

ELEC 3509

Electronics II

Professor Q.J. Zhang
Department of Electronics



How to use OrCAD

To Simulate a Circuit

plus

Two Exercise Examples for Practice

Login to DOE's Window Based Computer/Server

Step 1. Use your computer to connect to Carleton's VPN

Step 2. Use "Remote Desktop Connection" on your computer to connect to DOE's Window based server/computer

Instructions for Step 1: <https://carleton.ca/its/help-centre/remote-access/>

Instructions for Step 2:

<http://doe.carleton.ca/>

- > Computer Support

- > How to access DOE's Windows based servers remotely

- > How to Access a Local Desktop Computer using Remote Desktop Client

Start OrCAD

On DOE's Window based server/computer:

Start Menu

- > All Programs

- > Cadence Release

- > OrCAD Products

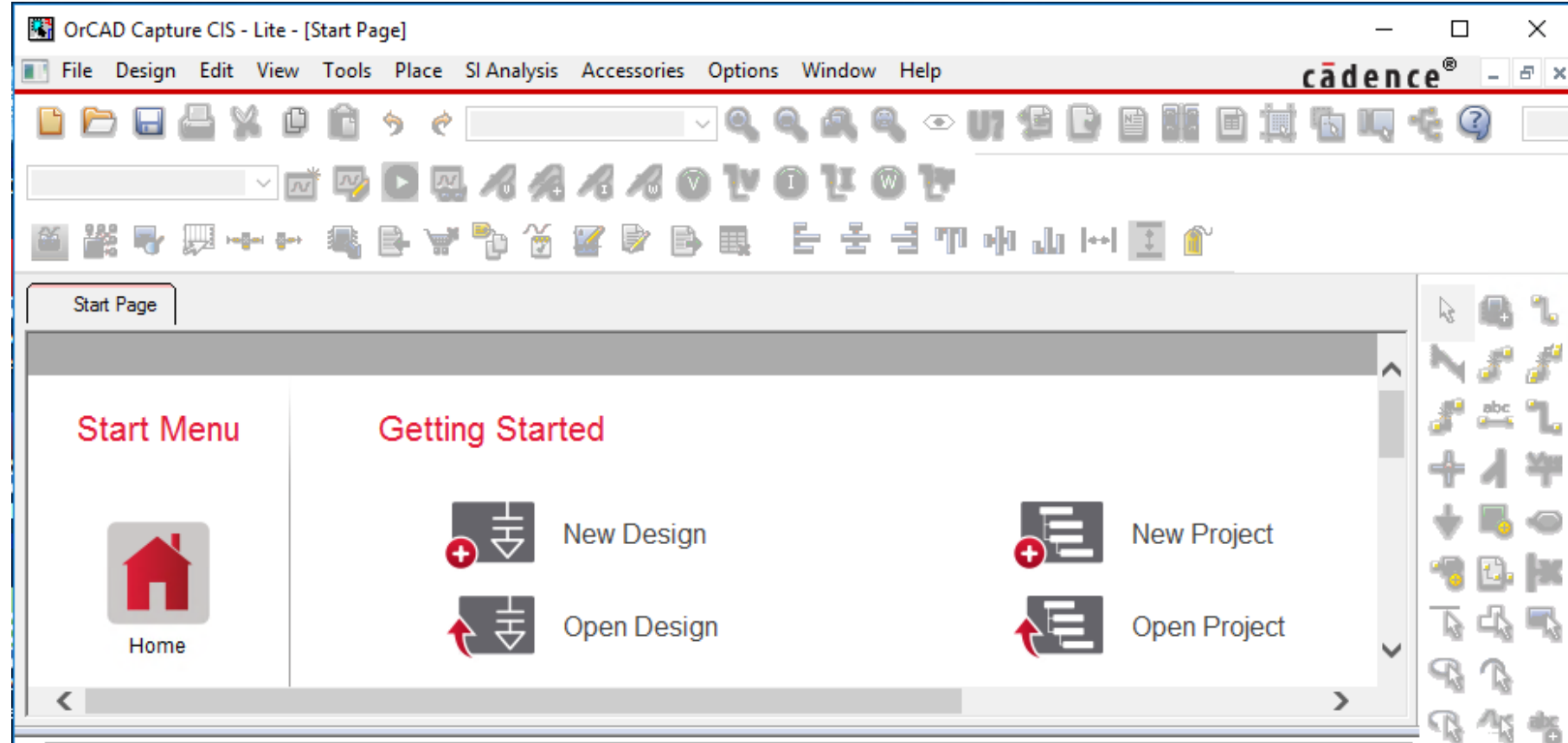
- > Capture CIS

- > Click "yes" as an answer the question:

- "Would you like to launch Capture using lite license?"

now you will have the Start Page of OrCAD Capture CIS - Lite -
(as shown next slide)

Start Page of OrCAD Capture CIS - Lite -



Create a New Project in OrCAD

-> Click “New Project” in the Start Page of OrCAD Capture CIS – Lite –

-> enter an Name, e.g., myExample_1

(Fig. 1)

-> choose “PSpice Analog or Mixed A/D”

(Fig. 1)

-> use “Browse” to select a folder in your H:\ drive

(Fig. 2)

-> In the pop-up window of Fig. 3, choose “Create a blank project”

(Fig. 3)

now you will have the Schematic Window of OrCAD Capture CIS (see next slide)

