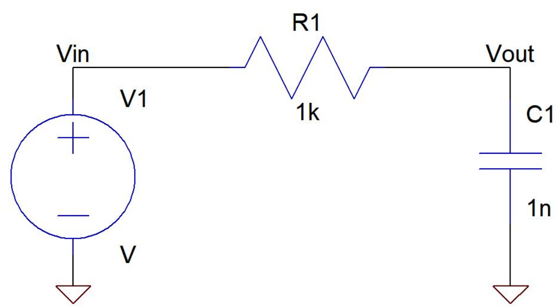
LTspice Basic Simulation Exercises

Alfred Festus Davidson
alfy7.github.io

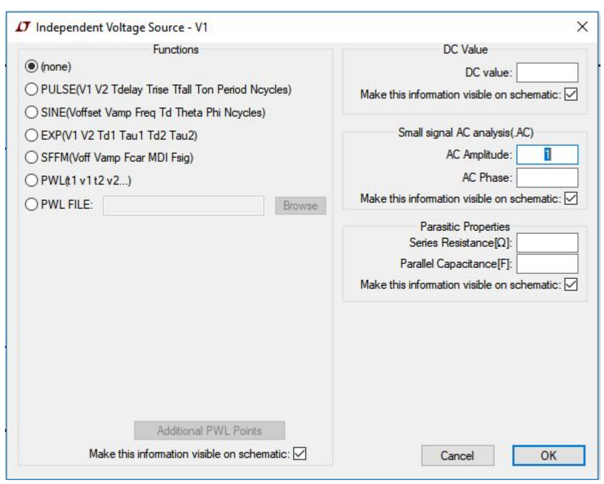
November 30, 2017

First Order Passive RC Filters

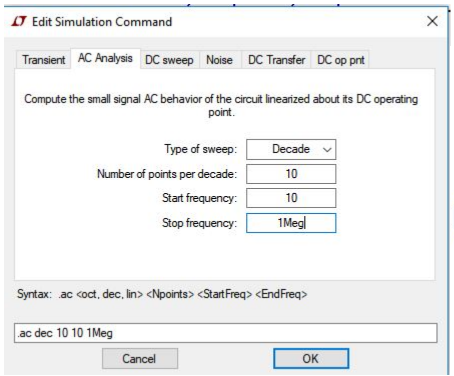
- Create a new schematic and draw the following circuit. Capacitors can be added either using the shortcut key *C* or its icon 

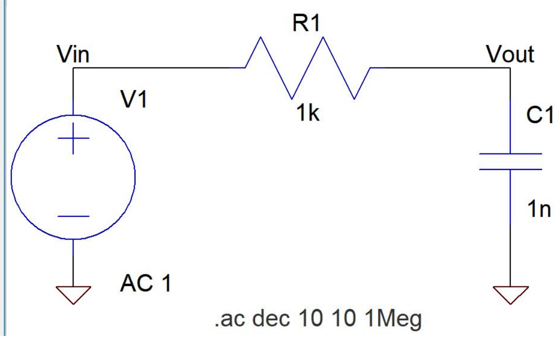


- This is an RC low pass filter. One feature of interest is its frequency response which is captured by a bode plot. We give an input voltage of varying frequency and see the amplitude and phase of the output voltage. To set voltage source *V1* as the input voltage, right-click it and give it an AC amplitude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

