University of Pennsylvania

Department of Electrical and Systems Engineering

ESE 216 LTSpice Quick Guide

Install the LTSpice from the webpage http://www.linear.com/LTspice and install it on your PC. It is free simulation software which has the SPICE simulator core and has many ideal and real device libraries available.

Note: There is an official guide from Linear Technology ® available at http://cds.linear.com/docs/en/ltspice/LTspiceGettingStartedGuide.pdf. You can read starting from page 13 to learn how to create a schematic design and simulate it using the LTSpice.

After installation, run the LTSpice software. Then create a new schematic by clicking on [icon] or choosing from the menu. You will be able to design a schematic for simulations.

Example

To demonstrate how to use LTSpice, let us take the example of designing and simulating an active 2nd-order Butterworth filter as shown in Fig. 1 and with the following specifications:

Parameter	Value
Cut-off Frequency	10kHz
$R_a(=R_b)$	1kΩ
C_a	22.5nF
C_b	11.25nF
Gain of Amplifier	Infinity (ideally)
Bandwidth of Amplifier	Much larger than then cut-off frequency
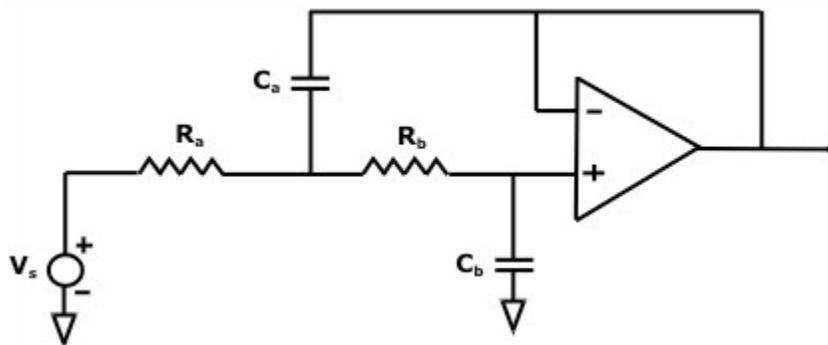
Fig. 1 The Schematic of an active 2nd-order Butterworth low-pass filter

1: Design the schematic

Fig. 2 Summary of Schematic Editor Toolbar

The design tool bar [toolbar image] has plenty of functions. You can move your mouse to each of the buttons to see the function.

We start with instancing the components into the schematic first by pressing the ⊅ button. And the screen as shown in Fig. 3 pops up.
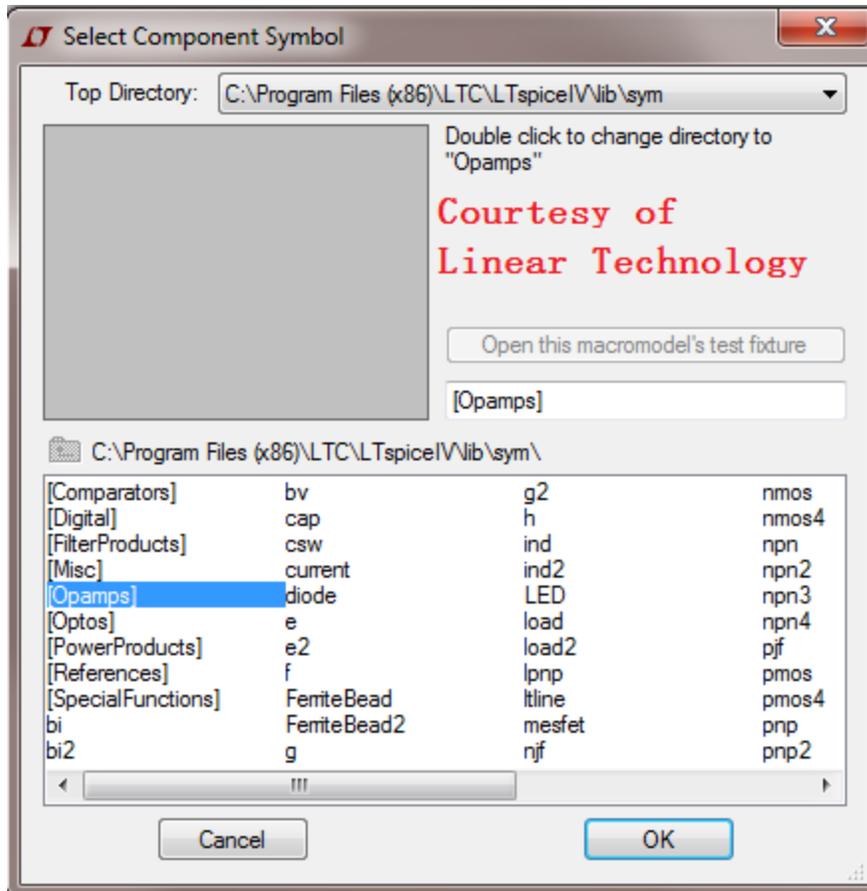
Fig. 3 Select Component Symbol window

