


# LTspice Basic Simulation Exercises

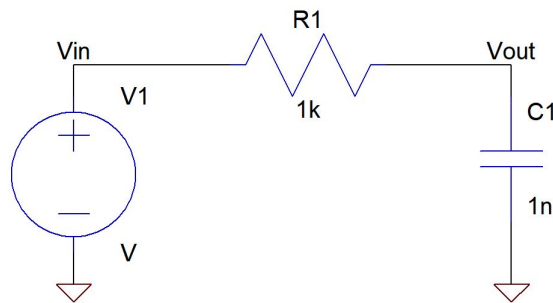
Alfred Festus Davidson

alfy7.github.io

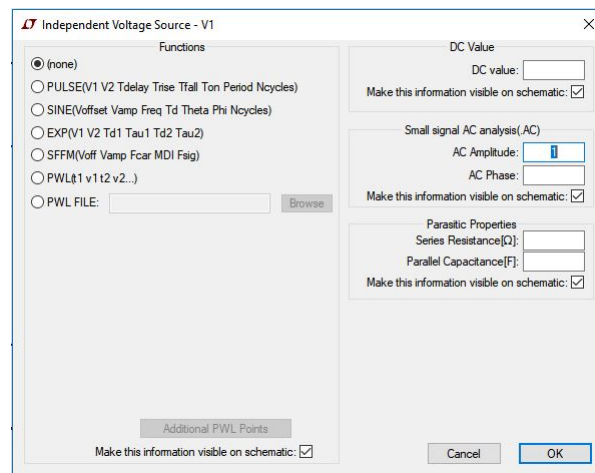
November 30, 2017

## First Order Passive RC Filters

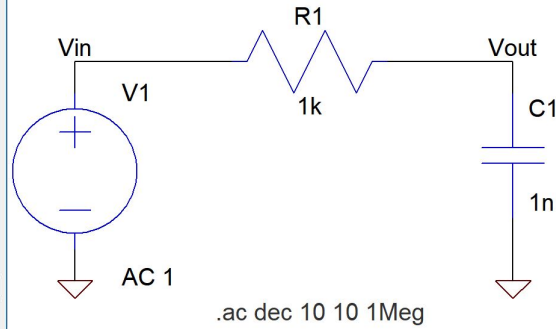
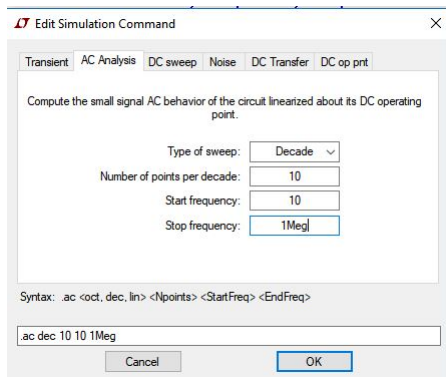
- Create a new schematic and draw the following circuit. Capacitors can be added either using the shortcut key  $C$  or its icon 



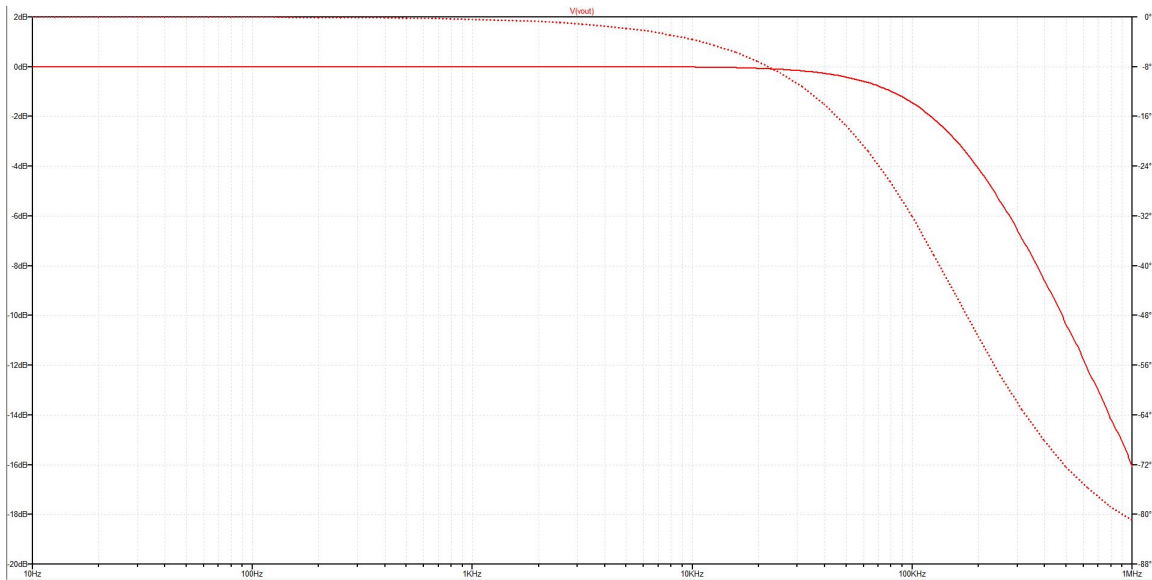
- This is an RC low pass filter. One feature of interest is its frequency response which is captured by a bode plot. We give an input voltage of varying frequency and see the amplitude and phase of the output voltage. To set voltage source  $V1$  as the input voltage, right-click it and give it an AC amplitude of 1.