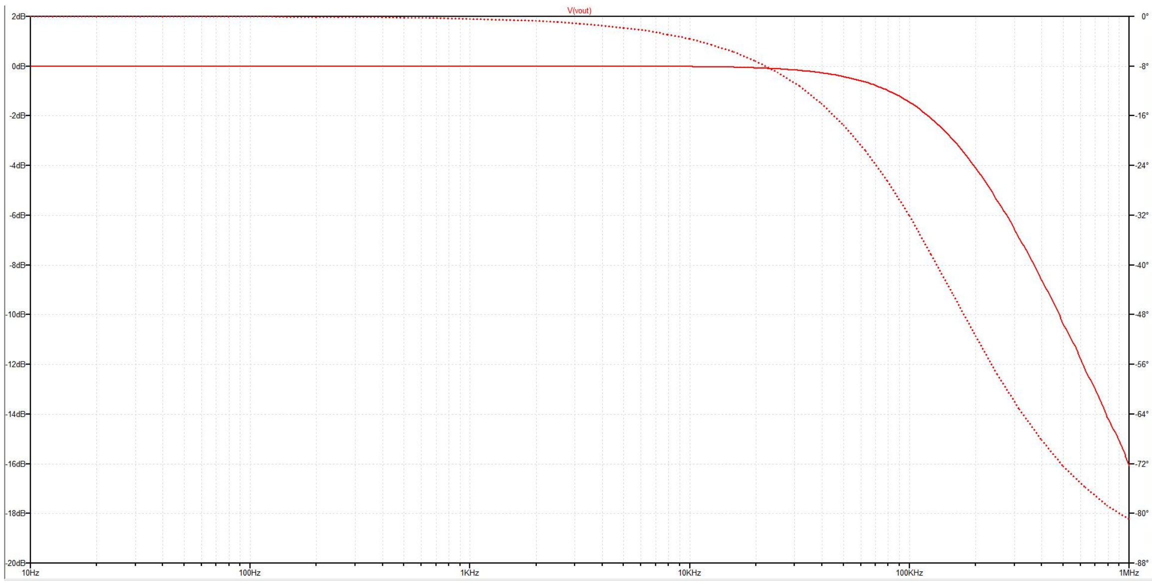


- Go to *Simulate->Edit Simulation Panel* and choose *AC Analysis* to get the bode plot. Note that *m* is for *milli* and *meg* for *mega* (case insensitive). Set the parameters as follows and place the resulting spice directive on the schematic. Note the syntax given at the bottom of the dialog box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

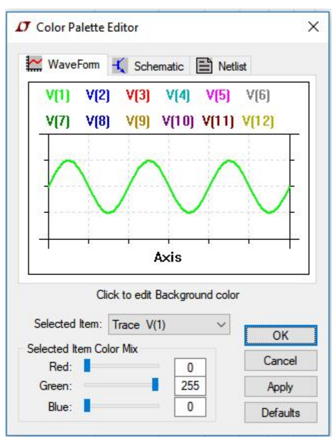




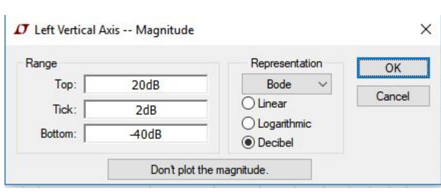
- Now run the simulation. Click on the node *Vout* in the schematic to probe it and see its appropriate waveform. The thick line is the magnitude plot while the dotted line is the phase plot.

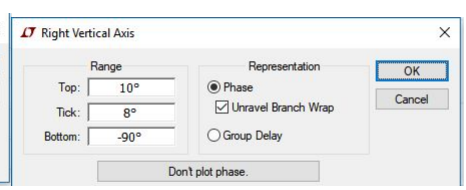


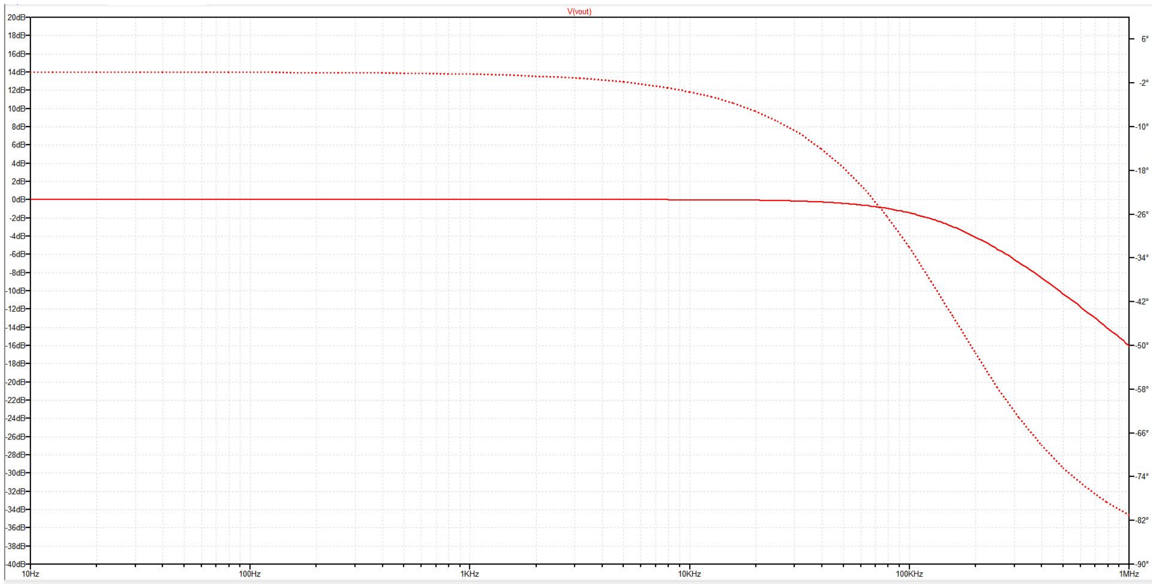
- You can change the color scheme by going to *Tools->Color Preference*, this time we edit the waveform tab.



