# LTspice Tutorial

By Dubari Borah

**Under the supervision of Dr. Dimitra Psychogiou and Dr. Zoya Popovic**

ECE Department, University of Colorado,Boulder

# Get started with LTspice

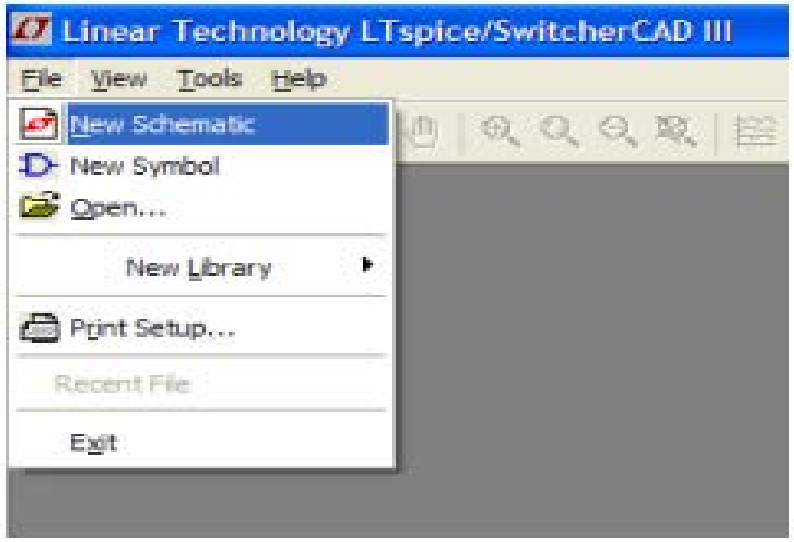
- Download and Install LTspice

Go to http://www.linear.com/designtools/software/ and you can follow the link https://www.youtube.com/watch?v=OTp9PP_qDcs to download and install the video.

# Drafting a Design using the Schematic editor

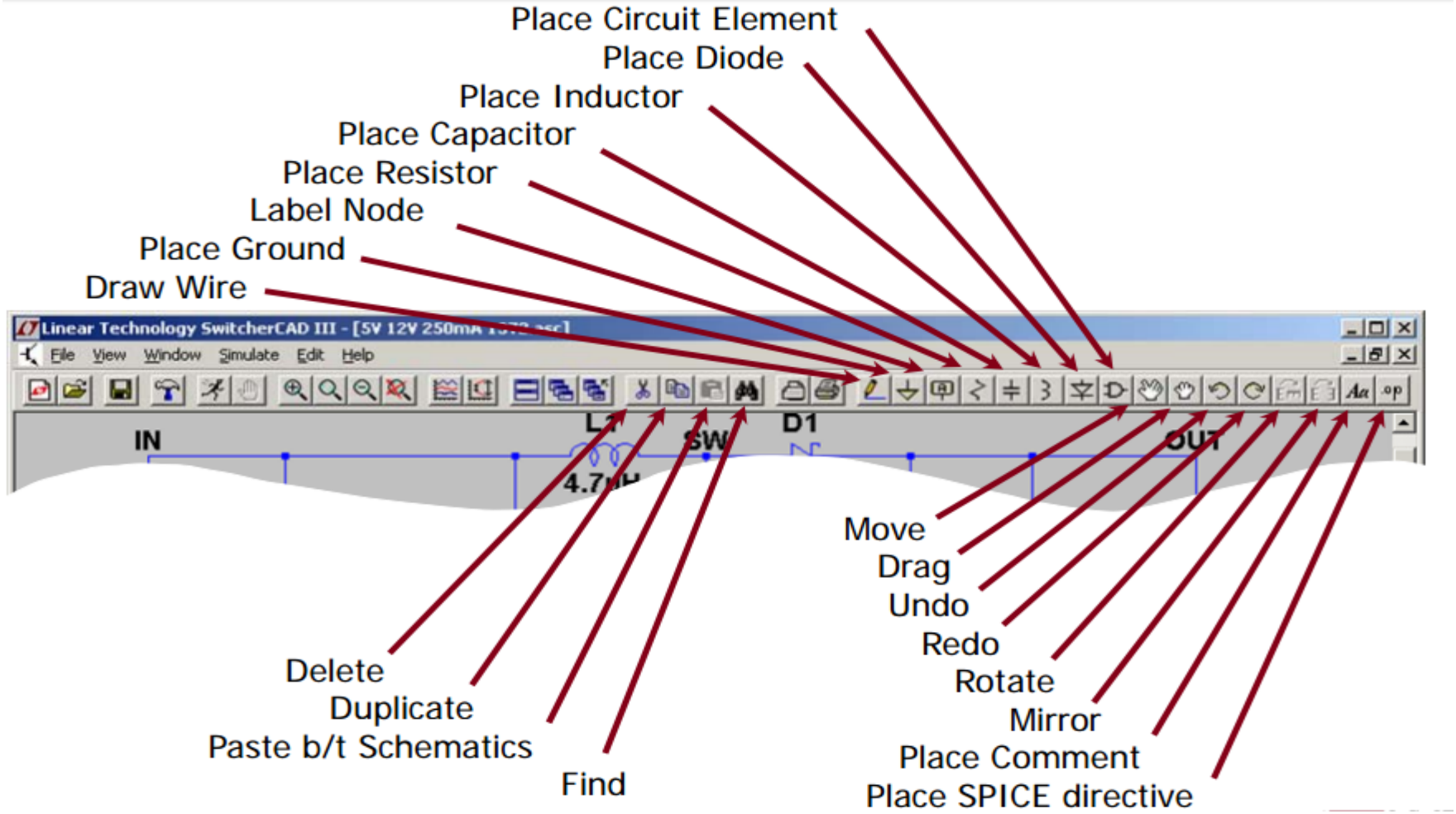
1. Create a new Schematic

⇒Select ‘File’ and ‘New Schematic’.

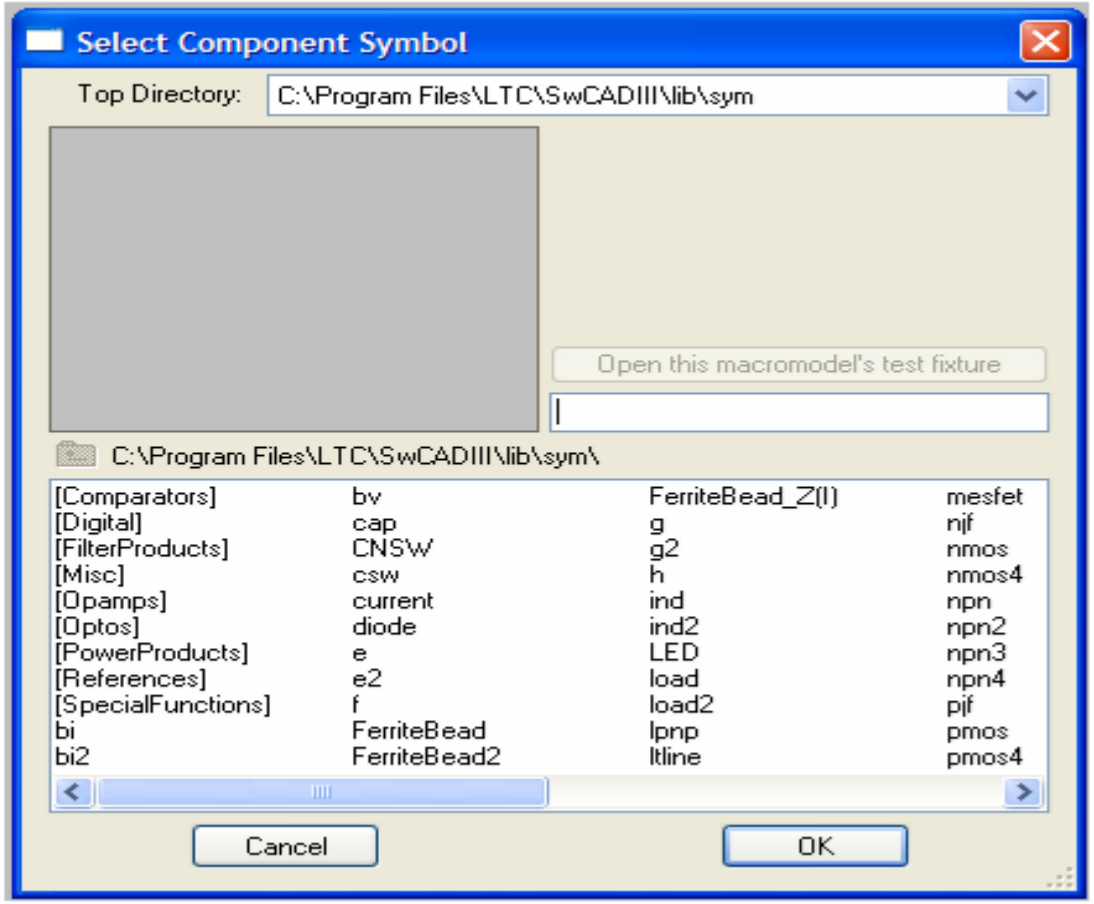


# 2. Add a Component

- To add a resistor, press ‘R’ or click the resistor button.
- Press ‘L’ or ‘C’ to insert an inductor or a capacitor respectively. Otherwise, click the respective buttons.