We can use the op-amp LT1224 from linear technology (same as the mini project). Just double click the [Opamps] folder to enter it. Choose LT1224 and press OK. Then you will be able to place the symbol of the LT1224 amplifier on the schematic. Similarly, you can add the capacitors and resistors into the schematic. The ideal models of capacitors and resistors are available in the top-level directory named "cap" and "res", respectively. (You can just use the ideal models of capacitors and resistors for your mini project). At last you need to place the sources and ground symbol into the schematic. The voltage source symbol is available in the top-level directory of the Select Component Symbol window named "voltage" and the ground symbol is available on the design tool bar as ⏚. The schematic should look similar to Fig. 5 after you have placed all the components.

Hint: In the Select Component Symbol window, to return to the higher hierarchical menu, you need to double click on the [..] icon (as shown in Fig. 4). And to rotate the component when you are placing them, use "Ctrl+R".
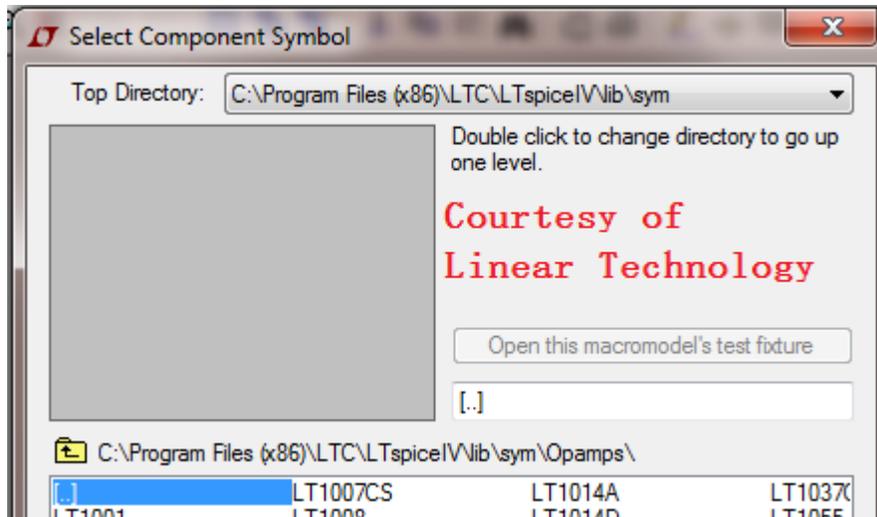
Fig. 4 Return to upper-level directory in the Select Component Symbol window
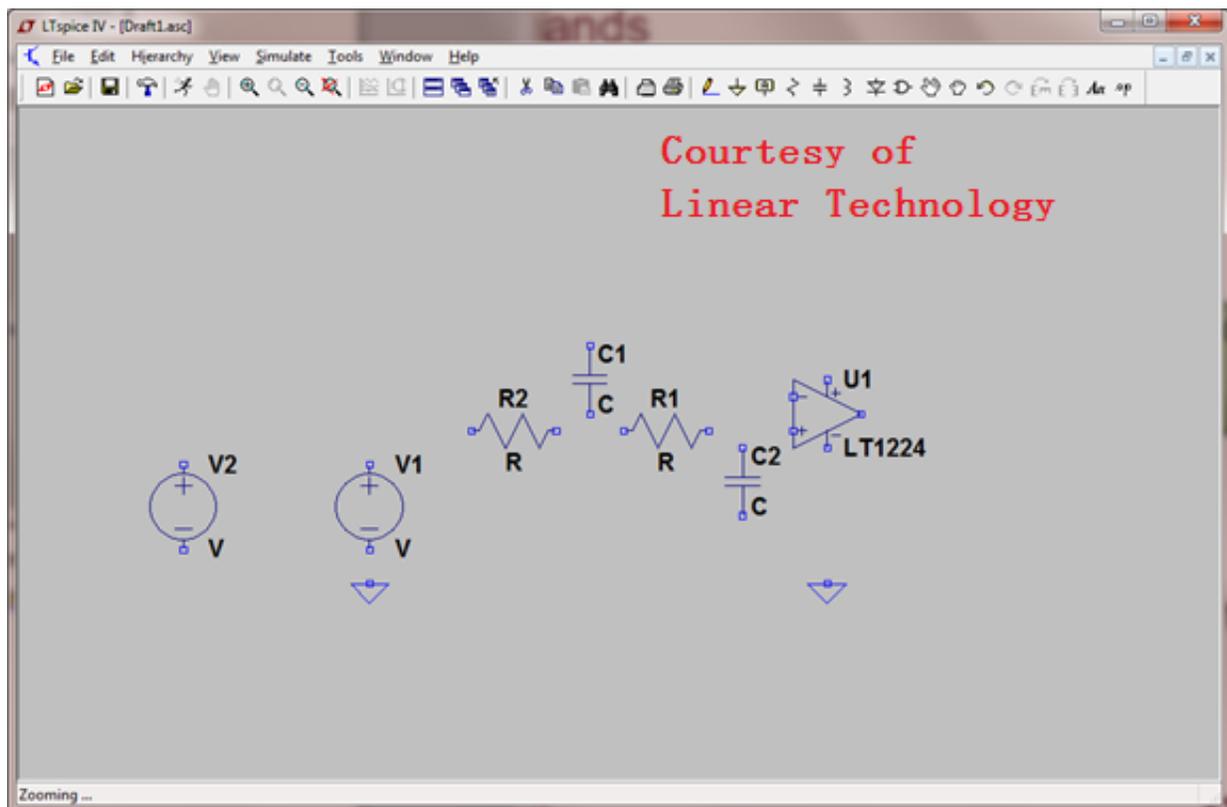
Fig. 5 Components for the filter design

