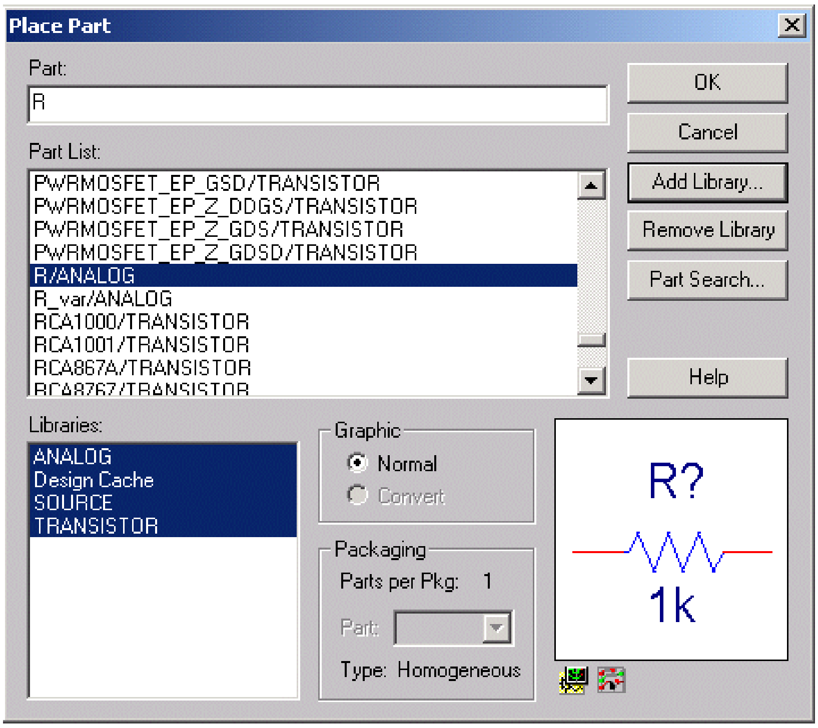

Fig. 5.

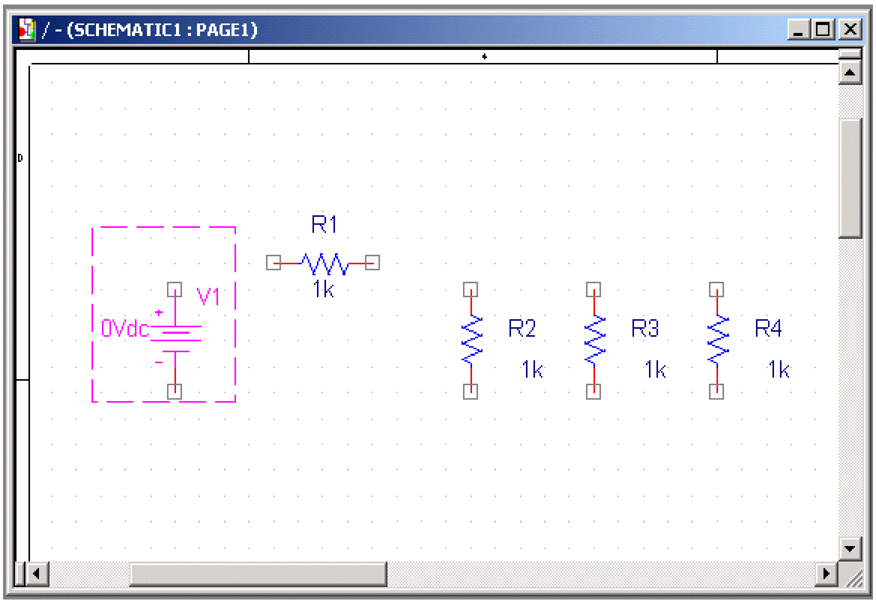

Fig. 6. Parts have been placed.