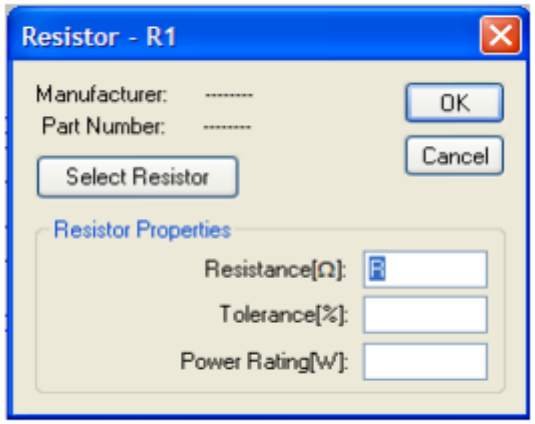


- For any other components, press F2 or click the component button. A ‘Select Component Symbol’ window will open and then from there, browse to get the desired component.

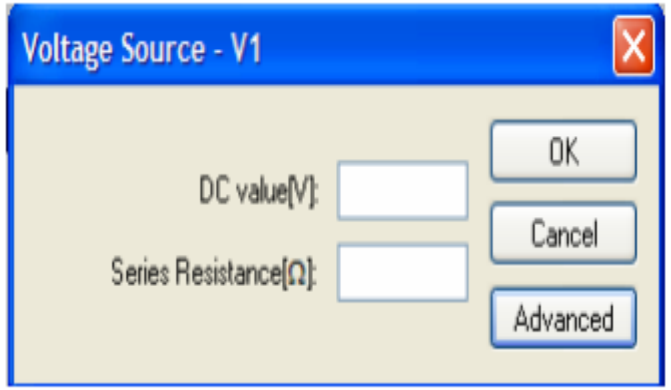


# 3.Modify Component Values

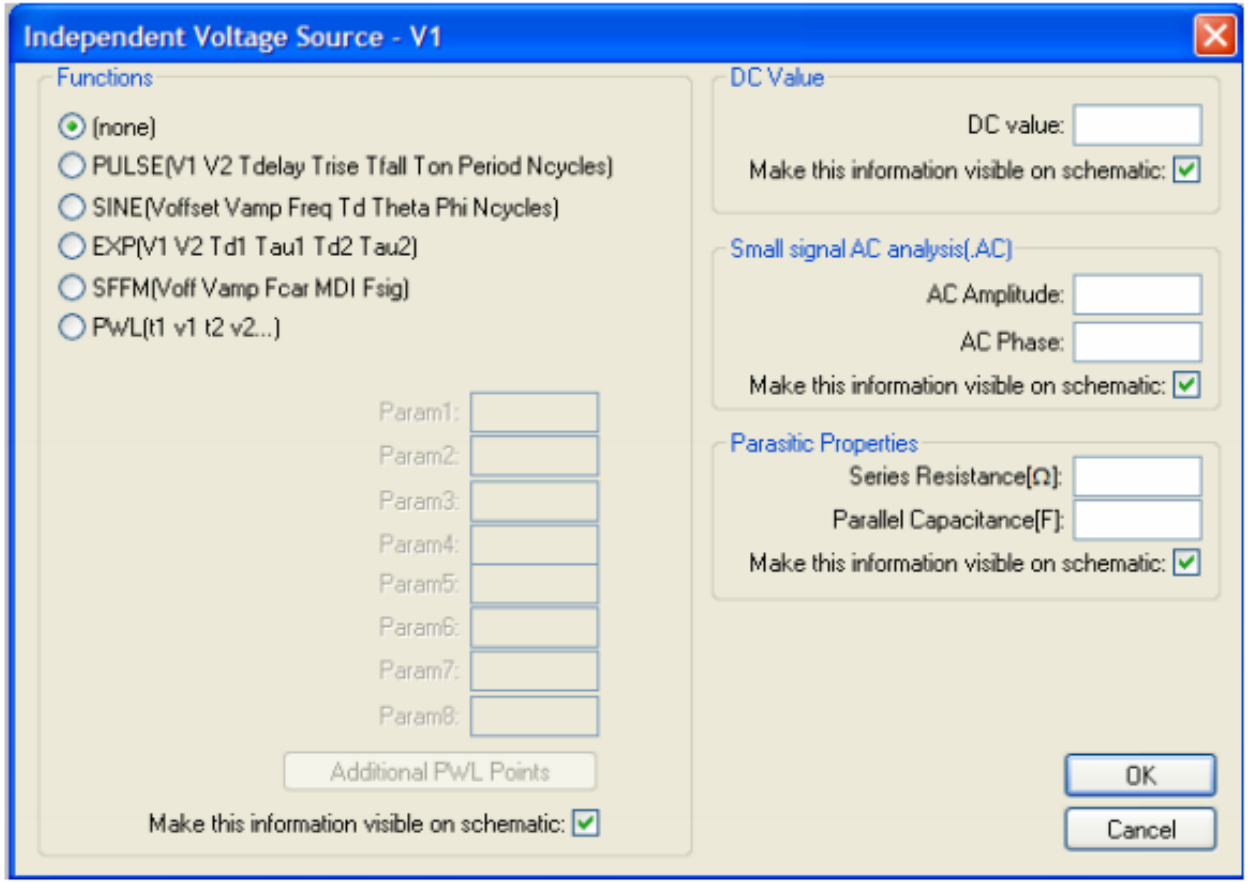
- Right click on the component and enter the value in default unit (Ω, H, F etc.)



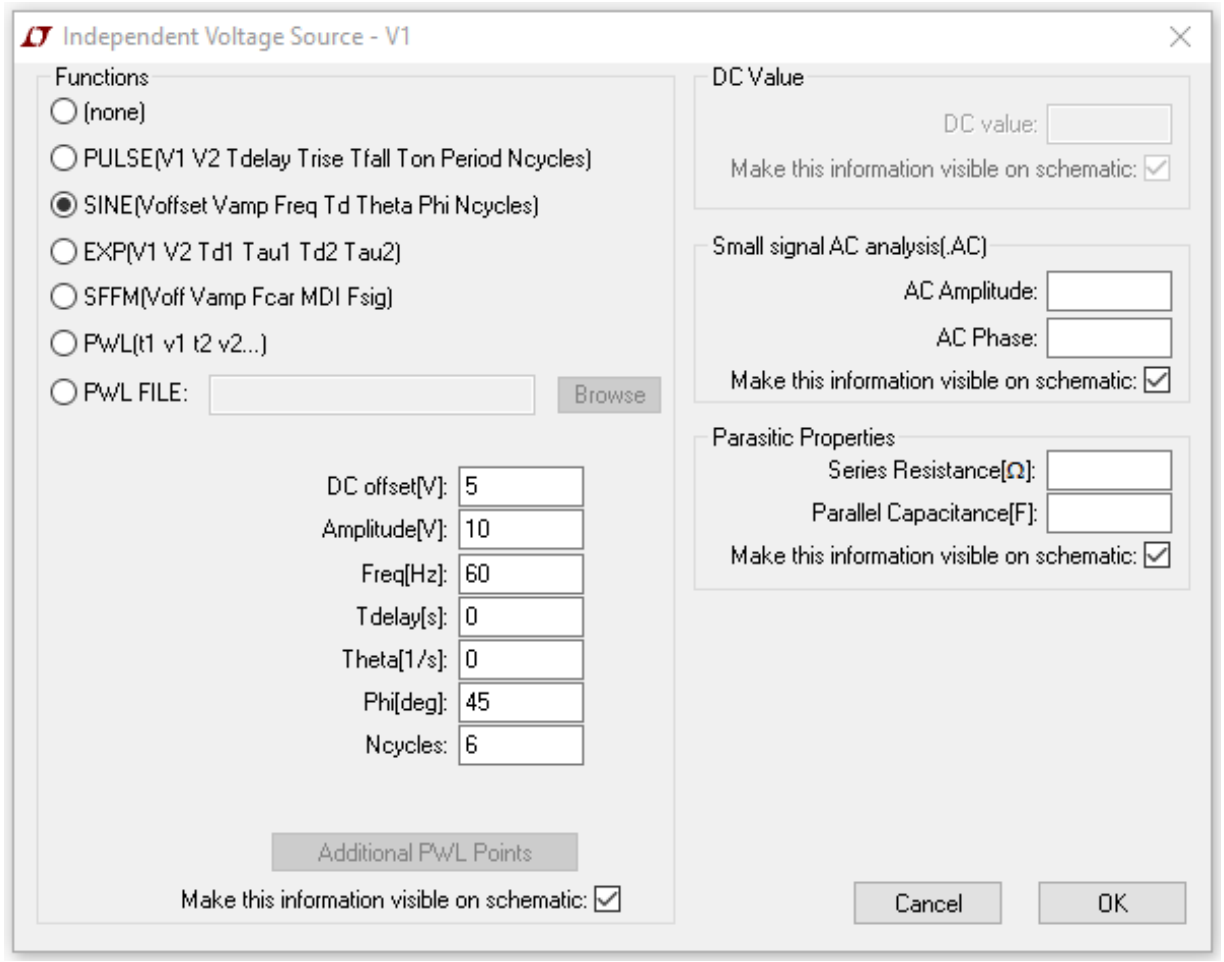

- For a DC Voltage source, enter the voltage and series resistance.



