EEE 111, ETE 111 lab: Electronics 1

Experiment No: 01
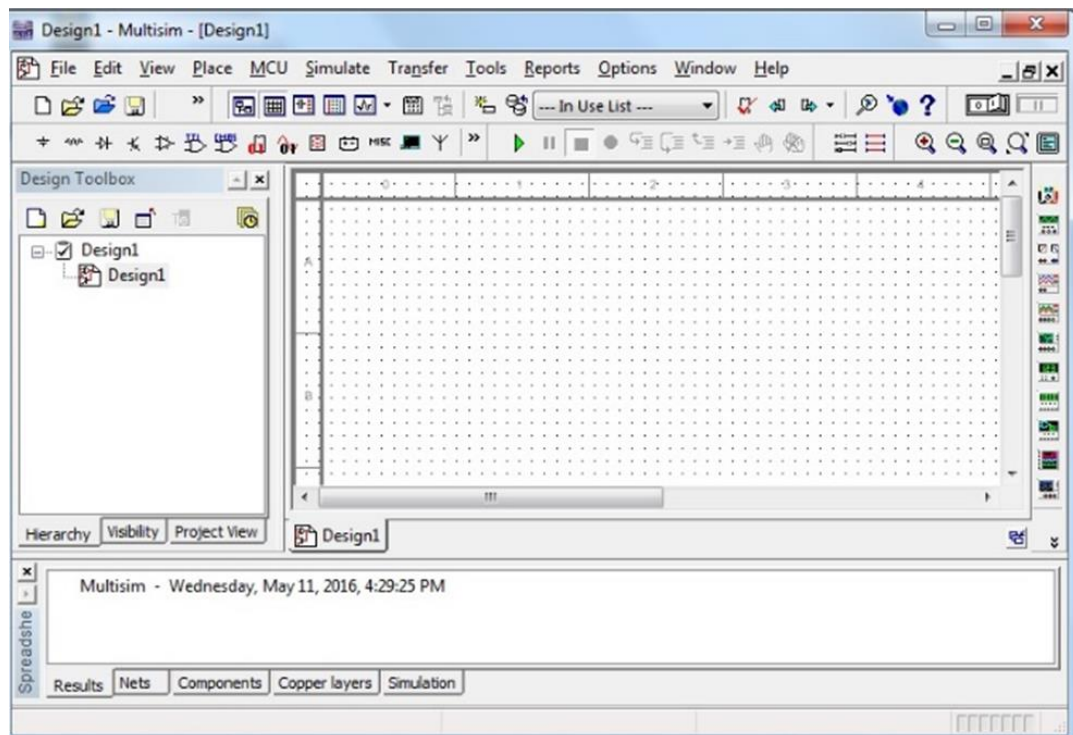
Name of the Experiment:

Introduction to MultiSim 12.0

Objectives:

To get an overview on how to use MultiSim software for circuit design and simulation. .

Study:

- Open **MultiSim 12.0** by clicking on its icon.
- Or go to *Programs > National Instruments Circuit > Design Suite 12.0 > Multisim 12.0*
- If any toolbox did not show, you can go to *View > Toolbox*
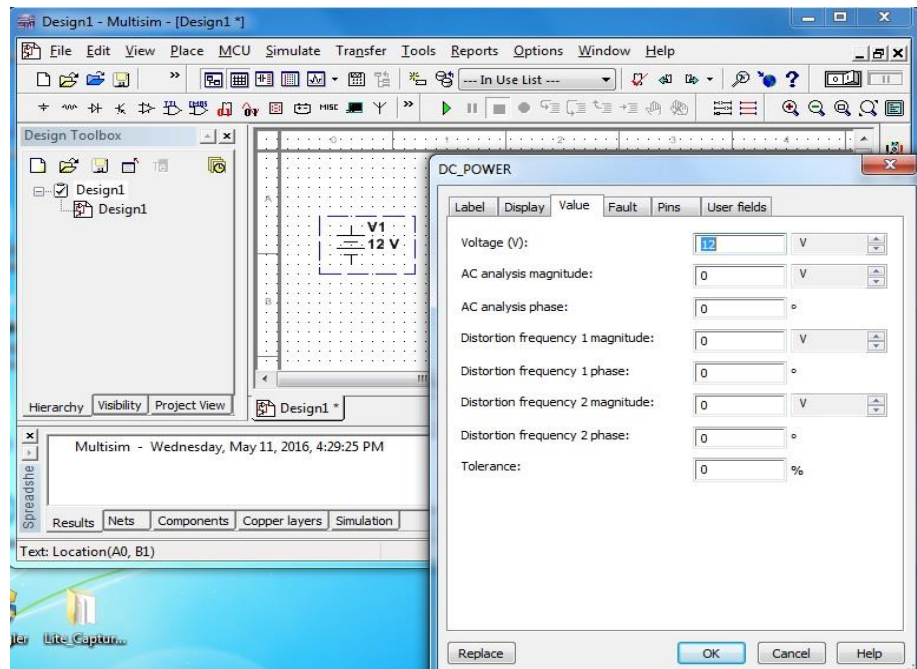- And check the desired toolbox

Unlike PSPICE, Multisim automatically starts a new project for you, and you can start drawing your circuit by placing the different components required.

Placing Components:

To place a new component, go to-
Place > Components

You will see a component selection window that pops up where you can place your components and sources.
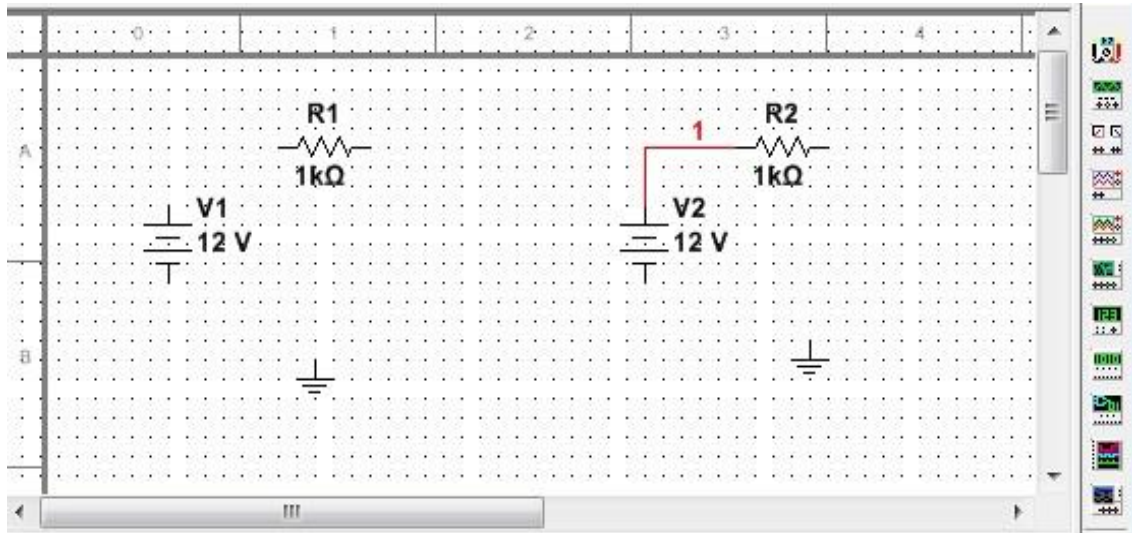