All of this might take a few tries. The steps (and some helpful tips) are:

a) Use the **Place -> Part** command <Shift+P> to bring up the window from which parts can be selected. If you know (or guess) the part's name (e.g., R for resistors, VDC for batteries, C for capacitors, etc.) you can simply type this name in the Part: box.
b) After selecting the desired part, click on OK. If necessary, press <Ctrl+r> to rotate the part to the desired orientation. Then, move the cursor to the desired location and left click to place the part. After all the parts of a given type are placed, click the right mouse button and select **End Mode** from the pop-up menu (pressing the "Esc" key does the same). If you want to place a different part, simply press <Shift+P> (no need to **End Mode**).
c) If you accidentally get an unwanted part, you can eliminate it after you leave the place-part mode. Select the unneeded part by placing the cursor on it and clicking the left mouse button. Then, press the "Del" key.
d) If a part has a wrong name (such as R6 instead of R1), change it after you leave the place-part mode. Double left click on the name and type in the correct name. (However, no two parts may have the same name when we do the PSpice analysis.)
e) A part can also be rotated after being placed. Select it and press <Ctrl+r>.

Wiring the circuit

Next, we use the **Place -> Wire** command <Shift+W> to change the cursor into a wiring tool. Move the cursor to the top end of the voltage source, click the left mouse button , move the cursor to the left end of R_1, and click to wire the voltage source to R_1. Continue wiring components until the circuit appears as shown in Fig. 7. Then, click the right mouse button and select **End Wire** from the pop-up menu (or just press "Esc" key). If you end up with unwanted wire segments, you can eliminate them after exiting the wiring mode. Click on the wire segment and press the "Del" key.

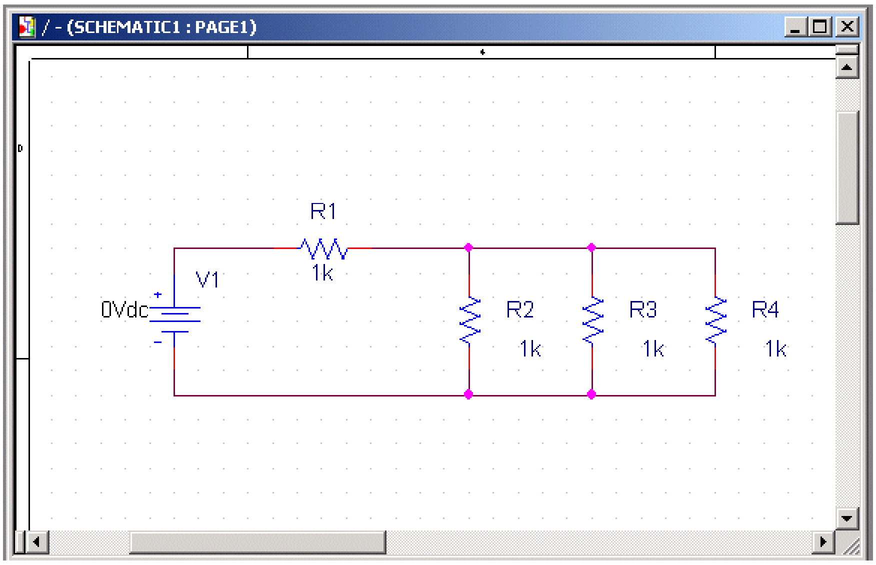

Fig. 7. Parts are wired.

Adding the ground node

All circuits analysed by PSpice must have a node designated as a ground or ***reference node***. The zero potential is arbitrarily assigned to this node and all other node potentials are calculated with reference to it. Although there are several ground symbols in PSpice, only the symbol `0` from the `Source` library forces the potential of the node to which it is connected to zero. The steps in adding the reference node are:

1. Left click on the ground tool on the right-hand side of the screen (se Fig. 8). This brings the *Place Ground* window.
2. Select the ground named `0/Source` and place it at the bottom of the circuit, as shown in Fig. 8. After placing the ground symbol, right click and select **End Mode** from the pop-up menu (or press the "Esc" key).
3. Use the **Place** -> **Wire** command <Shift+W> to change the cursor into a wiring tool and wire the ground to the circuit as shown in Fig. 8.