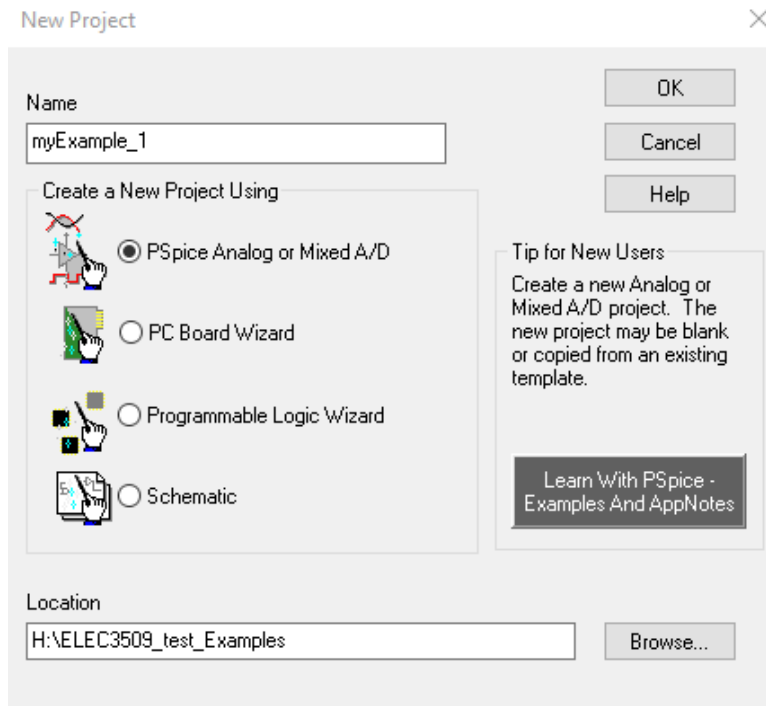


Fig. 1

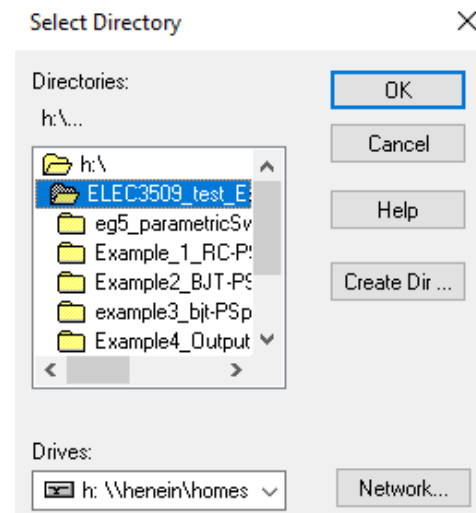


Fig. 2

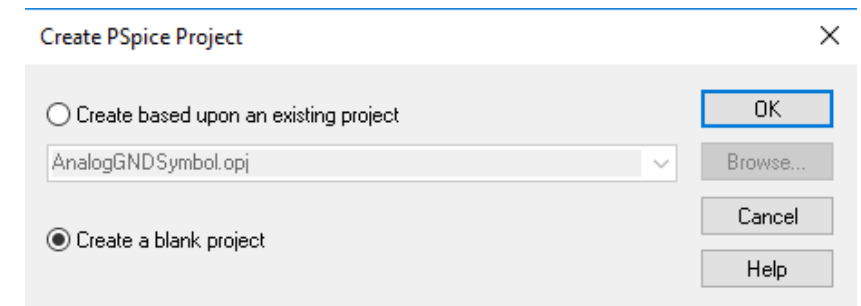
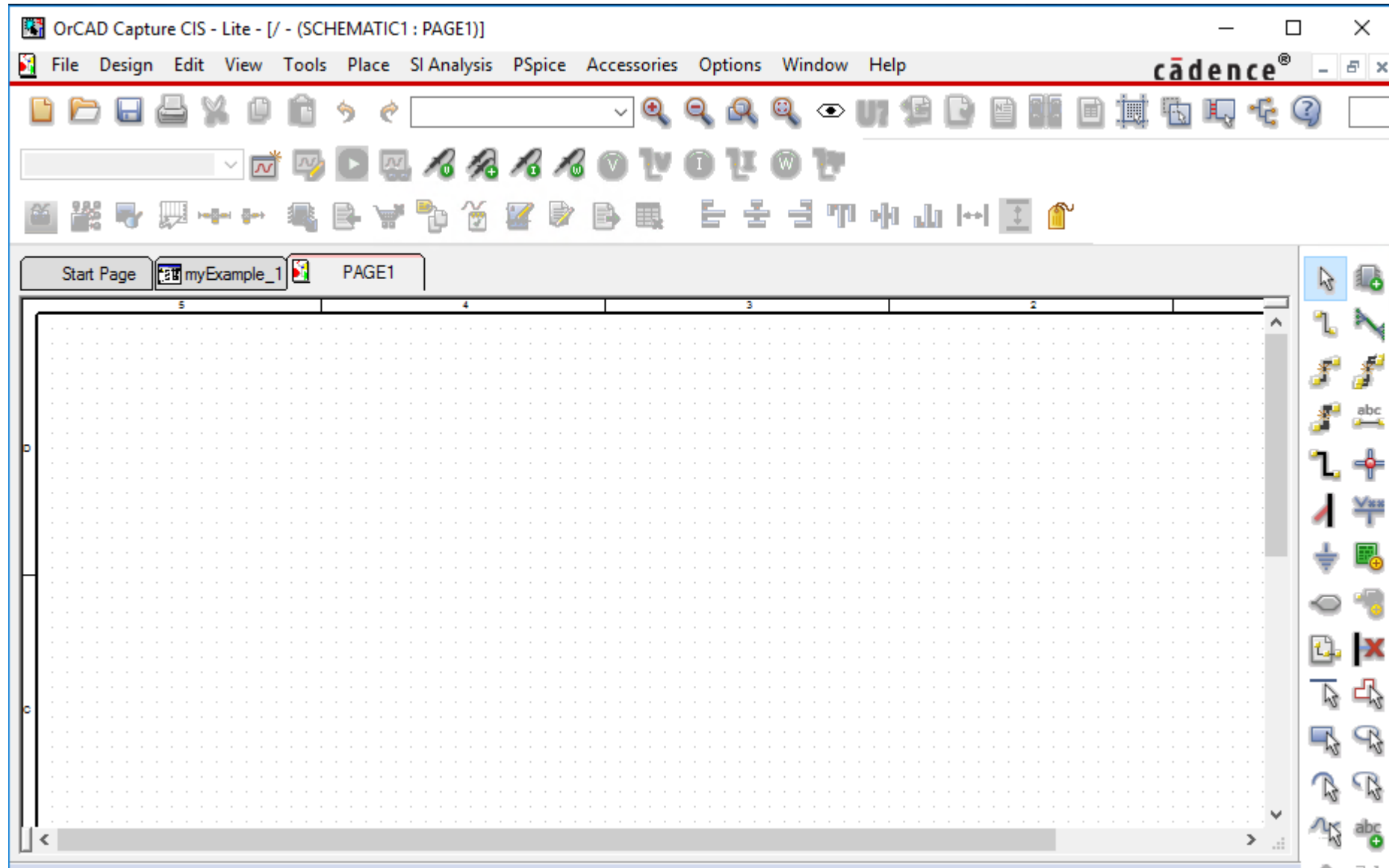


Fig. 3

Schematic Window of OrCAD Capture CIS



Place a Resistor into the Schematic Window

-> “Place” menu

-> “PSpice Component”

-> Resistor

-> Left-Click on “Resistor”

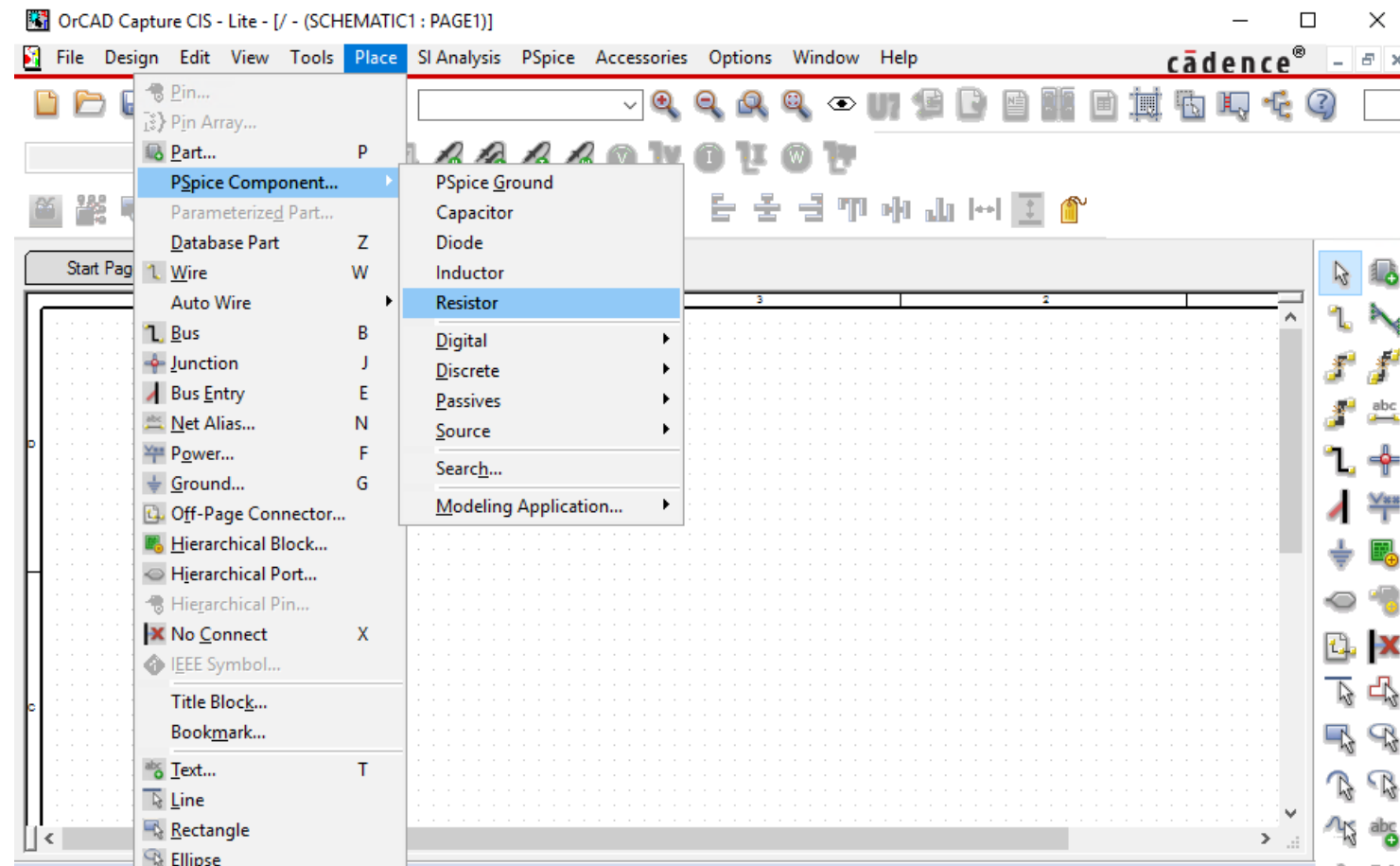
then move cursor into the schematic window

