



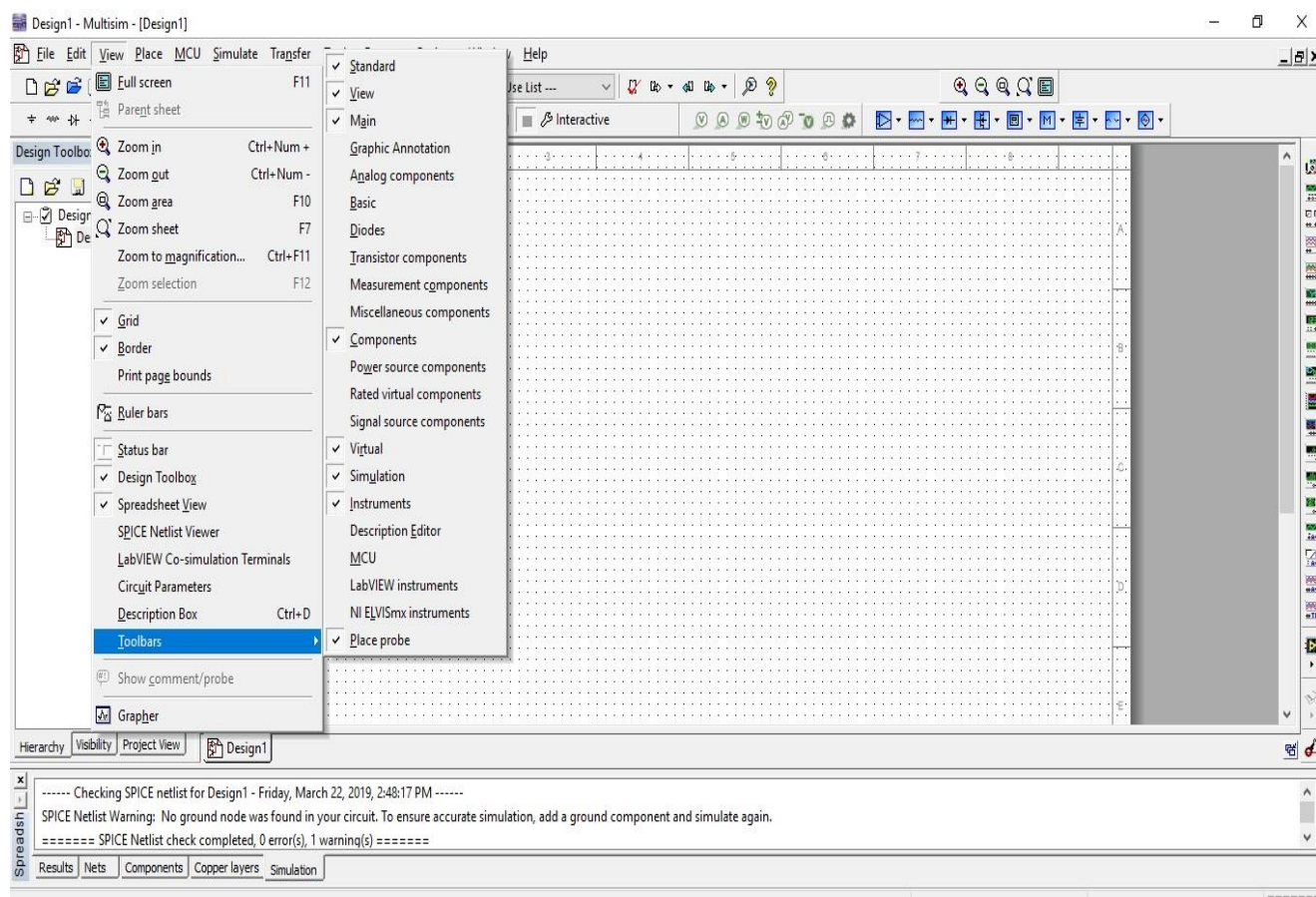
Introduction to MultiSim 14.0

Objectives:

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

Study:

- Open **MultiSim 14.0** by clicking on its icon.
- Or go to *Programs > National Instruments Circuit > Design Suite 14.0 > Multisim 14.0*
- If any toolbox did not show, you can go to *View > Toolbox*
- And check the desired toolbox

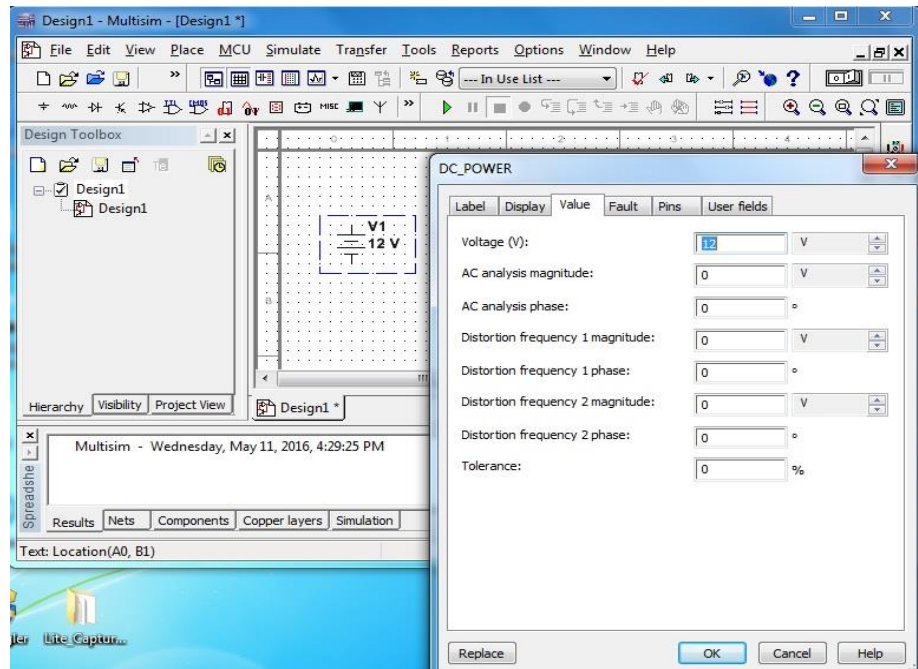


Unlike PSPICE, Multisim automatically starts a new project for you, and you can start drawing your circuit by placing the different components required.

Placing Components:

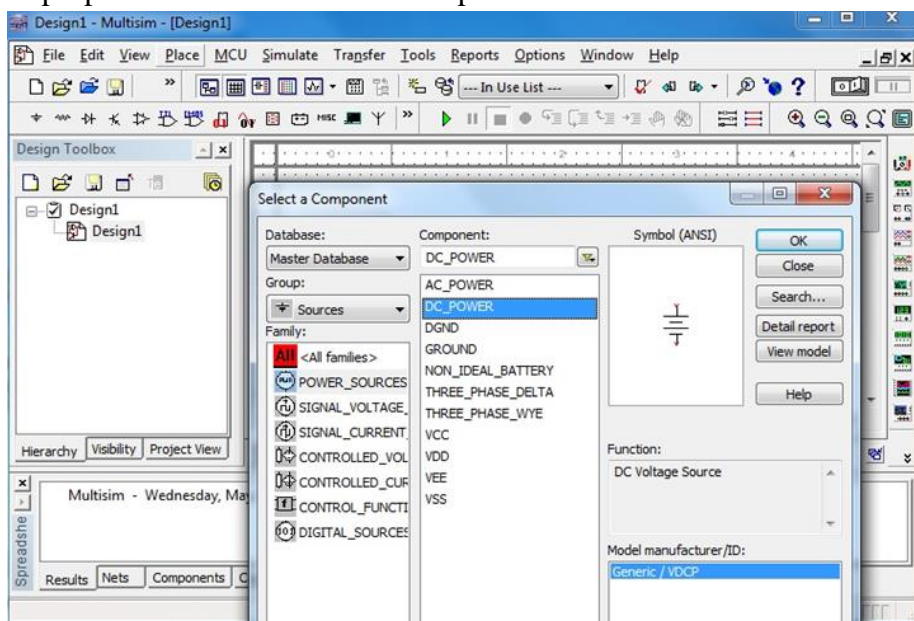
To place a new component, go to-
Place > Components

