|	
From:	"CircuitLab Academy" <support@circuitlab.com>
To:	"Metha Kerdkeaw" <chon7540@chontech.ac.th>
Date:	11/4/2023 10:56:10 PM
Subject:	Filters = Frequency response (Bode plot)

Hi Metha,

Amplifiers and filters require thinking in the frequency domain. Flip through the screenshots below to learn how to display a Bode plot in just a few clicks.
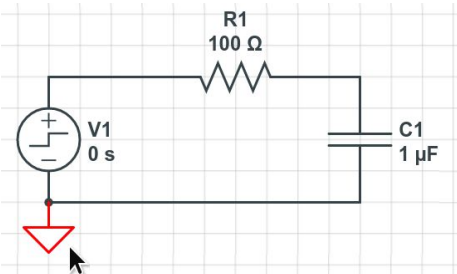
From the toolbox, click and drag a voltage step source, a resistor (press R to rotate horizontal), and a capacitor onto your schematic:
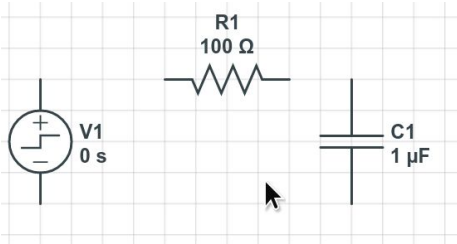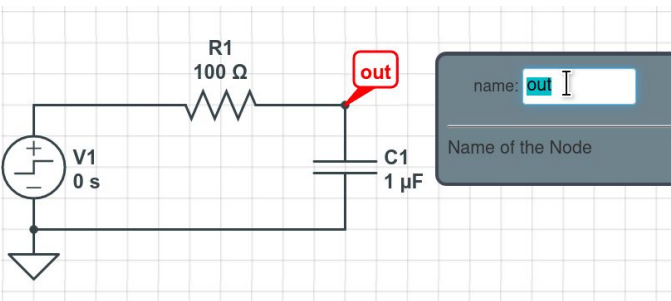

Drag from component endpoints to connect the circuit with wires:
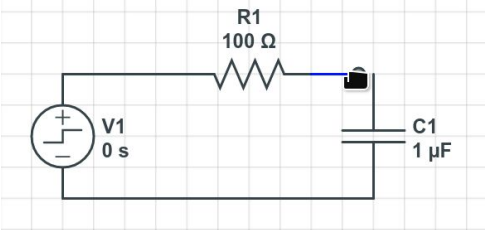
Press G and click to insert a ground node at the bottom of the voltage source:

Press N and click to insert a node name label at the top of the capacitor. Double-click and rename it "out":

Great! We've finished drawing an RC low-pass filter. Let's have the simulator generate the Bode plot.

Click Simulate at the bottom of the window to open the simulation settings panel, and then click Frequency Domain:

Click the Input field and an autocomplete appears. Click to select V1:

This tells the simulator that V1 is our driving source, and it should show us the small-signal response relative to changes in that source.

Next, change Points/Decade to 1000:

This tells the simulator to give us more resolution in the frequency domain, at the cost of taking more time to run the simulation.

Click on the node name "out" and observe that two expressions are added to the simulation settings Outputs list:
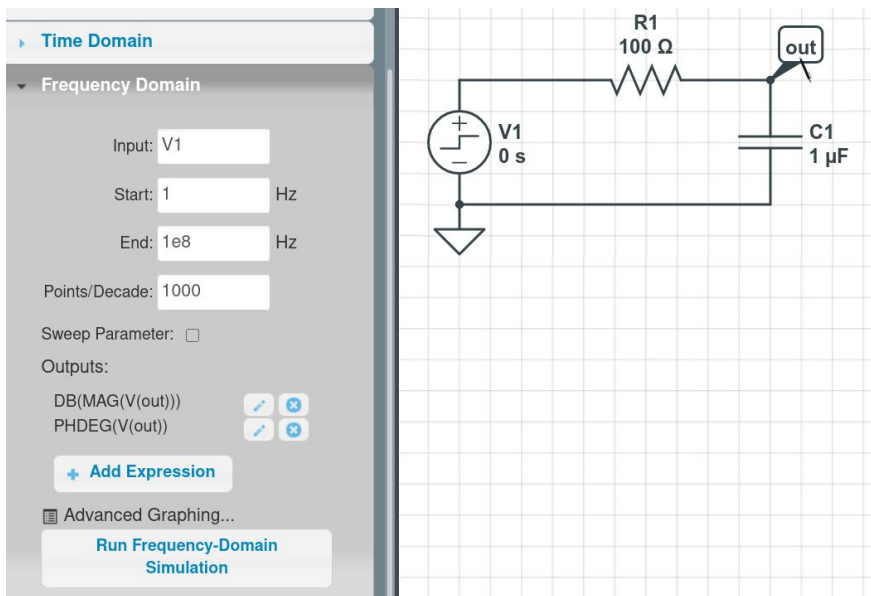
The first expression is DB(MAG(V(out))), which asks for the magnitude in dB. The second is PHDEG(V(out)), which asks for the phase in degrees. These are the key ingredients of a Bode plot.