Changing Component Parameters:

To change the values or names of the component, you can either right click on the component and select properties, or double click on the component, and a screen will pop up allowing you to change the properties and values of the component.
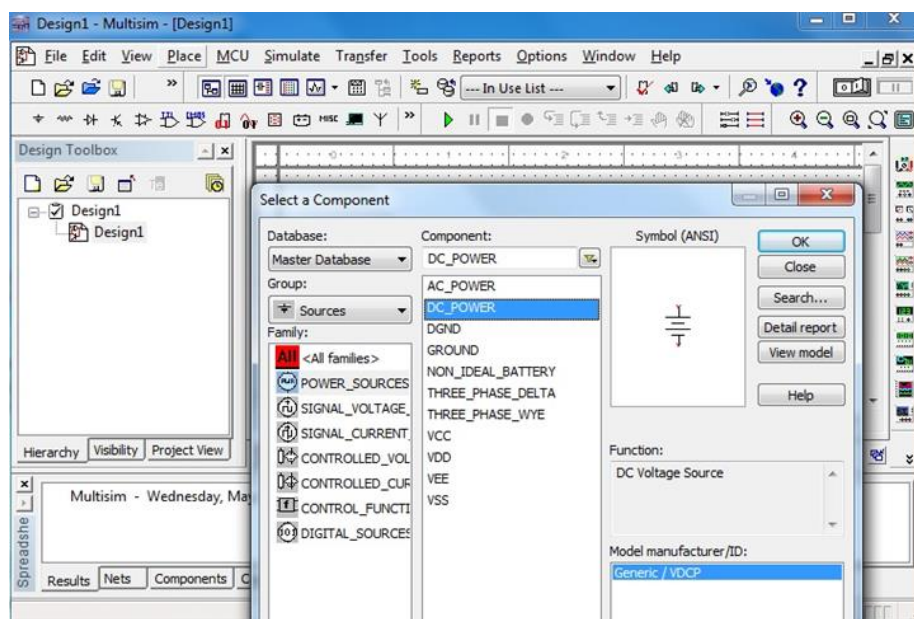
Wiring:

When wiring, place cursor at the corners of component and you should see a black dot appearing, click the mouse once and drag your wire to the edge of another component where a red dot should appear, click again to connect.
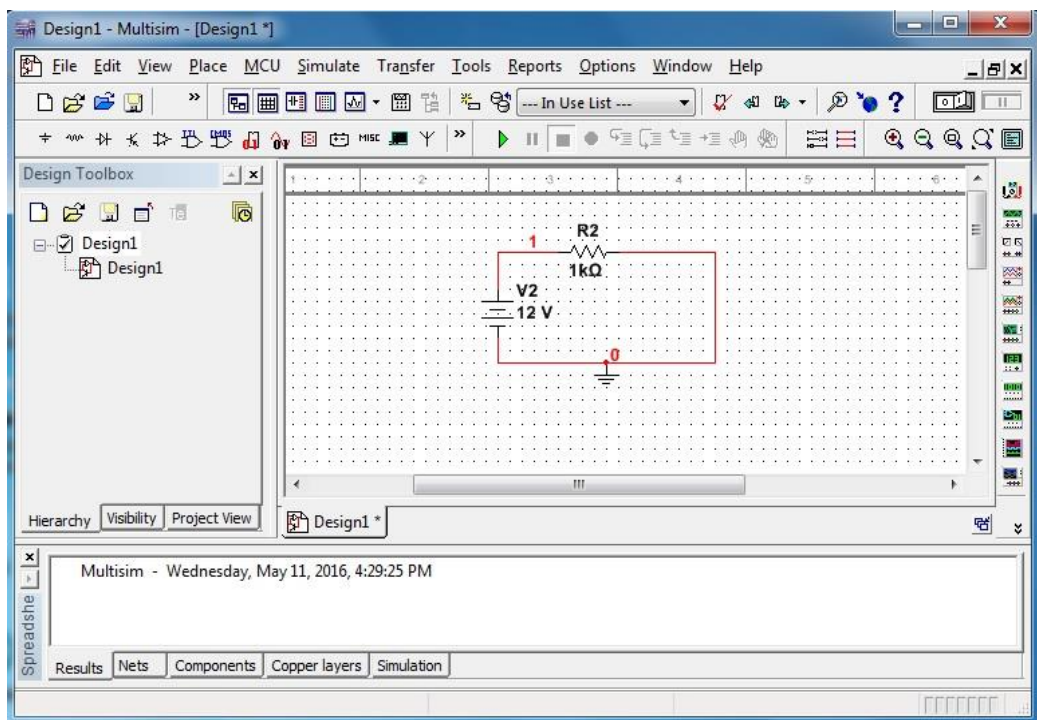
Displaying Node numbers/ Wire names:

To show the net names of the wires, select the desired wire and right click Properties, and a Net Settings window should pop up. Checking the <u>show net name</u> box will allow one to see the wire names/net on the circuit.
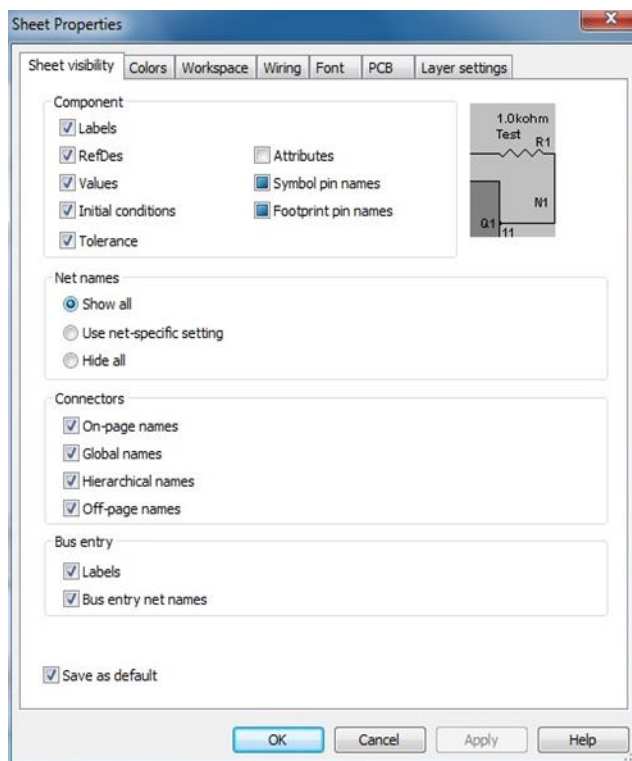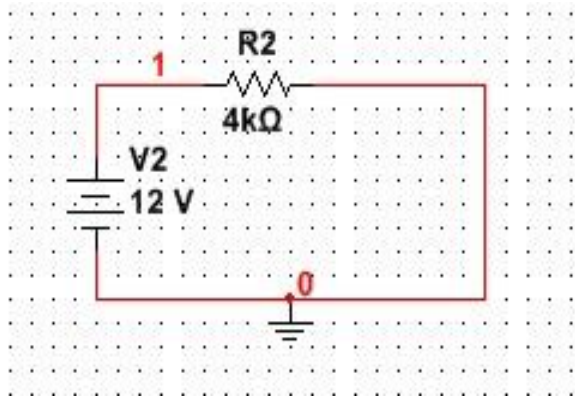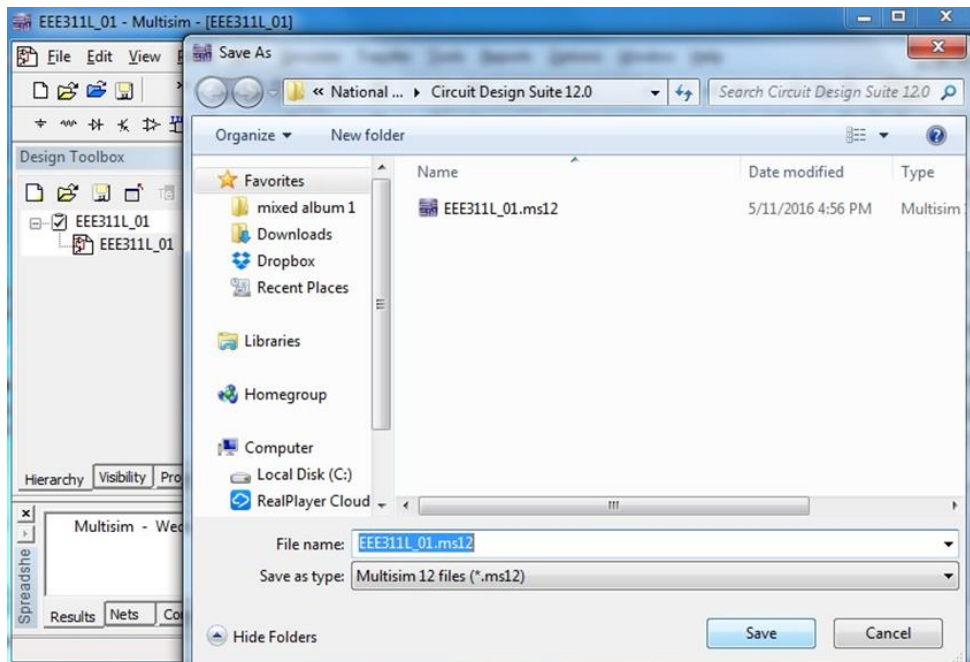

One can also do Options > Sheet Properties > Circuit > Net Names: check "Show All".
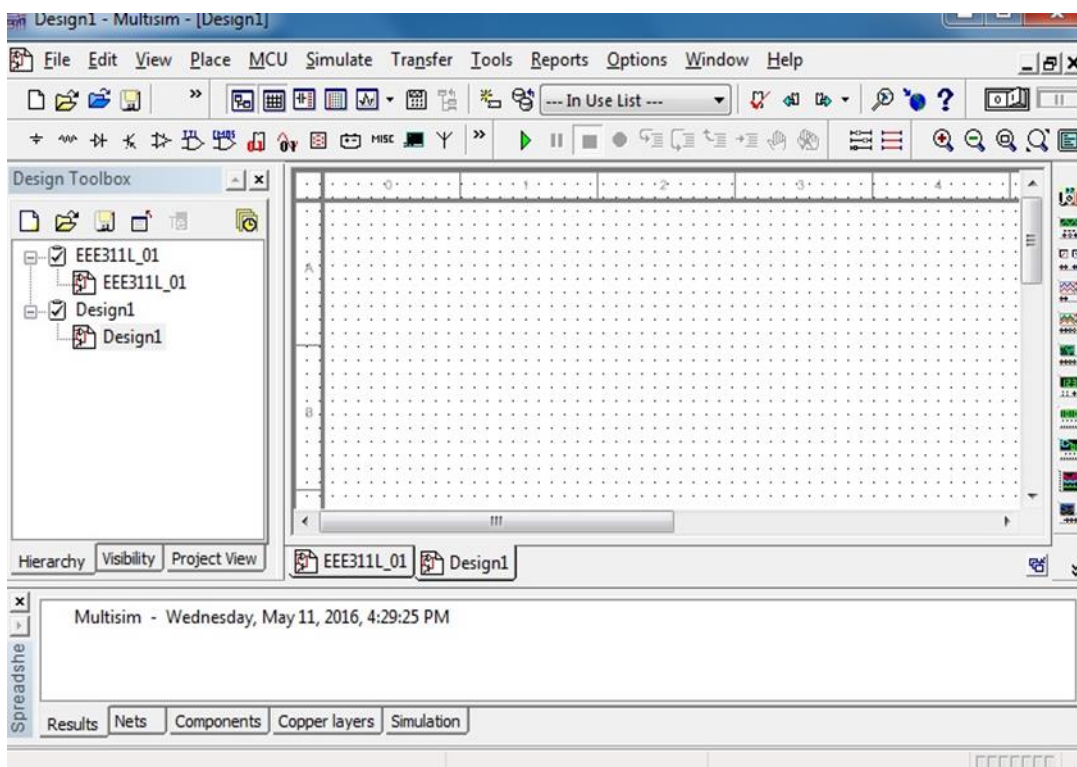
Saving:

To save your project, go to-
File > Save as > Name your design

Once saved, you will see the name appearing on top of the window and the original name Design1 will change to your new project name.
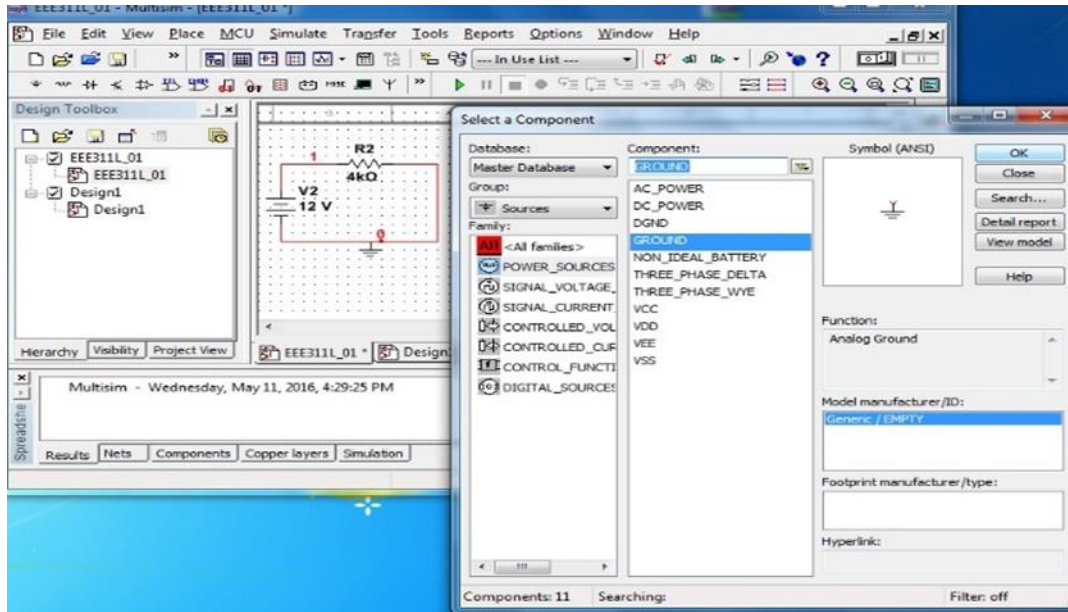
You can also create more than one project in one setting by-
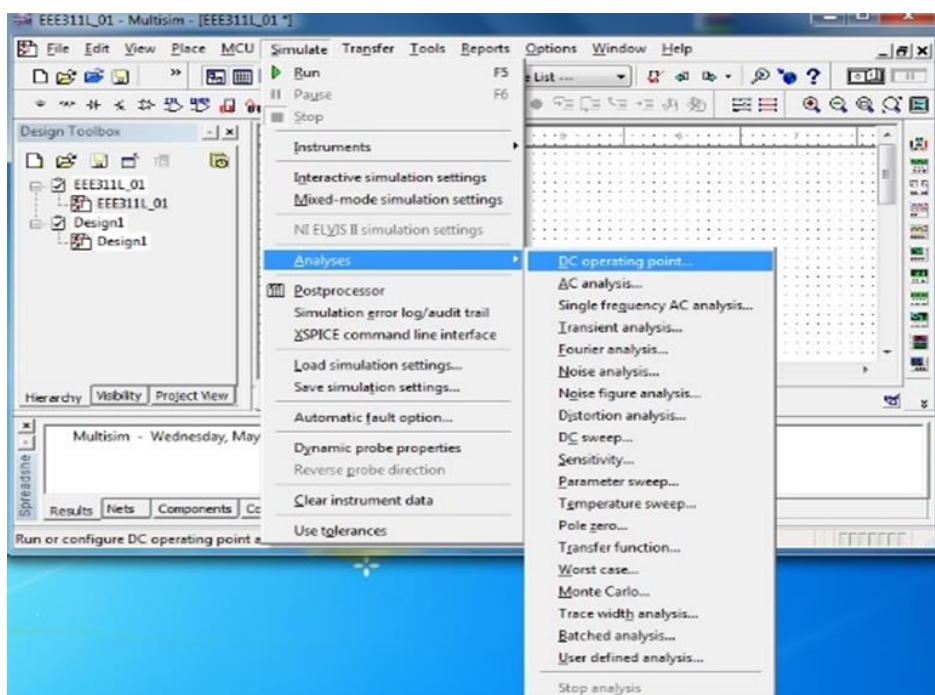*File > New > Design*

Simulation Analysis:

1. First, draw the desire circuit:
2. *Place > Components*
3. In the <u>Select Component</u> windows, under Group, select Sources for DC_POWER and GROUND.
4. In Group, select Basic and chose RESISTOR.
5. Connect the wires to complete the circuit.

DC Operating Points:

To see the DC Bias points, go to-
Simulate > Analyses > DC Operating Point

The DC Operating Point Analysis window will pop up with three tabs, Output, Analysis options and Summary. For the Output tab, one can select the desired variables by Adding or Removing. Click Simulate when done. You can also change the simulation title name in the Analysis options tab.
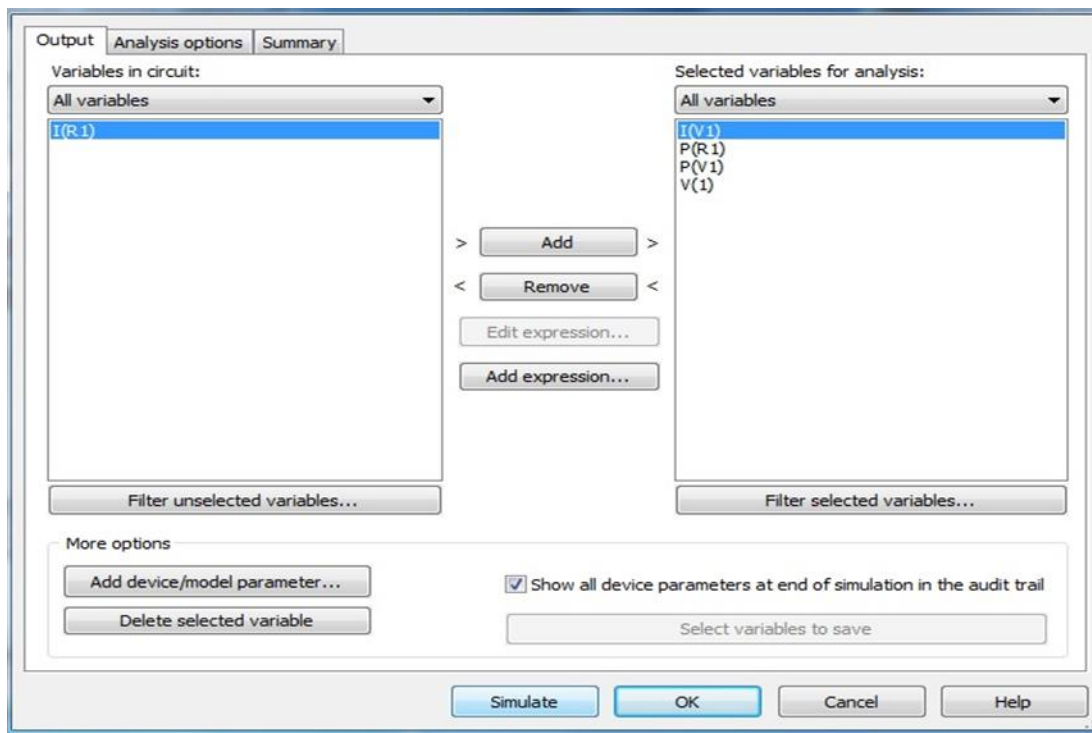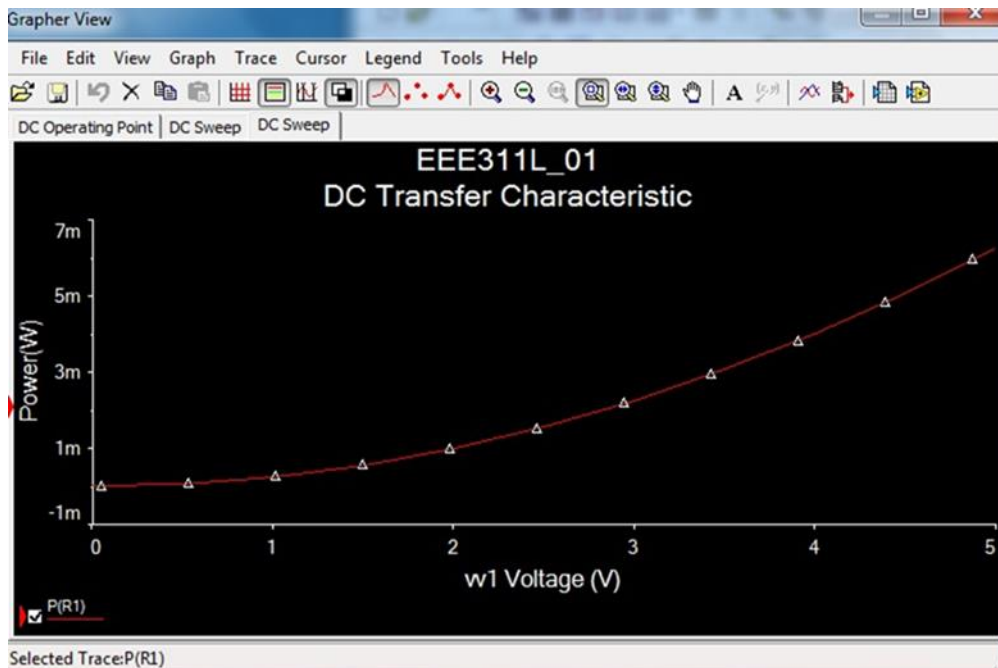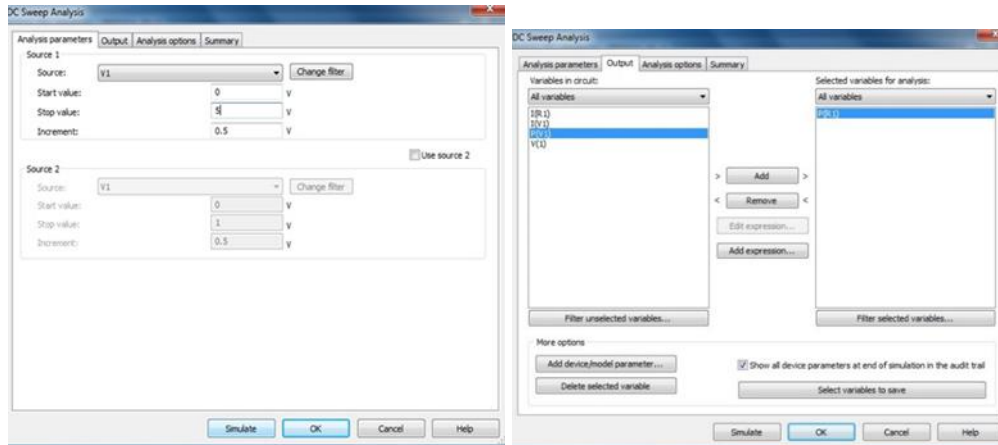

A Grapher View window will pop up after clicking on the Simulate button, showing the DC operating points corresponds to the components in the circuit.

DC Sweep:

Go to *Simulate > Analyses > DC Sweep*.
Select the desired variable to sweep over, in this case, we want the Power of Resistor 1, and click Simulate.

Parametric Sweep:

To see the different Current values when we change the Resistor values, we can do a parametric sweep, go to-
Simulate > Analyses > Parameter Sweep

In the Parameter Sweep window, change the following parameters:

After changing the parameters in the
Analysis parameters, go to the Output
tab and select the desired output to I(R1).