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

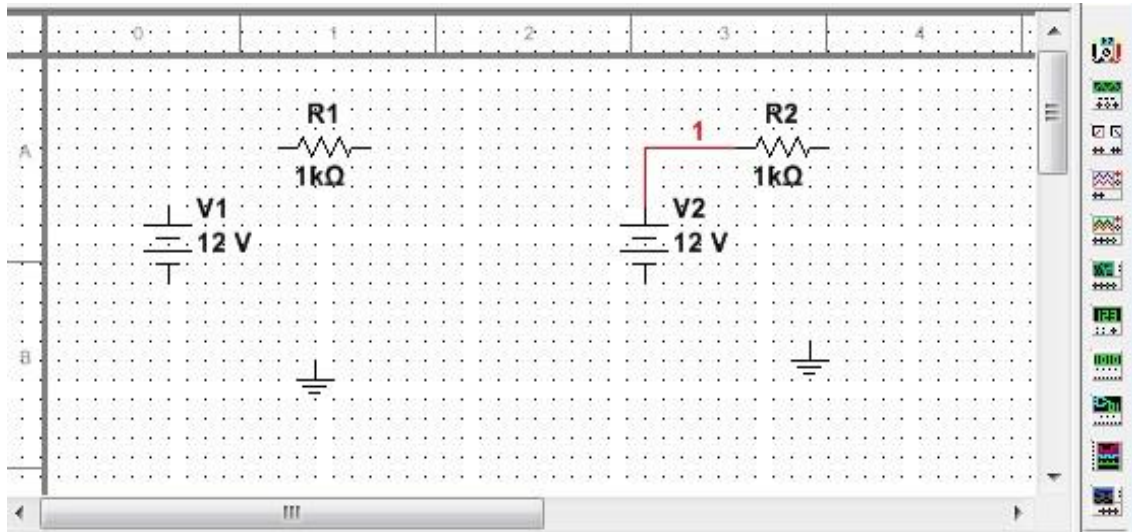


Changing Component Parameters:

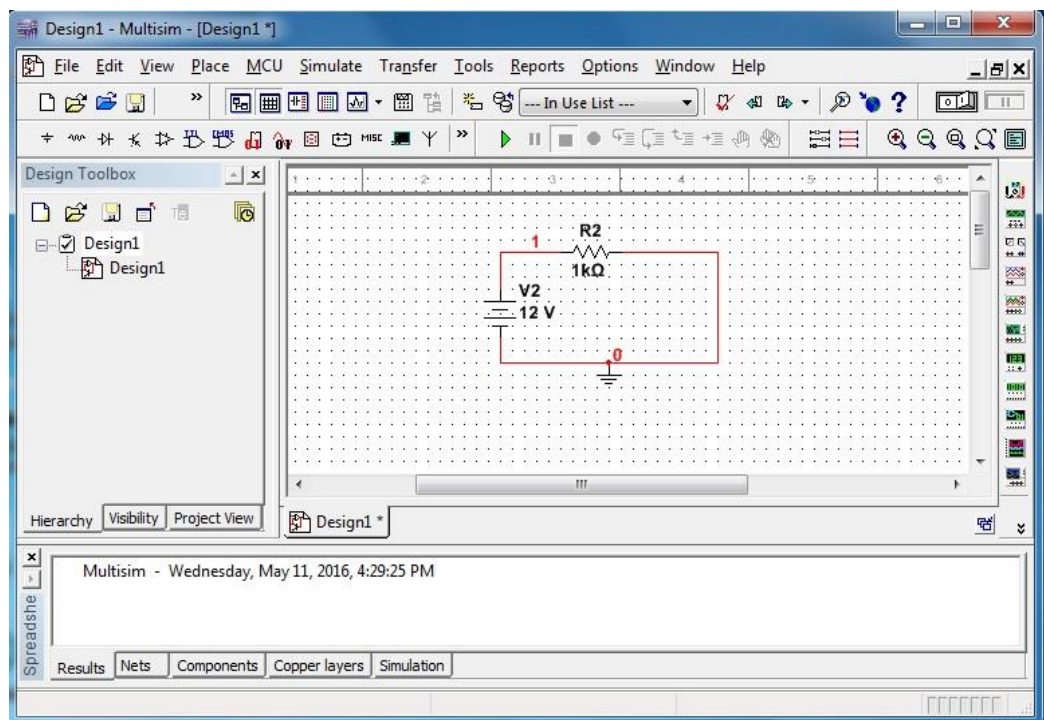
To change the values or names of the component, you can either right click on the component and select properties, or double click on the component, and a screen will pop up allowing you to change the properties and values of the component.



Wiring:

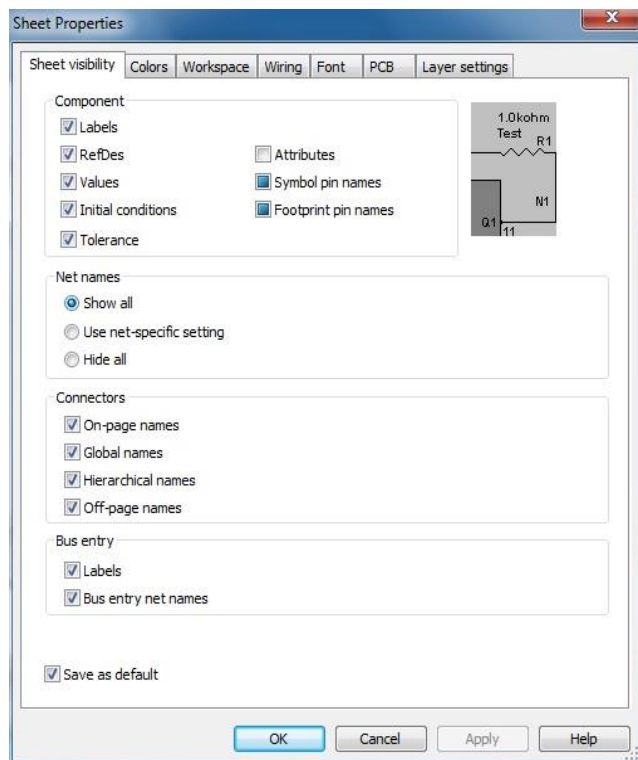
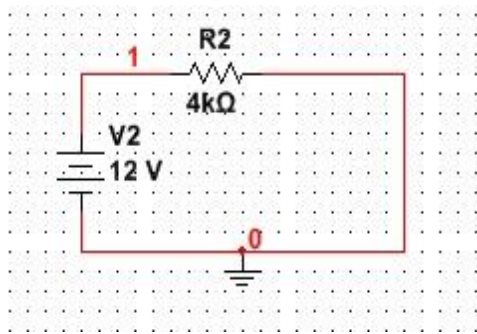


When wiring, place cursor at the corners of component and you should see a black dot appearing, click the mouse once and drag your wire to the edge of another component where a red dot should appear, click again to connect.



Displaying Node numbers/ Wire names:

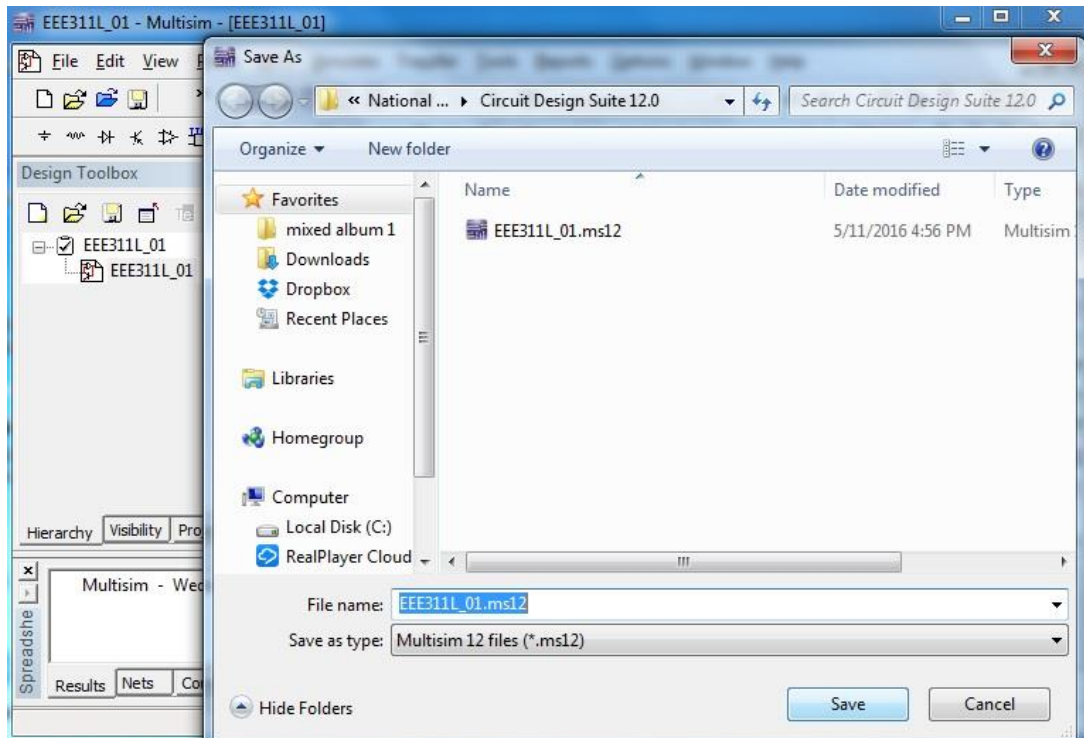
To show the net names of the wires, select the desired wire and right click Properties, and a Net Settings window should pop up. Checking the show net name box will allow one to see the wire names/net on the circuit.



