

How to Start OrCAD

For Labs of ELEC 3509

Carleton University

Summer 2020

(Q.J. Zhang)

Step 1: Log in on a computer in the Department of Electronics (DOE) undergraduate lab.

To do this remotely, please follow the instructions in the DOE website under “Computer Support”. Once you succeeded in remotely connecting to DOE’s Window based computers, a Window of the “DOE’s computer screen” will appear on your screen. Let this Window be called “the virtual screen”.

All the subsequent steps in this document will be performed within this “virtual screen”, (i.e., the screen of the DOE’s computer). For example, “Select the Start menu” means “Select the Start menu in the virtual screen”.

Step 2: Select the Start menu, “All Programs”, then “Cadence Release”,

then “OrCAD Products”,

then “Capture CIS” (and wait for a response, be patient while waiting).

A pop-up window will appear with the message “OrCAD Capture license was not found. Would you like to launch Capture using lite license?” Click “Yes”.

A window of the “OrCAD Capture CIS - Lite -” will appear.

Step 3: Now you can create a new circuit using OrCAD, or open an existing circuit project previously saved from OrCAD; and perform simulations of the circuit.

How to Use OrCAD

To Build a Circuit and Simulate It

For Labs of ELEC 3509

Carleton University

Summer 2020

(Q.J. Zhang)

Start the “OrCAD Capture CIS” program.

Select “New Project” – a dialog window will pop-up.

Type a name for the new project.

Select the option of “PSpice Analog or Mixed A/D”.

Select a folder in your “H:” drive for the “Location” field.

Click “OK”.

In the next dialog window, select the option of “Create a blank project”, Click “OK”.

Now you will be in the schematic window.

At the top of this schematic window, you will see various menus items: “File”, “Edit”, “**Place**”, “**PSpice**”, etc.

Draw the schematic of a new circuit:


To get a resistor (or capacitor, inductor, ground) and place it into your circuit: Select “Place” menu, then “PSpice Component”, then “Resistor” (or Capacitor, Inductor, PSpice-Ground). Left-click on “Resistor”, then:

Move the cursor into the schematic window. Left click, and the resistor will be placed into your circuit.

Right click to end this mode of operation.

There are 3 parts of this resistor schematic:

a label, e.g., “R1”

a symbol, e.g., 

a value, e.g., “1k”

Double click the label if you want to rename the resistor.

Double click the value if you want to change the value of the resistance.

Single click the symbol to select it, then right-click it, then select “Rotate” – if you want to rotate the resistor.

To get a voltage source (or current source, or controlled source): Select “Place” menu, then “PSpice Component”, then “Source”.

To get a wire to connect the components together: Select “Place” menu, then “Wire”.

Define Simulation Profile:

Select the “PSpice” menu, then “New Simulation Profile”. Supply a name for the New Simulation (with “Inherent From” option being “none”). A window (called “Simulation Settings” window) will appear (see ** at bottom of this page).

In this “Simulation Settings” window:

Select the Analysis type to be “DC Sweep” or “AC Sweep”.

Define the Label for the sweep variable.

Define the Start value, end value and increment for the sweep.

To edit an existing Simulation Profile: select the “PSpice” menu, then “Edit Simulation Profile”. A window (called “Simulation Settings” window) will appear (see ** at bottom of this page).

Perform Simulation, and view results:

Select the “PSpice” menu, then “Run”. The circuit is now been simulated.

A new window will appear for graphical display of solutions (see ** below).

In this graphical display window, click “Trace” menu, then “Add Trace”. Select an output, e.g., voltage at a terminal of R1 or current through R2, to be plotted graphically as a trace.

You will then see the graphical display showing the curves of selected voltages or currents.

Save the project.

** If the expected window does not appear, then look carefully: it may be behind the main schematic window; Sometimes it may appear as an icon (a minimized window) in the taskbar at the bottom of the screen (the virtual screen when you run the program remotely).

Further Exercise: Place a BJT (e.g., Q2N3904) into your circuit:

Firstly, you need to load the BJT library:

From the schematic window, select the “Place” menu, then “Part”.

On the right-hand-side of the schematic window, a “Place Part” section will appear. Under the “Libraries” subsection, find the “Add Library” icon (one of 3 small icons). Click the “Add Library” icon, and a list of libraries will be shown. Scroll down the list to “bipolar”. Add the “bipolar” library.

Remark: If you cannot find “bipolar” while scrolling down the list, make sure you are in the correct folder: you should scroll in the “library\pspice\” folder, i.e.,

C:\Cadence\SPB_172\tools\capture\library\pspice

Now the Bipolar library is loaded.