After clicking on Simulate, the Grapher View should pop up showing the Graph with 4 different curves.   Click on the show Cursors will allow you to see the different current values for different resistance values.
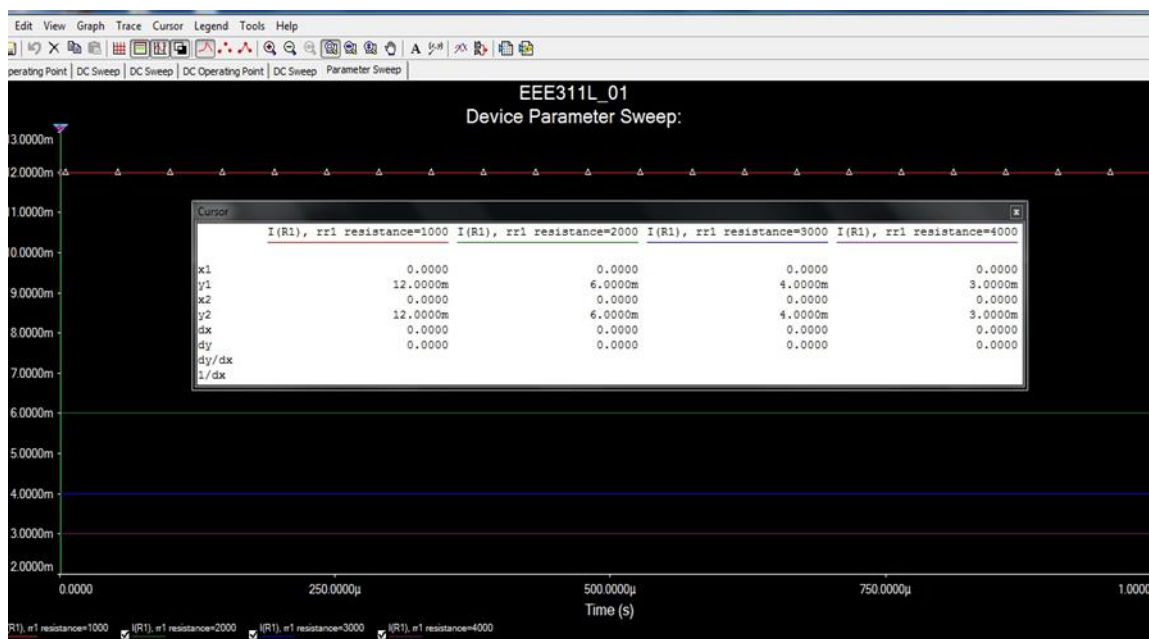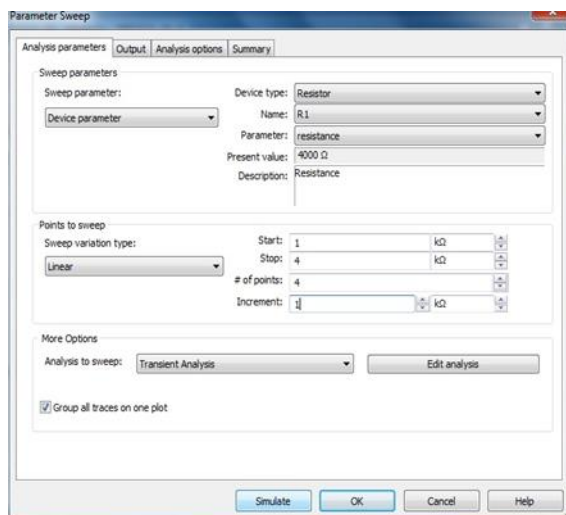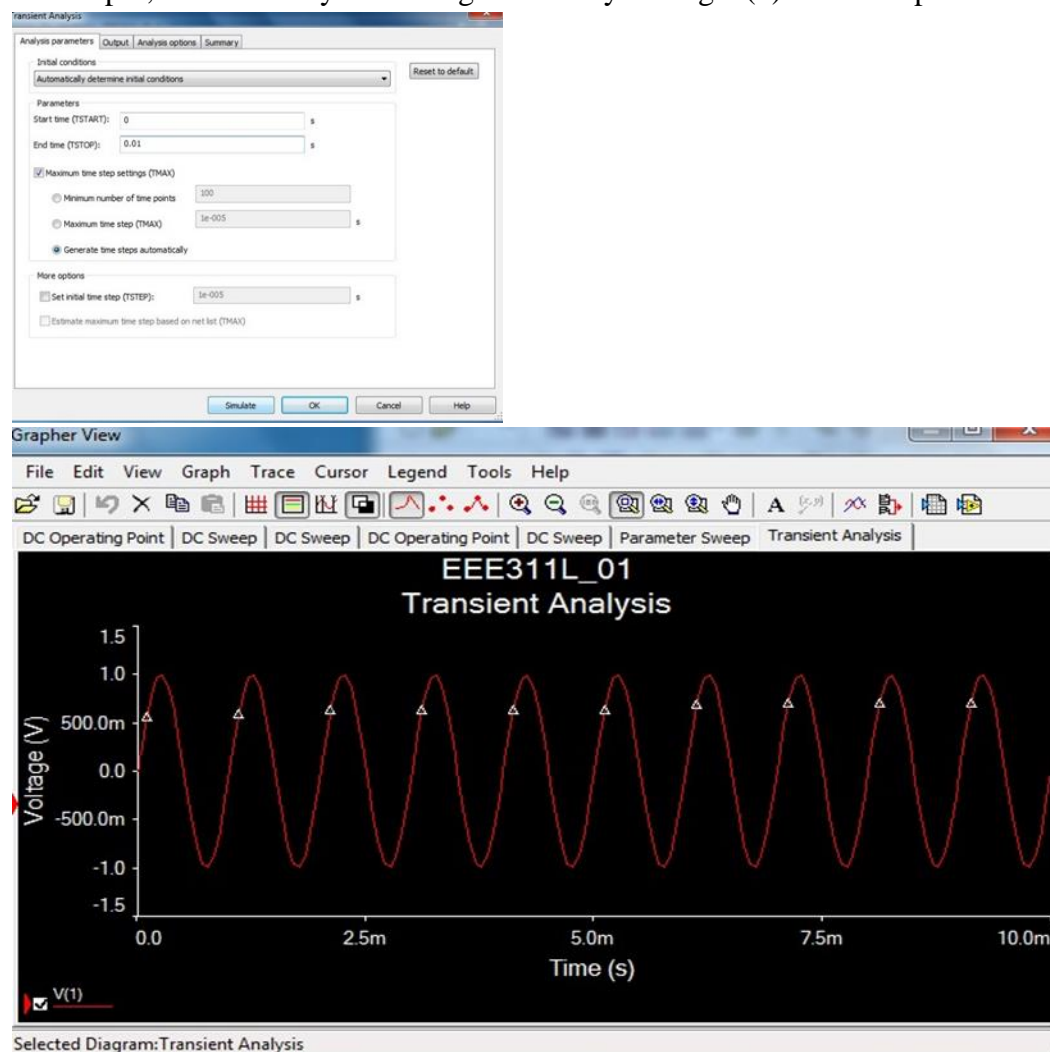
Transient Analysis:

For transient analysis, we can change the voltage source to an AC_VOLTAGE under the
SIGNAL_VOLTAGE_SOURCES.
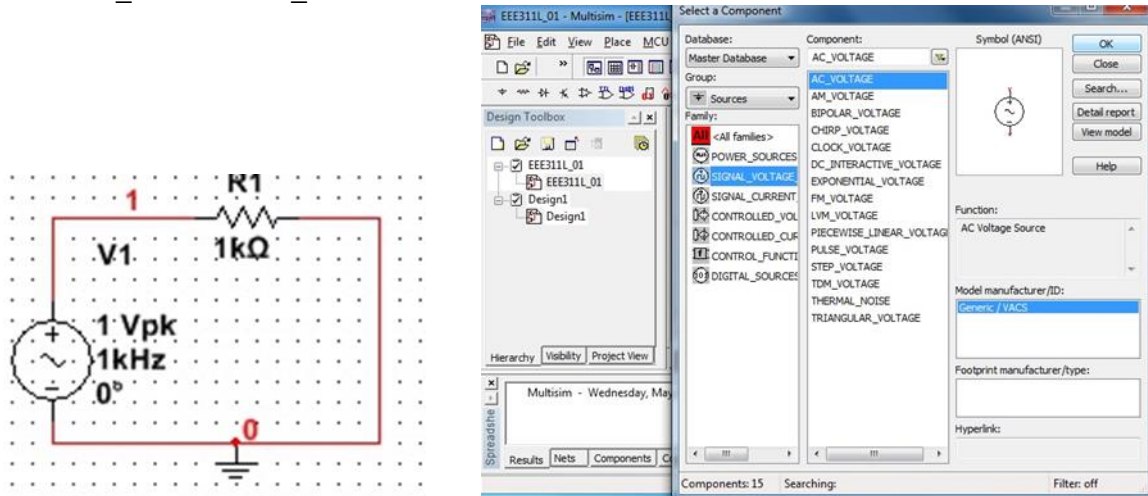
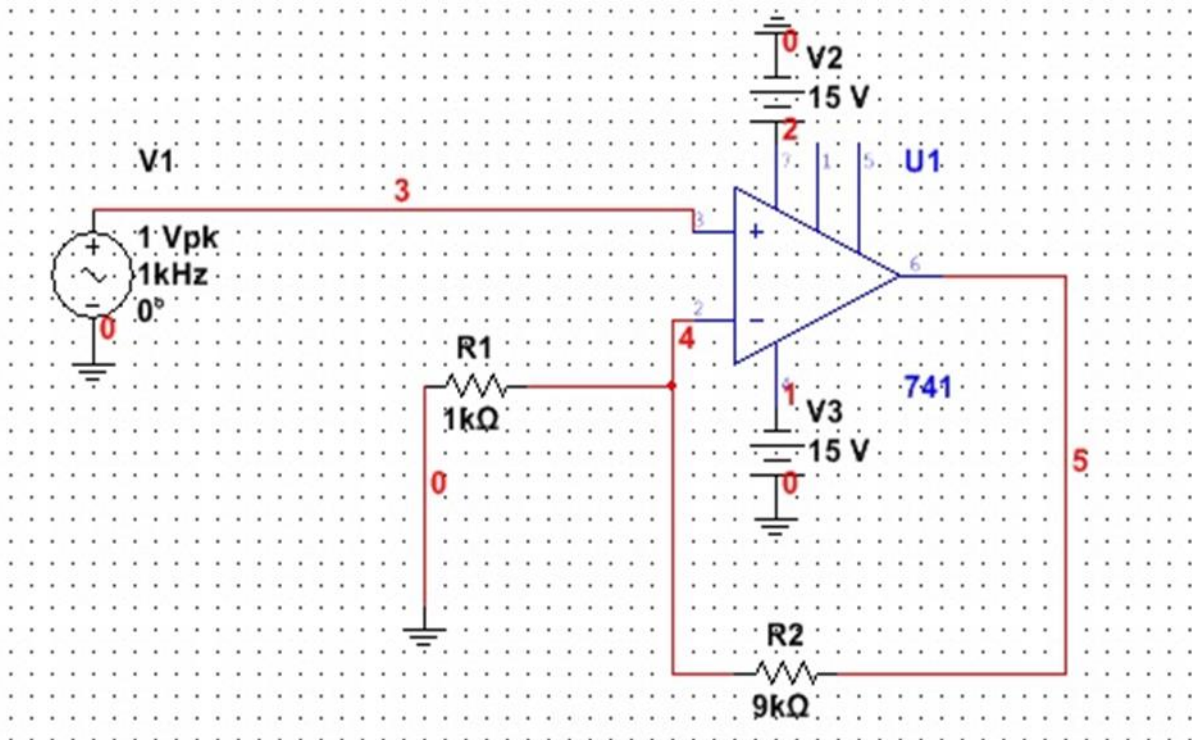Go to Simulate > Analyses > Transient Analysis
Change the following parameters

For output, we can verify our voltage source by adding V(1) to our output

AC Analysis:

Draw the following circuit (closed loop inverting amplifier), for the amplifier, under <u>Group</u>, select Analog and select the OPAMP under <u>Family</u> and type in 741 for the amplifier part.


Simulate > Analyses > AC Analysis

A window for the AC analysis should pop up, change to the following parameters:
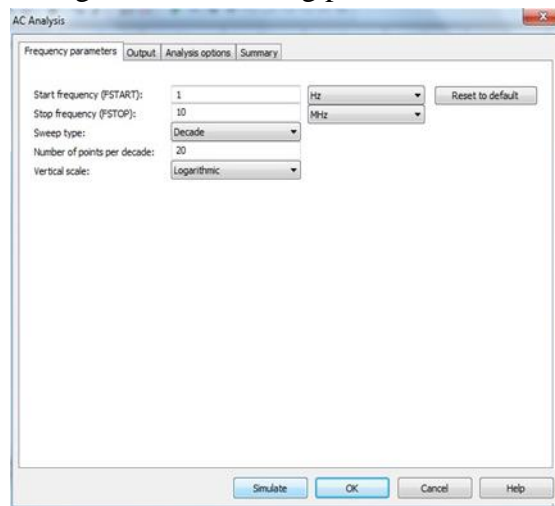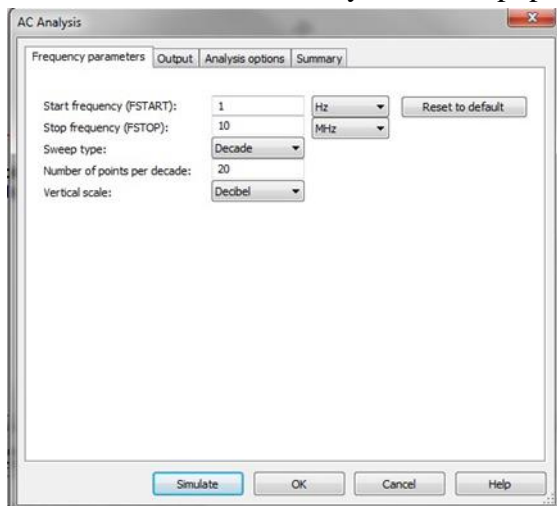

For the Output tab, choose V(6), the output of the amplifier.