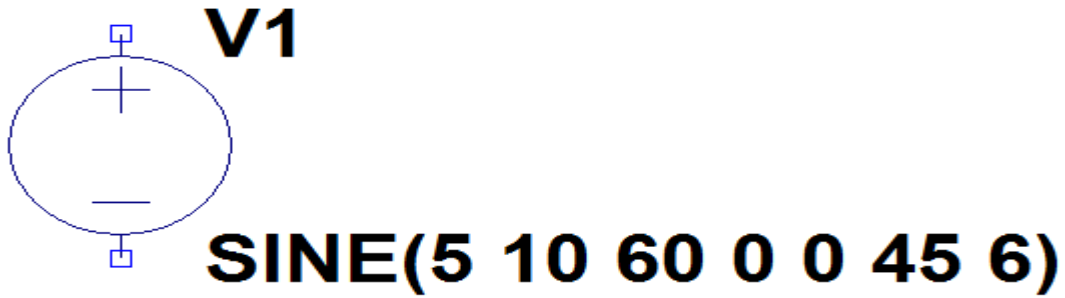
For all other voltage sources, click ‘Advanced’ and select the type of voltage source and enter the required parameters.



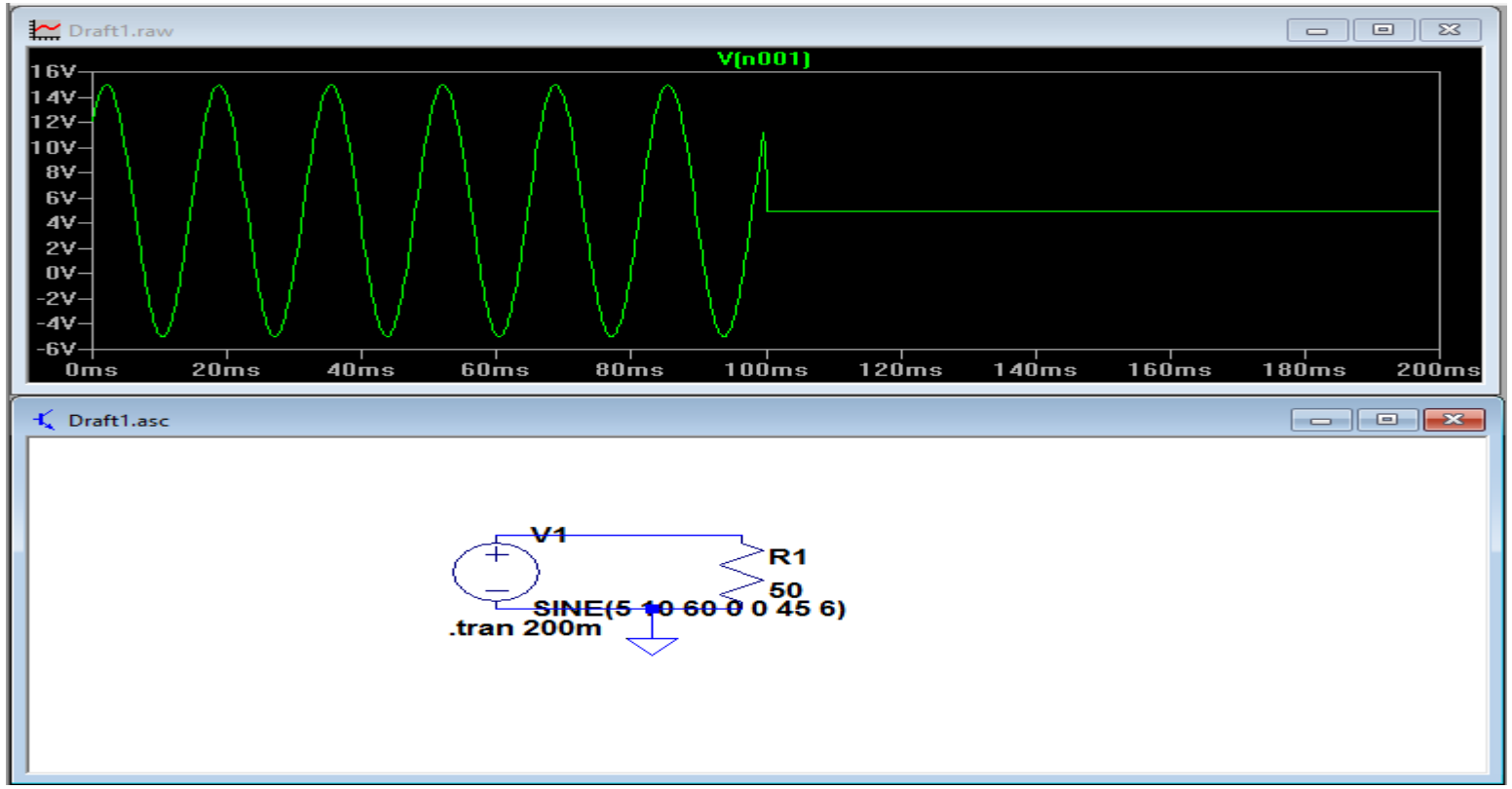
Example 1: simulate v(t)=$5 + 10\sin(2 * \pi * 60 + 45^{\circ})$V for 100ms as a transient analysis.



- This information will be seen on the schematic view as follows and to edit the input information, you can right click on the component as before.




- Now, this source is connected to a 50Ω load to see the waveform as shown below.



## 4. **Delete a Component**

- Click the scissors in the toolbar and press ‘esc’ to quit deleting.

## 5. **Move an element**

- To move a component, click the move button and left click on the component you want to move. Move the component to the new location and left click to place the component. Press ‘esc’ to quit moving components.

## 6. Add a wire

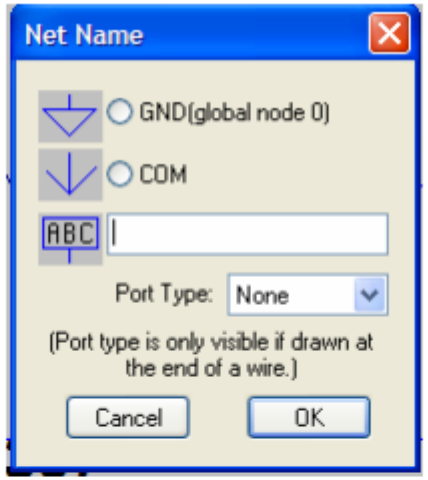
- Click the add wire button. Left click on the starting location, move to the ending location and left click again.  Press ‘esc’ to quit adding wires.

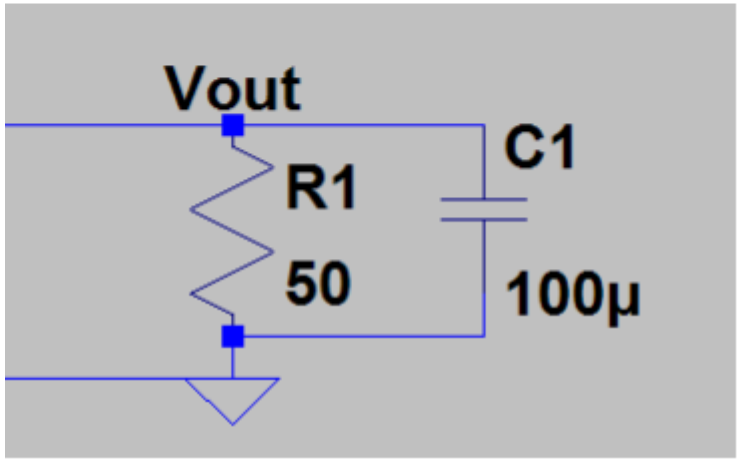
## 7. Add ground

- Press ‘G’ or click the ground button and click on the schematic to add a ground. Press ‘esc’ to quit adding grounds.

# 8.Label a net

- Press ‘F4’ or click the add net button. Type the net name and click ‘OK’. Click to add the net to the schematic.

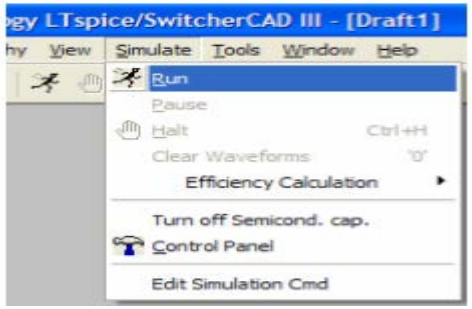


# Run and Probe a Circuit

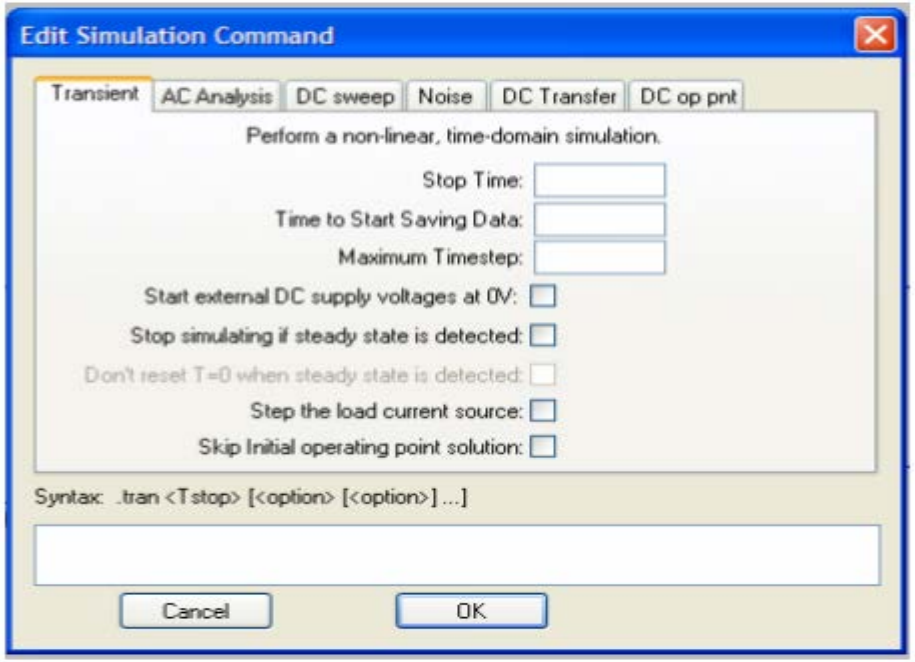
## 8.Simulate

- To run a circuit, select ‘Simulate’ from the file menu and ‘Run’ or click the run button.