After placing all the components, you can do the wiring by clicking the ✎ button. Click on the starting and stop points of the wires. You can also do the connection by naming the wires in the same net by using 🄰 . The connected schematic should look similar to Fig. 6.

Note: To name a net, just click on the ⒜ button, input the net name and attach the name to the wires which you want to connect in the same net.
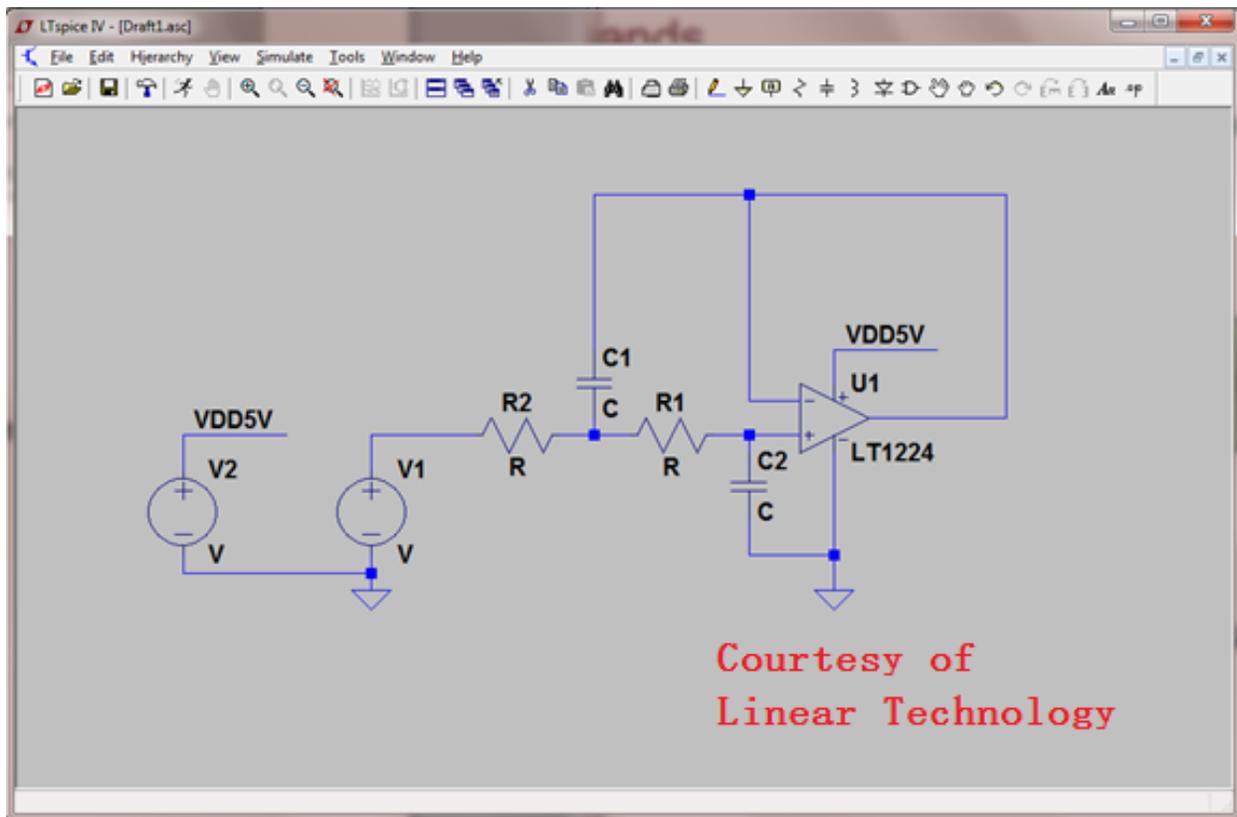
Fig. 6 Connected schematic