Two graphs should be show in the Grapher View, where the first graph shows the magnitude of the gain of the amplifier, and the second graph shows the Phase vs. Frequency (we will usually look at only the first graph). Again, if we would like to see or plot points on the graph, we can choose Cursors, and move or type in our desired location.
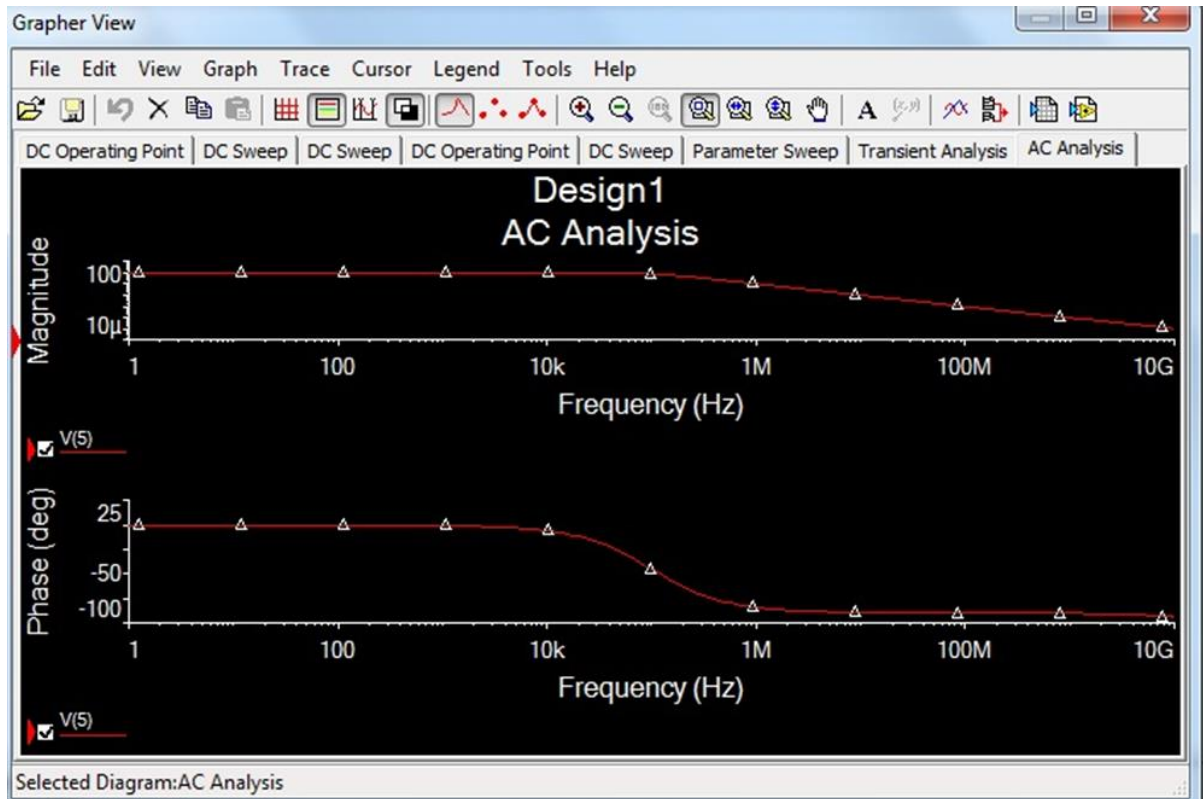
Copy Graphs and Circuits:

- To copy circuit schematic: Select the entire circuit using the mouse and *right click > copy*
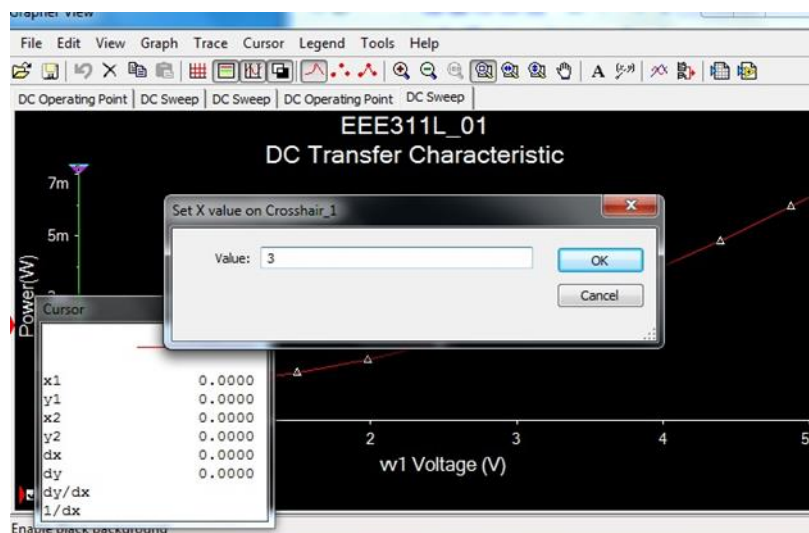- To copy the graph desired from Grapher View: *Edit > copy graph*
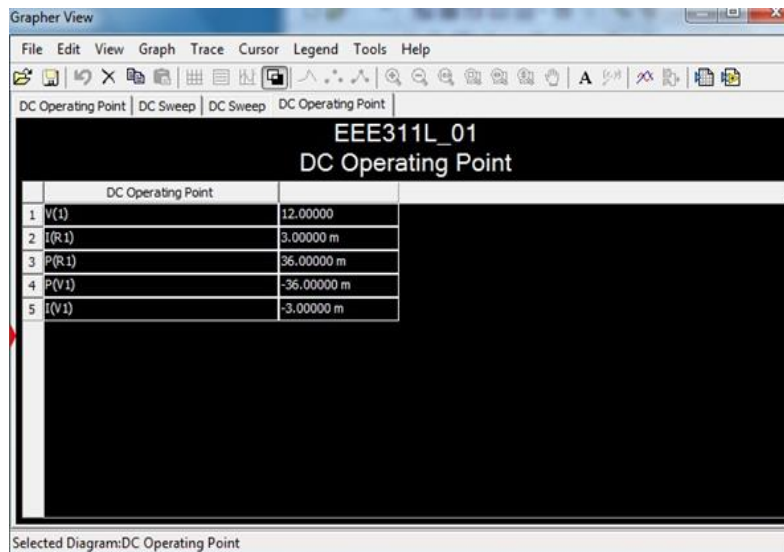
Cursor:

To see different point values or the use of **Cursors** in the Grapher View window**, go to-** *Cursor > Show Cursors*

A cursor window should pop up and two cursors should show up

Set X value (a Set X value window should pop up)
- Set value to 3 and click OK
- This sets the Voltage value to 3 and this allow us to see what is the y value, or Power value when the voltage is at 3 volts