left-click and the resistor is placed

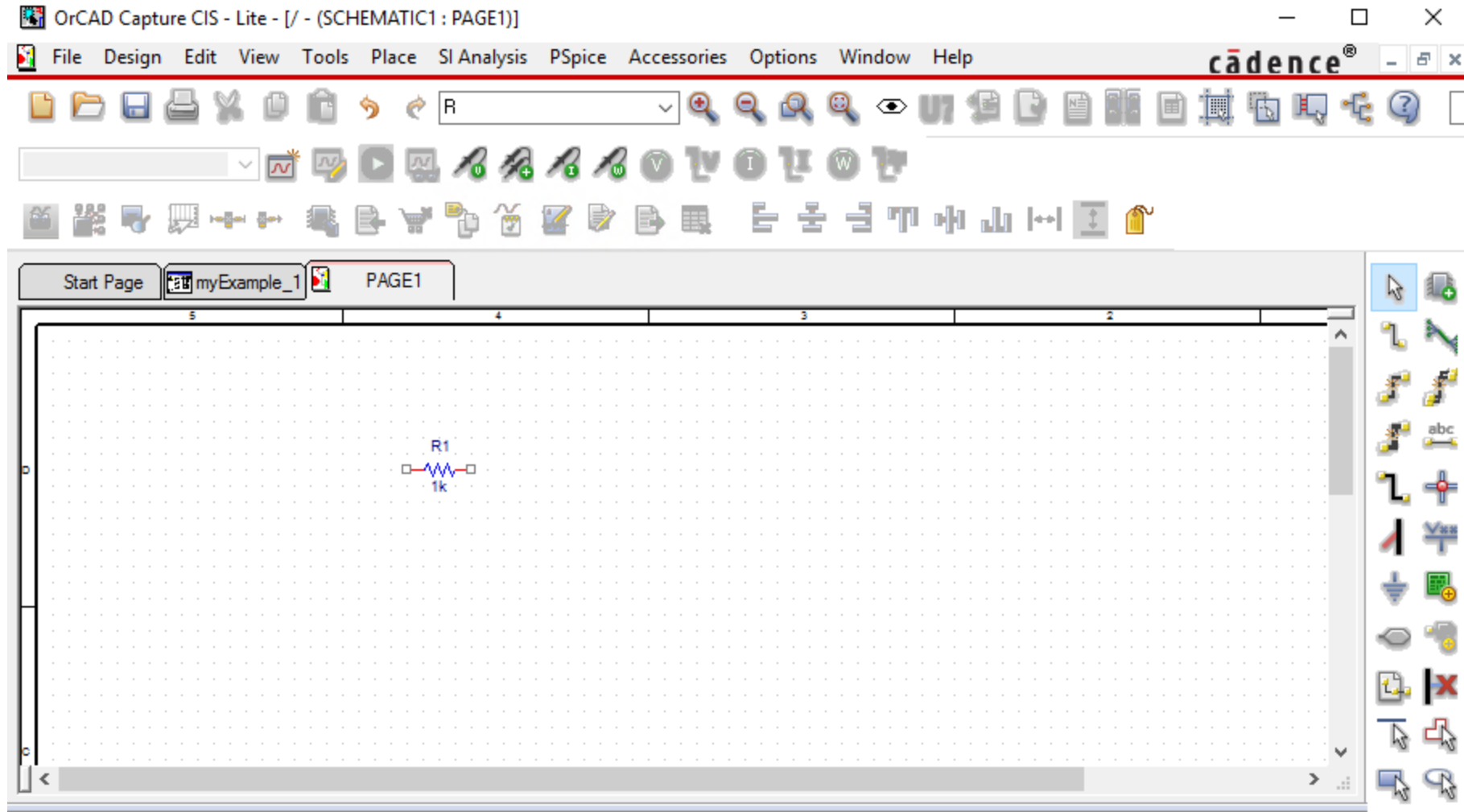
into the schematic

right-click and choose “End Mode”


now you have a resistor in your circuit schematic (see next slide)



A Resistor is Placed into the Circuit Schematic



Modify the Value and Label of the Resistor

- > left click on the resistor symbol  to select it
- > Right-click and then choose “Rotate” to rotate the resistor symbol from horizontal to vertical orientation
- > double click on the value, to edit the value of the resistance, e.g., change 1k to 2k
- > double click the label of the resistor, to rename the resistor, e.g., rename it from “R1” to “R_Load”

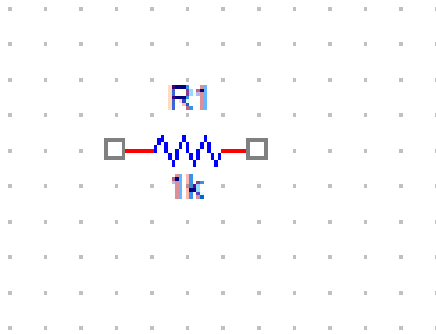


Fig. 1: Original resistor

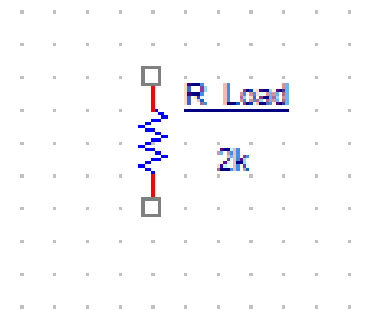


Fig. 2: The resistor is rotated (now in vertical orientation)
Resistor label is renamed to R_Load
Resistor value is changed to 2K

Create a Complete Circuit (simple RC example)

- > place a resistor into the schematic
- > place a capacitor into the schematic
- > place an AC source into the schematic
 - from “Place” menu,
 - > PSpice Component -> Source -> Voltage Sources -> AC
- > place a ground into the schematic
 - from “Place” menu
 - > PSpice Component -> PSpice Ground
- > use wire to connect the components
 - from “Place” menu
 - > Wire
 - click on Wire
 - then move the cursor to the component terminals
 - left-click to connect the wire to the component
 - connect the wire to another component
 - right-click to stop further connections