Changing component values

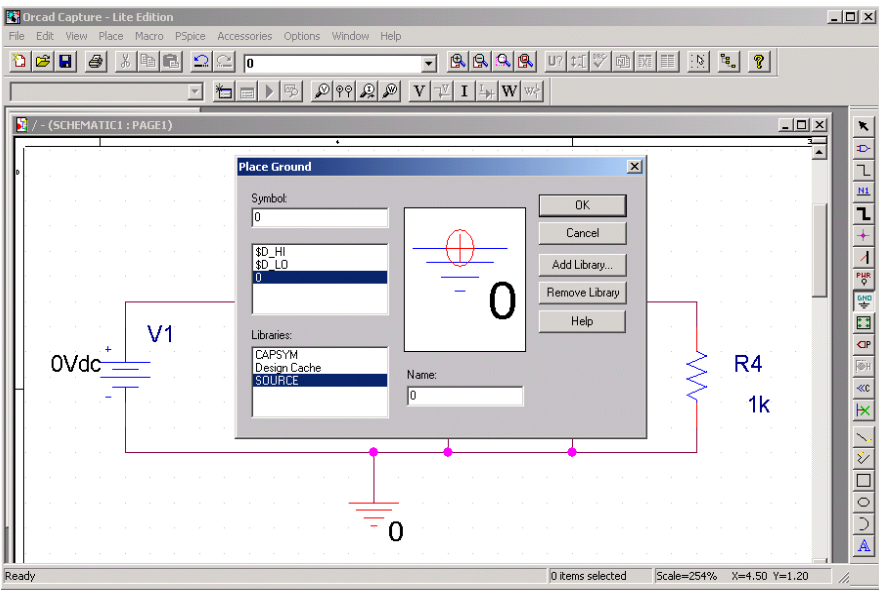

Fig. 8. Placing the ground.

Next, we change the default component values to those shown in Fig. 4. Double click on `0Vdc` next to the voltage source to bring up the *Display Properties* window, shown in Fig. 9. Enter `20V` as the new value and then click on **OK** to close the window.

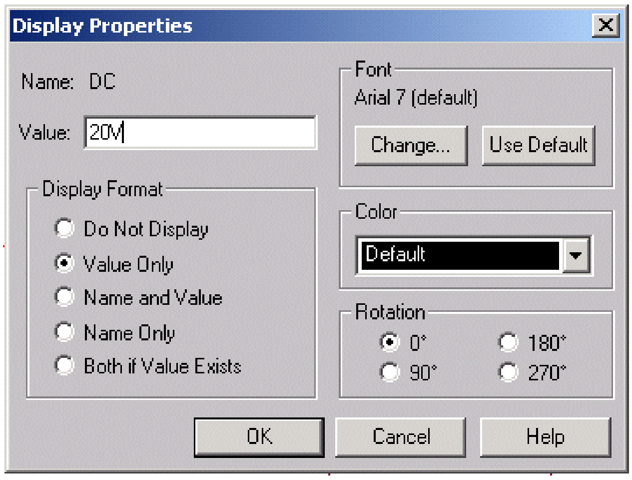

Fig. 9.

In the similar fashion, change the values of the resistances so that they match those in Fig. 4. You can also change the names of the elements. For example, `V1` may be changed to `Vs`. Component labels and values can be repositioned on the diagram, to improve appearance; it is done by selecting a label or value and dragging it into a desired position. Then, the circuit should appear as shown in Fig. 10.