One can also do *Options > Sheet Properties > Circuit > Net Names: check “Show All”*.

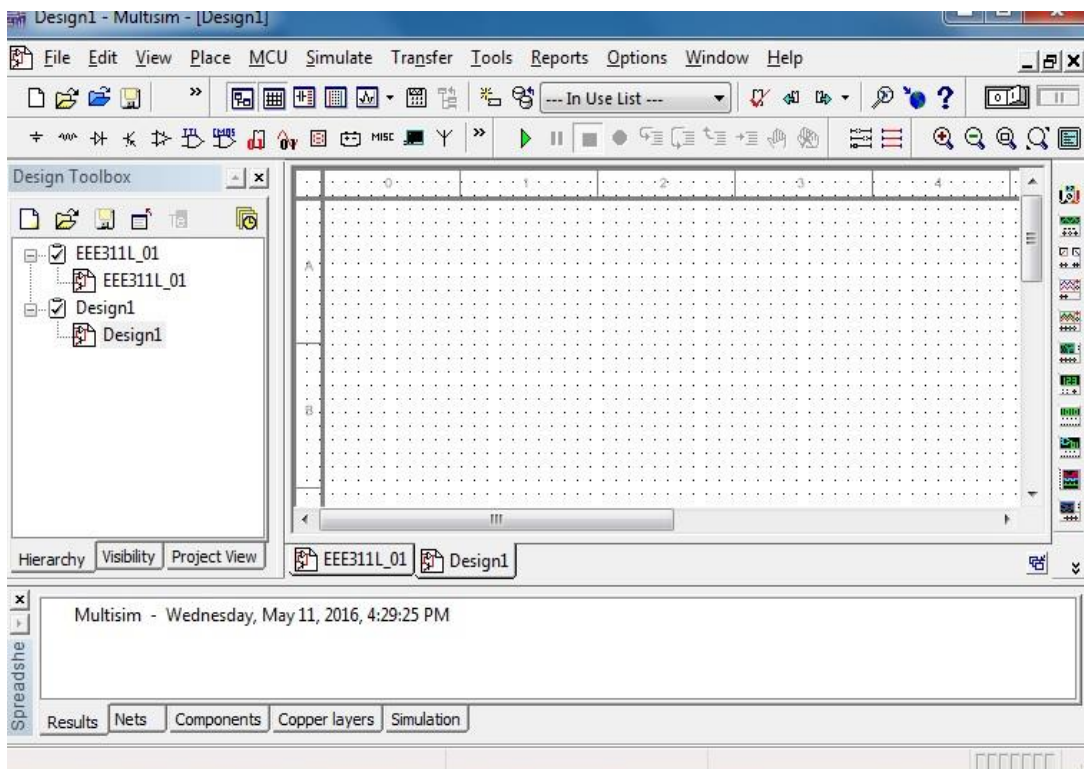
Saving:

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.

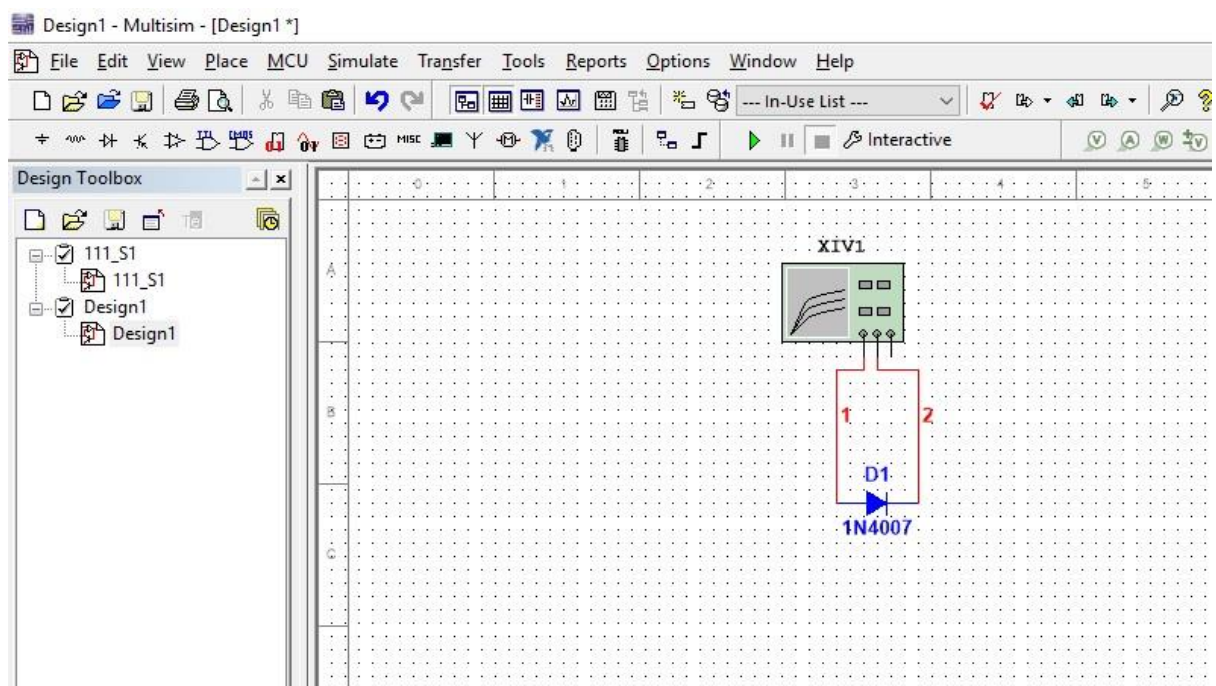
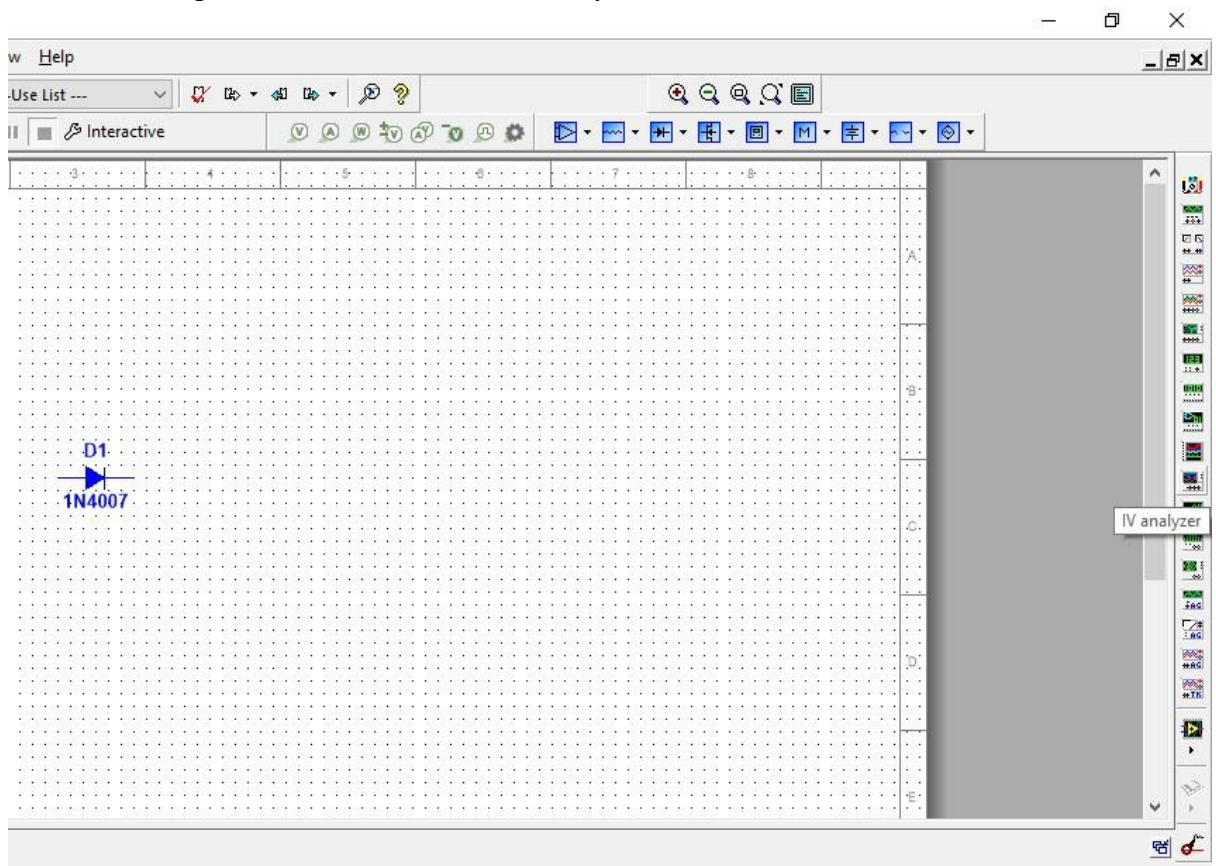
You can also create more than one project in one setting by-
File > New > Design