Click Run Frequency-Domain Simulation and a plot window appears:
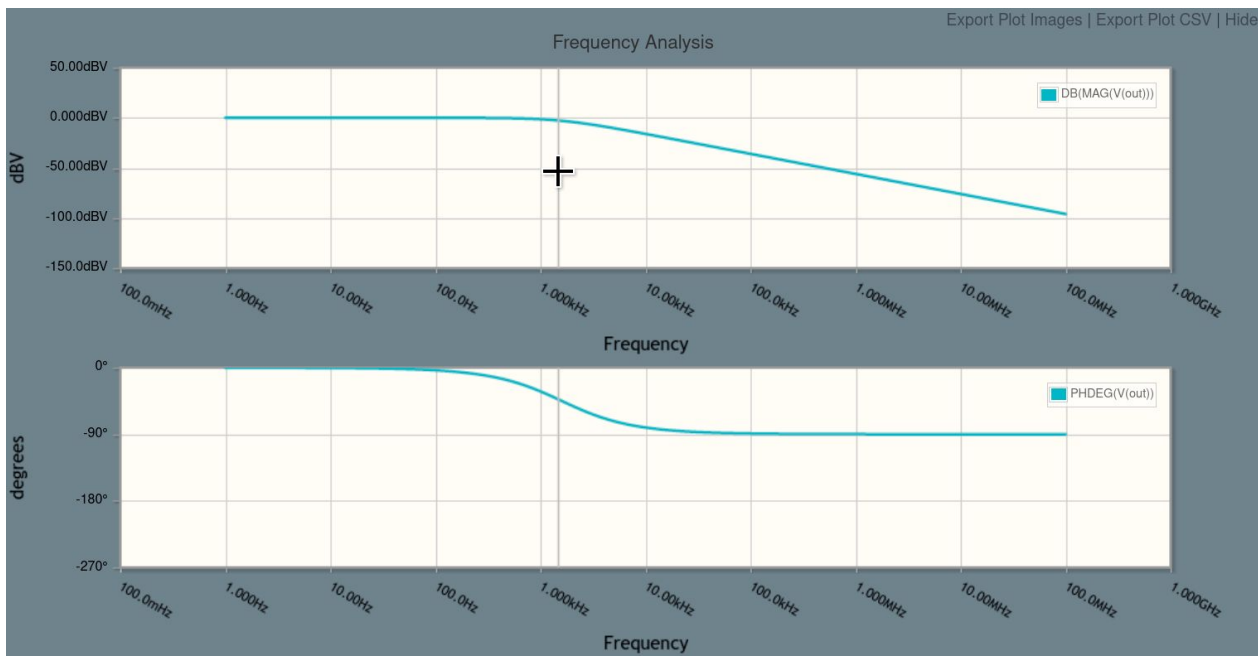
This is a classic Bode plot, magnitude and phase subplots, with a logarithmically-scaled x-axis.

