

LTSpice Parameter Sweep Tutorial

ECE 202 Signals and Systems I

Andrew Herdrich

Department of Electrical and Computer Engineering

Portland State University

January 7, 2007

Version 2

AC sweep analyses in LTSpice can be used to analyze the frequency response of a circuit with fixed parameter values. If several parameter values need to be examined, the designer can either manually enter the values and simulate the circuit several times to view the response, or use the SPICE dot directive “.STEP”. The .STEP directive allows up to three parameters to be swept across an arbitrary range of values in a single simulation run. Note, however, that each new parameter sweep can drastically increase the simulation time for the circuit if parameter sweep ranges are not carefully chosen.

As an example of the parameter sweeping capabilities of LTSPICE, consider the following elementary high-pass filter circuit:

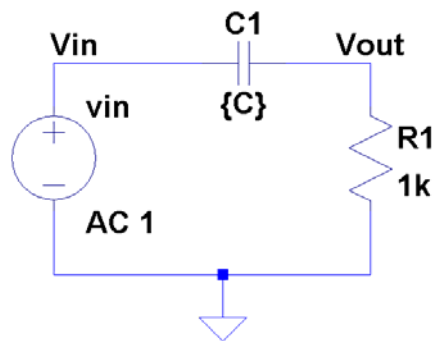


Figure 1. A simple HP filter used to test LTSPICE parameter sweeps.

Note that in Figure 1 above that the voltage source value has been changed to “AC 1” in preparation for an AC sweep to create a Bode plot, and the value of *C1* has been changed to *{C}*, which allows the substitution of external values. Students familiar with various SPICE simulators may note that this procedure is quite similar to that used to add parameters to PSPICE simulations.

Adding the SPICE directive “.ac dec 10 1 100k” to the circuit and the parameter definition “.param C=1uF” allows an AC sweep to be performed, as shown below. The AC directive performs a logarithmic sweep from 1Hz to 100kHz, with 10 points per decade, and the PARAM directive inserts a temporary value for the capacitor of 1 μ F.

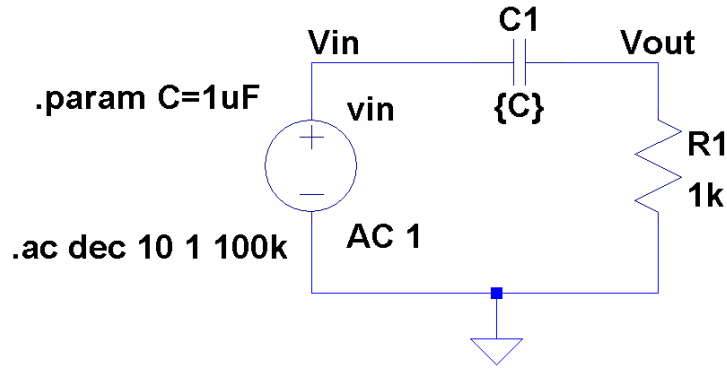


Figure 2. The HP filter set up for an AC sweep to generate a Bode plot.

Performing a simulation on the circuit above (Simulate Menu->Run or Alt-S-R) yields the Bode plot shown below. Note that the cutoff frequency appears to be around 160Hz, and the frequency axis is shown on a logarithmic scale.