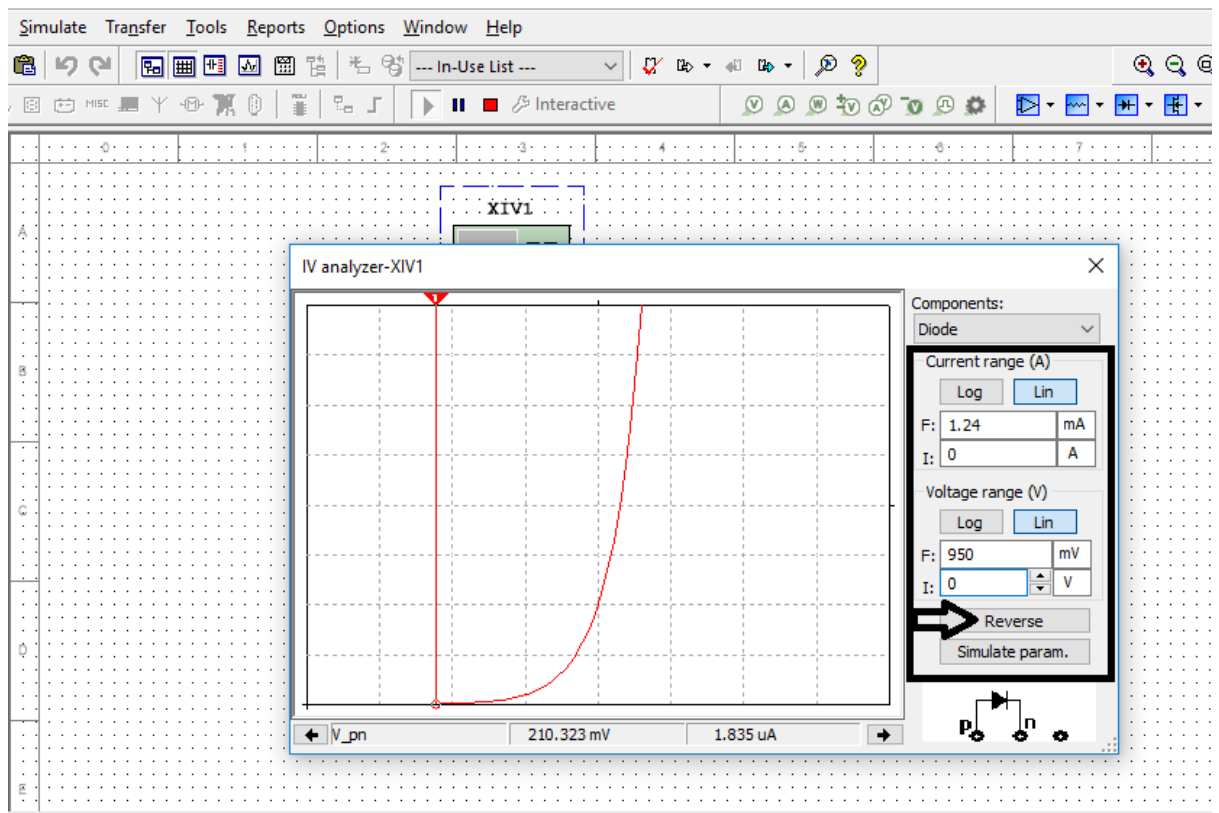
Simulation Analysis:

Experiment-1:

- ❖ You can use the IV analyzer to get the diode characteristics (I_d vs V_d) curve. To do so, first place the diode and the IV analyzer and connect them like this.

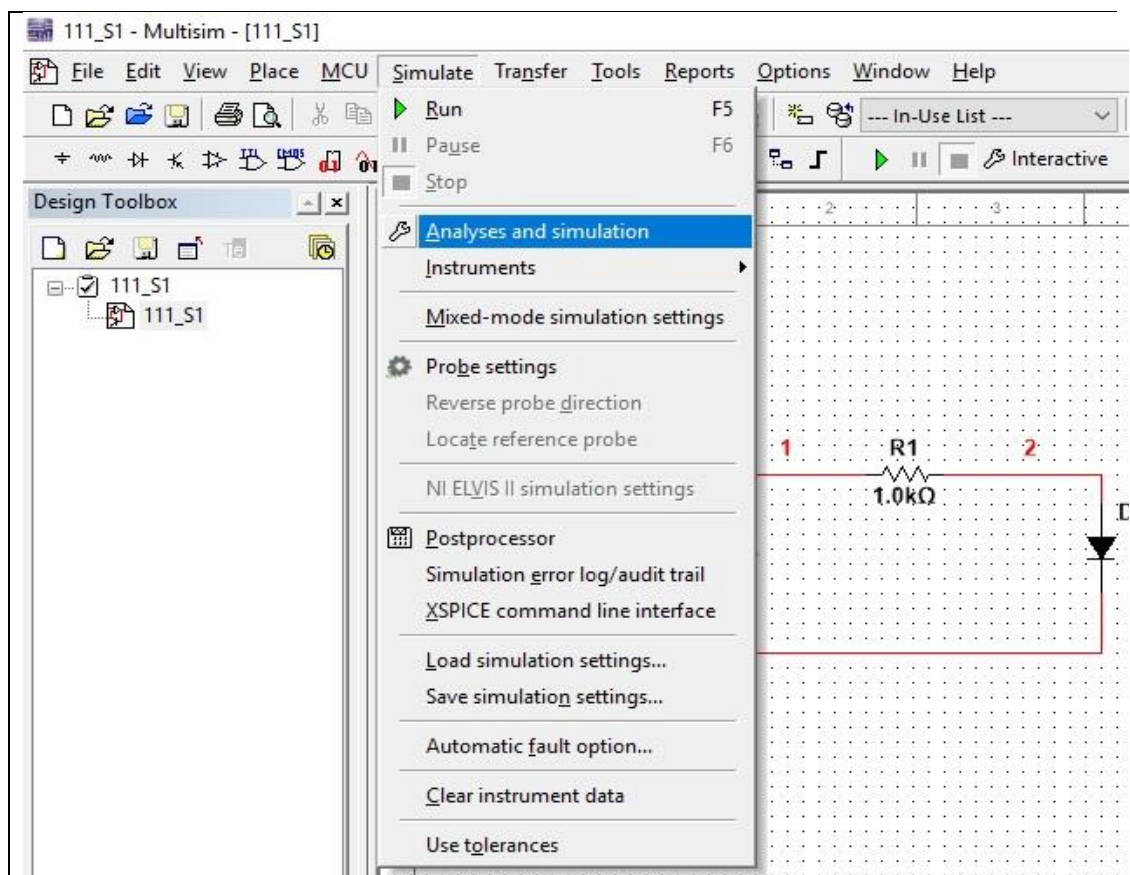


Then double click on the analyzer and set the following values before running the simulation.