Hover over the magnitude plot to identify the -3 dB point:
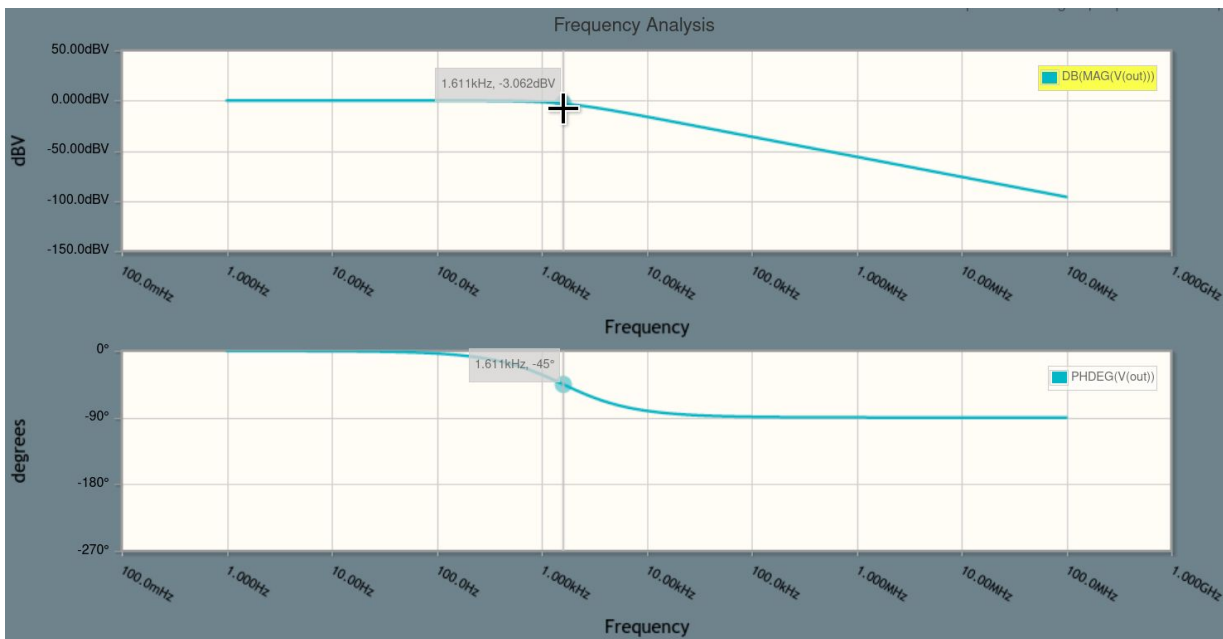
Unsurprisingly, at the -3dB frequency, the linked plot cursors show us that the phase is -45 degrees.

Click to focus the simulation settings window and click the checkmark next to Sweep Parameter:

Type "C1.C" in this Parameter box, change Sweep Type to "Custom", and enter "1u, 10u" in Values:

Click Run Frequency-Domain Simulation and a plot window appears:

This new plot lets us quickly compare the frequency response of the circuit with two different capacitor values for C1, with 1uF in blue and 10uF in orange. The larger capacitance clearly has a lower cutoff frequency (10x larger capacitor means 10x lower bandwidth).
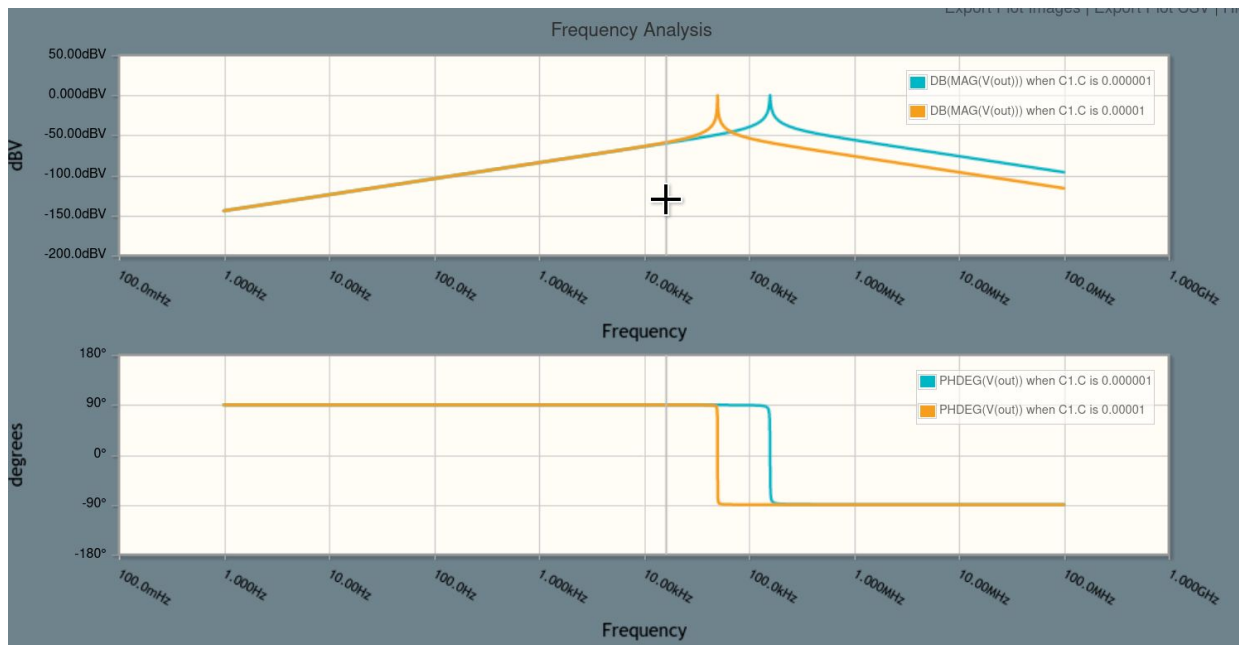
Click Build at the bottom of the window to go back to editing the circuit:
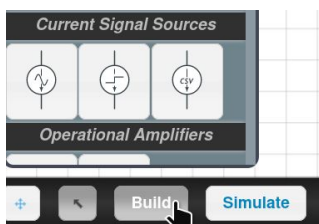
Drag an inductor from the toolbox and drag to wire it in parallel with C1:
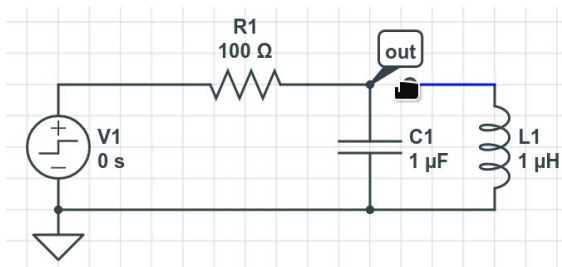
Press F5 to run the simulation again:

Is this what you expected to happen by adding an inductor? L1 and C1 now form a parallel LC circuit and have a resonance frequency, shown by the sharp peak in the magnitude plot.

At the resonance frequency, the parallel LC looks like an overall high impedance, and the signal passes through the filter at almost no loss (0dB). At lower frequencies, the inductor reduces the gain, and at higher frequencies, the capacitor does so.

Also, when you compare the plots for the two capacitor values, note that for a 10x change in the capacitance, the resonance frequency does not move by a factor of ten -- in fact the frequency moves by a square root of ten.

As a bonus, let's show the step response of this filter. Click to focus the simulation settings window, then click Time Domain:
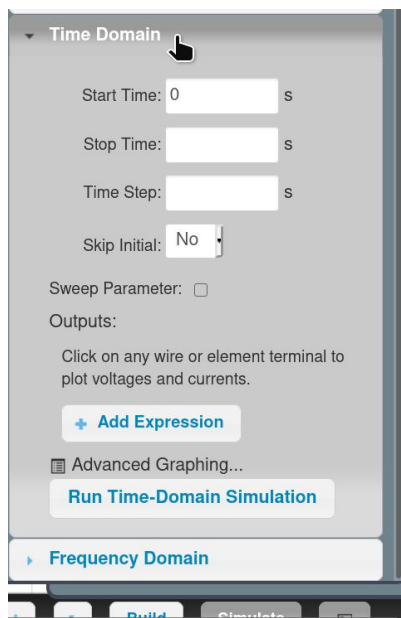
Enter a Stop Time of "500u" and a Time Step of "1n", and click the "out" node name to add V(out) to the Outputs list:

Start Time: 0 s

Stop Time: 500u s

Time Step: 1n s
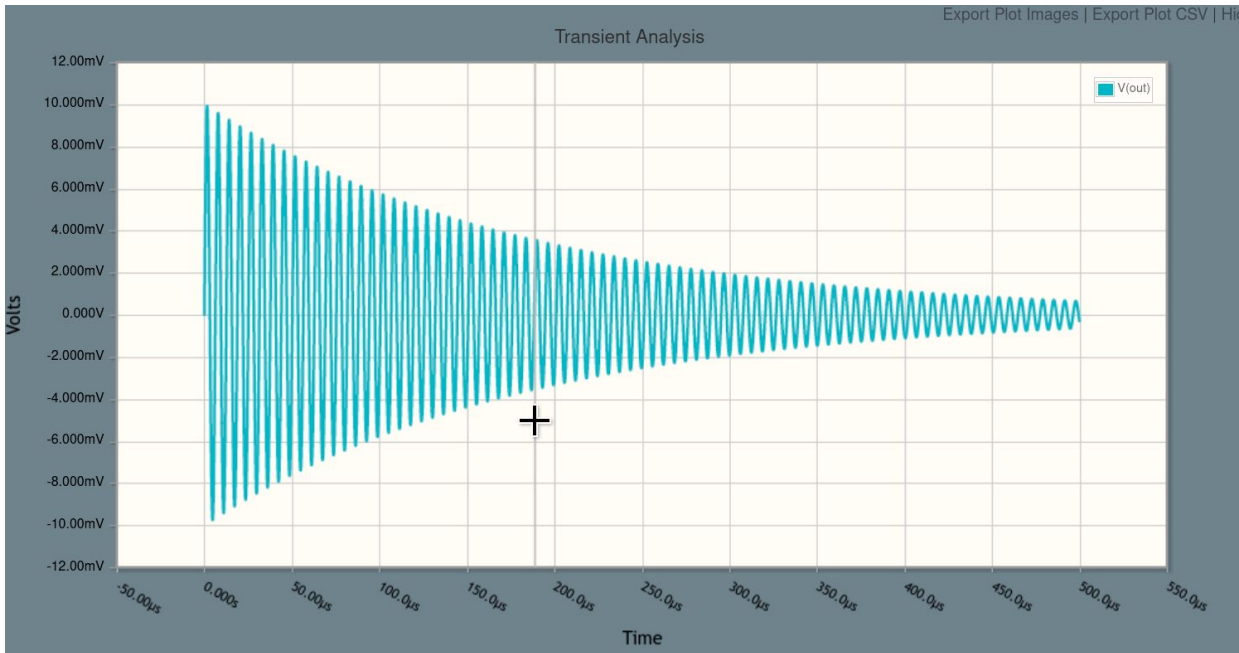
Skip Initial: No

Sweep Parameter: ☐

Outputs:

V(out)

+ **Add Expression**

▤ Advanced Graphing...

Run Time-Domain Simulation

Click Run Time-Domain Simulation. After a few seconds of running the simulation, a plot window appears:
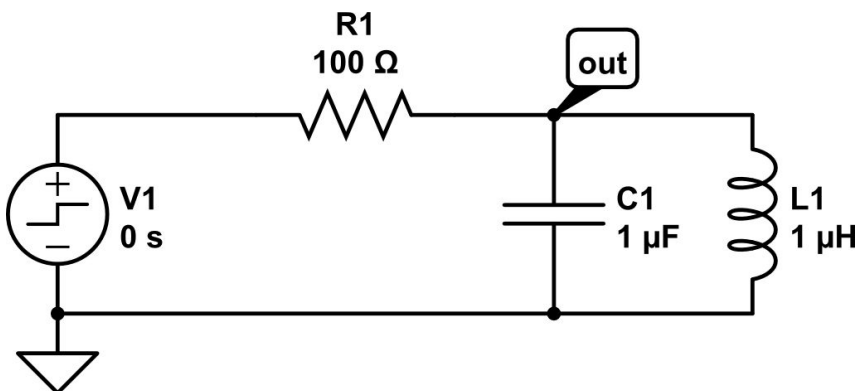

Wow! That's a lot of ringing in response to an input step.

You can modify inductor L1 and change its R_ESR to see the effects of series resistance inside the inductor. We'll leave that as an exercise for you.

That's it! You're now free to simulate the frequency response of filters and amplifiers.

Click below to open the final circuit , or try it yourself from scratch (recommended).