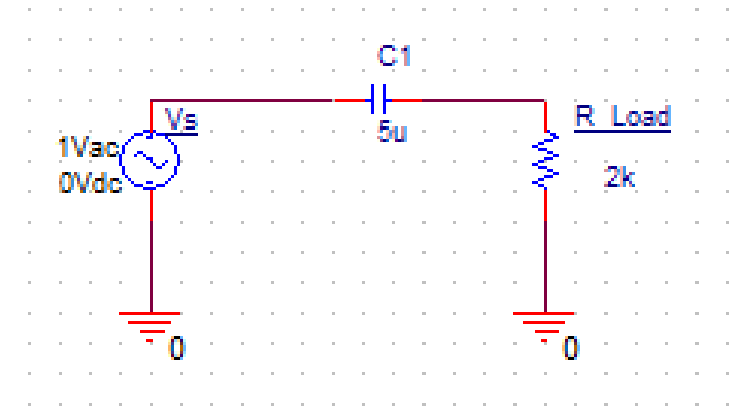


Fig. 1: schematic of the RC circuit

Now you have the complete circuit schematic as shown in Fig. 1

Define Simulation Profile

From PSpice menu

- > New Simulation Profile (Fig. 1)
- > Enter a name for your simulation profile (Fig. 2)
- > if you encounter a question (pop-up window) “would you like to launch Simulation Manager using Lite license ?” click “Yes”

You will then get a “Simulation Settings” window (Fig. 3)

- > Select “AC Sweep/Noise”
 - > enter values for “Start Frequency”, “End Frequency” and Points/Decade
- Notice that “logarithmic” is the default scale for frequency sweep

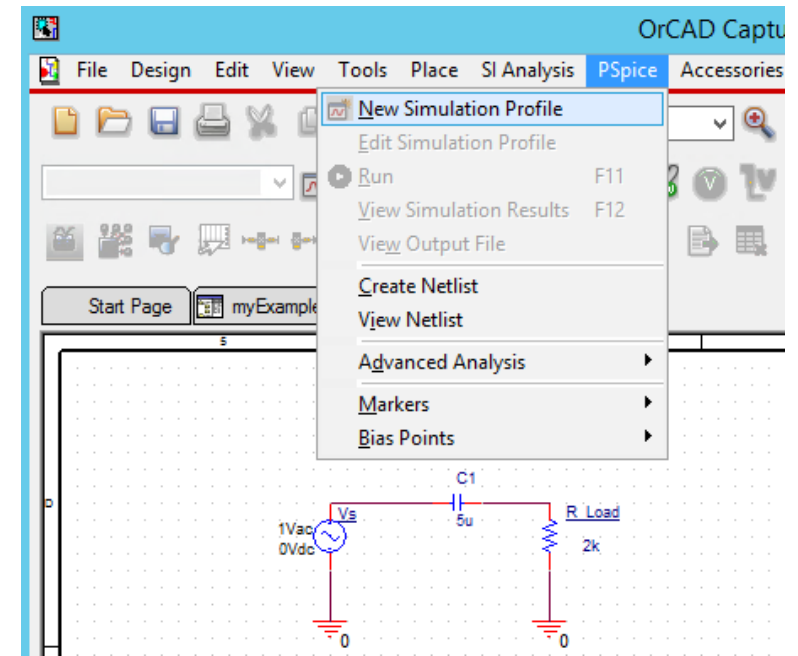


Fig. 1

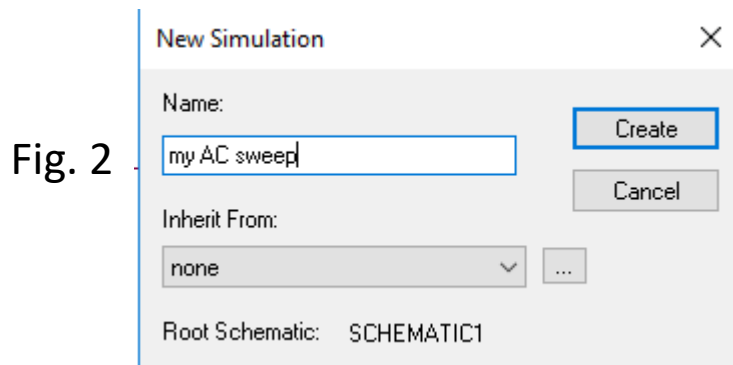


Fig. 2

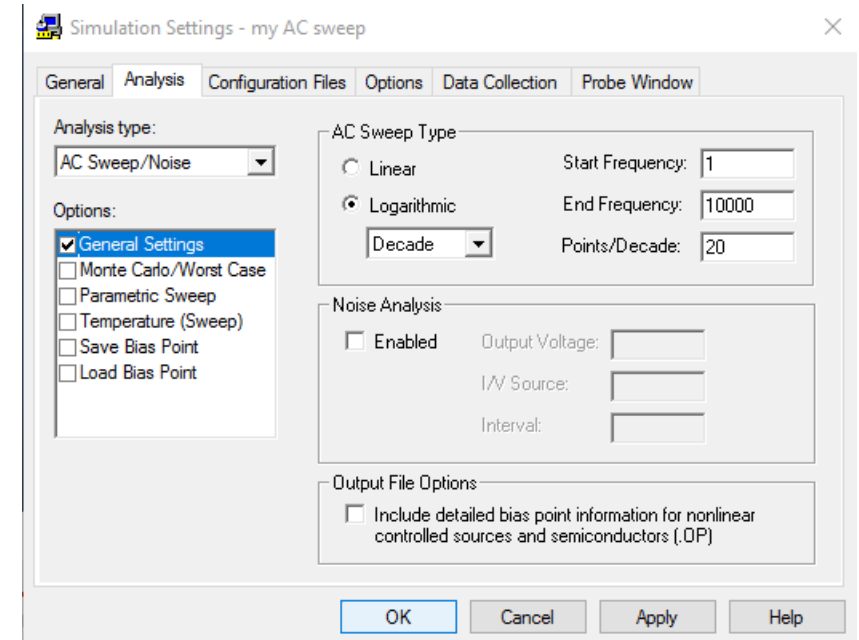


Fig. 3

Perform Simulation and Display Results

From PSpice menu:

-> Select “Run” (Fig. 1)

Wait until simulation is finished, and a new window appears (Fig. 2)

If you cannot see the new window, it may be behind the main schematic window.

In the new window, select “Trace” menu, then “Add Trace” (Fig. 3)

Select V2(R_Load) which means voltage at terminal 2 of R_Load (Fig. 4)

Now the curve for voltage V2(R_Load) versus frequency will be plotted (next slide)