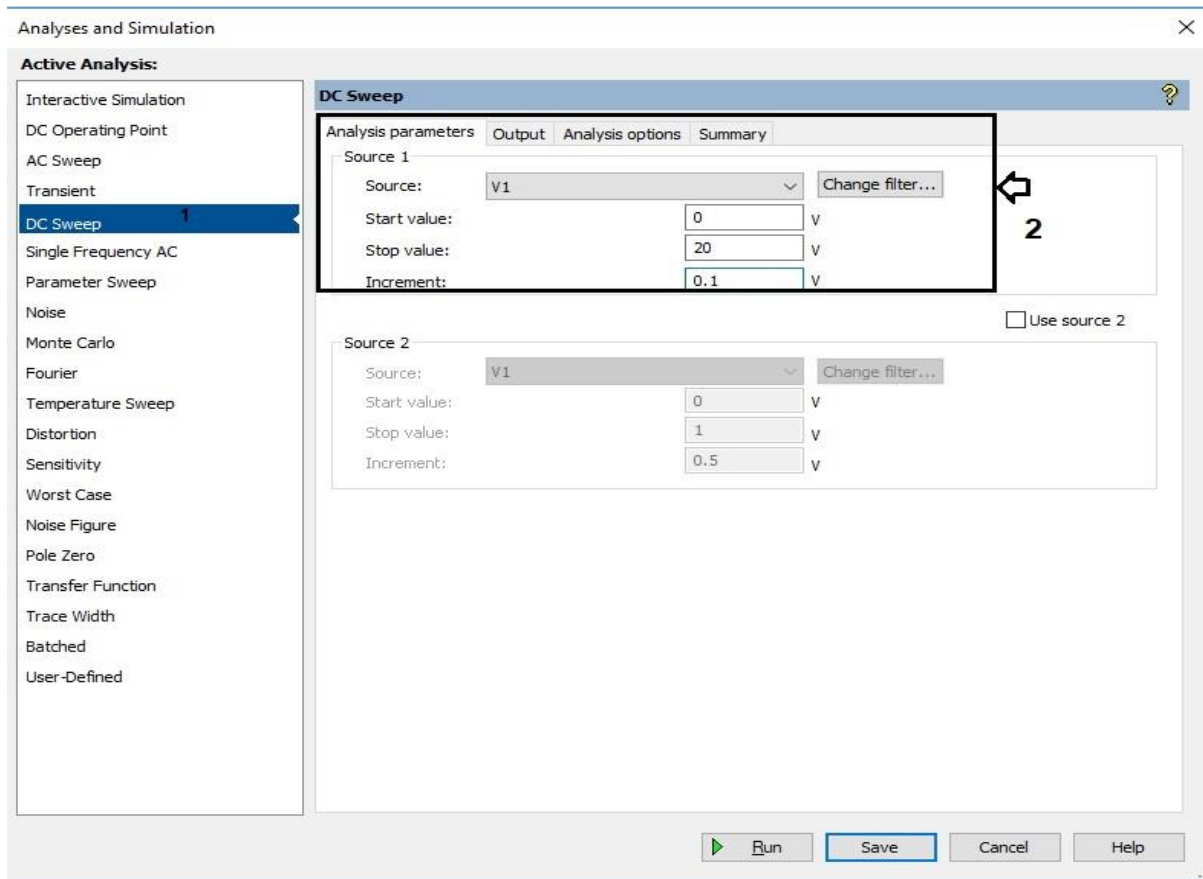


This curve represents the IV characteristics of the diode.

- ❖ We can also do **DC Sweep** to find out the Diode Voltage, Resistor Voltage and Diode Current. First construct the desired circuit and go to Simulate >> Analyses and simulation.

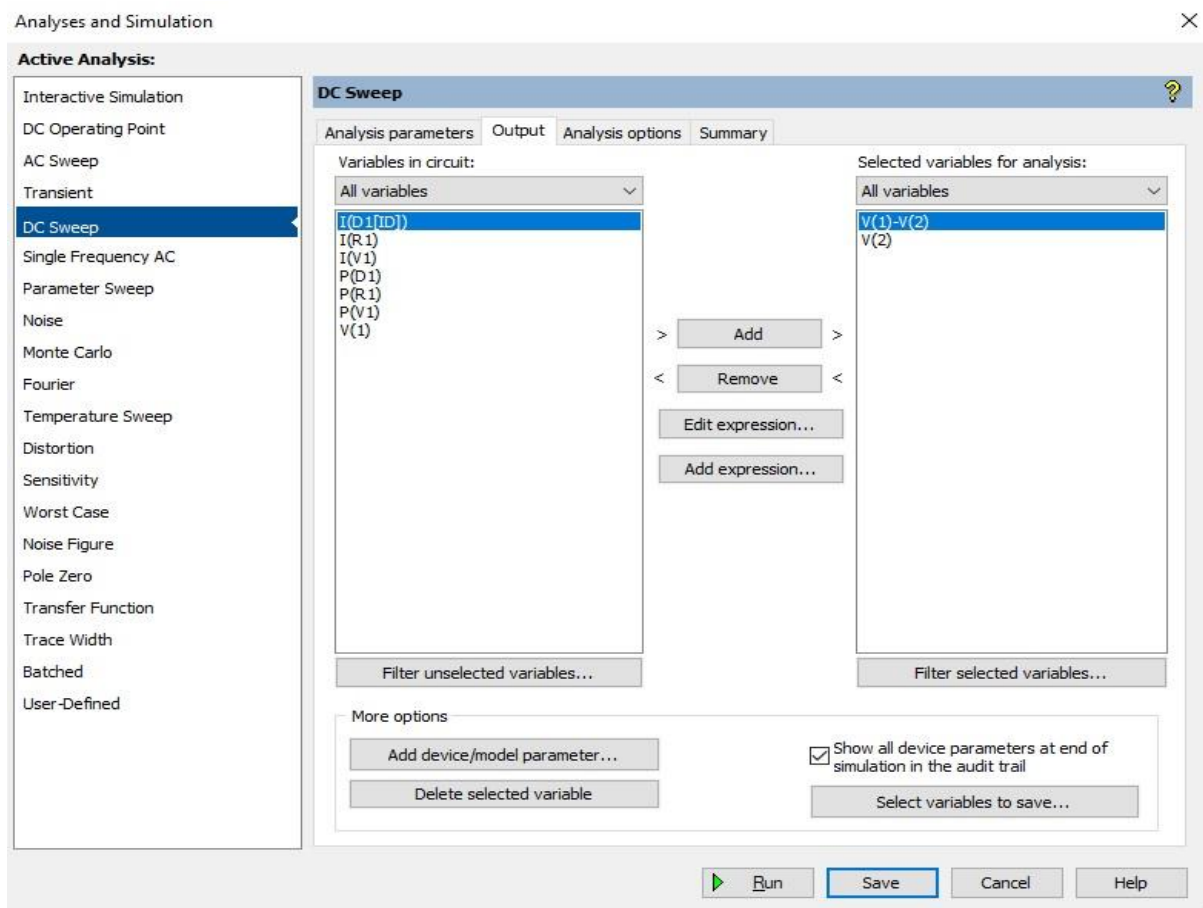


Then follow the following sequence:

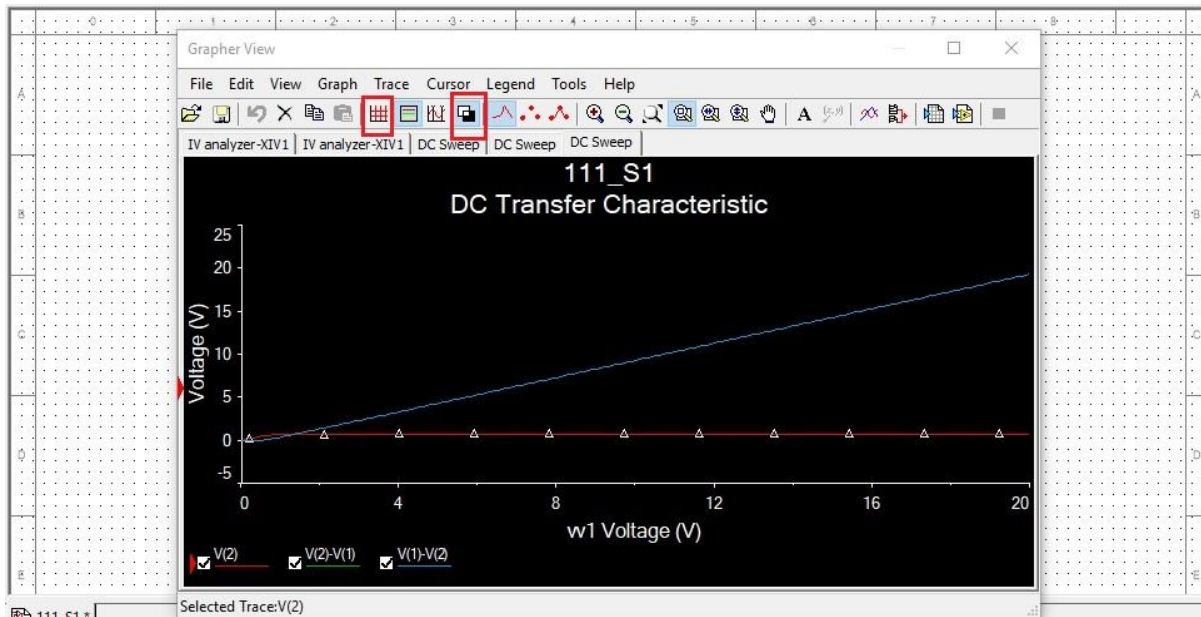


To see the V_R as Output, use the add expression method to subtract node 1 voltage from node 2 voltage and Node 2 voltage will be the diode voltage for the circuit drawn here.