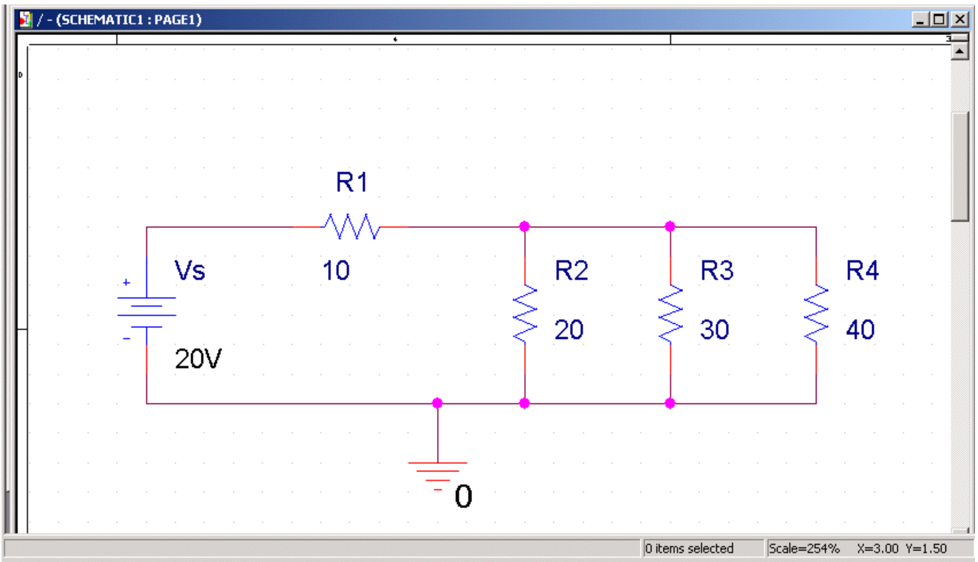

Fig. 10. Completed circuit.

Setting up a new simulation profile

Simulation profiles tell PSpice what type of analysis is desired. Use the **PSpice -> New Simulation Profile** command to bring up the window shown in Fig. 11. Type in `DC Solution` as the name of the simulation profile, make sure that the *Inherit From* window contains the word `none`, and left click on **Create**. This brings up the window shown in Fig. 12. Use the pull-down menu and select `Bias Point` as the analysis type. Then, click **OK**. The simulation profile we have created instruct PSpice to compute the voltages at the circuit nodes (with respect to the reference node - ground), the current for each element and the power dissipated in each element.

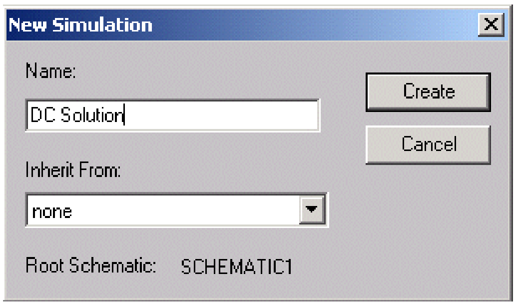

Fig. 11.

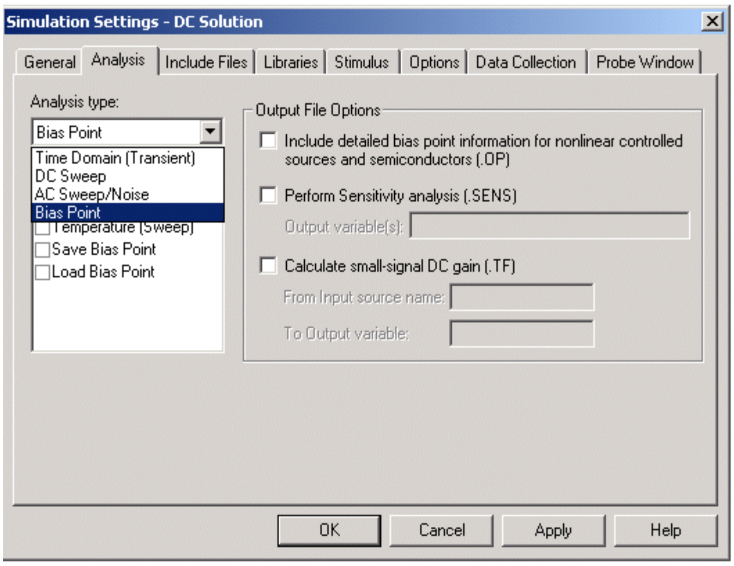

Fig. 12.