In the Bipolar library, you can search your “Part”, e.g., the part being Q2N3904 (which is NPN transistor) or Q2N3906 (which is PNP transistor). Double click the “Q2N3904” in the Part List, then move the cursor into the schematic window to place the transistor into the circuit schematic.

In this way, you can create a circuit including BJT, resistors, capacitors, sources, ground etc.

Further Hints: There are various short-cut icons for your convenience, such as placing parts, placing wire, placing grounds, edit simulation profile, run simulation, etc. Feel free to explore them optionally.

For more instructions on how to use OrCAD/PSPICE, see

http://www.doe.carleton.ca/~nagui/spice/PSpicePrimer_17.pdf

OrCAD Simulation:

Illustrative Examples/Exercises

Preparation for Labs of ELEC 3509

Carleton University

Summer 2020

(Q.J. Zhang)

Example 1: AC Simulation of a RC Circuit

Create the circuit schematic as shown in Fig. 1. Notice that the source is an AC voltage source. The value of the capacitor is $5\mu\text{F}$.

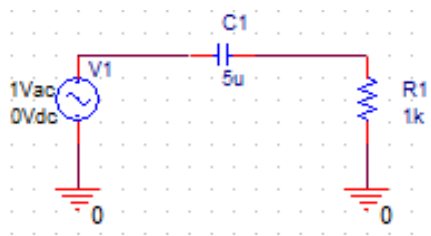


Fig. 1. The schematics of an RC circuit

Define “New Simulation Profile” as shown in Fig. 2. Notice that the “AC Sweep” is selected as the Analysis type. The sweep variable is frequency. The logarithmic scale is used for frequency (as in Bode plot). The frequency range is set to be from 1Hz to 10KHz. We use 20 frequency points per decade.

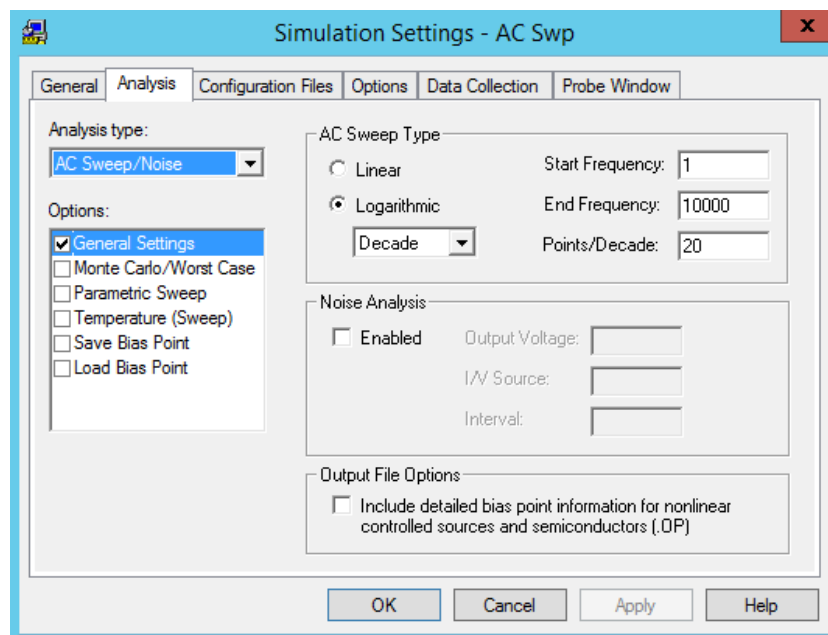


Fig. 2. The Simulation Profile for the RC circuit.

Perform simulation and display results as shown in Fig. 3. Notice that we have selected trace V(R1:2) for the plot. It means the voltage at terminal 2 of resistor R1. In general, each resistor has 2 terminals (terminals 1 and 2) in a circuit. Because our source voltage is 1V, the plot for the V(R1:2) is equivalent to the plot of the transfer function (or gain) of the circuit.