The final step is to add the values to the components. Just right click on the components to change the values in the pop-up window. Please change the values of the input voltage source according to Fig. 7. The final schematic should be looking like Fig. 8.
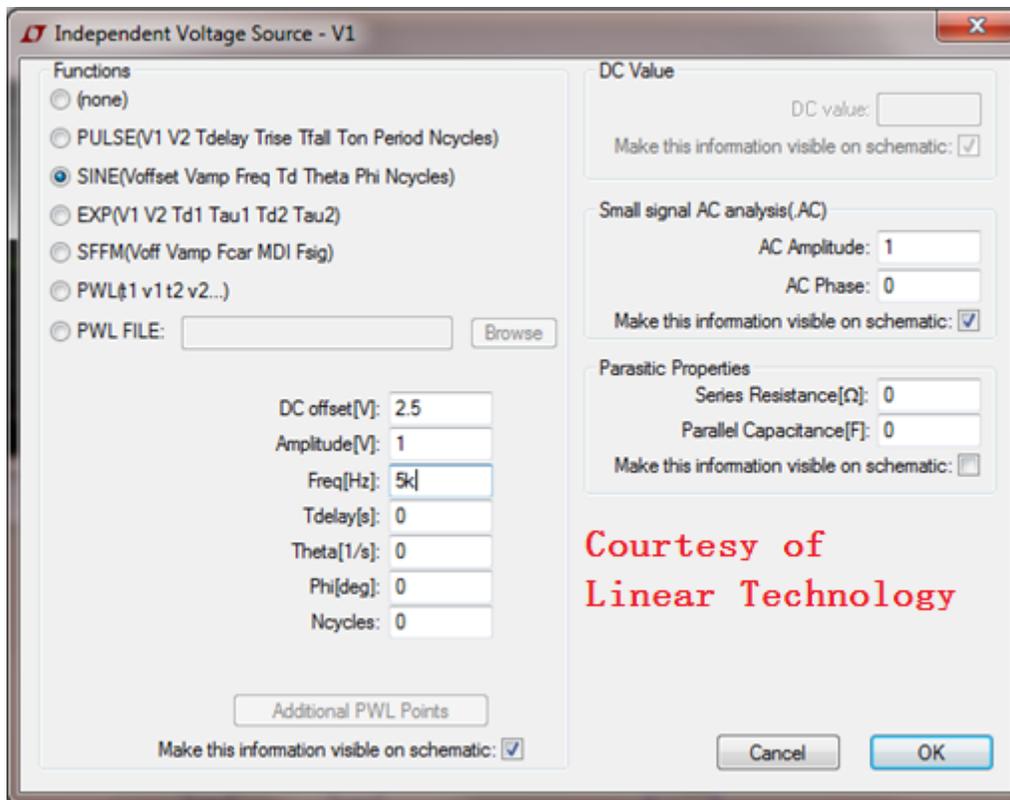
Fig. 7 Voltage source parameters setup
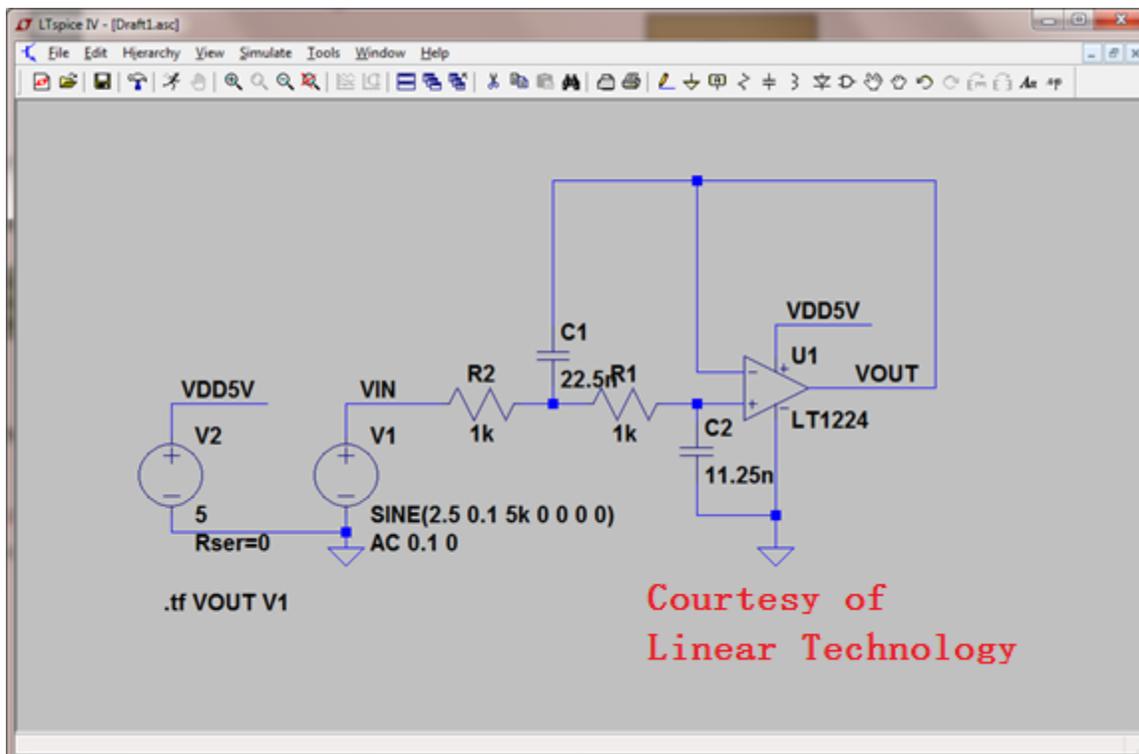
Fig. 8 Finished Schematic