➤ *Note that the node numbers may be different for your design/ circuit.*

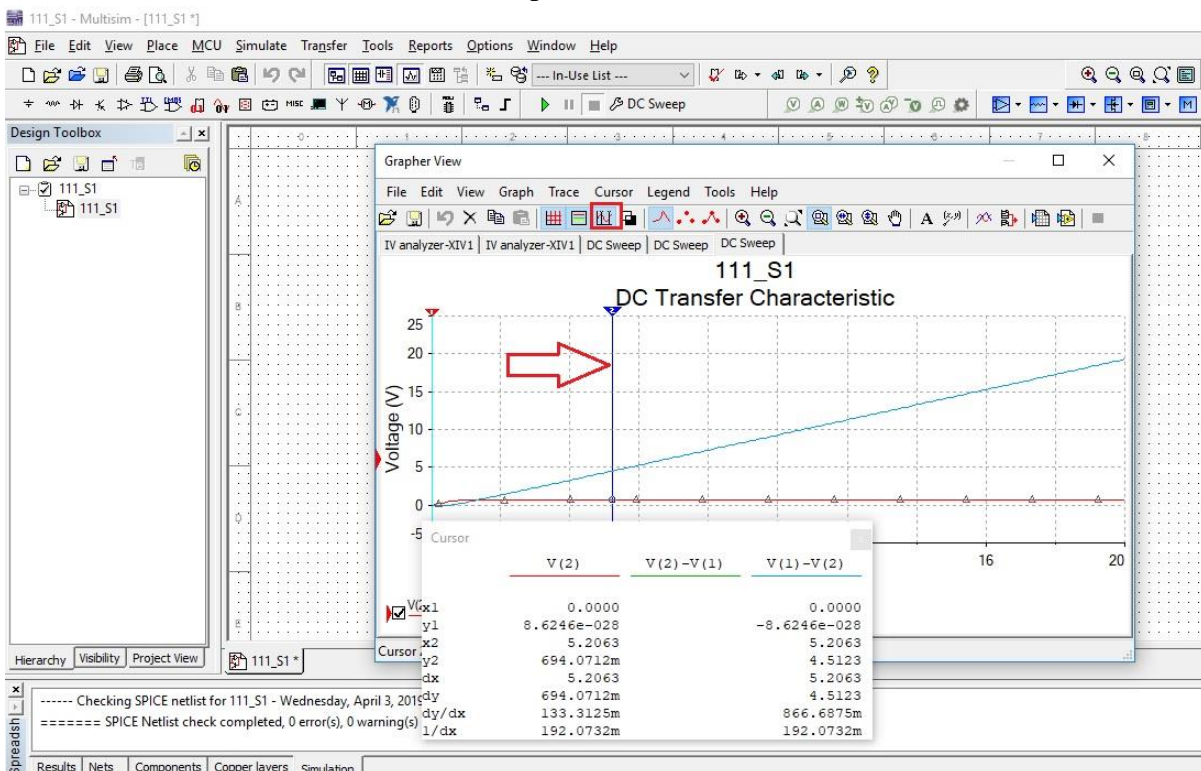


You will get the following output:



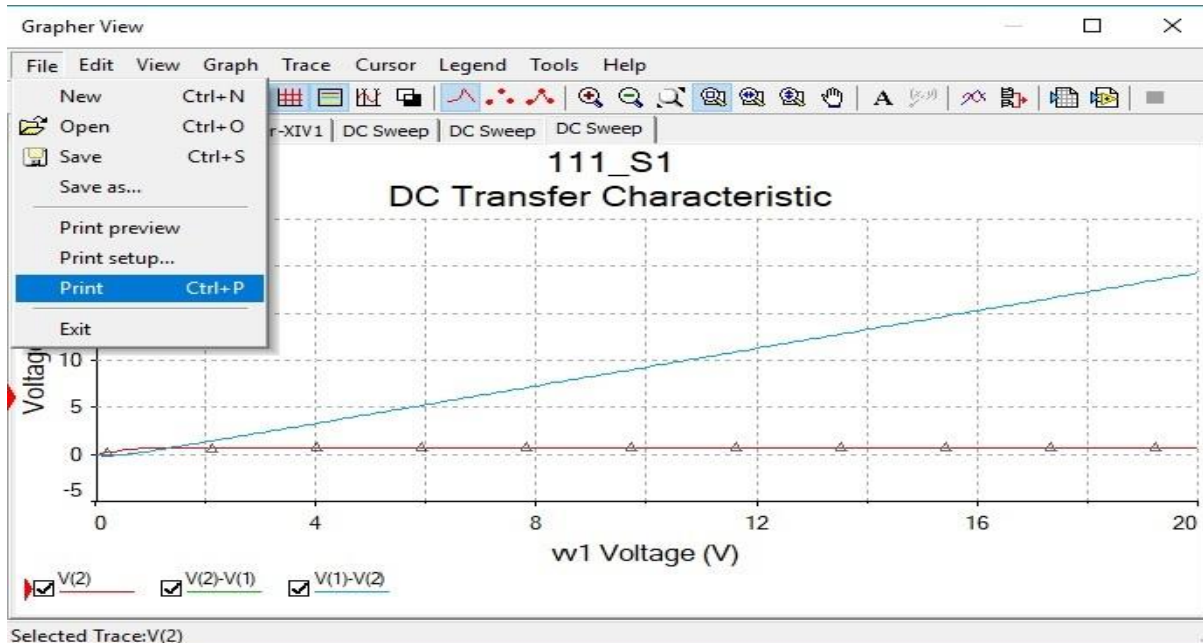
Click on the marked boxes to turn on the grid and to make the background white.

Also click on the cursor option to turn on the cursor and drag the cursor over your graph to see the Y-axis values for various X-axis points.



Printing the Output:

To get the PDF copy of your Output graphs, go to the file option of the Grapher View and select print. Check your desired graphs and finally use Microsoft print to PDF option to save the output graph(s) as PDF.

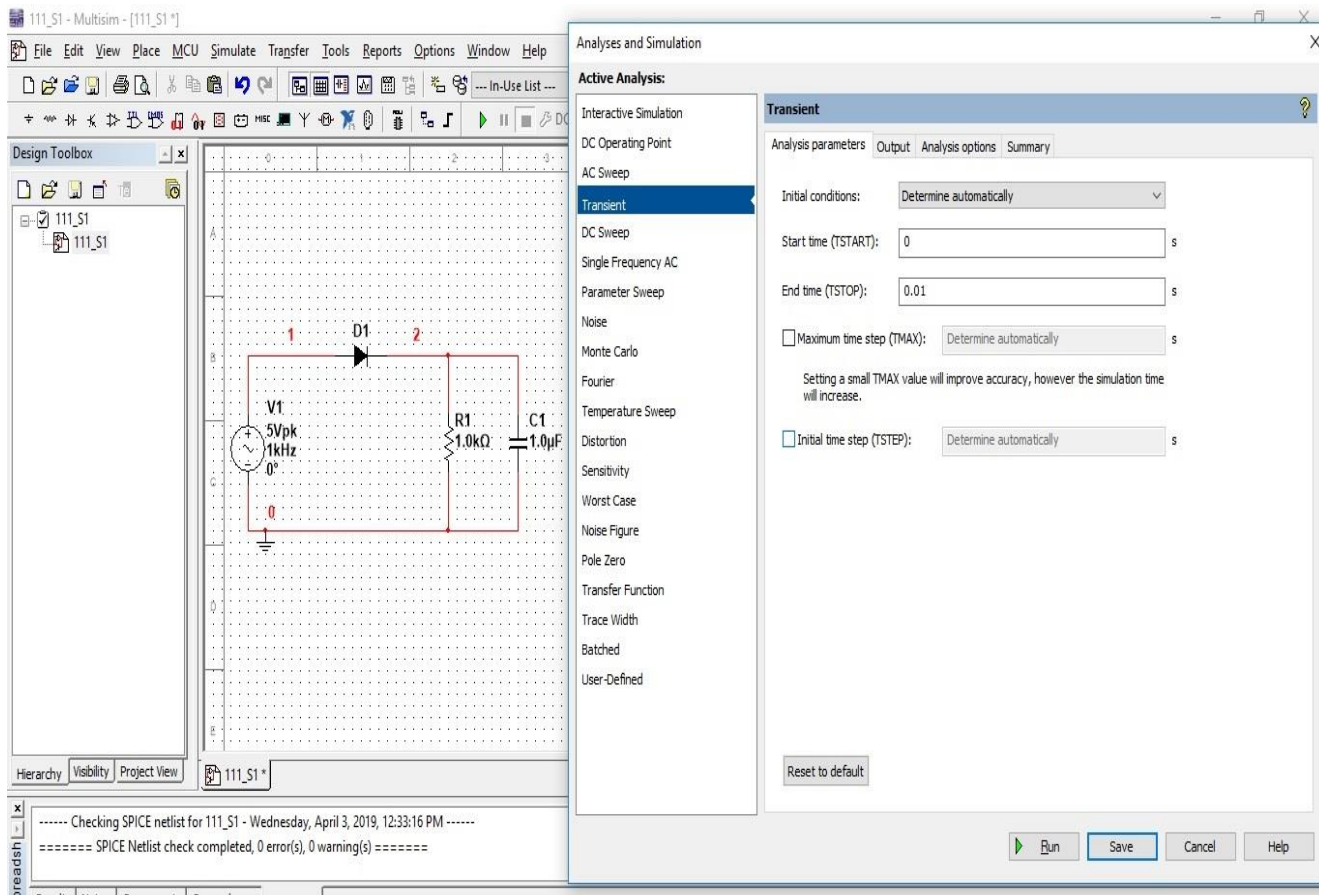


Experiment -2:

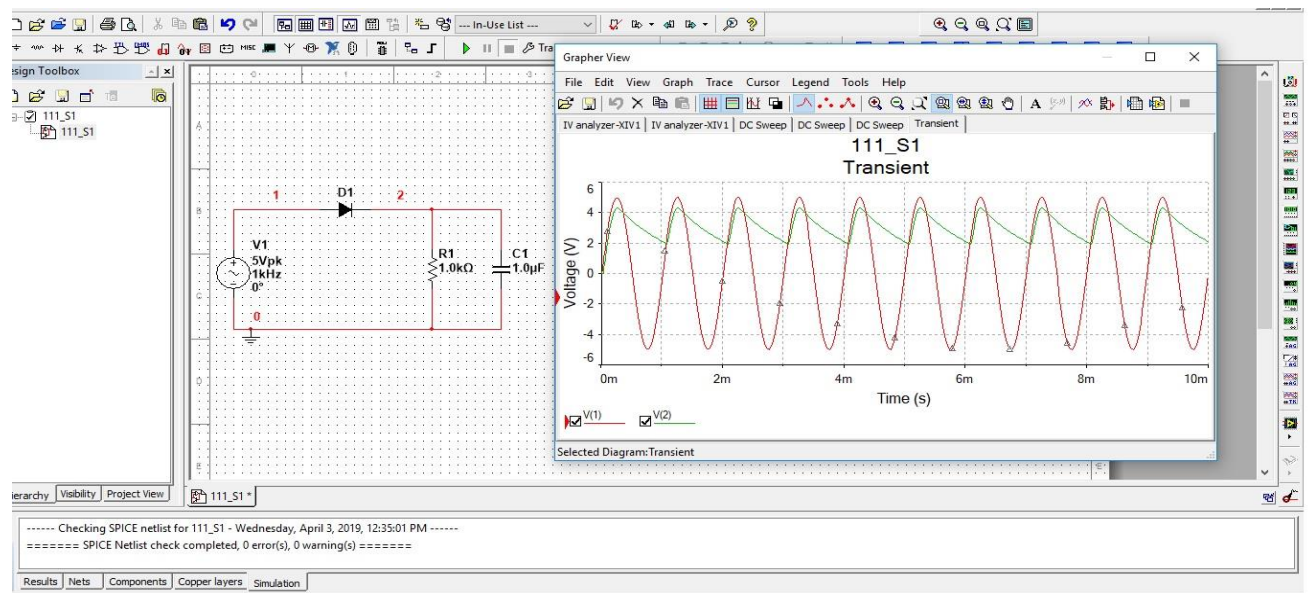
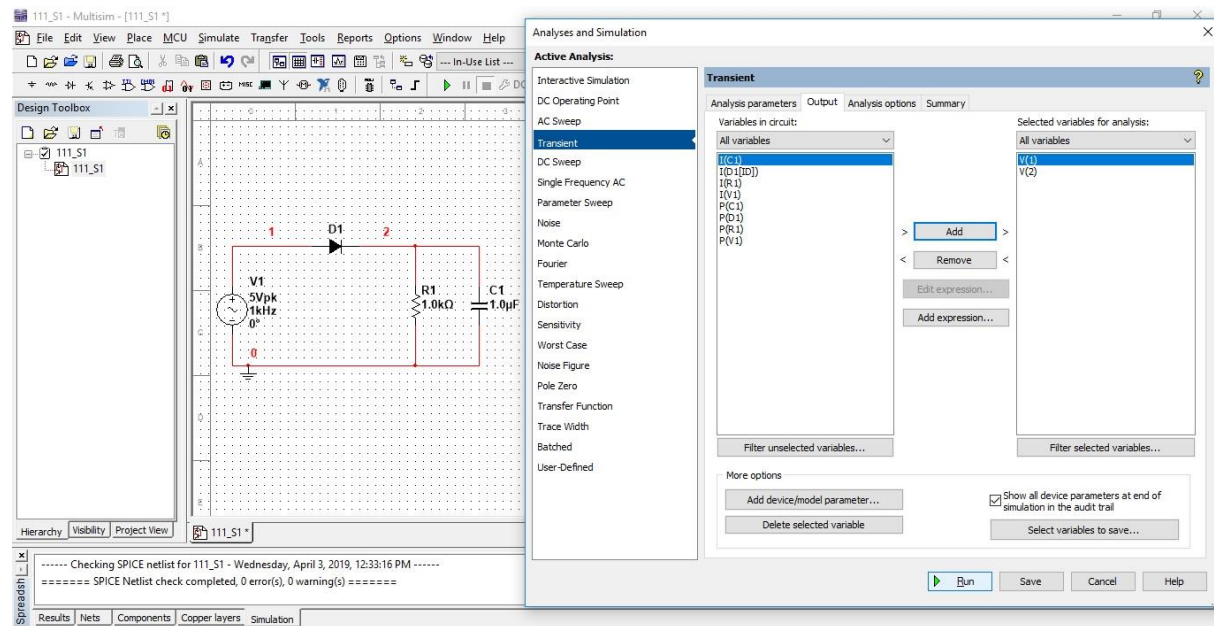
Please note that the component values shown here may vary from the values of your circuit.

Transient Analysis:

For an AC source, if both the input voltage and frequency remains unchanged over time, we use the simulation method, "Transient analysis" to show the circuit outputs.



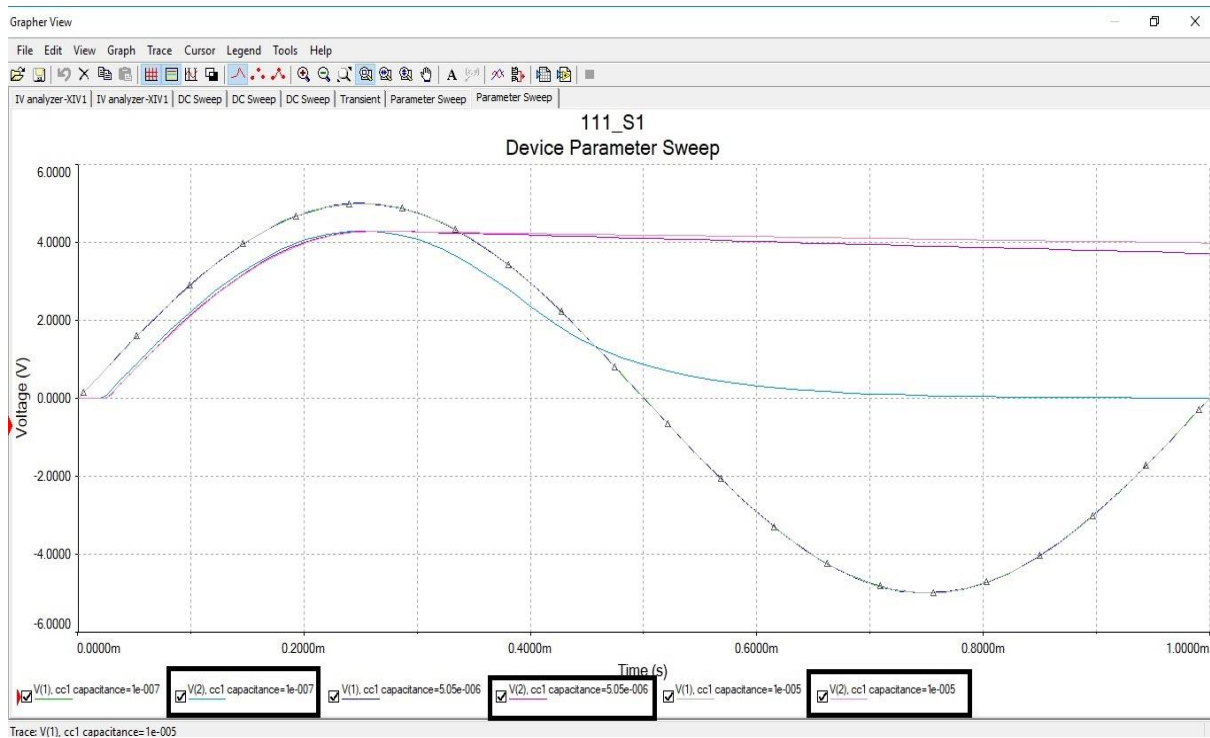
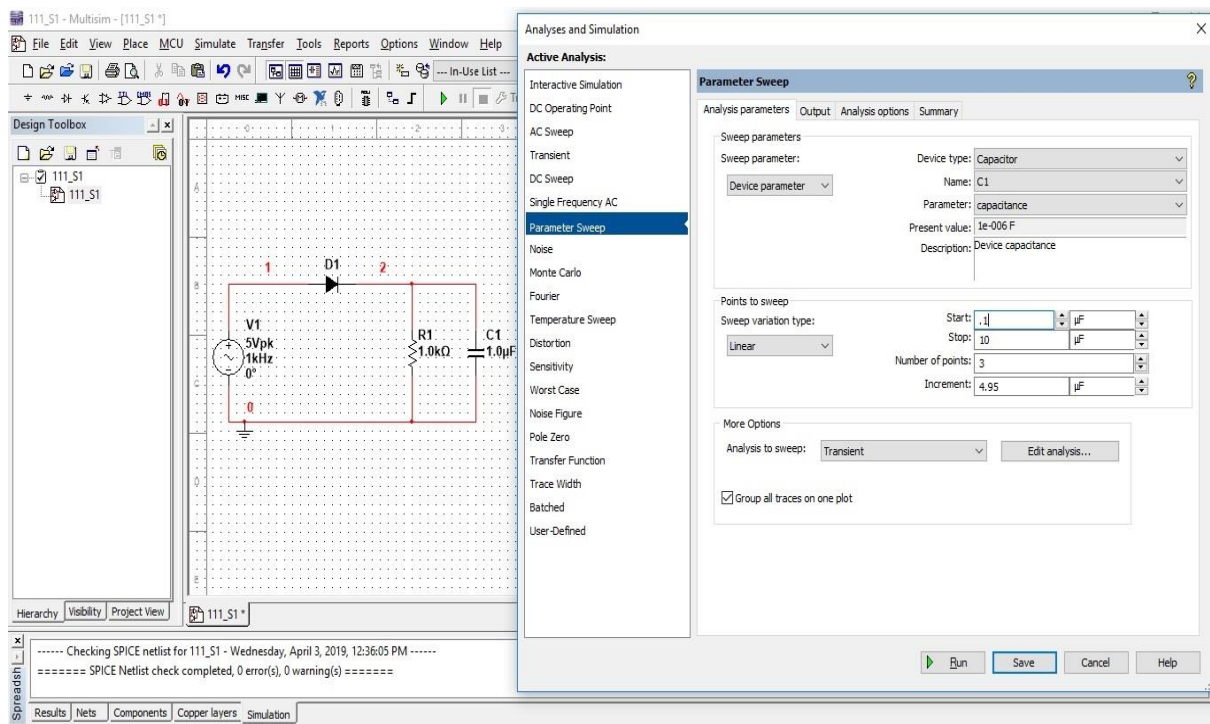
The “End Time (TSTOP)” should be given large enough to show multiple cycles of the output signal.