Running the PSpice simulation

After creating the new simulation profile, we are ready to run the simulation. It is done by using the **PSpice -> Run** command. After successful simulation, close the *SCHEMATIC - DC Solution* window that appears. Next, click on the V, I and W icons (visible in Fig. 13) to toggle the display of voltages, currents and powers on and off. If required, these labels can be repositioned or deleted to achieve better picture clarity. Note that in Fig. 13 power displayed next to the voltage source is negative. It means that the source **delivers** power.

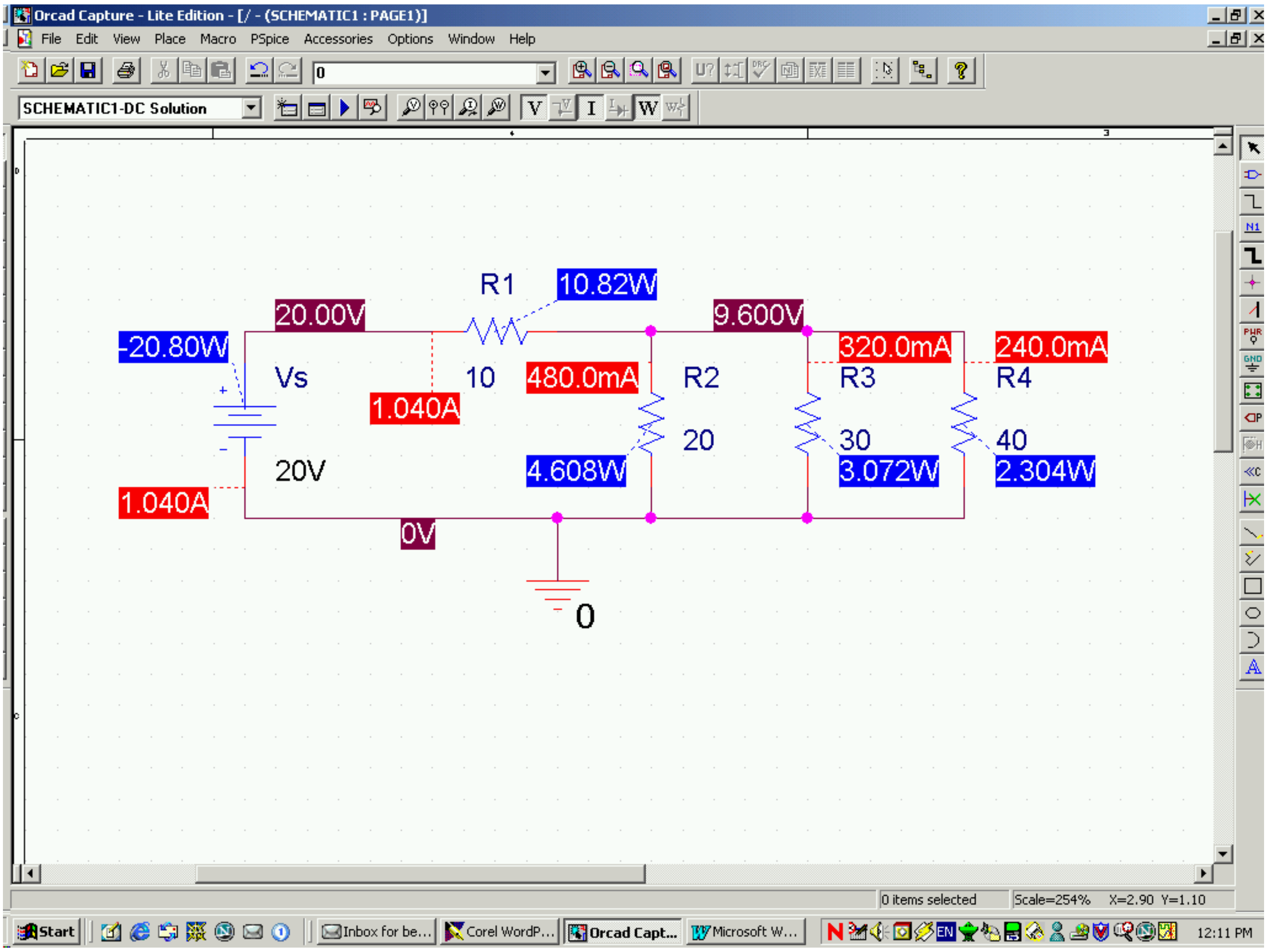

Fig. 13.