2. Simulate and Plot the Schematic

Fig. 9 Available simulations in the LTSpice

In the LTSpice, there are 6 different types of simulations (analyses) available. To simulate the transfer characteristics of the filter, we should use the Small signal AC simulation function. To add a simulation in the schematic, choose "simulation" in the menu bar, and then choose "Edit simulation cmd". Select the "AC Analysis" page and set the parameters as shown in Fig. 10. Then click "OK".
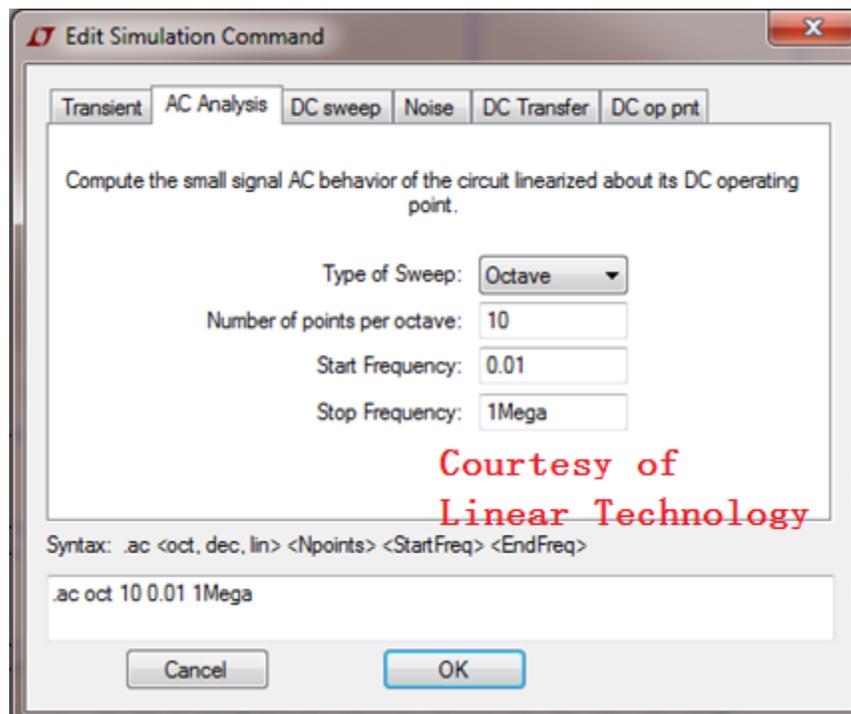
Fig. 10 Edit simulation command window
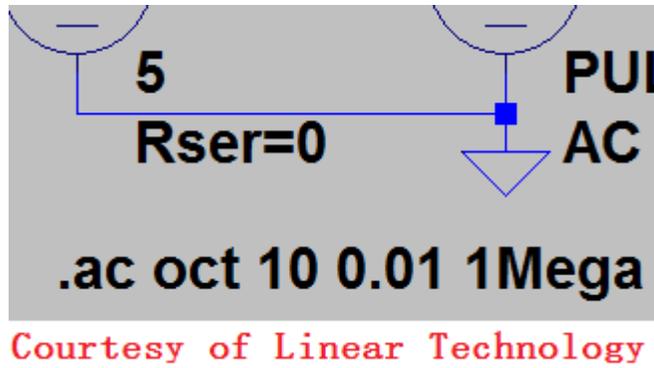
Fig. 11 Place the simulation option in the schematic