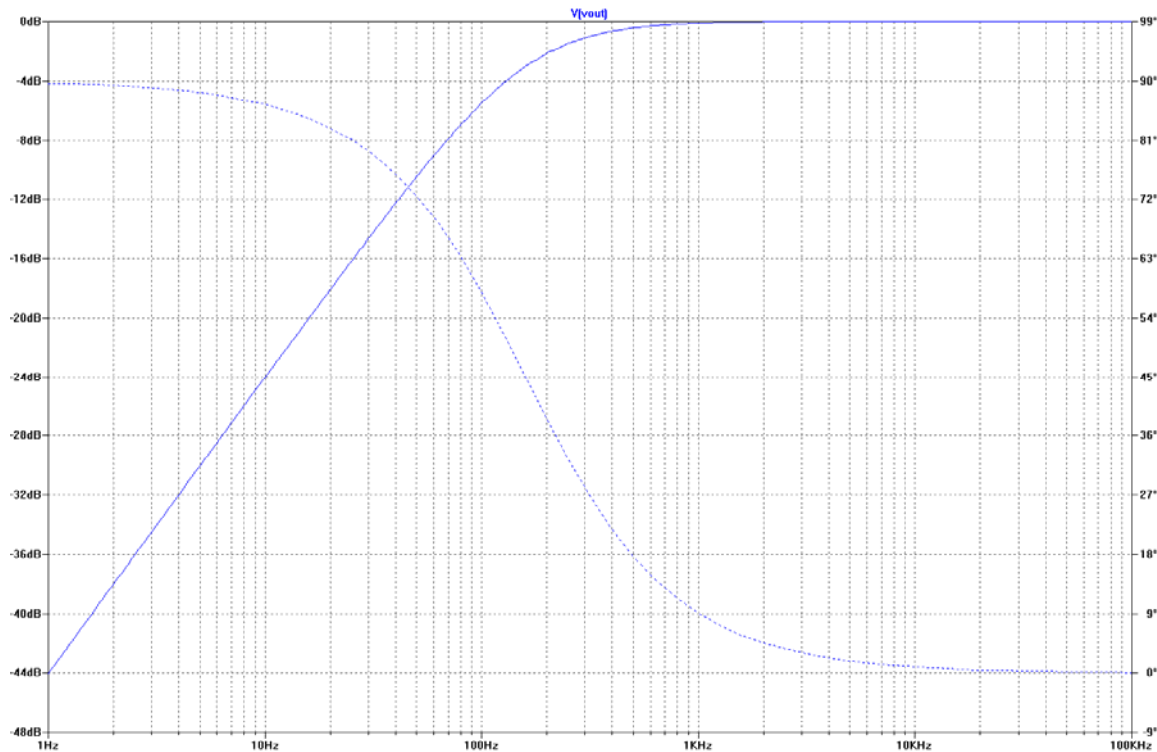


Figure 3. A Bode plot for a simple RC HP filter. Phase is shown as a dotted line.

The above simulation works well for a single component value, but what if 5 capacitor values need to be tested? One solution would be to change the PARAM directive that we set up earlier five times and run the simulation repeatedly to create five separate Bode plots. A more elegant solution is to use the SPICE dot directive STEP to automatically insert the parameter values for us. For example, to test capacitors from 1 μ F to 10 μ F in 2 μ F steps, the directive “.step param C 1uF 10uF 2uF” could be used. This steps the parameter C through the requisite value range in a linear fashion. The directive can be added to the circuit diagram as follows:

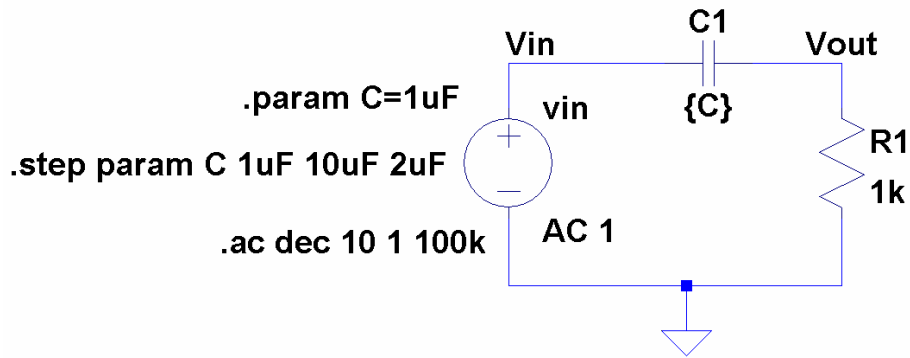


Figure 3. The RC HP filter set up for a parameter sweep with values from 1μF to 10μF in 2μF steps.