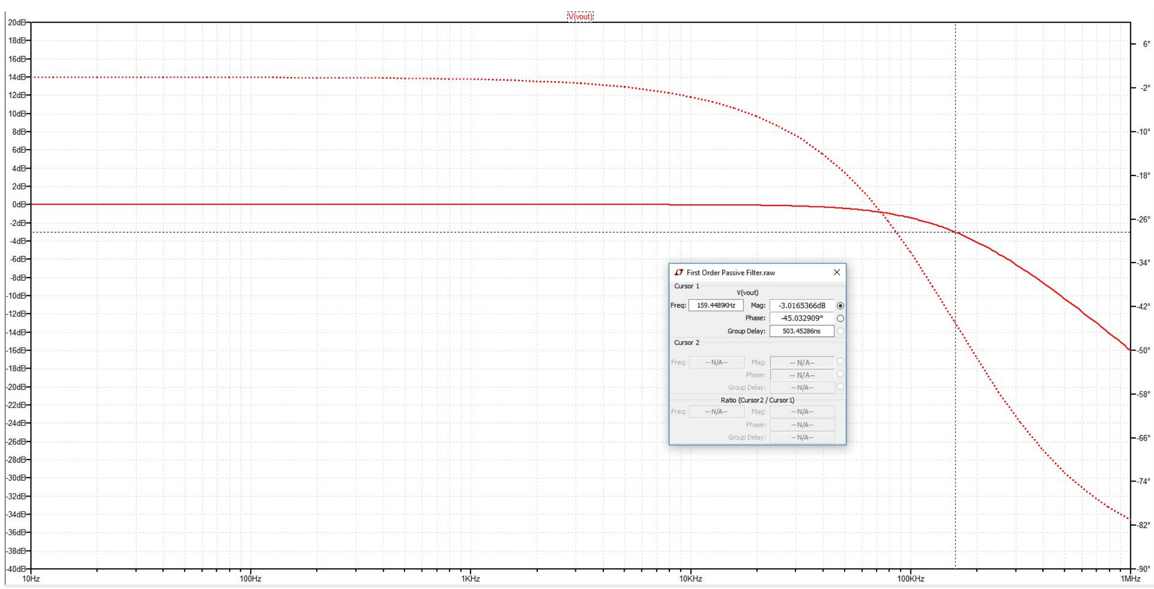
- Right click the axis to change limits and tick.