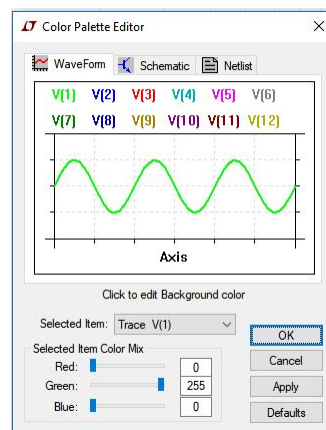
- Go to *Simulate->Edit Simulation Panel* and choose *AC Analysis* to get the bode plot. Note that  $m$  is for *milli* and  $meg$  for *mega* (case insensitive). Set the parameters as follows and place the resulting spice directive on the schematic. Note the syntax given at the bottom of the dialog box to directly write the spice directive directly in future.



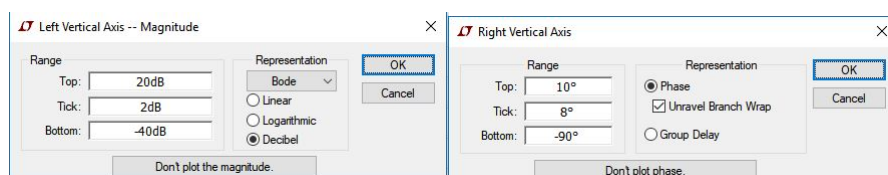
- Now run the simulation. Click on the node *Vout* in the schematic to probe it and see its appropriate waveform. The thick line is the magnitude plot while the dotted line is the phase plot.

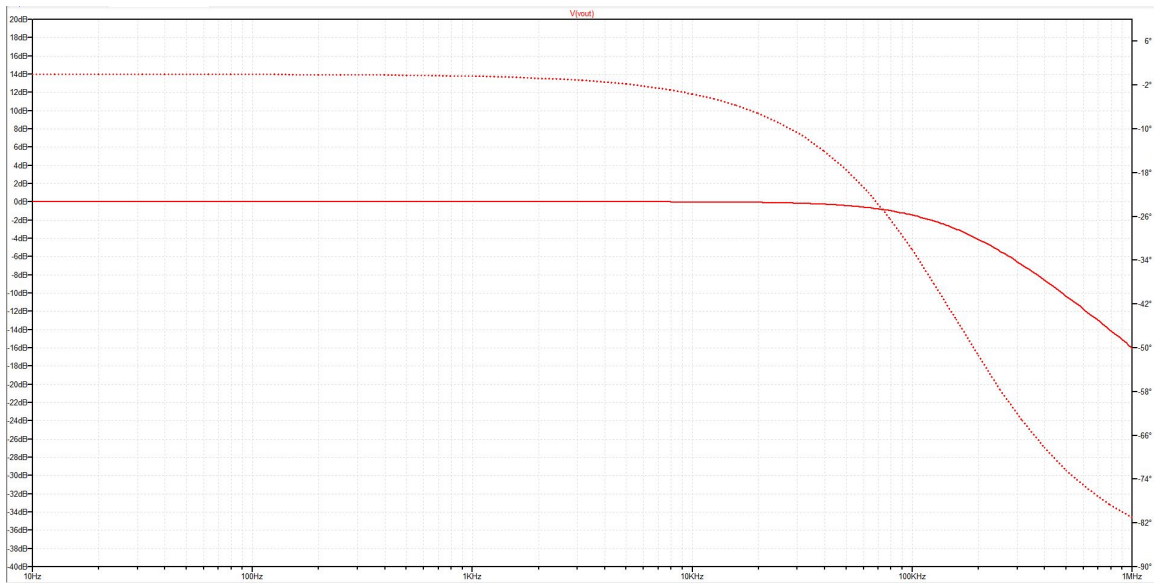


- You can change the color scheme by going to *Tools->Color Preference*, this time we edit the waveform tab.