Simulating the circuit with the STEP directive produces the following Bode plot:

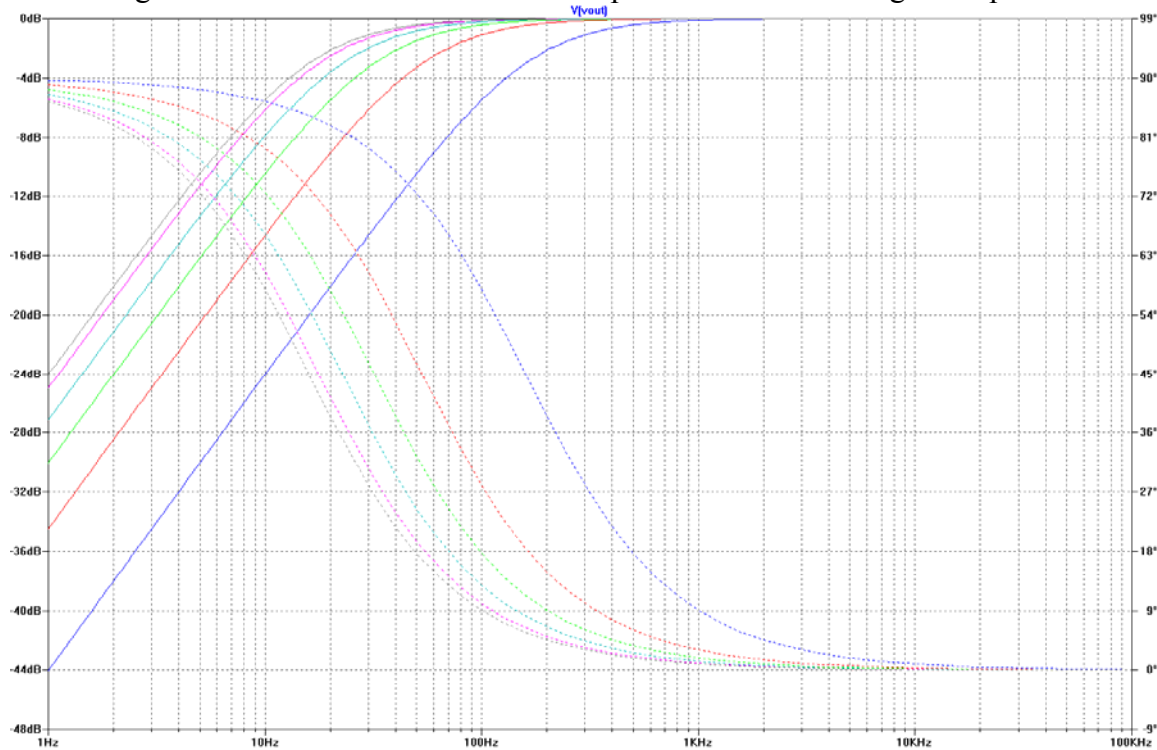


Figure 4. The Bode plot created by sweeping capacitor values. Phase is indicated with a dashed line.

It is apparent from the above plot that as the capacitor value is increased, the cutoff frequency decreases, from around 160Hz to around 16Hz.

This is a useful result. Other parameter sweep types can also be created, such as sweeping a component through a range logarithmically over a decade or octave. For instance, the capacitor can be swept logarithmically through the range of 100nF to 1μF with 8 points per decade using the directive “.step dec param C 100nF 1uF 8”, as shown below:

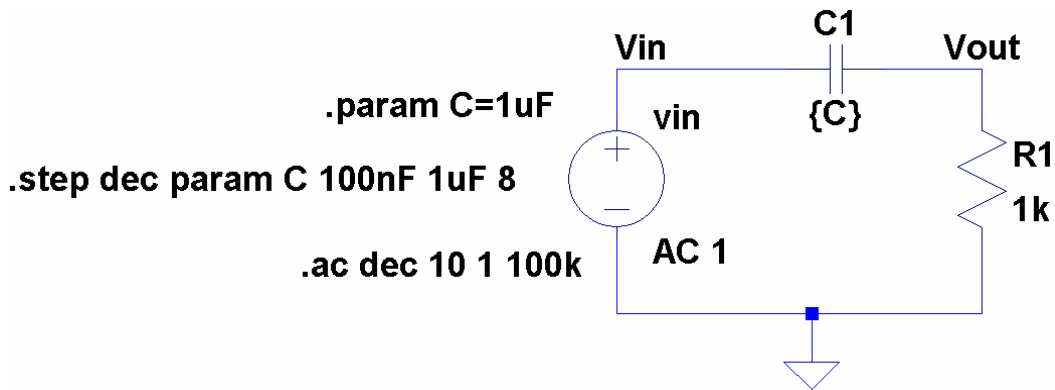


Figure 5. Using the SPICE STEP directive to sweep the capacitor through the range of 100nF to 1μF logarithmically with 8 steps per decade.

The result of the logarithmic parameter sweep is shown below:

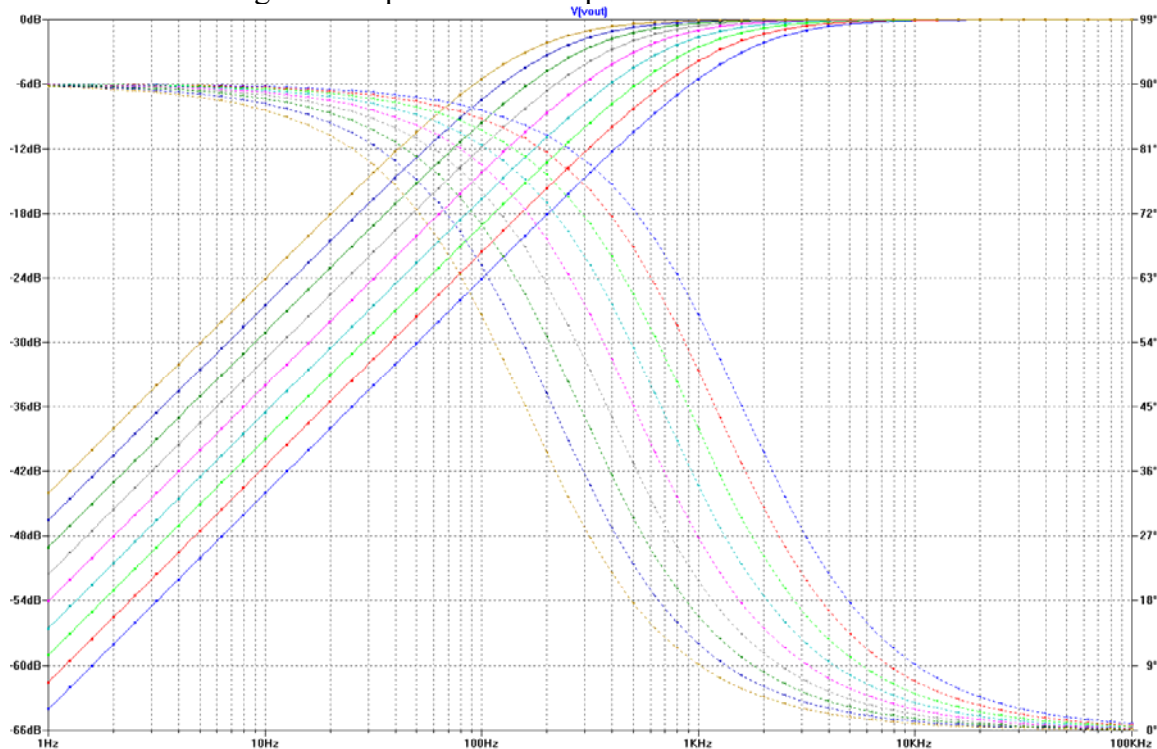


Figure 6. The result of logarithmically sweeping the capacitor through 9 values.