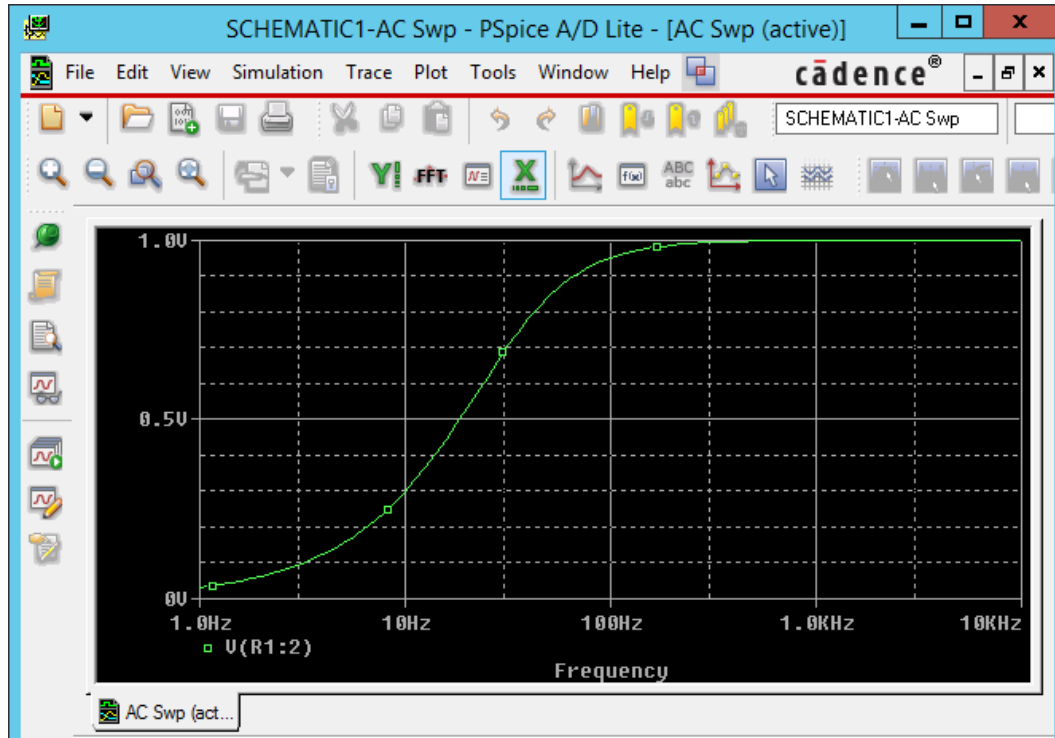


Fig. 3. Graphical display of simulation results for the RC circuit. Notice that the vertical axis is V(R1:2) which means the voltage at terminal 2 of resistor R1. An alternative label for the same voltage is V2(R1) when adding traces to the graph.

Example 2: DC Simulation of an NPN BJT

Load the bipolar library. Fig. 4 shows that the NPN transistor part “Q2N3904” can be found in the BIPOLAR library.

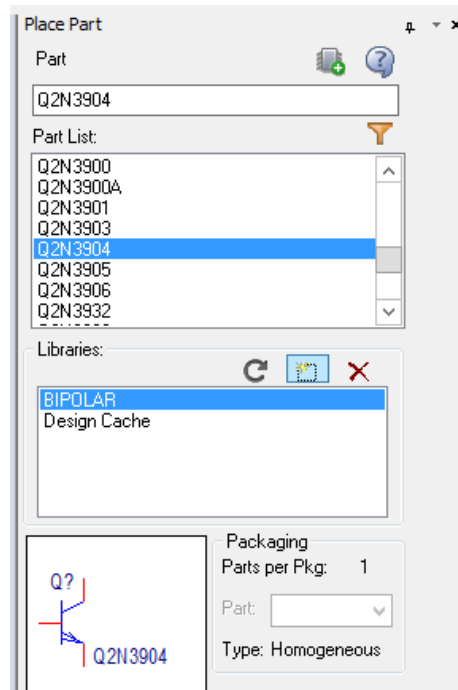


Fig. 4. Load the bipolar library. The BJT part “Q2N3904” can be seen in the “Part List” of the BIPOLAR library.

Create the circuit schematic as shown in Fig. 5. Notice that the sources are DC voltage sources. We have renamed the two sources as VB and VC. VB is set to 3V. The value of the resistor R1 is set to 230K Ω .

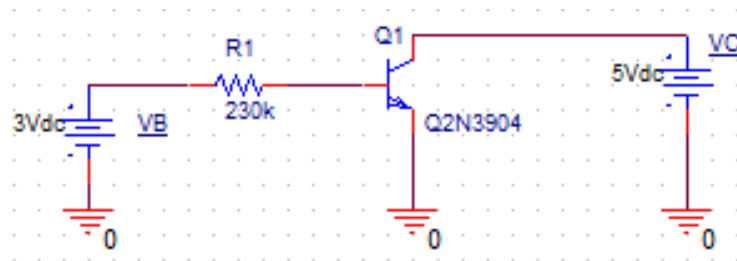


Fig. 5. The schematics of a BJT circuit

Define “New Simulation Profile” as shown in Fig. 6. Notice that we selected DC sweep. We use linear scale for voltage parameter VC. The range for sweeping VC is from 0V to 10V with step size of 0.1V.