Place the simulation option tag in the schematic and then press the "run" ![run] button. Then the simulation will run and a plot window will appear in the LTSpice program as shown in Fig. 12.
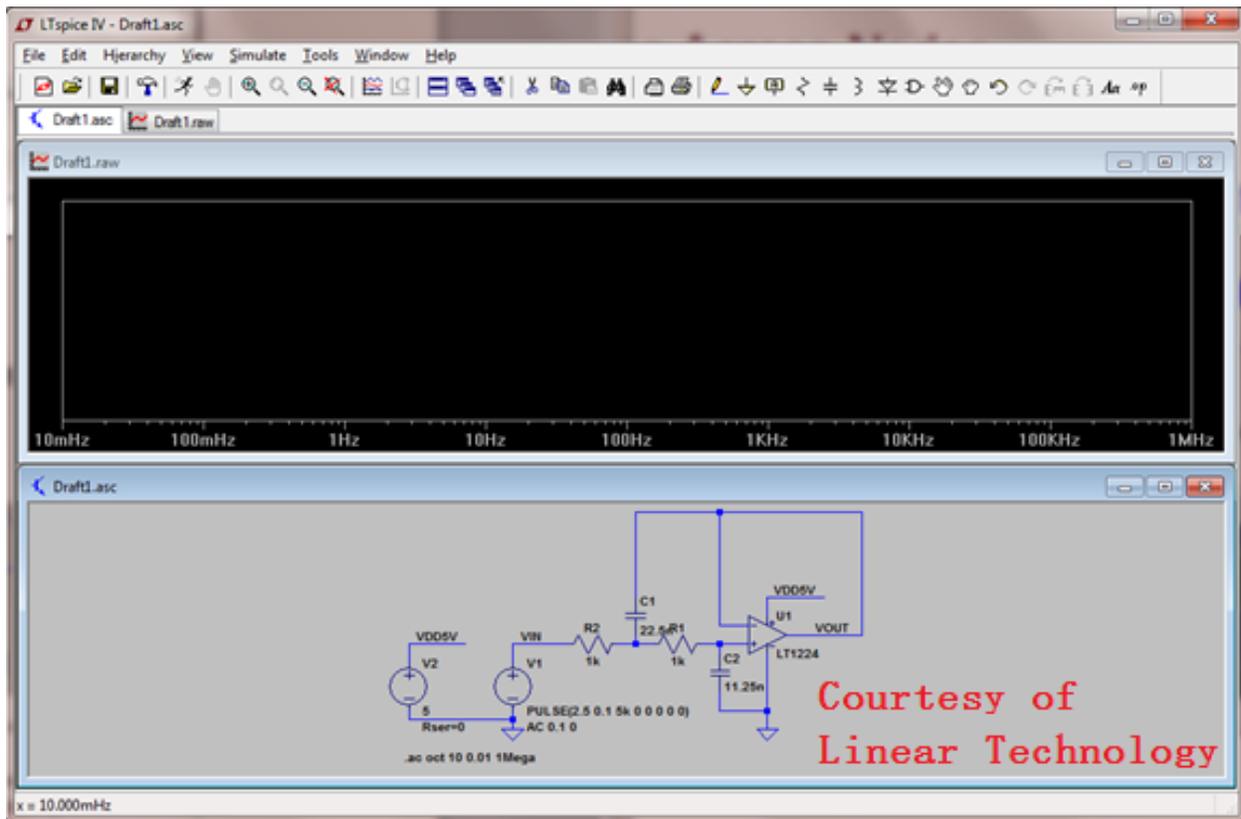
Fig. 12 LTSpice window after running ac analysis simulation