Controlled (dependent) sources

PSpice can handle circuits containing all four types of controlled (dependent) sources. These are:

Voltage-Controlled Voltage Source (VCVS) - PSpice name: E
Current-Controlled Current Source (CCCS) - PSpice name: F
Voltage-Controlled Current Source (VCCS) - PSpice name: G
Current-Controlled Voltage Source (CCVS) - PSpice name: H

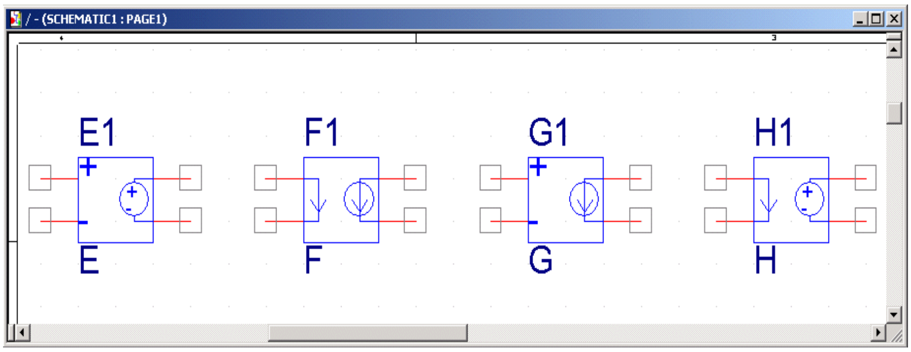

Fig. 14. Controlled sources.

The symbols (in their default orientation) are shown in Fig. 14. As the symbols are oriented in Fig. 14, the controlling variable should be wired to the terminals on the left-hand side, and the source terminals are on the right-hand side. Of course, we can also rotate and flip (mirror) each symbol to simplify the diagram.

As an example, lets solve for i_y the circuit shown in Fig. 15.

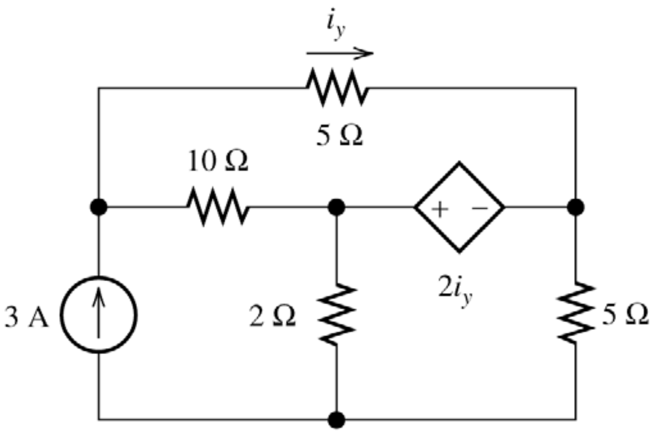

Fig. 15. Example circuit with a CCVS.

To simplify the diagram, the current-controlled voltage source must be rotated and then flipped horizontally (use <Ctrl+r> three times and then **Edit** -> **Mirror** -> **Horizontal** command). After placing the resistors and the (independent) current source (its PSpice part name is IDC), wiring, grounding and assigning the values, your circuit should look like the one shown in Fig. 16.