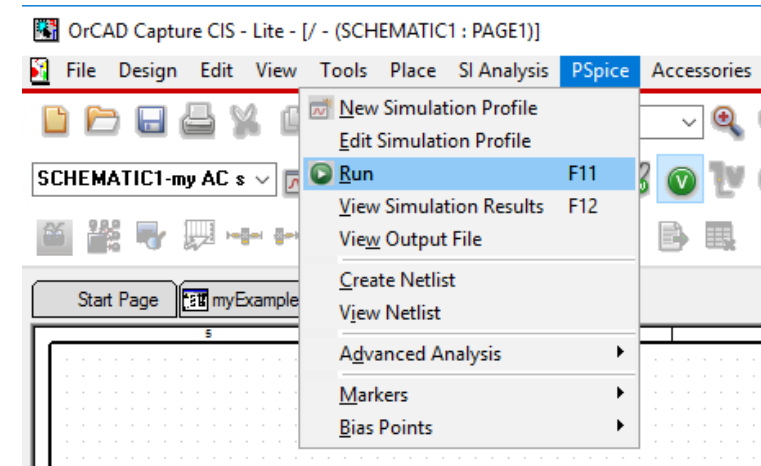


Fig. 1

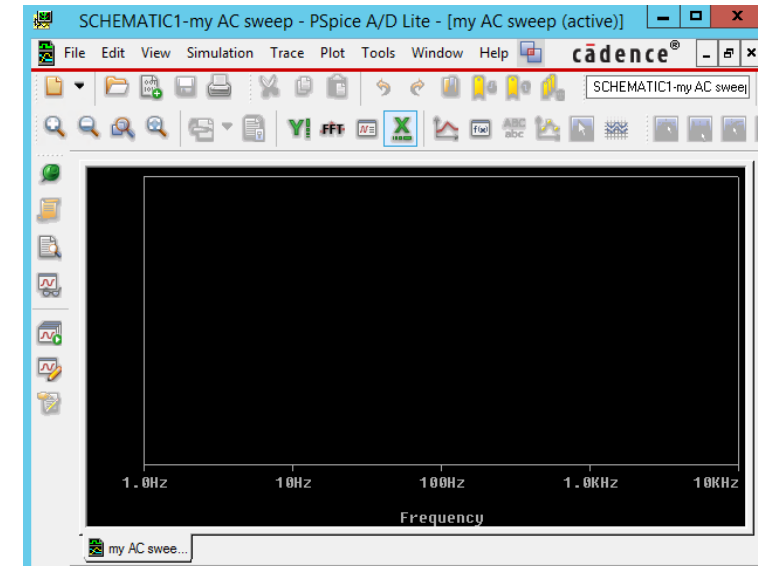


Fig. 2

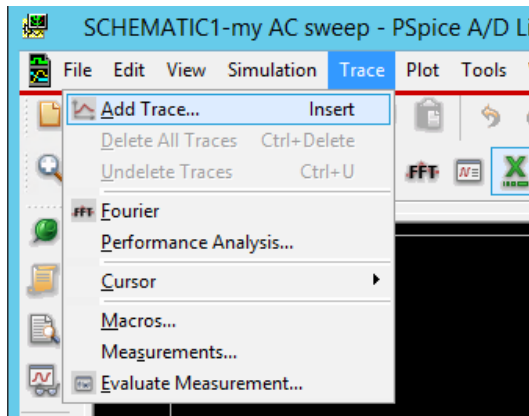


Fig. 3

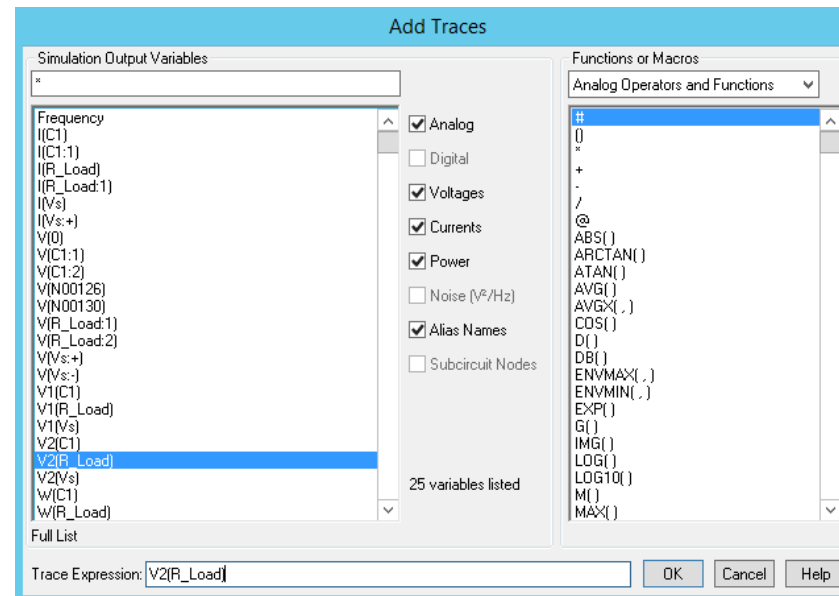
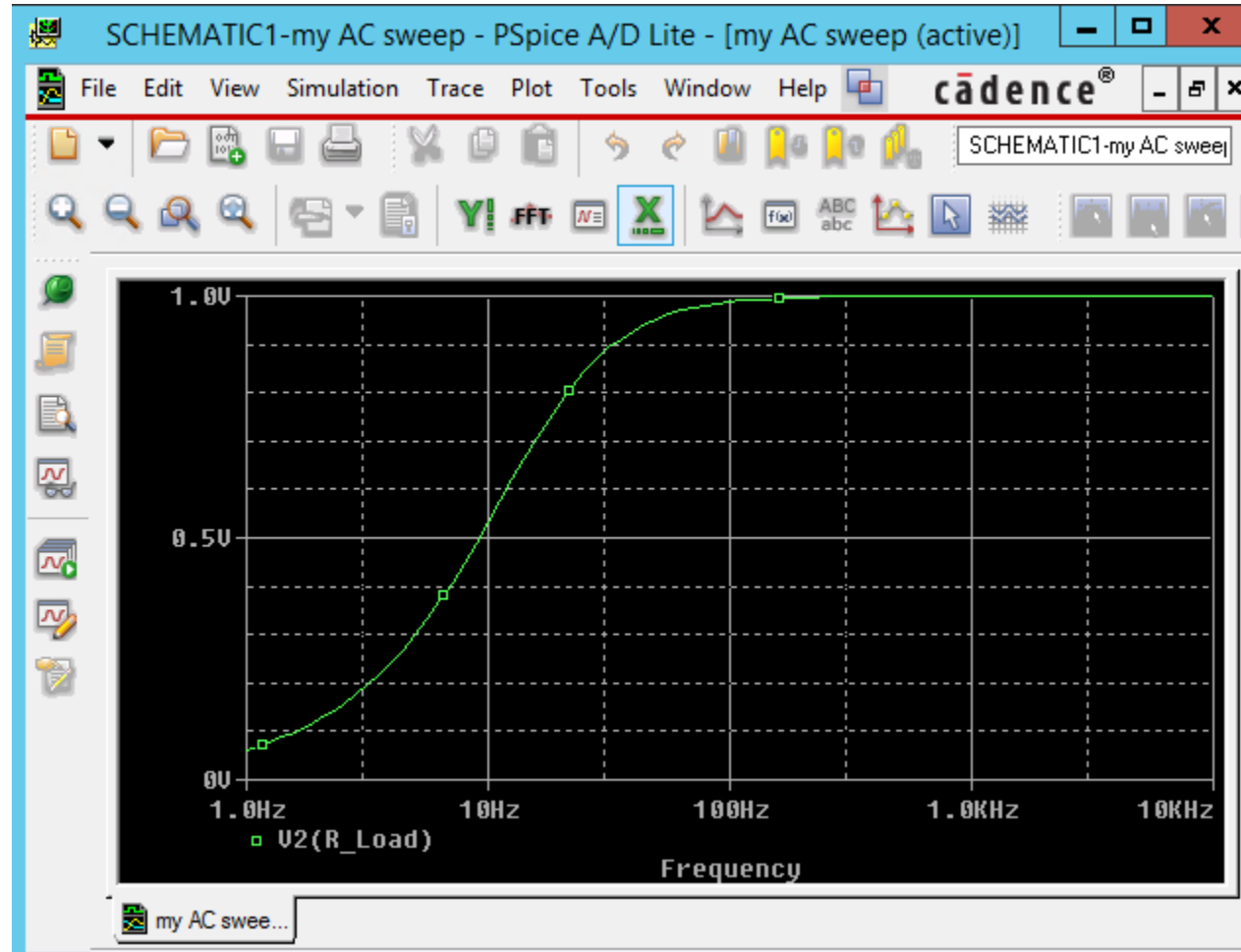


Fig. 4

Display of Response Solutions

The curve for voltage V2(R_Load) versus frequency is shown. The horizontal axis is frequency in log scale. The frequency is from 1Hz to 10KHz. The vertical axis is Voltage at R_Load.



How to Delete a Component in the Schematic

To delete an unwanted component (a resistor, capacitor, ...):

- > single-click on the symbol of a particular component (to select the particular component)
- > press the “delete” key on your keyboard

Similarly, you can delete any unwanted wire, and unwanted ground.

To delete many components within a rectangular area in the schematic, use cursor (left-click-and-hold, move cursor, release the hold) to draw a rectangular area, then press Delete Key. In this way, everything in the rectangular area (all components, all wires, all grounds) are deleted.