## 9. Analysis

- Select the appropriate tab for simulation and enter the simulation options and click ‘OK’.



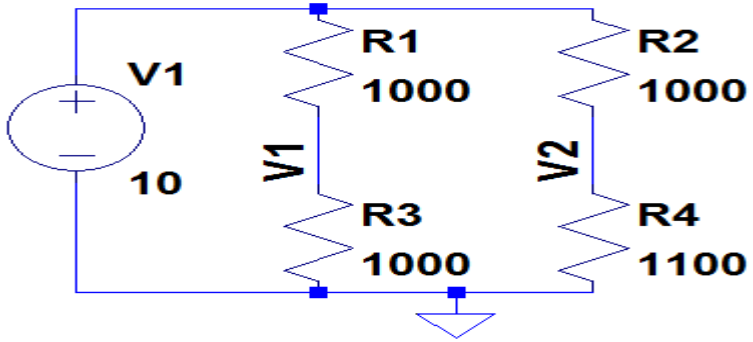
# 10.Measurement

- When making any measurement, a single click will add the measurement to the plot, while a double click will erase all existing measurements and plot the selected measurement by itself.
- To measure a voltage on a node with respect to ground, move the mouse over the node to be measured and click. The voltmeter probe cursor will appear when the mouse cursor is above an appropriate node.

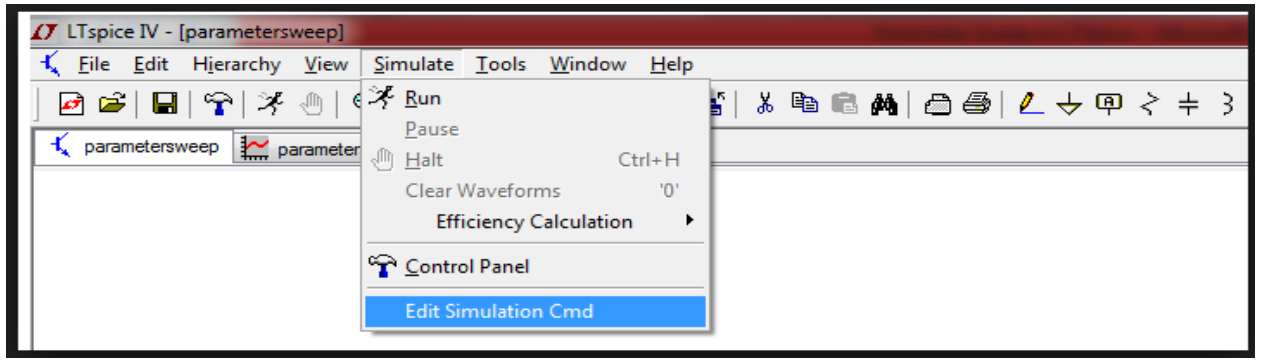
- To measure voltage across a component, click on the positive node to be measured, drag the mouse to the negative node and release the mouse.

- To measure the current through a component, move the mouse cursor over the component and click. The clamp ammeter will appear when the mouse cursor is above an appropriate component.

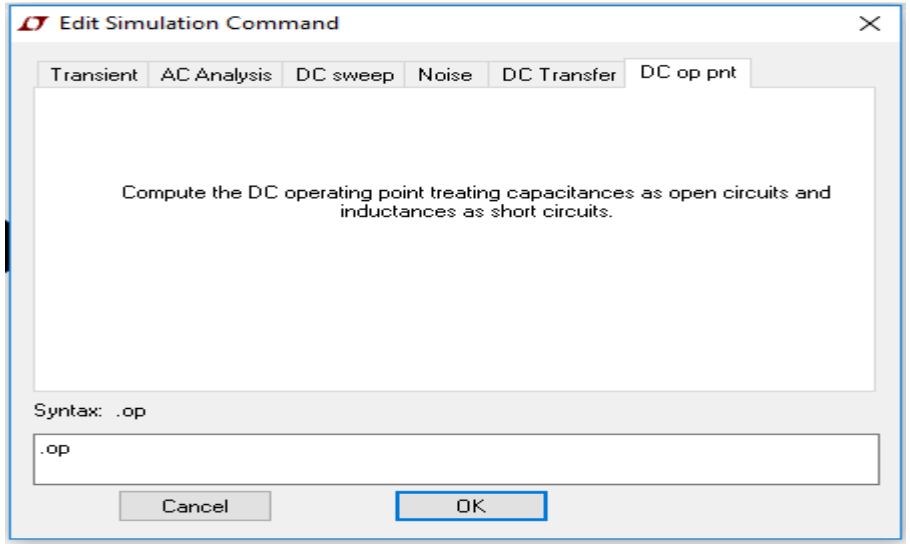
# Example 2: Basic DC analysis in LTspice



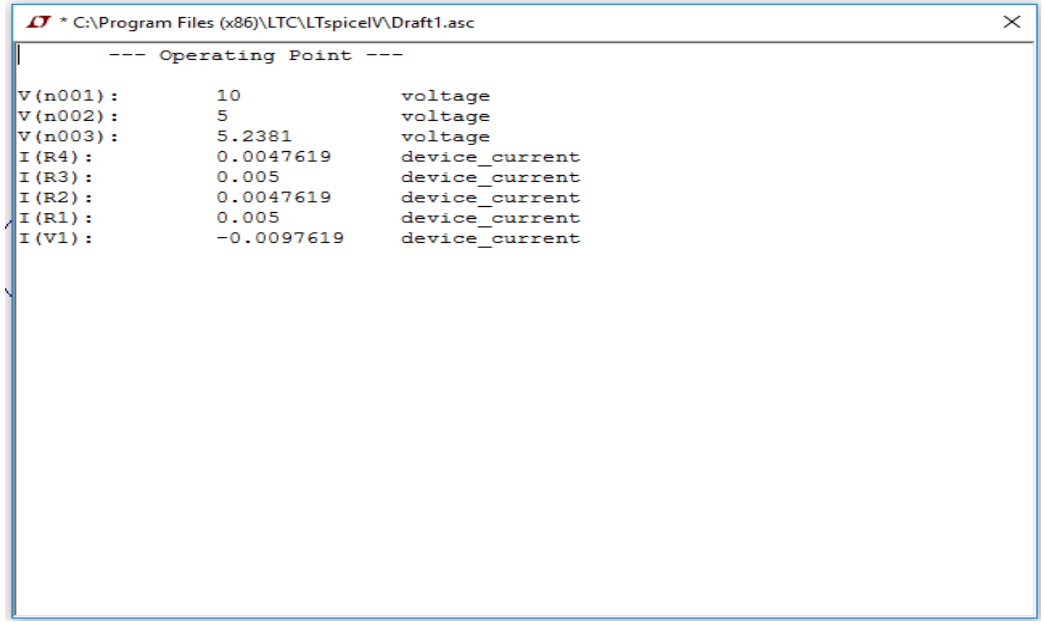

1. All the components are added as mentioned before.
2. The values of V1, R1,R2,R3 and R4 are modified to give the desired values.
3. The net names ‘V1’ and ‘V2’ are used as shown in the figure below to measure voltages at those points.
4. Select ‘Simulate’ from the menu and click the ‘Edit Simulation Cmd’ button to see the different kind of analysis that you can perform here. These are Transient, AC Analysis, DC sweep, Noise, DC Transfer and DC op pnt.



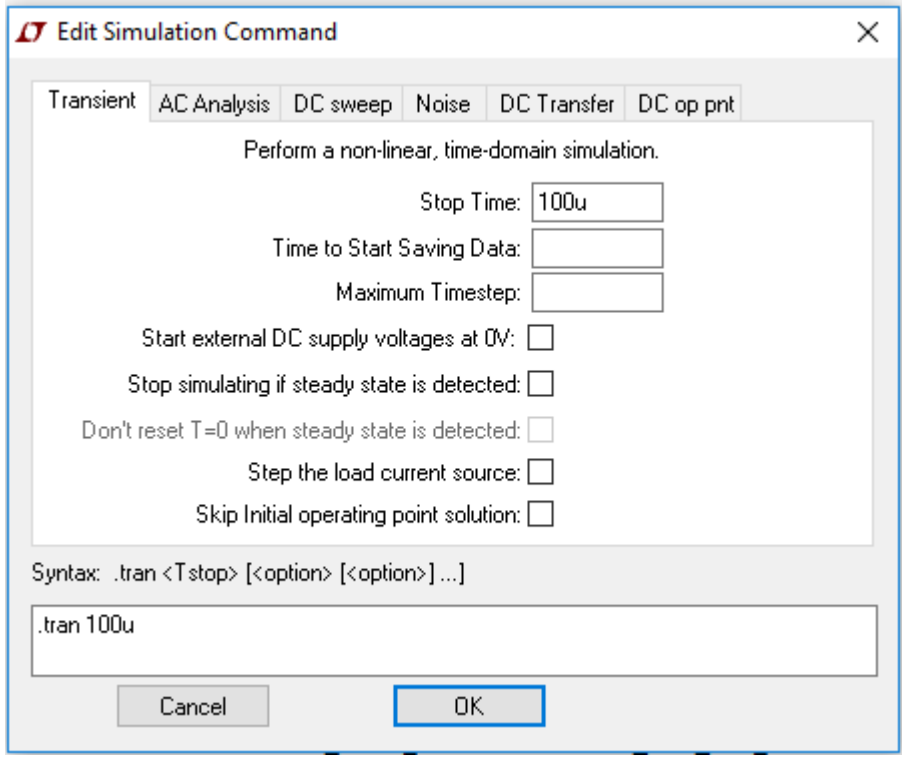
5.From these, select ‘DC op pnt’ and press ‘OK’ as shown below.