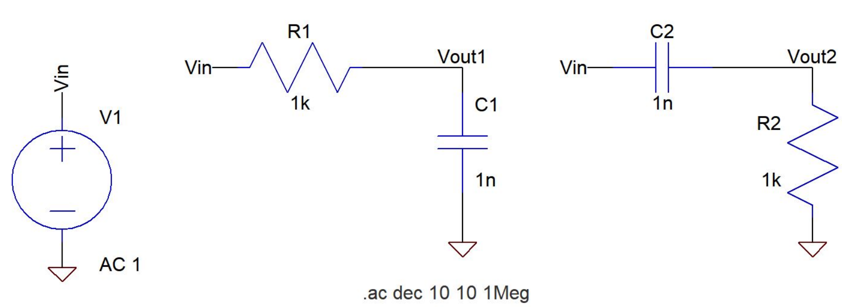




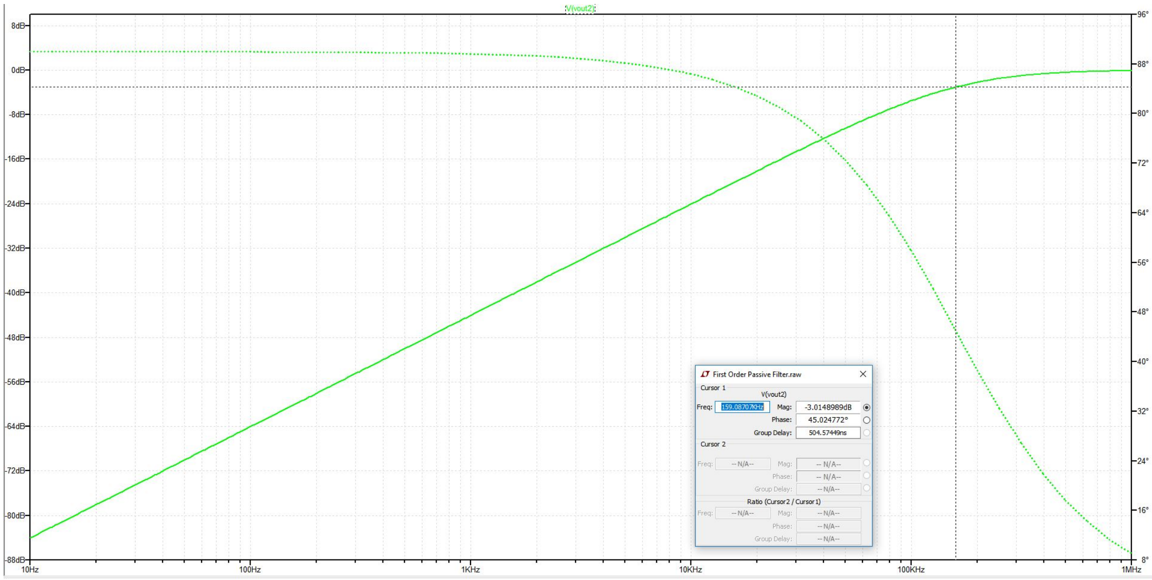
- Click on the waveform name ($Vout$ in this case) at the top of the waveform to get the cursor. You can scroll through data points and view points of interest (say the $3dB$ cutoff frequency in this case)



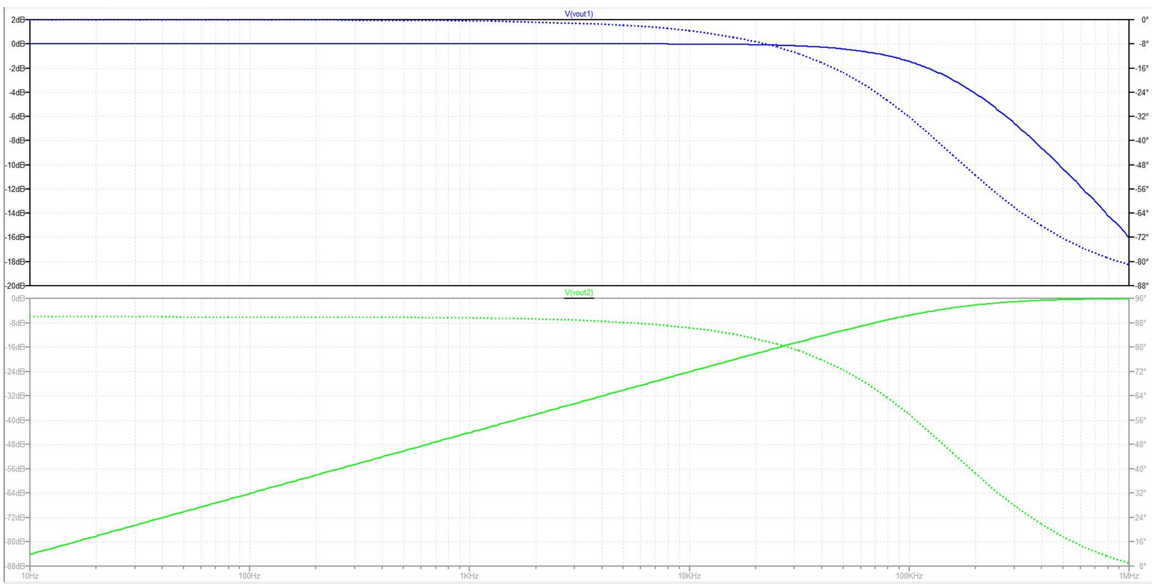
- Now make the following changes in the schematic. Note that all the 3 nodes labeled Vin are connected even though there are no wires. Notice that the output nodes have been named differently to avoid shorting them together.