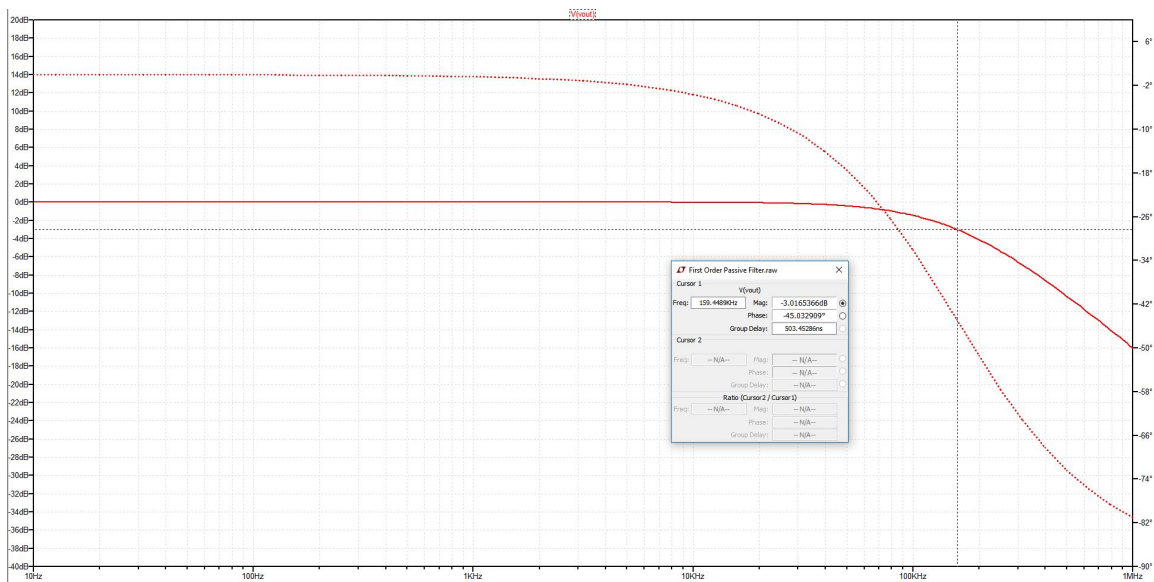


- Right click the axis to change limits and tick.