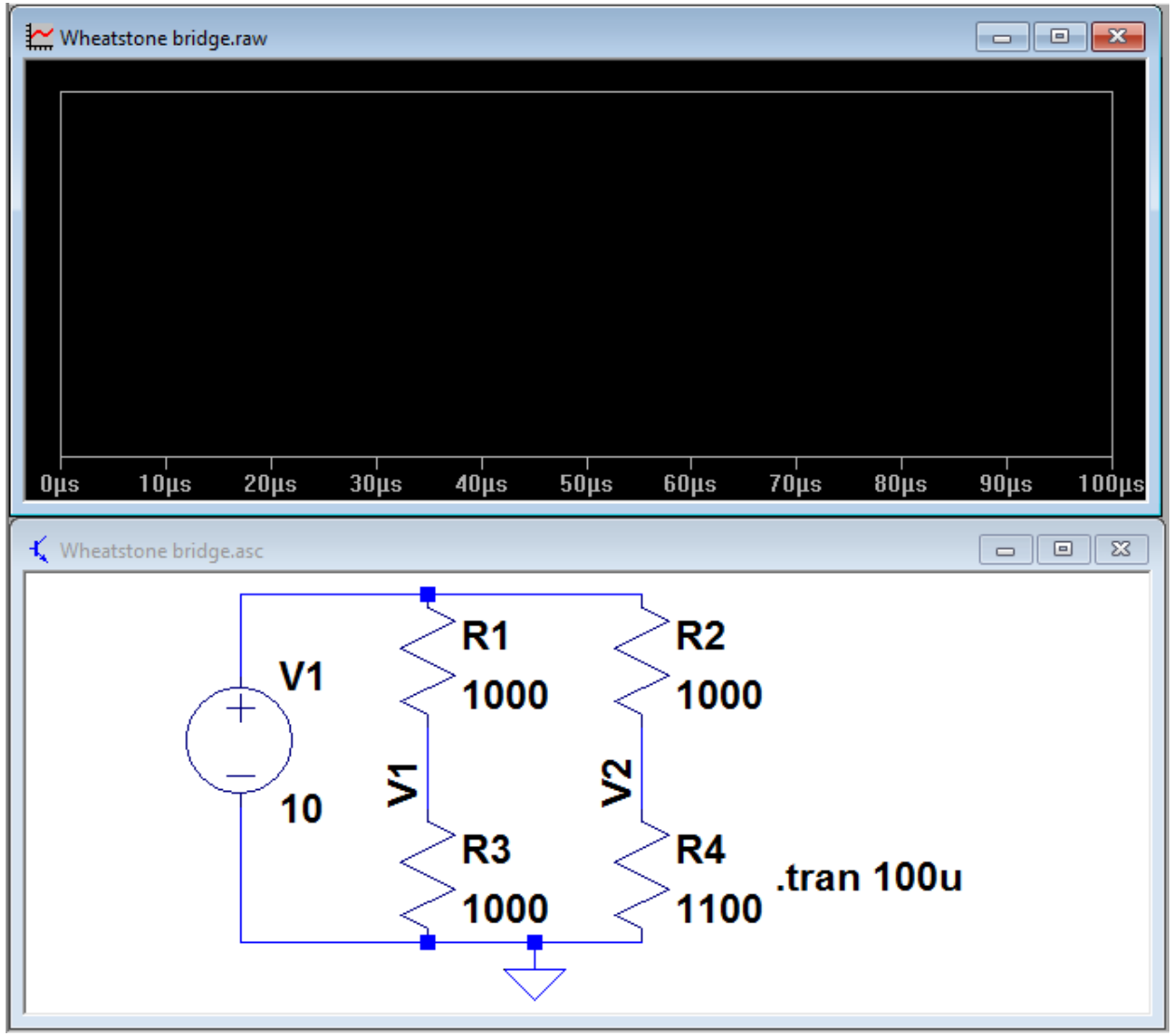

This will give voltages at different nodes and currents flowing through different elements in tabular form.



6.Next, we will do a ‘Transient’ analysis.  For that, again select the ‘Simulate’ button from the menu and click the ‘Edit Simulation Cmd’ button. Select ‘Transient’ from the window, set the ‘Stop Time’ as 100µs and click ‘OK’.



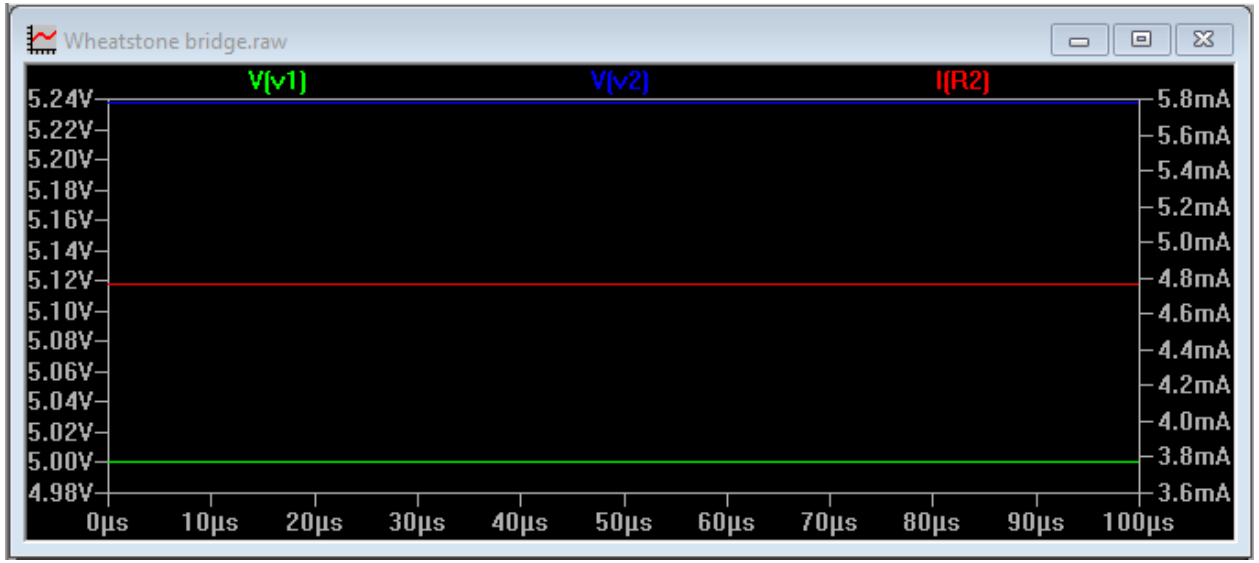
# Now click the ‘Run’ button to see the following panel.



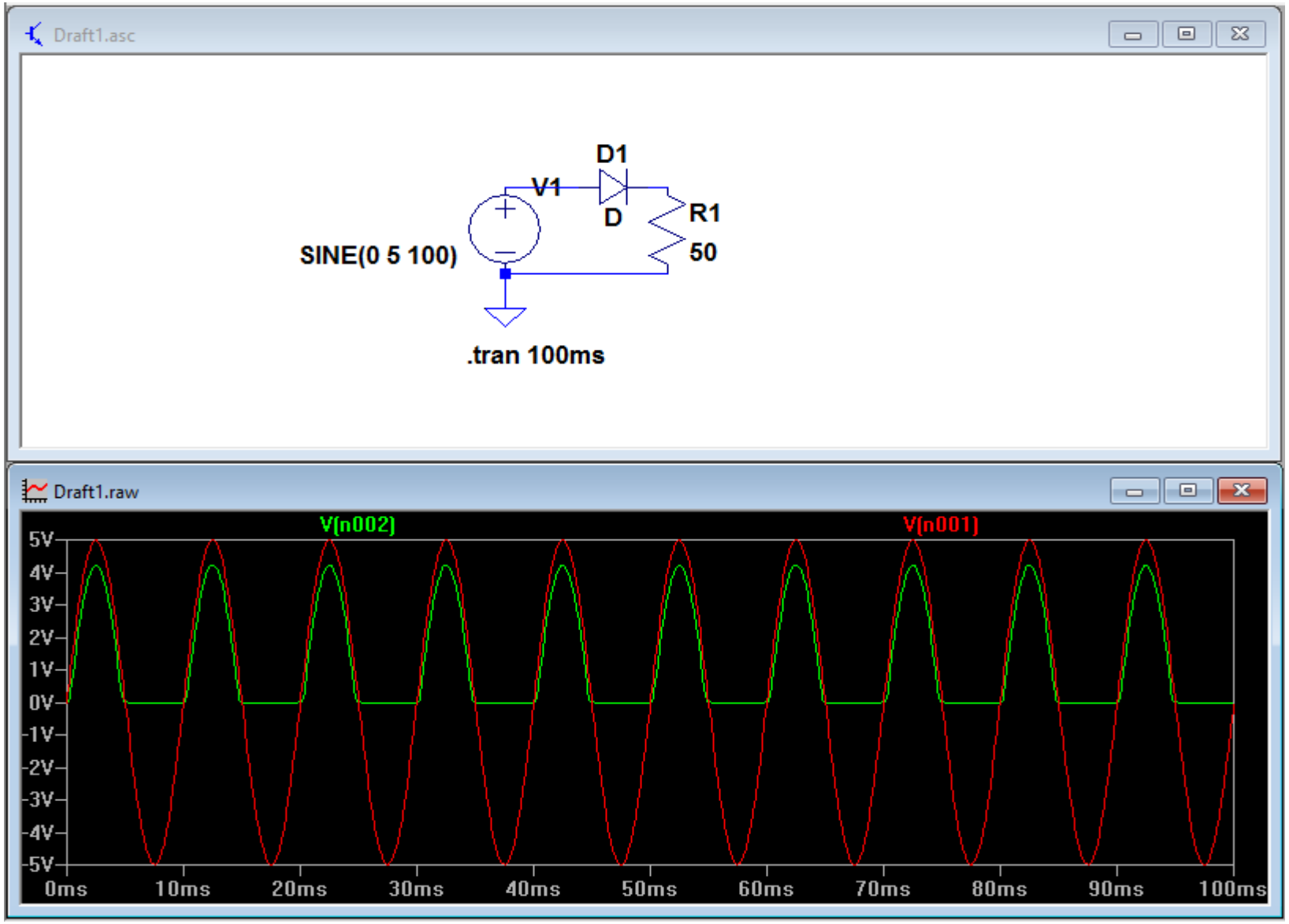
7. To measure voltage at any point of the circuit, take the cursor there and the cursor should change to a voltmeter probe as shown below.