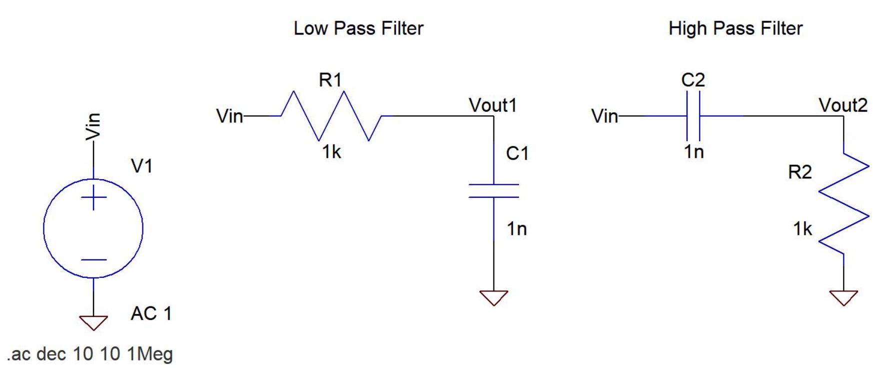
- Run the simulation and probe $Vout2$. Adjust the limits if needed.



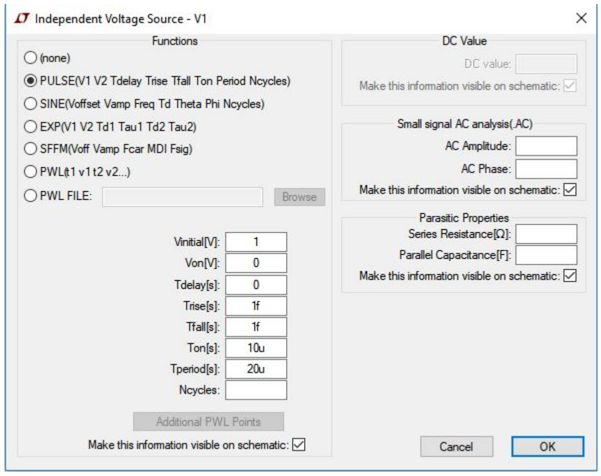
- Right-click the plot pane and click *Add plot pane.* Now you have two plot panes sharing the same x axis. Right click the new pane and click *Add Traces.* Choose *Vout1.* Now you can compare both the circuits.



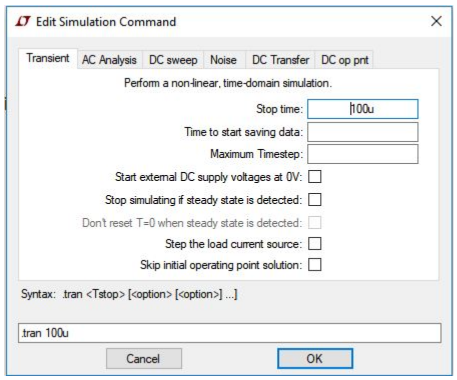
- Add proper titles to the schematic.



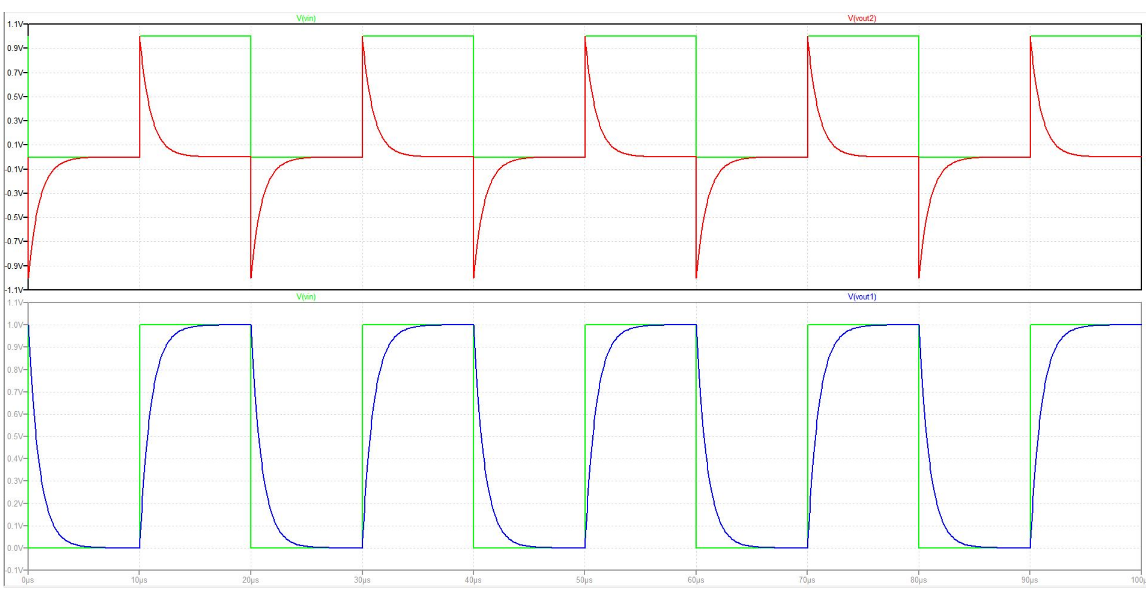
- Another interesting feature of this circuit is its response to a square wave. First we need the voltage source to generate a square wave. Right-click it, and view advanced options. Choose the *pulse* mode and use the following settings. The rise and fall times are chosen to be arbitrarily small. (f stands for femto)