How to Load the Bipolar Transistor Library

From “Place” menu, select “Part” (Fig. 1)

A new section named “Place Part” will appear at the right hand side of the window (Fig. 2)

Click the “Add Library” icon (Fig. 2)

Scroll to “bipolar” library, and select the “bipolar” (Fig. 3)

The BIPPLAR library is successfully loaded into the libraries (Fig. 4)

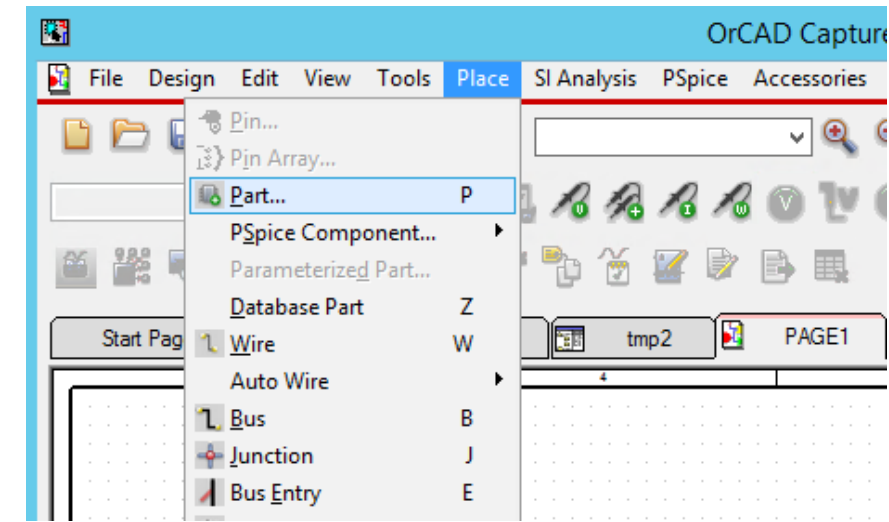


Fig. 1

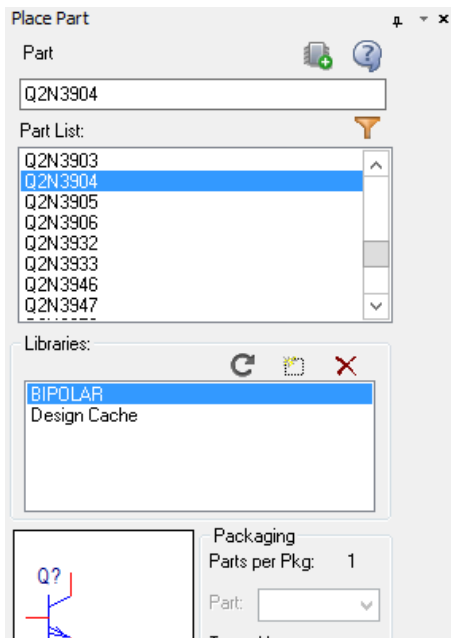


Fig. 4

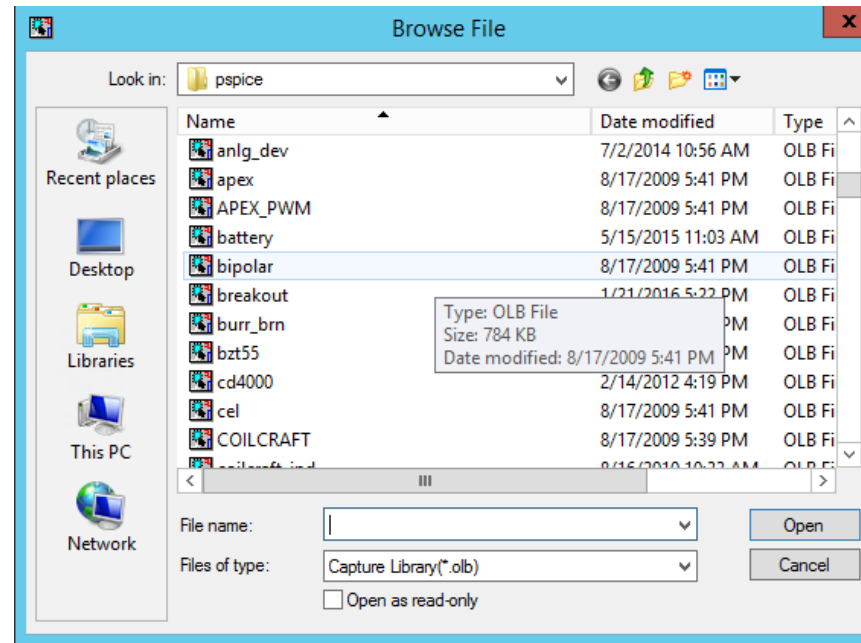


Fig. 3

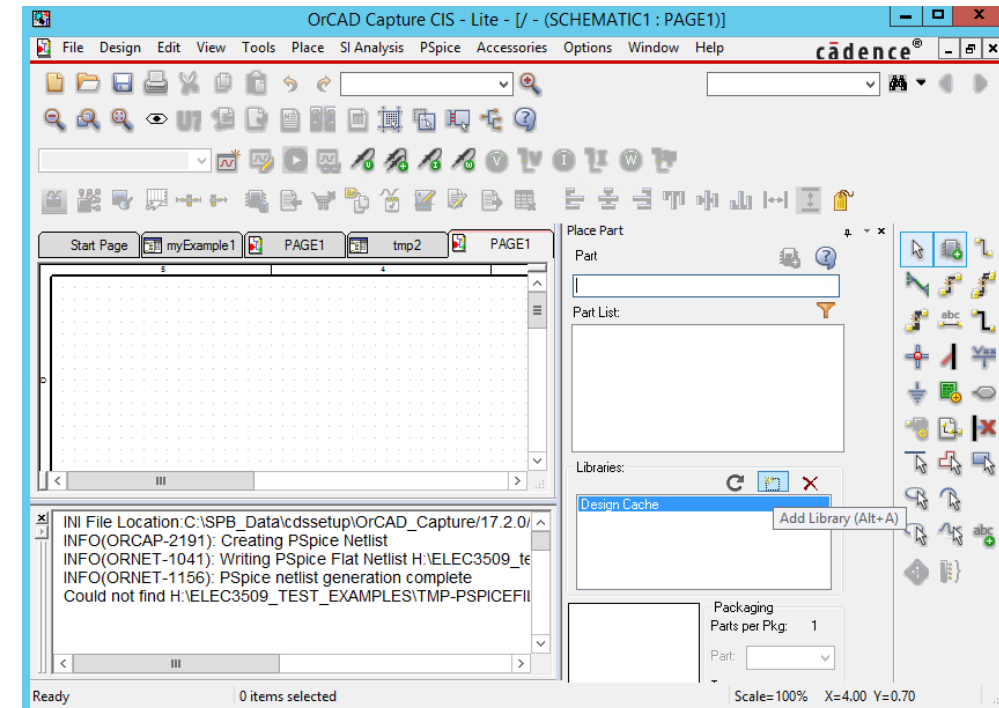


Fig. 2

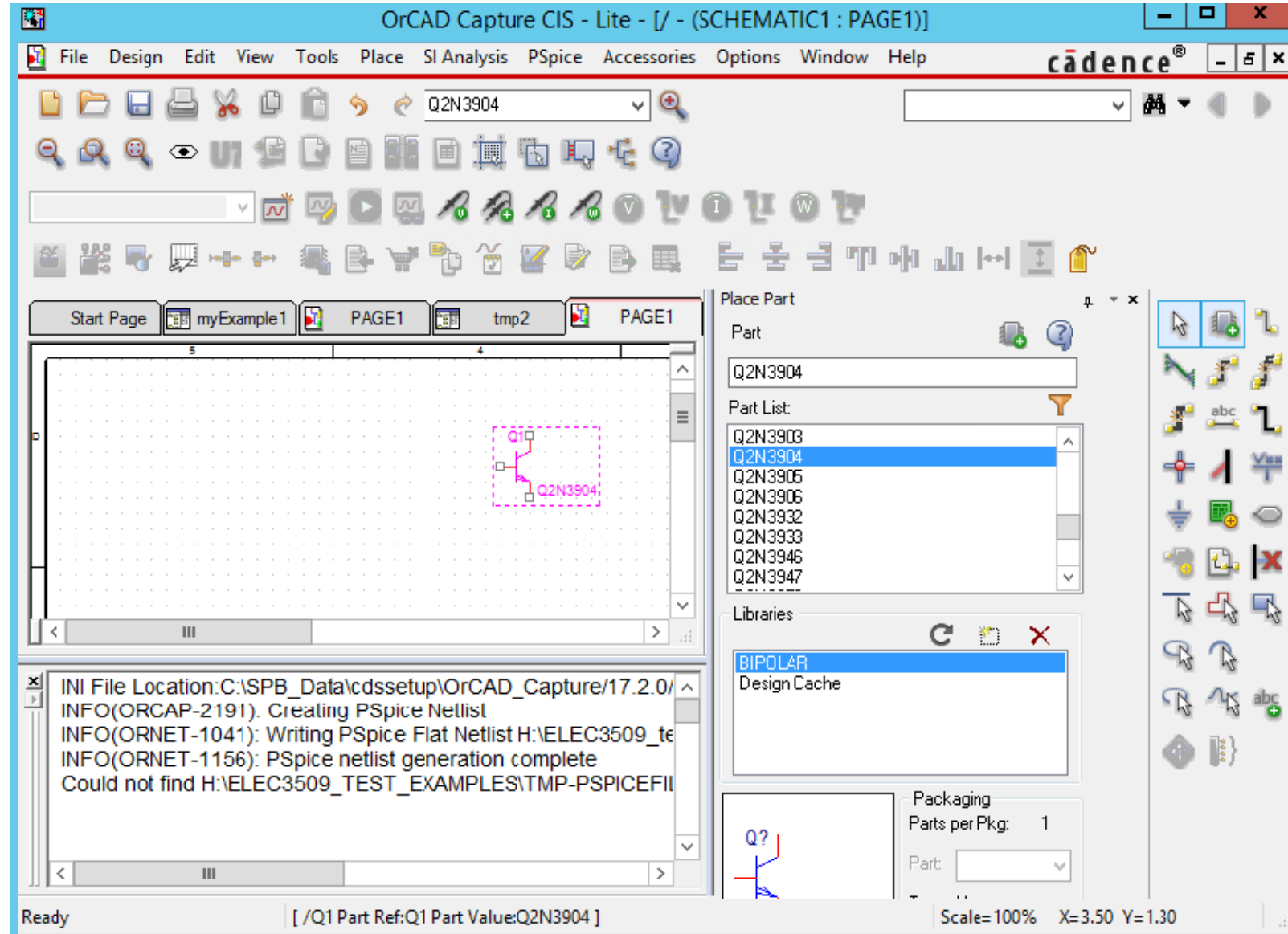
How to place a BJT into the Circuit

In the Libraries, select
“BIPOLAR”

From Part List, find the part
Q2N3904 (NPN transistor) or
Q2N3906 (PNP transistor)

Double click Q2N3904, then
move the cursor to the
schematic window

Left-click, and the transistor is
placed into the schematic



Exercise 1: RC Circuit Example

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

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 frequency range is set to be from 1Hz to 10KHz. We use 20 frequency points per decade.