To measure current through a component in the circuit, take the cursor there and it should change to a clamp ammeter as shown below.

In the following plot, the voltages at points V1 and V2 as well as current through R2 are shown.



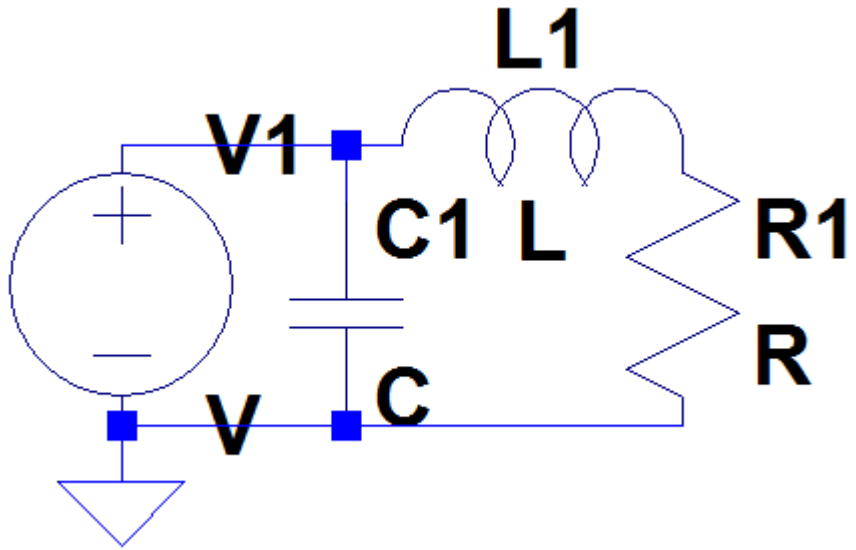
# Example 3: Half-Wave Rectification



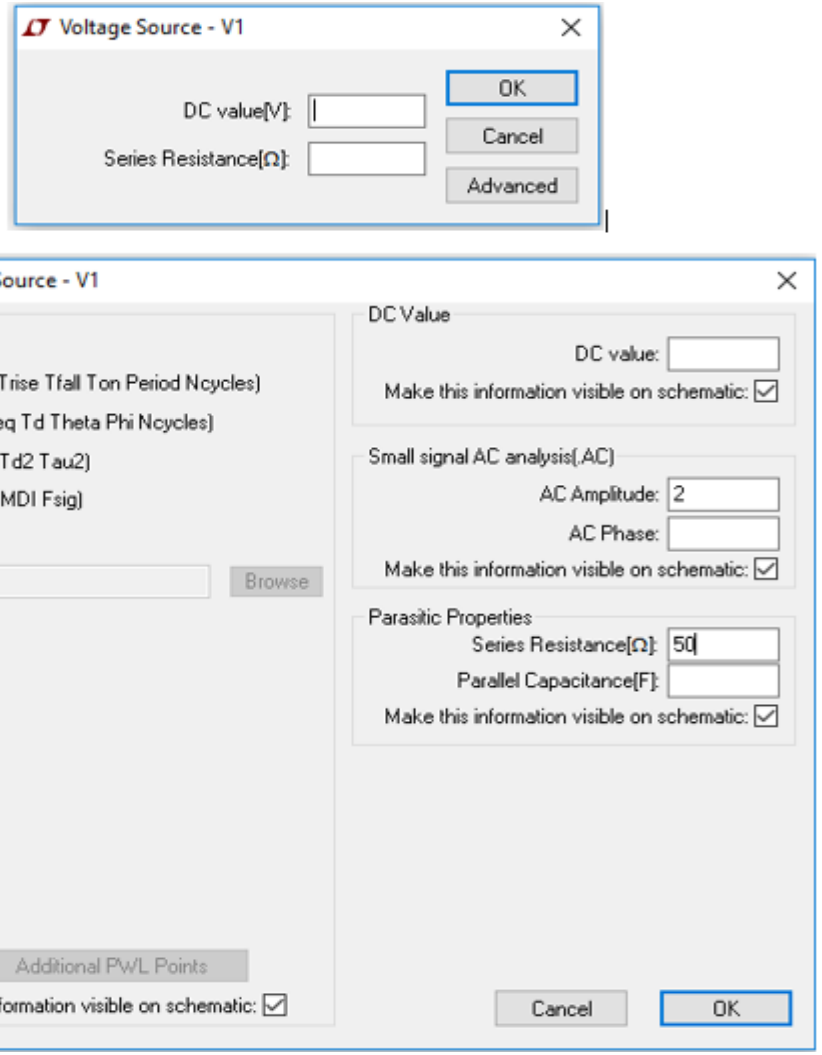
# AC Analysis

## Example 4-Design a maximally flat low pass filter with a cutoff frequency of 2GHz and plot the response for f=0 to 5GHz.

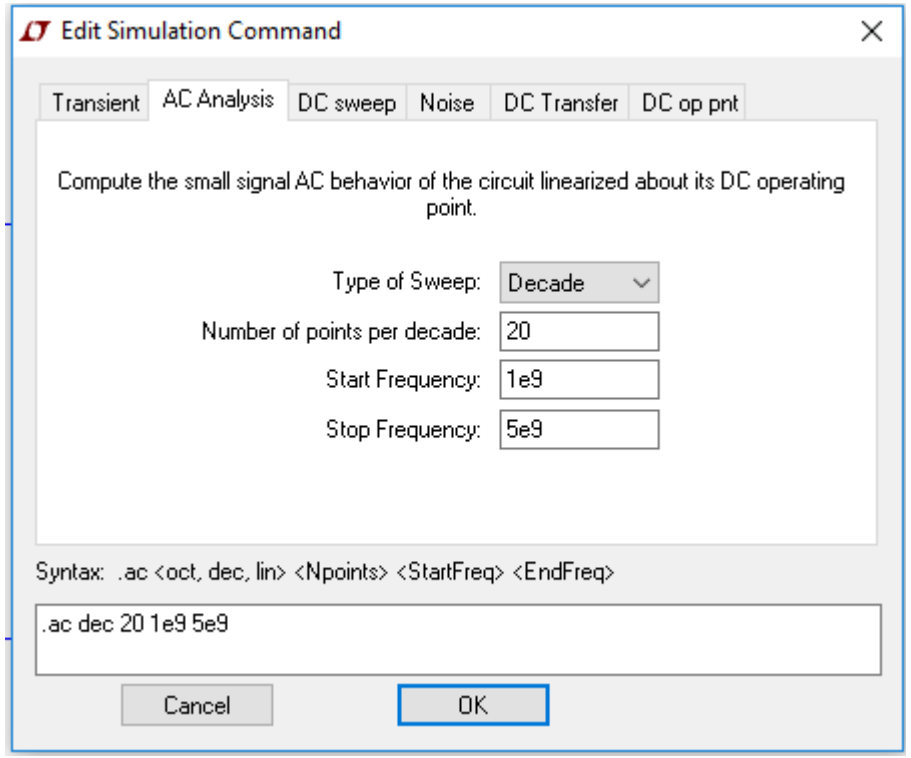
Let's design a 2nd order low pass filter as shown below which has a cutoff frequency of 2GHz. For this, all the required components are added in the schematic as it was done in the previous example. Then, change the element values to those obtained in calculation.