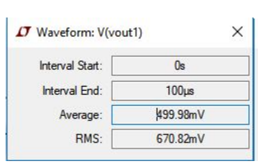


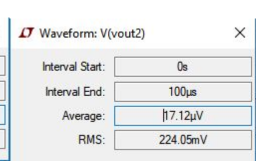
- Now we wish to see the *transient* response and not the AC response. We make this change in the *Simulate->Edit Simulation Command* and choose the *Transient* tab. Enter the following settings.



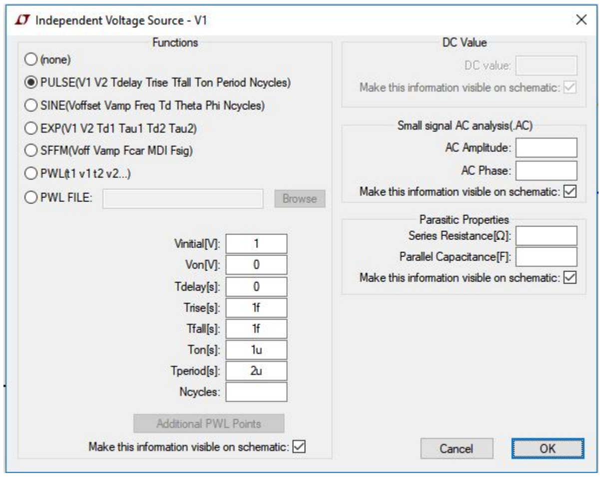
- Now, run the simulation. Add another plot pane. On each plot pane, add *Vin*. Add *Vout1* on one pane, and *Vout2* on another. Again, you can change the colors, adjust the axis settings and use the cursor to scroll through the data points. Here, the frequency of the square wave is chosen to be much smaller than the $3dB$ cutoff frequency of the filters. You can hold *Ctrl* and left click the waveform name (*Vout1* or *Vout2* in this case) to see the average and RMS value (over the time window visible)