After simulation, you can plot the results by pressing the "Pick Visible Traces" ![traces] button. A pop-up window showing the plottable traces will appear. As shown in Fig. 13, all the voltages and currents and "Ix" are available for plotting. We care about the transfer function of the filter, so we need to plot the output voltage of the filter versus frequency (since the input voltage is not changing with frequency).

Note: Since there is no built-in gain (Vout/Vin) plotting function available in the software, we can just plot the output voltage by setting the input voltage amplitude equal to 1 for the AC analysis. To get the transfer function, you can just subtract the output voltage dB value by 0dBV, which indicates the dB value of the output voltage equals to the gain.
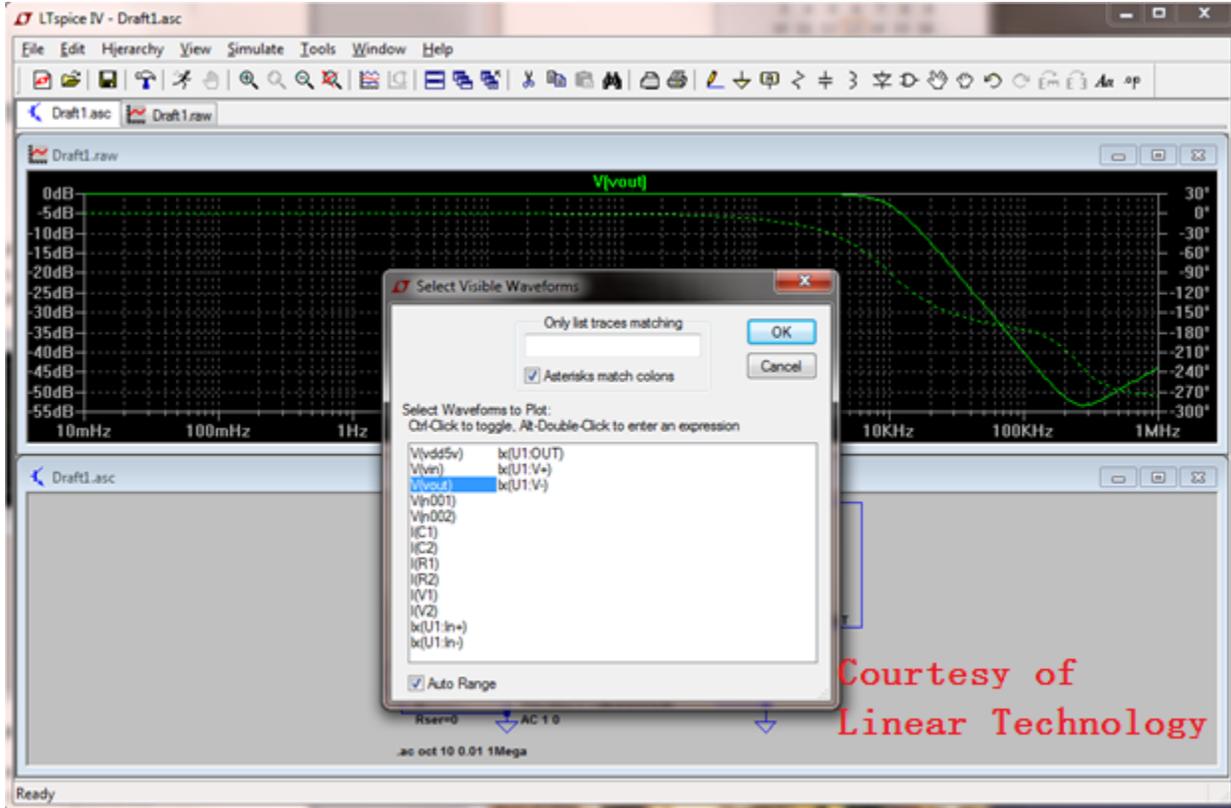
Fig. 13 Plot of the amplitude of the filter output voltage

The plot of the output voltage amplitude is shown in Fig. 13. The solid line is the magnitude and the dotted line is the phase. You can see at low frequency, the amplitude is 0dBV which equals to the input voltage amplitude, and it decreases with frequency.