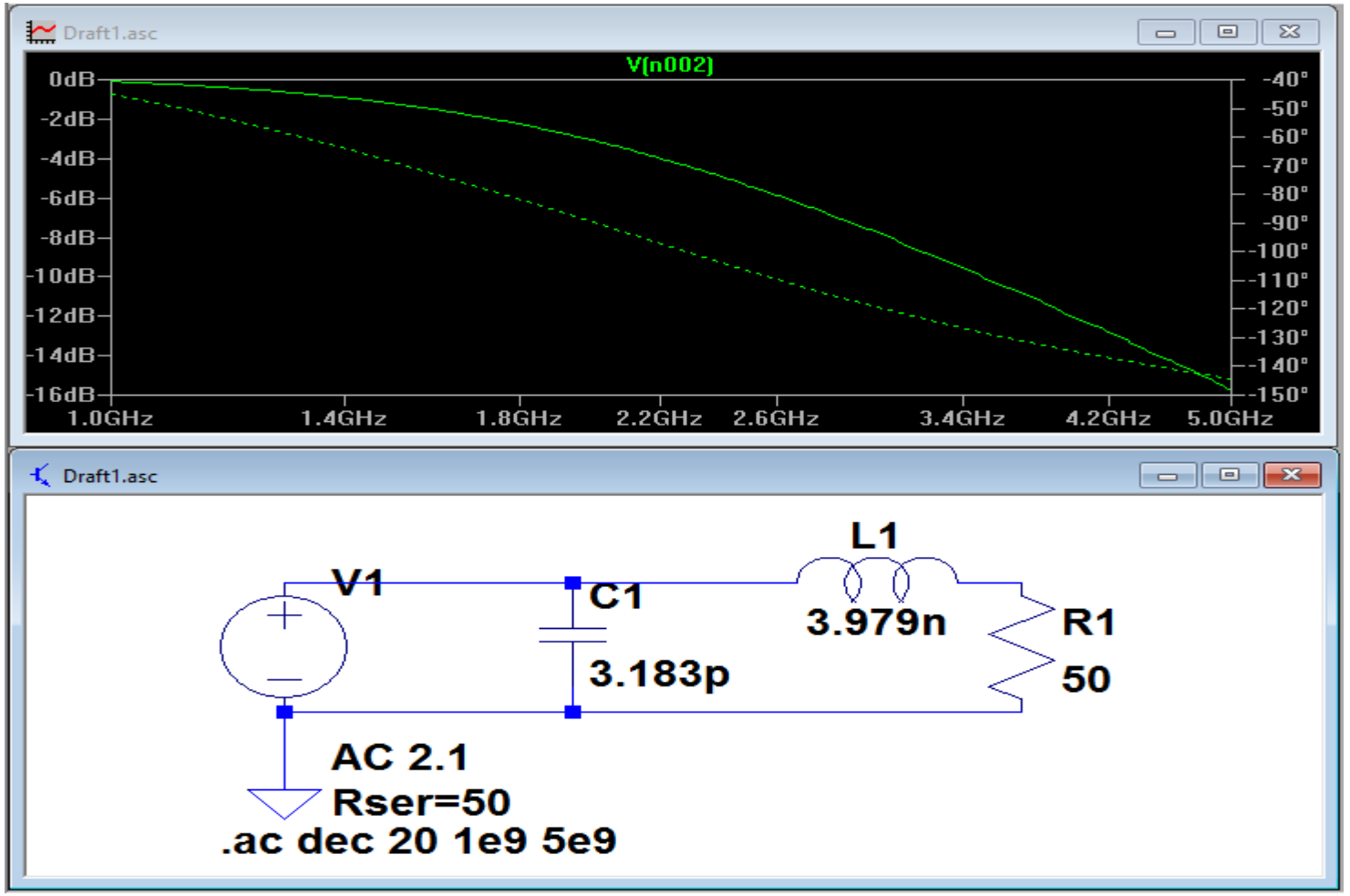
- To assign a value to the voltage source, right click on the voltage source and then select ‘Advanced’ as we will do AC analysis. Assign values for AC Amplitude and Series Resistance. You can also assign values for ‘AC Phase’ and ‘Parallel Capacitance’ if necessary.

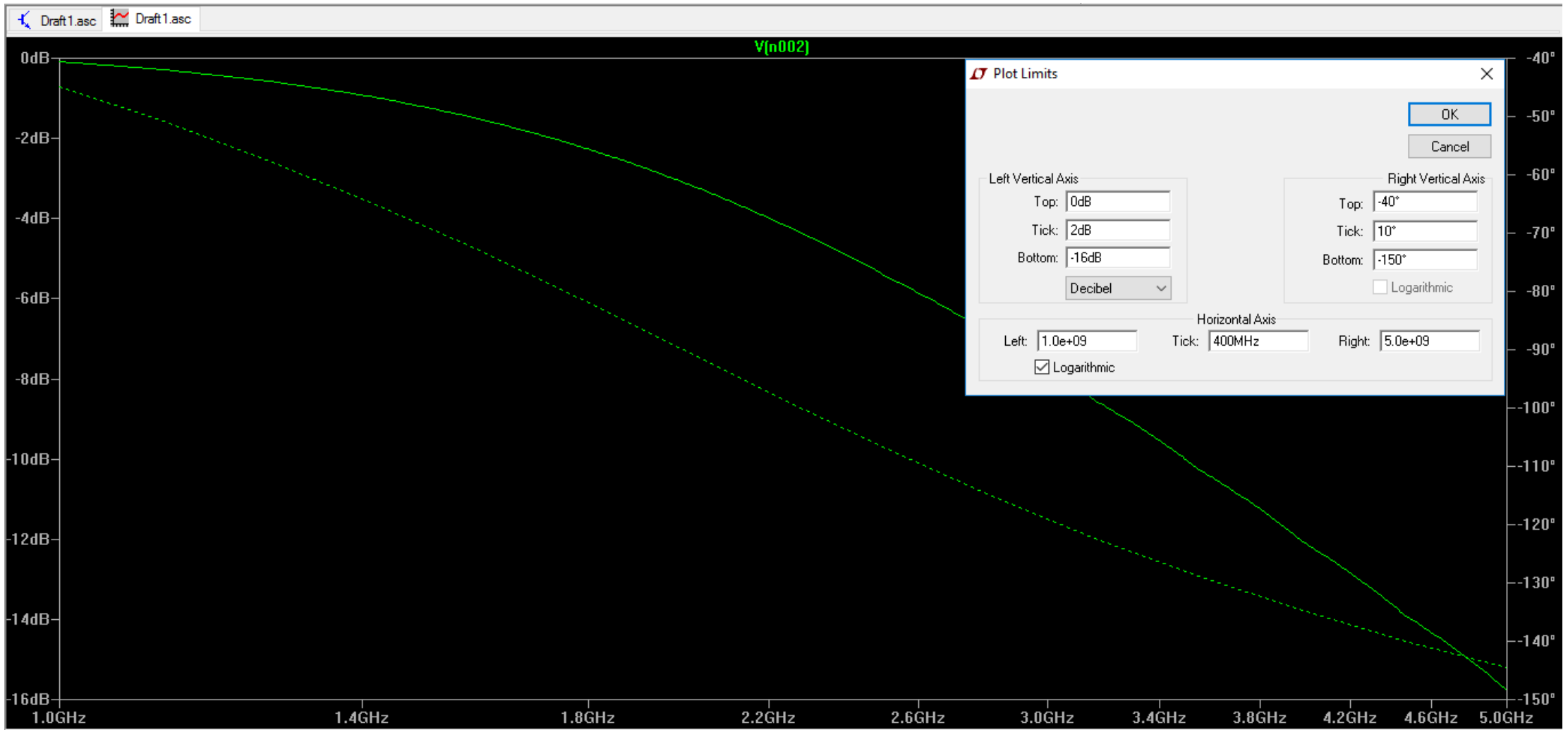


- Click on the ‘Simulate’ button and select ‘ Edit Simulation Cmd’ as in the previous example.
- Select the appropriate tab (in this case ‘AC Analysis’) for simulation, enter the simulation options and press ‘OK’.