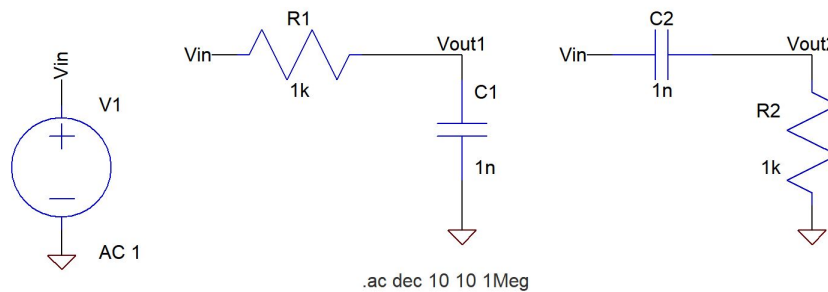




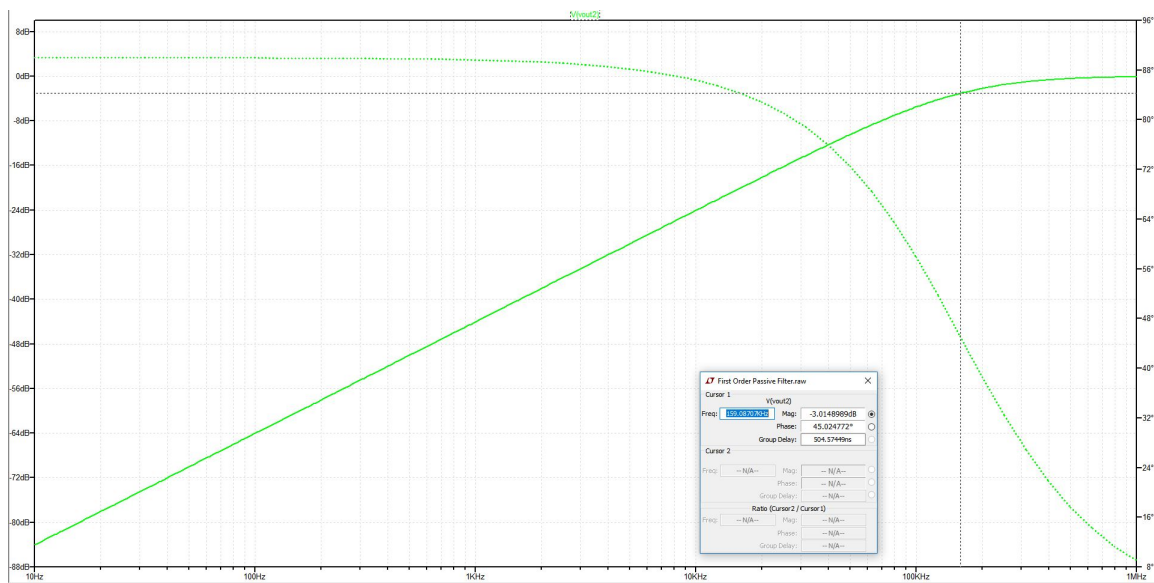
- Click on the waveform name (*Vout* in this case) at the top of the waveform to get the cursor. You can scroll through data points and view points of interest (say the *3dB* cutoff frequency in this case)



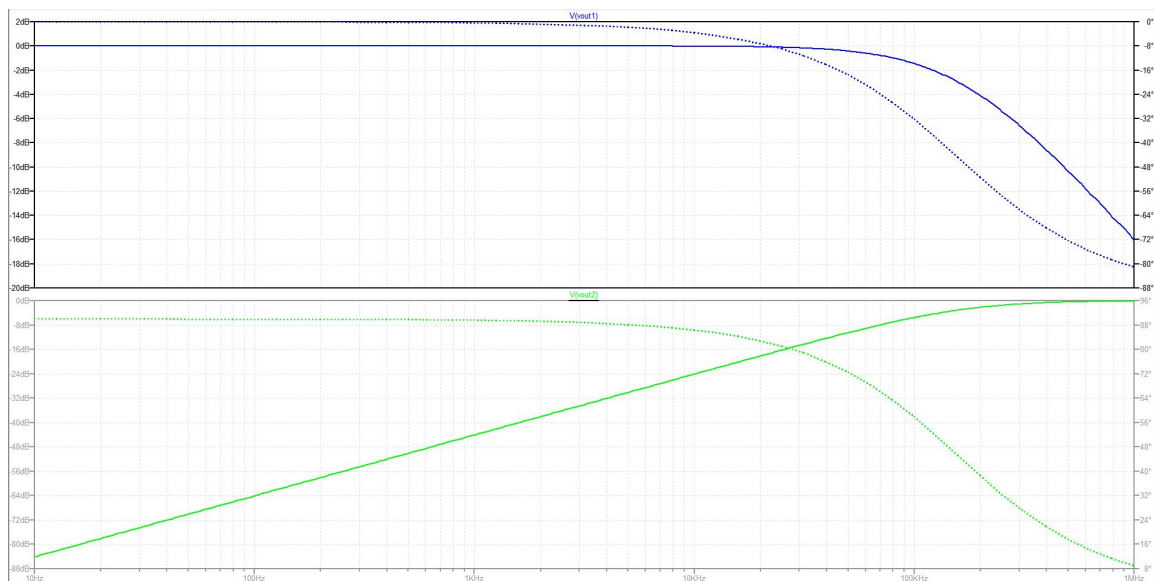
- Now make the following changes in the schematic. Note that all the 3 nodes labeled *Vin* are connected even though there are no wires. Notice that the output nodes have been named differently to avoid shorting them together.



- Run the simulation and probe *Vout2*. Adjust the limits if needed.



- Right-click the plot plane and click *Add plot plane*. Now you have two plot planes sharing the same  $x$  axis. Right click the new plane and click *Add Traces*. Choose *Vout1*. Now you can compare both the circuits.



- Add proper titles to complete the schematic.