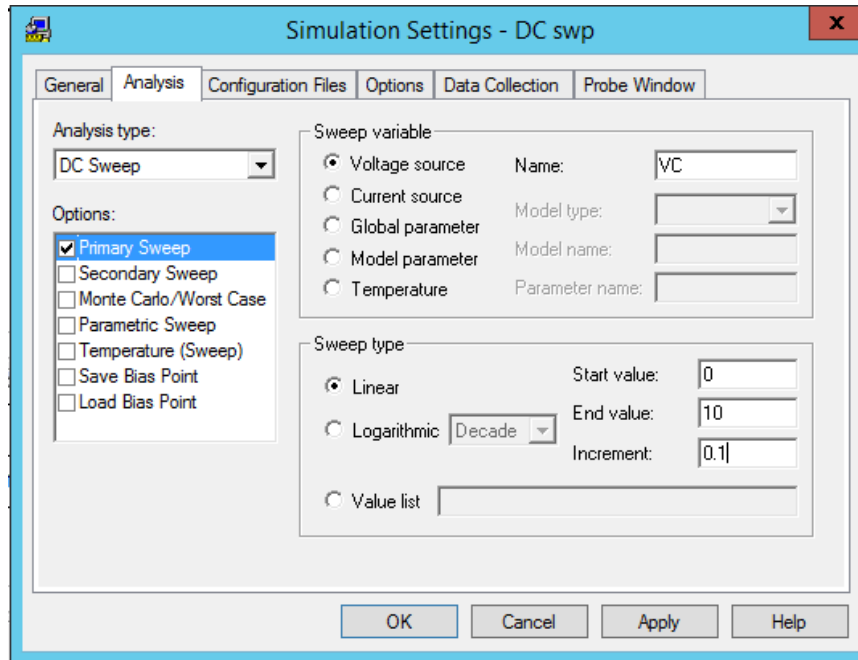


Fig. 6. The Simulation Profile for the BJT circuit.

Perform simulation and display results as shown in Fig. 7. Notice that we have selected trace $I(Q1:c)$ for the plot. It means the Current through terminal c of transistor Q1 (i.e., collector current). In general, each BJT has 3 terminals (terminals b, e, c) in a circuit. This figure shows one curve in the family of IV curves of this transistor.

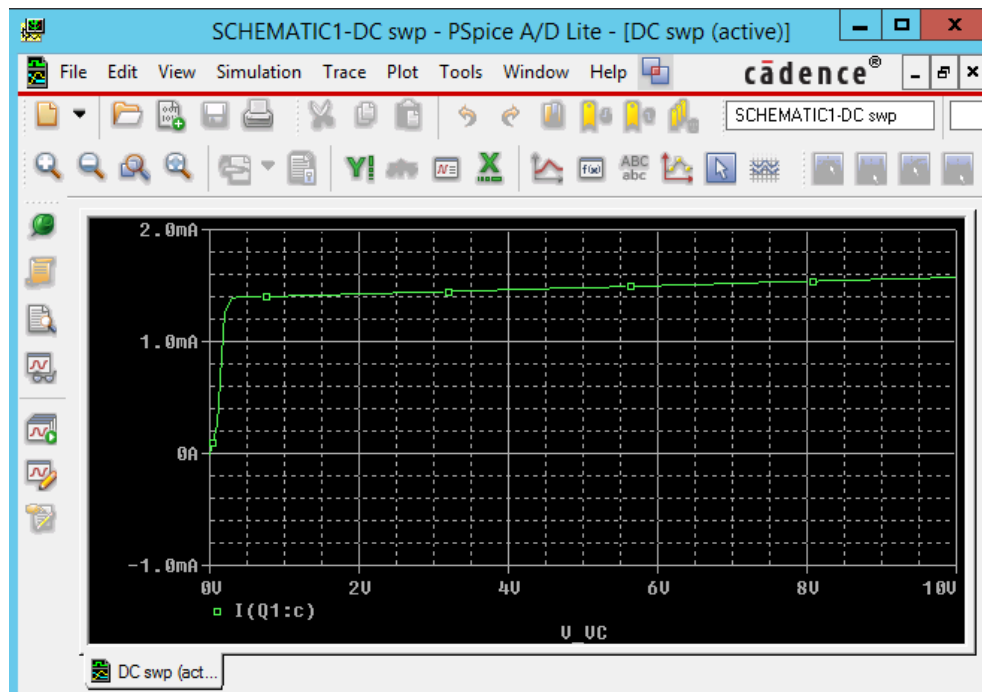


Fig. 7. Graphical display of simulation results for the BJT circuit. Notice that the vertical axis is $I(Q1:c)$ which means the collector current of Q1. An alternative label for the same current is $I_C(Q1)$ when adding traces to the graph.