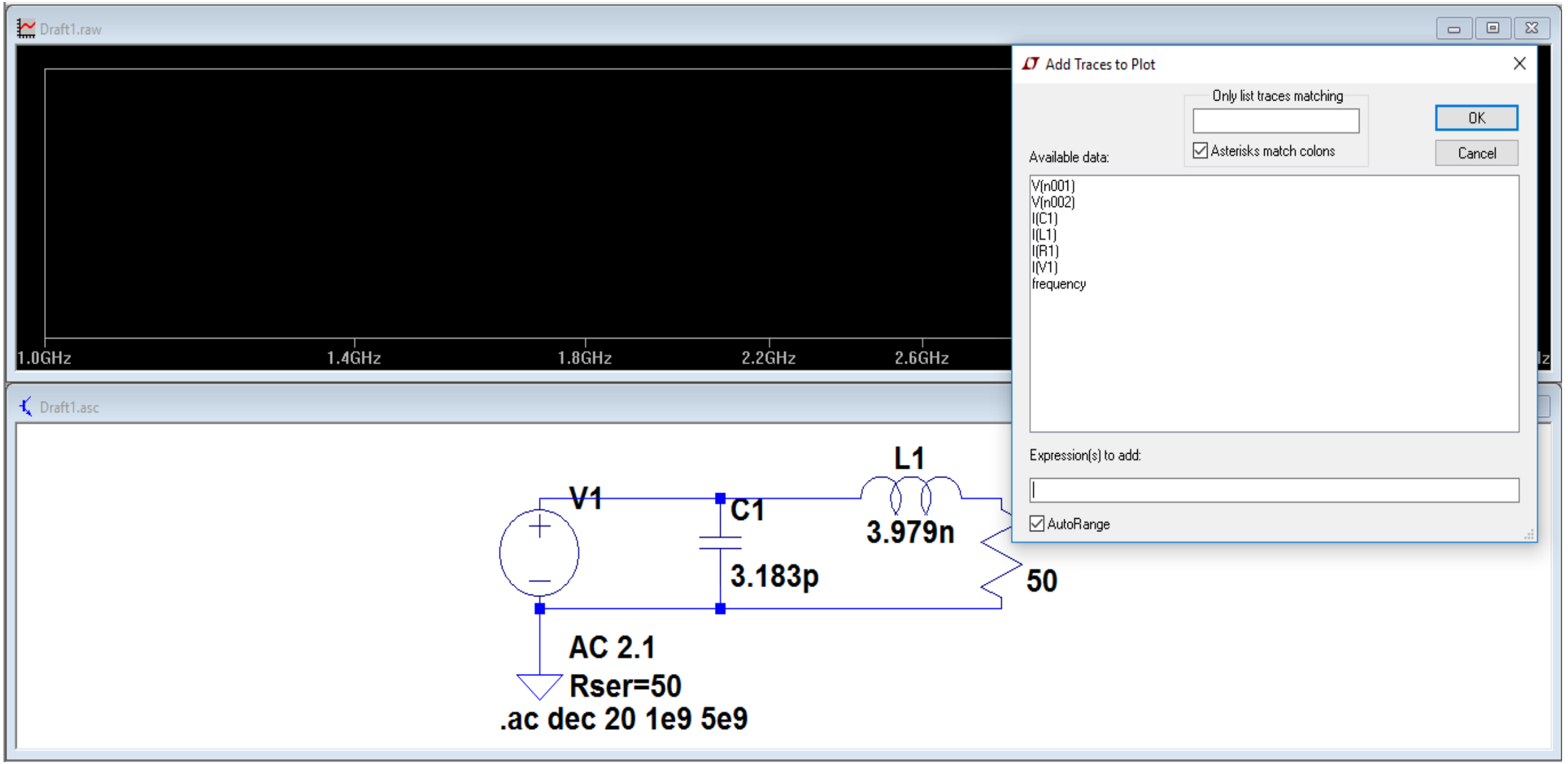


- Then, click the ‘Run’ button and place the voltage probe at the output which will give an output as shown below. To modify the settings for all the axes, right click on the graph and go to ‘Manual Limits’. It will give an window as shown in the next slide where you can change the units, range etc.

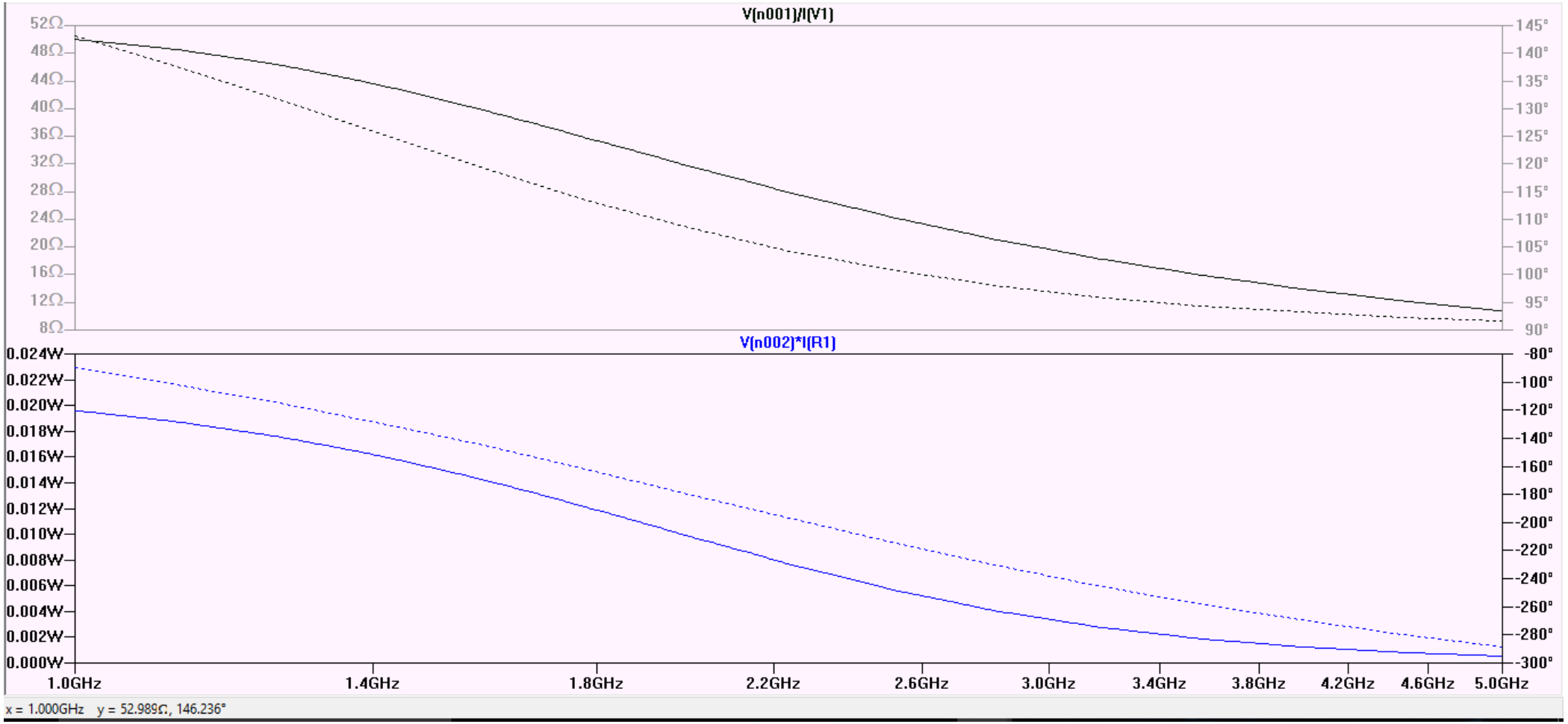



Draft1.asc
Draft1.asc
V(n002)
0dB
-2dB
-4dB
-6dB
-8dB
-10dB
-12dB
-14dB
-16dB
-40°
-50°
-60°
-70°
-80°
-90°
-100°
-110°
-120°
-130°
-140°
-150°
1.0GHz
1.4GHz
1.8GHz
2.2GHz
2.6GHz
3.0GHz
3.4GHz
3.8GHz
4.2GHz
4.6GHz
5.0GHz
Plot Limits
OK
Cancel
Left Vertical Axis
Top: 0dB
Tick: 2dB
Bottom: -16dB
Decibel
Right Vertical Axis
Top: -40°
Tick: 10°
Bottom: -150°
Logarithmic
Horizontal Axis
Left: 1.0e+09
Tick: 400MHz
Right: 5.0e+09
Logarithmic

- To plot the input impedance of the filter, right click on the plot and select ‘Add Trace’ as shown below.

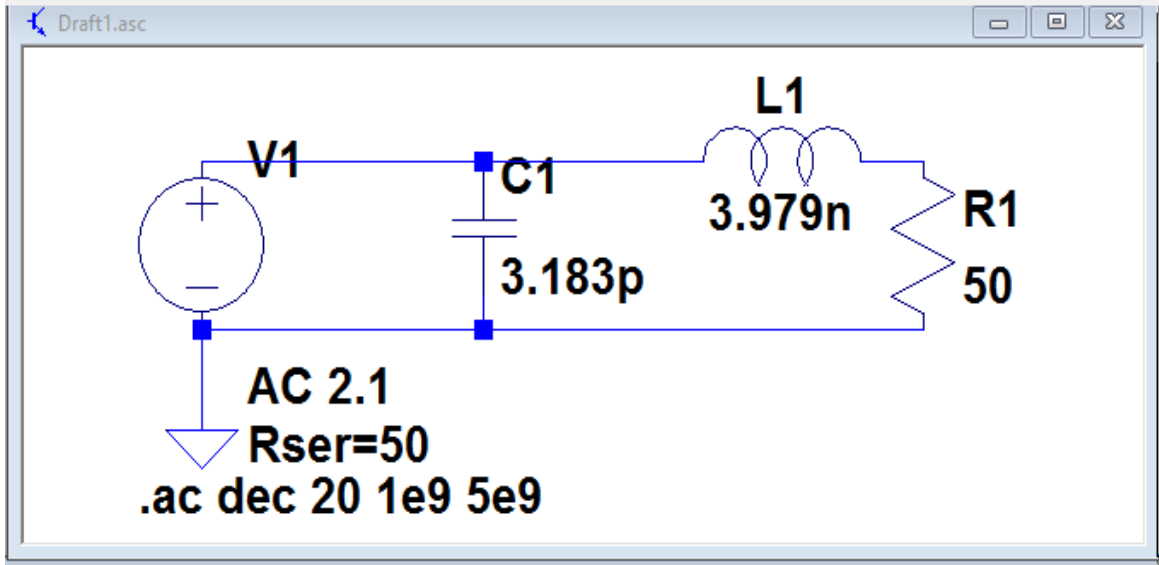


In ‘Expressions to add’, write V(n001)/I(V1) to measure the input impedance. If you want to measure the dissipated power or loss by the load R1, write the expression V(n002)*I(R1).



# Comparison between 2nd order LPF and 5th order LPF

## 2nd order LPF





## 5th order LPF