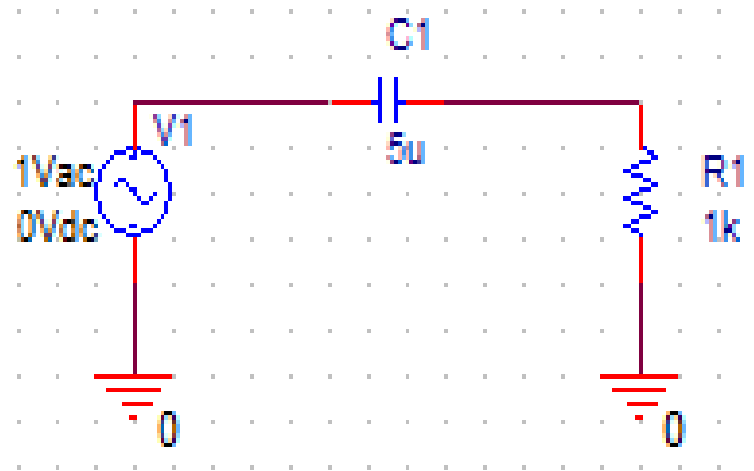


Fig. 1

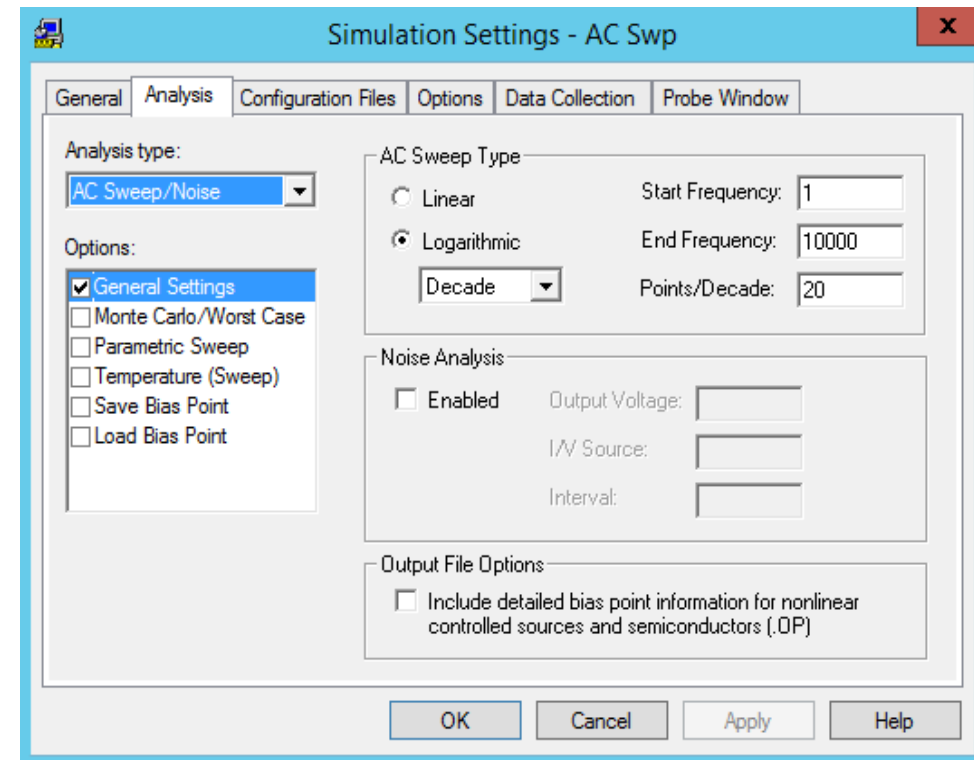


Fig. 2

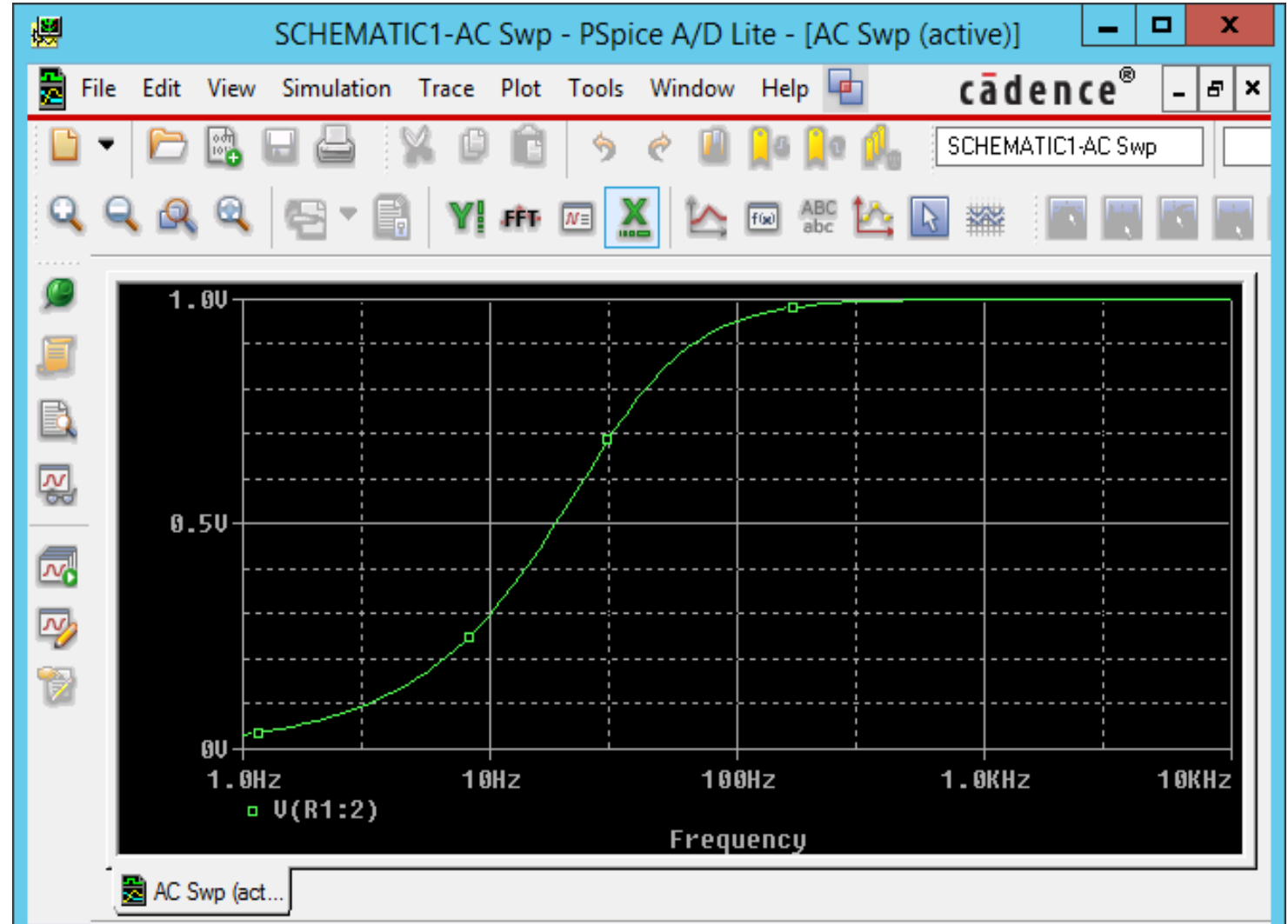
Exercise 1: RC Circuit Example

Perform simulation and display results as shown.

Horizontal axis is frequency from 1Hz to 10KHz in log scale

Vertical axis is $V(R1:2)$, which is the voltage at terminal 2 of resistor R1. An alternative symbol for the same voltage is $V2(R1)$.

Because the 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.



Exercise 2: BJT DC Curve

Create the circuit schematic as shown in Fig. 1. 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 .