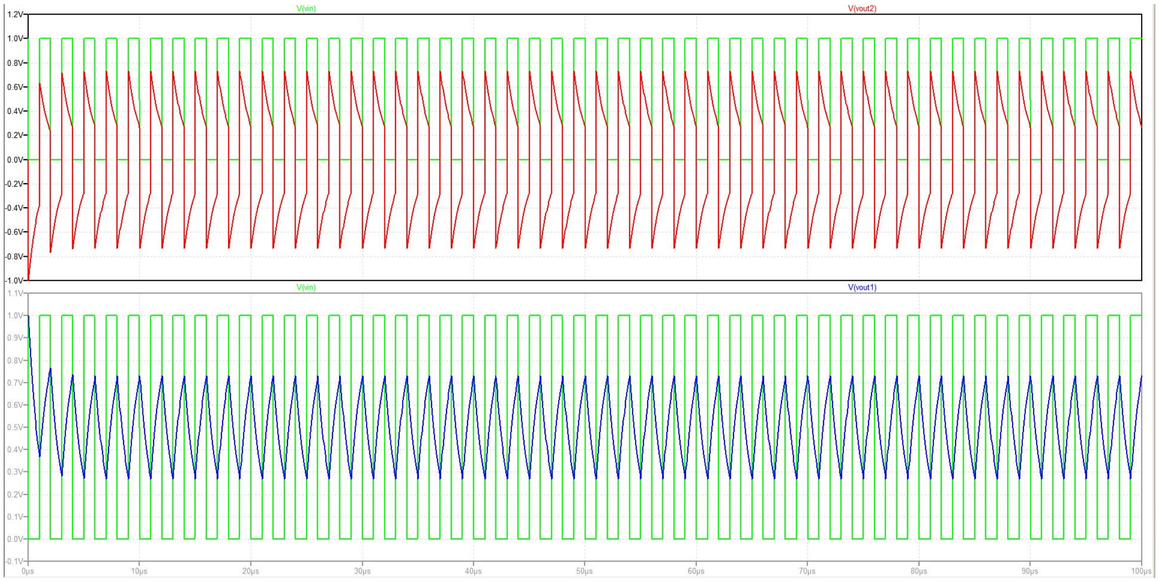




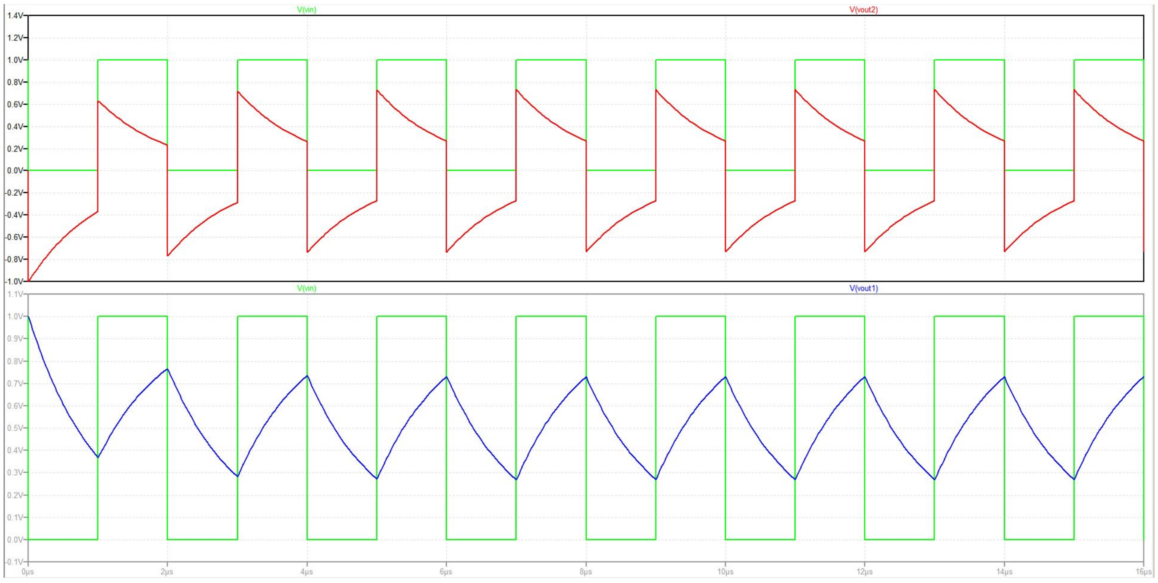
- Now, modify the square wave characteristics as follows.



- Run the simulation

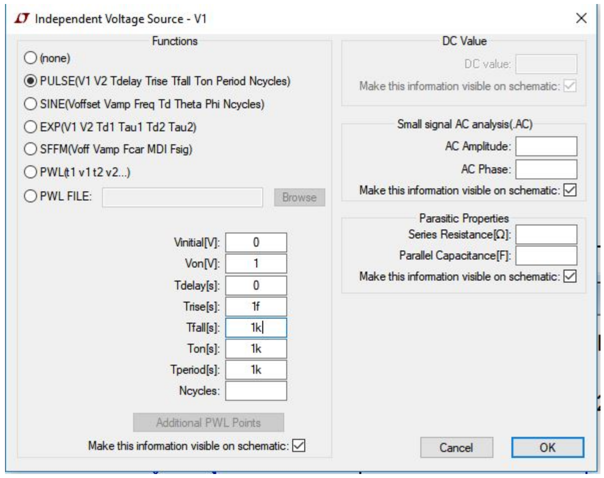


You can either change the transient simulation time or magnify the plots using the magnification tools.