Parameter Sweep:

We use “parameter sweep” when we want to see the outputs for multiple values of the same circuit component. In this type of simulation the source(s) are kept as same as the set values. But the output shifts/varies by varying any single component values.

For example, in this Lab we used multiple capacitor values to get different outputs for the same input. This kind of cases can be simulated by using the method “Parameter Sweep”.

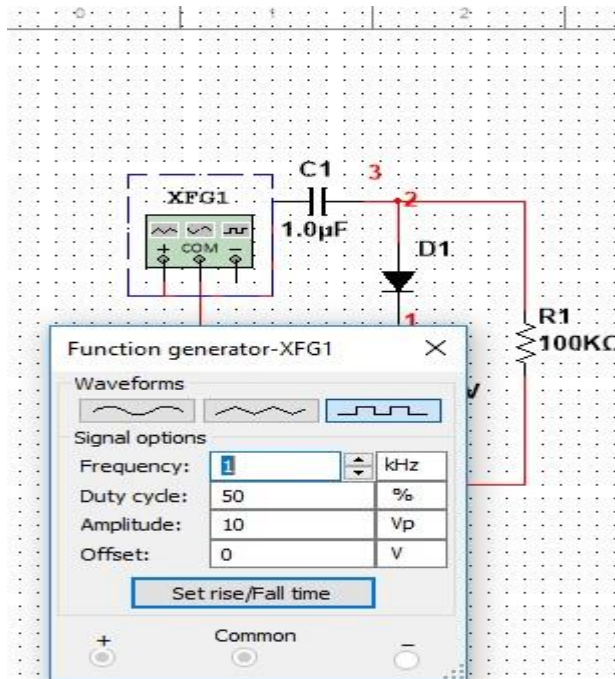


In the above picture, the boxed lines indicate the output voltage for various values of the capacitor. These help you understand the effect of the varying capacitor values on the output.

Experiment-3:

For the circuits of this experiment you will need to do a “Parameter Sweep” on the DC voltage source.

Use the function generator to get the desired type of signal for this simulations. Double click on the function generator to set the signal type and values.



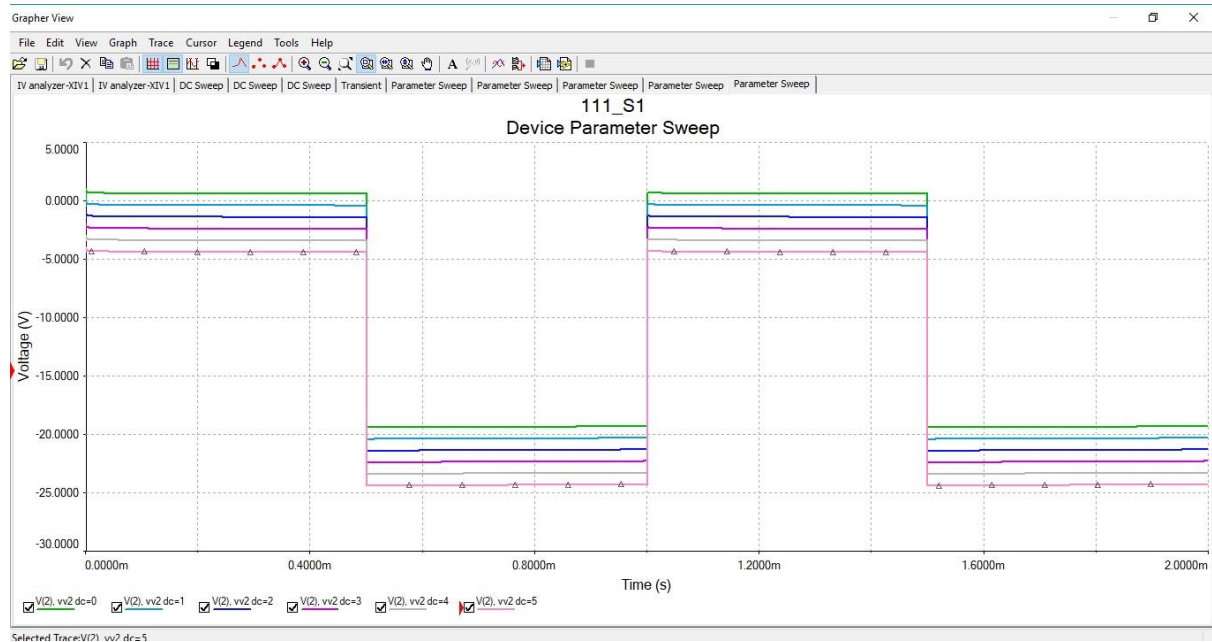
For the Parameter sweep, carefully set the Device type, Name and Parameter accordingly. Then set the start value, the end value and the total number of data points between them.

The image shows the "Parameter Sweep" dialog box in a simulation software. The "Active Analysis" list on the left includes "Parameter Sweep". The "Parameter Sweep" dialog box has the following settings:

- Analysis parameters: Output, Analysis options, Summary
- Sweep parameters:
 - Device type: Vsource
 - Sweep parameter: Device parameter
 - Name: V2
 - Parameter: dc
 - Present value: 2.5 V
 - Description: DC value of source
- Points to sweep:
 - Sweep variation type: Linear
 - Start: 0 V
 - Stop: 5 V
 - Number of points: 6
 - Increment: 1 V
- More Options:
 - Analysis to sweep: Transient
 - Group all traces on one plot: ☒

The bottom of the dialog box has buttons for Run, Save, Cancel, and Help.

The output will be similar to this one. From this output, you can clearly see that V_{p-p} remains unchanged here but V_{max} varies as we vary the DC voltage source.



Experiment-4:

For this experiment, do a DC sweep for the circuit of figure 4.3 of your lab manual.

Do a Parameter sweep for the circuit of figure 4.4 of your lab manual. For the Parameter sweep, carefully set the Device type, Name and Parameter accordingly. Then set the start value, the end value and the total number of data points between them.

Do a DC sweep for the circuit of figure 4.5 of your lab manual.

Experiment-5:

Use 2N2222 BJT to simulate the circuits.

For, input characteristics, set $V_{CC}=1.3V$ to get $V_{CE}=1V$ and $V_{CC}=5.3V$ to get $V_{CE}=5V$

For, output characteristics, set $V_{BB}=2.8V$ to get $I_B=20\mu A$ and $V_{BB}=3.027V$ to get $I_B=30\mu A$

Apply DC sweep for both the circuits.

Experiment-6:

As shown in the class.

For Fig 1 and Fig 2, apply the simulation method “DC Operating Point” to get the values for I_C and V_{CE} .

Beta value changing procedure is given in the lab manual.

For the last circuit, for every value of R_4 , apply “Single frequency AC” analysis method to get I_B , I_C and V_{CE} . Also do “Transient analysis” to get the waveform across R_5 (Output).