Define “New Simulation Profile” as shown in Fig. 2. DC Sweep is selected for Analysis type. The sweep variable is voltage source VC. The range for sweeping VC is from 0V to 10V with step size of 0.1V.

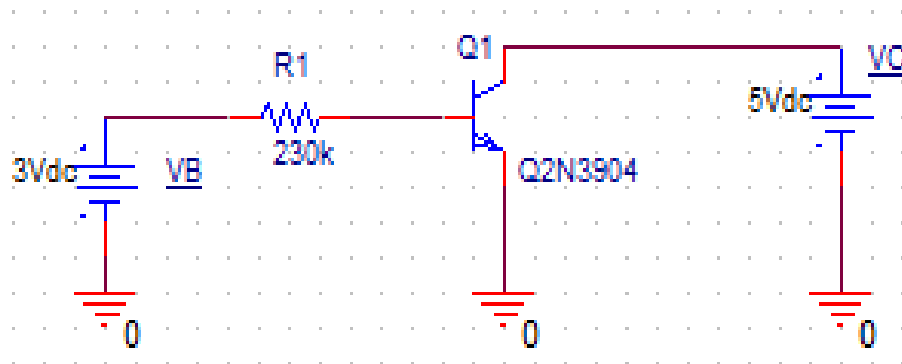


Fig. 1

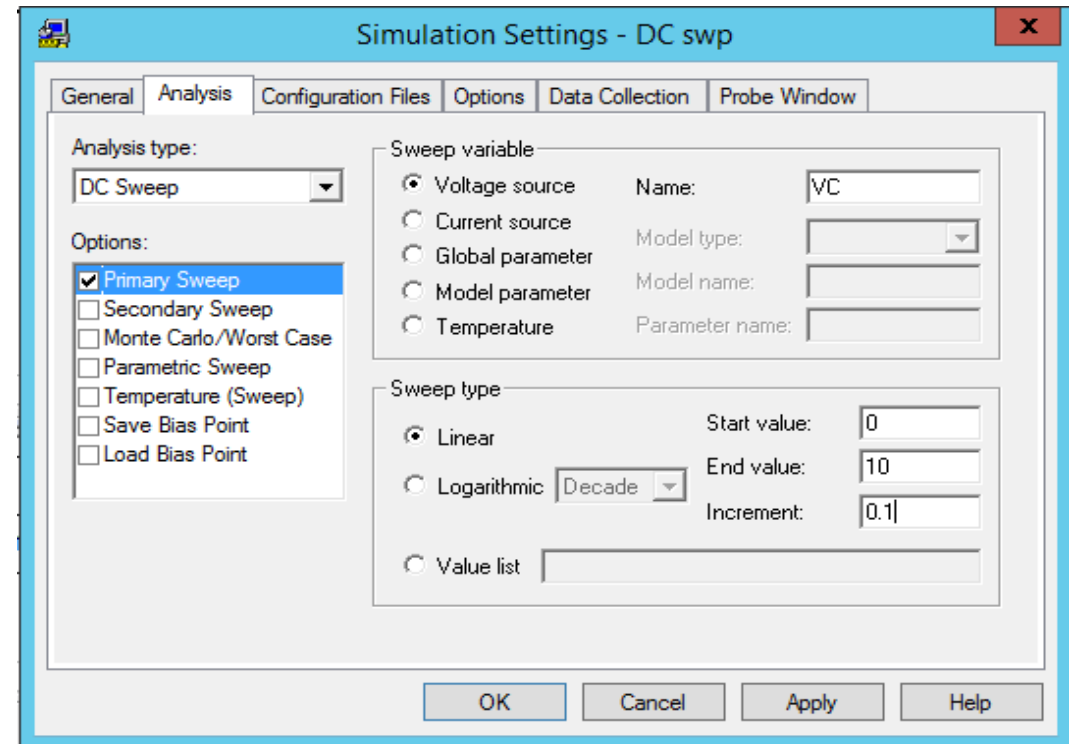


Fig. 2

Exercise 2: BJT DC Curve

Perform simulation and display results as shown.

Horizontal axis is voltage V_C

Vertical axis is selected as $I(Q1:c)$. It means the Current through terminal c of transistor Q1 (i.e., collector current). An alternative symbol for the same current is $I_C(Q1)$.

This figure shows one curve in the family of IV curves of this transistor.