Note: You can zoom in on the plot by pressing the left button of your mouse and drag to include the area that you want to zoom in. And you can move the mouse along the plotted curve and read the corresponding value at the bottom of the window.

To read the data points on the plot, click on the green "V(vout)" title on plot and a cursor and a pop-up window showing the readings will appear (as show in in Fig. 14). Drag the cursor to the desired points on the plot to get the data.

Therefore, in this quick guide, we have designed and simulated the schematic of an active 2nd-order Butterworth filter. We have also plotted the output voltage amplitude to interpret the transfer function of the filter.

Note: In this example, the gain of the filter is unity. This means the input voltage and output voltage are changing between (2.5V-1V) = 1.5V and (2.5V+1V) = 3.5V, which is within the range of supply to the LT1224 op-amp (0V~5V). In your mini project, you are required to design the filter with a gain equals to 16, which means, if you set the input voltage amplitude to 1V, the output will be varying in the 2*16*1V = 32V range. The 32V range may exceed the supply range of the op-amp in real experiments. However, in the ac simulation, due to the nature of the AC simulation method, this is not a problem (this is not the case for transient simultions). So you can still using the same dc biasing conditions (5V VDD and 0V GND) to run the simulation and get the correct results even the actual output voltage range is larger than the supply range of the circuit. You just need to keep in mind that you should not allow the signal level exceed the supply range in real experiments.
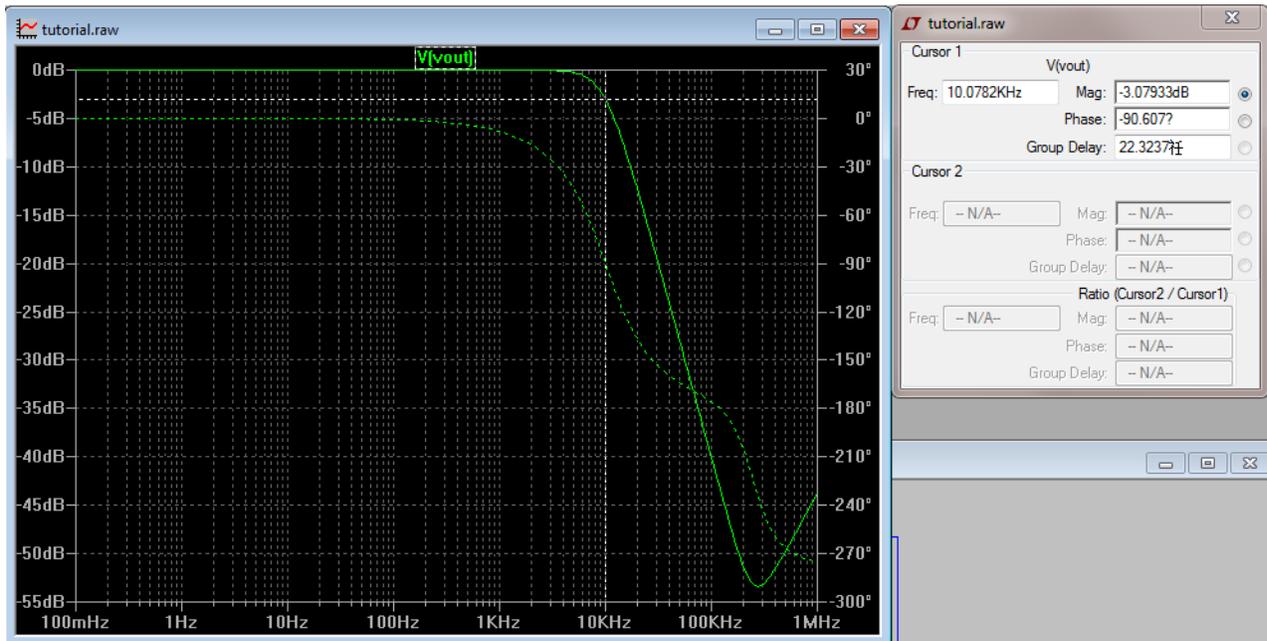
Fig. 14 The reading on the plot