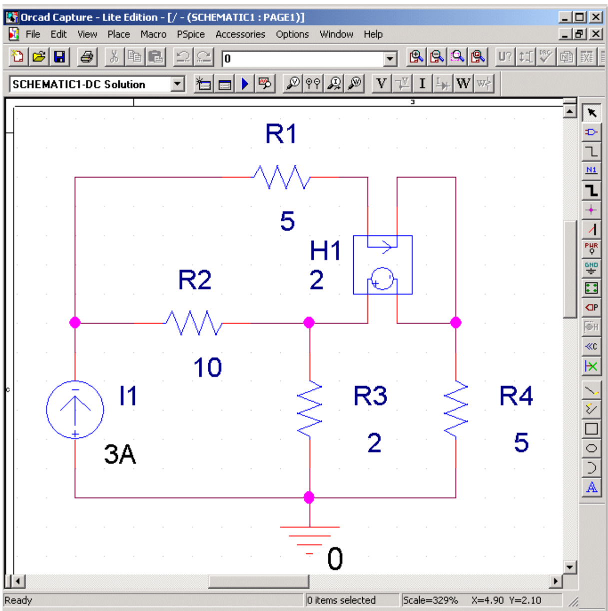

Fig. 16.

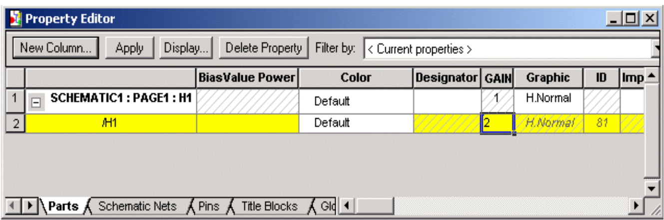

Fig. 17.

Setting the gain parameter (transresistance) of the CCVS cannot be done by changing the value displayed on the diagram (as it is done for resistors and sources). Instead, double click on the controlled-source symbol to bring up the *Property Editor* window, as shown in Fig. 17. Here, enter the gain as 2, click on **Apply** and close the window. Then, set up the simulation profile like in the previous example and run PSpice. The result should be i_y = 2.308 A, as shown in Fig. 18.

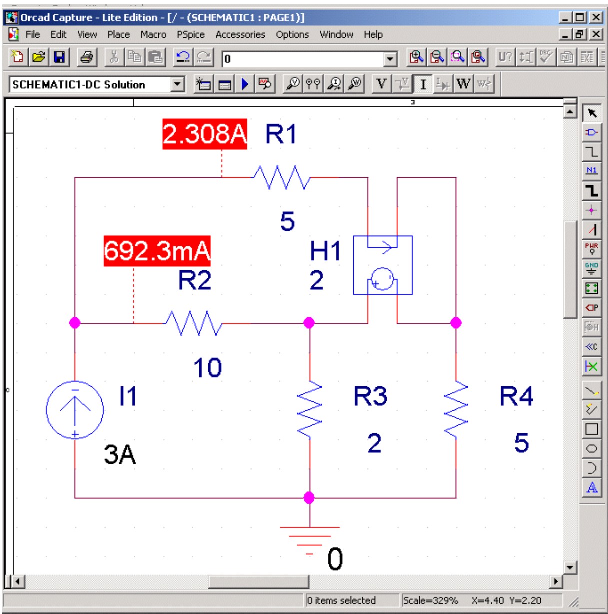

Fig. 18.

Exercise:

Use Capture and PSpice to solve the circuit in Fig. 19 for v_{oc}. Answer: v_{oc} = 4.62 V

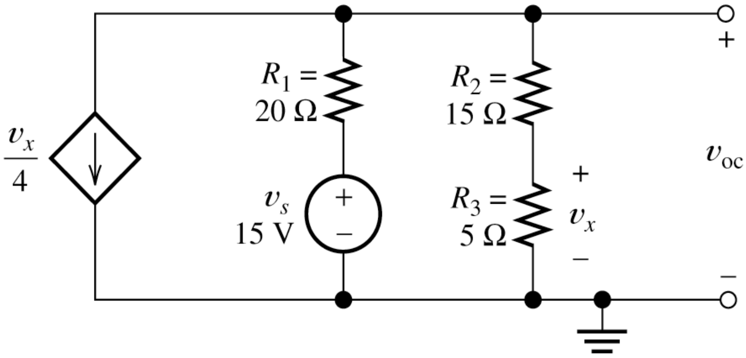

Fig. 19. Circuit with a VCCS.