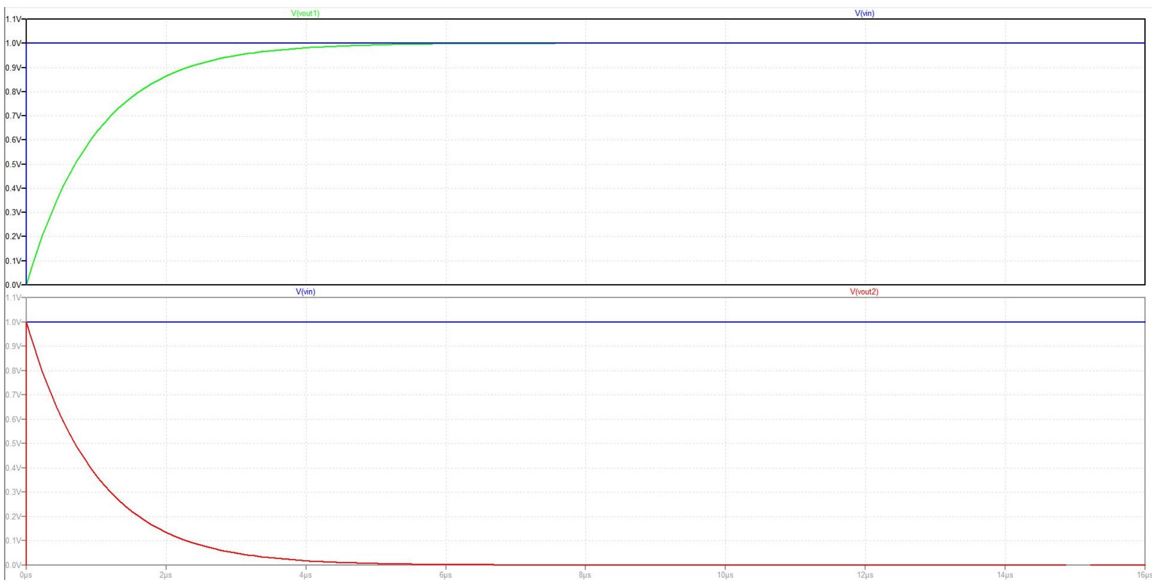


Here, the frequency of the square wave is chosen to be much larger than the $3dB$ cutoff frequency of the filters.

- Yet another feature used to characterise this circuit is its step response. One way to get the voltage source to generate a step input is shown below.

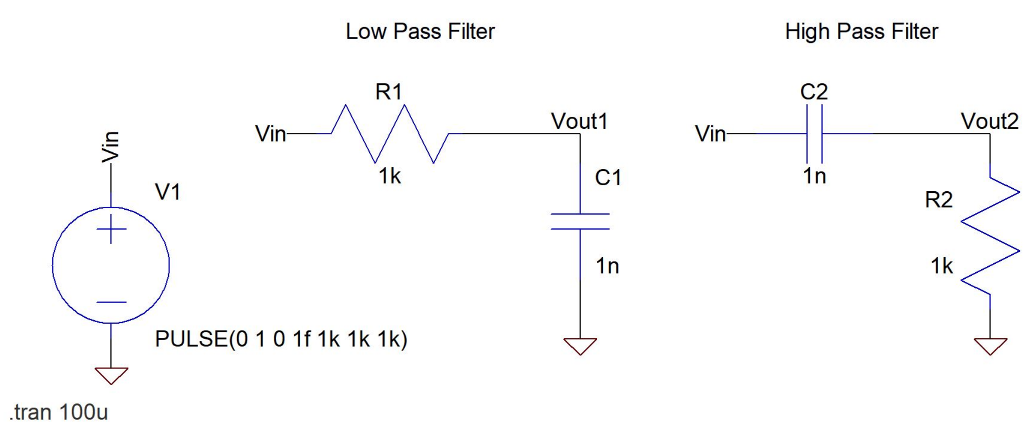


The idea here is to make the fall time arbitrarily large so that the pulse looks like a step. The output is as follows.



As you can see, one can clearly state which circuit is low pass and which one is high pass.

- The final schematic is shown below



- As a side note, the *Add Trace* dialog box also supports expressions and certain functions like absolute value and derivative, which can provide additional insights into the circuit behaviour (the time derivative of the step response will give the impulse response).