As is shown in the plot above, the corner frequency ranged from around 160Hz to around 1.6kHz. A simple list can also be used to sweep parameter values. For instance, the directive “.step param C list 0.47uF 1uF 2.2uF 10uF” can be used to test the capacitor values 0.47μF, 1μF, 2.2μF, and 10μF, which we just so happen to have on hand. This is practically useful, since it lets us view the range of possible corner frequency settings. The SPICE directive can be implemented as follows:

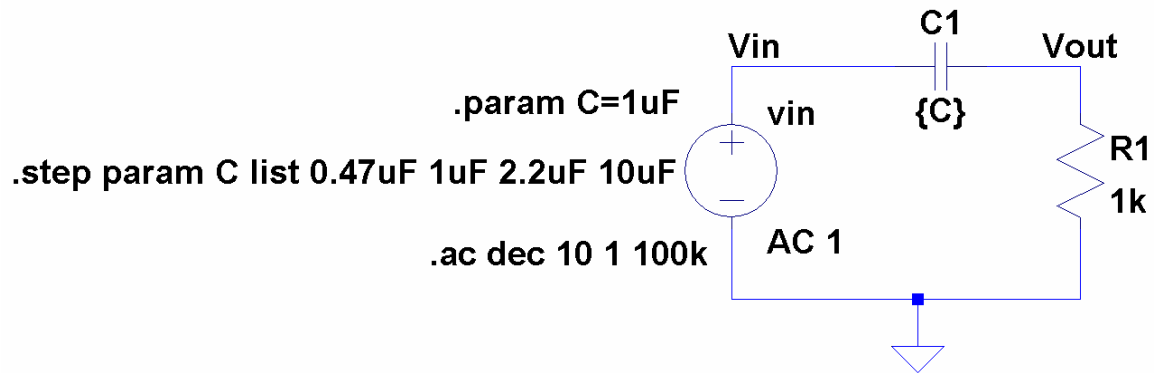


Figure 7. A simulation set up to test the capacitor values $0.47\mu\text{F}$, $1\mu\text{F}$, $2.2\mu\text{F}$, and $10\mu\text{F}$ in the filter circuit above.

The results of the simulation are shown below:

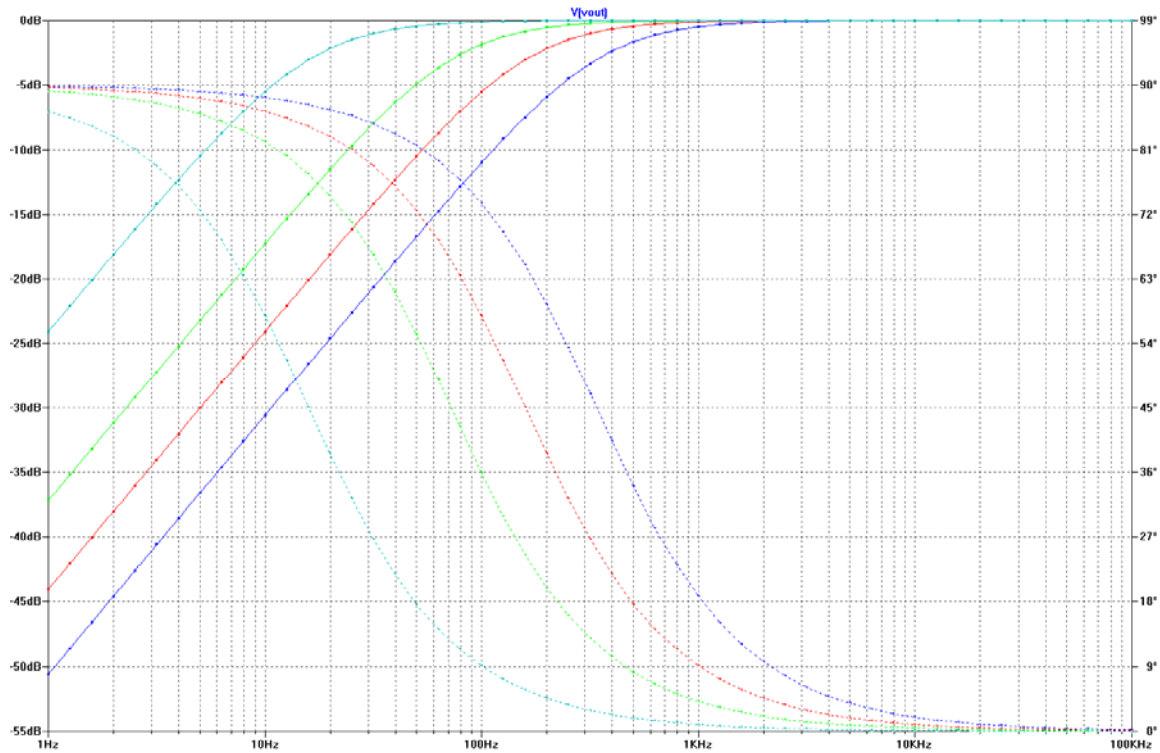


Figure 8. The corner frequencies created through the use of the capacitor values $0.47\mu\text{F}$, $1\mu\text{F}$, $2.2\mu\text{F}$, and $10\mu\text{F}$.

The above example shows how to sweep through a list of capacitor values to determine where possible corner frequencies lie. It is also possible to simultaneously test both resistor and capacitor values, which can be useful when designing a circuit, as is shown in the example below.

Example:

Suppose that you have been asked to design a first-order RC HP filter, and have the following resistor and capacitor values available: $0.47\mu\text{F}$, $1\mu\text{F}$, $2.2\mu\text{F}$, and $10\mu\text{F}$, 10Ω , $1\text{k}\Omega$, and $1\text{M}\Omega$. Create a Bode plot showing all possible corner frequencies that can be implemented.

Solution:

Use two simultaneous sweeps to determine the approximate corner frequencies. The first SPICE directive will be a list, “.step param C list 0.47uF 1uF 2.2uF 10uF”, and the second will be another list, “.step param R list 10 1k 1meg”. The two simultaneous sweeps can be conducted since the STEP command operates recursively (up to three levels deep), and the simulation will still execute quickly, since the number of parameters to test is small. The test circuit diagram is shown below.

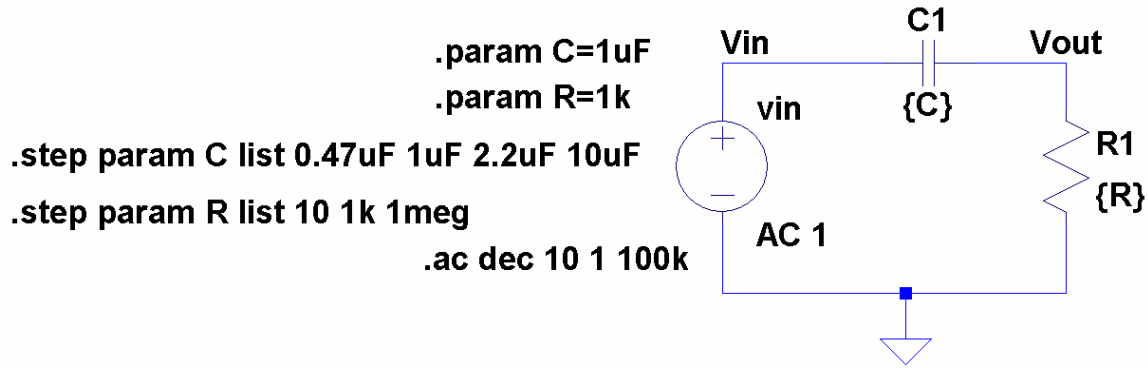


Figure 9. The directive sequence used to sweep two parameters simultaneously, in this case, R and C , through a range of values.

The resulting Bode plot showing the possible corner frequencies is shown below:

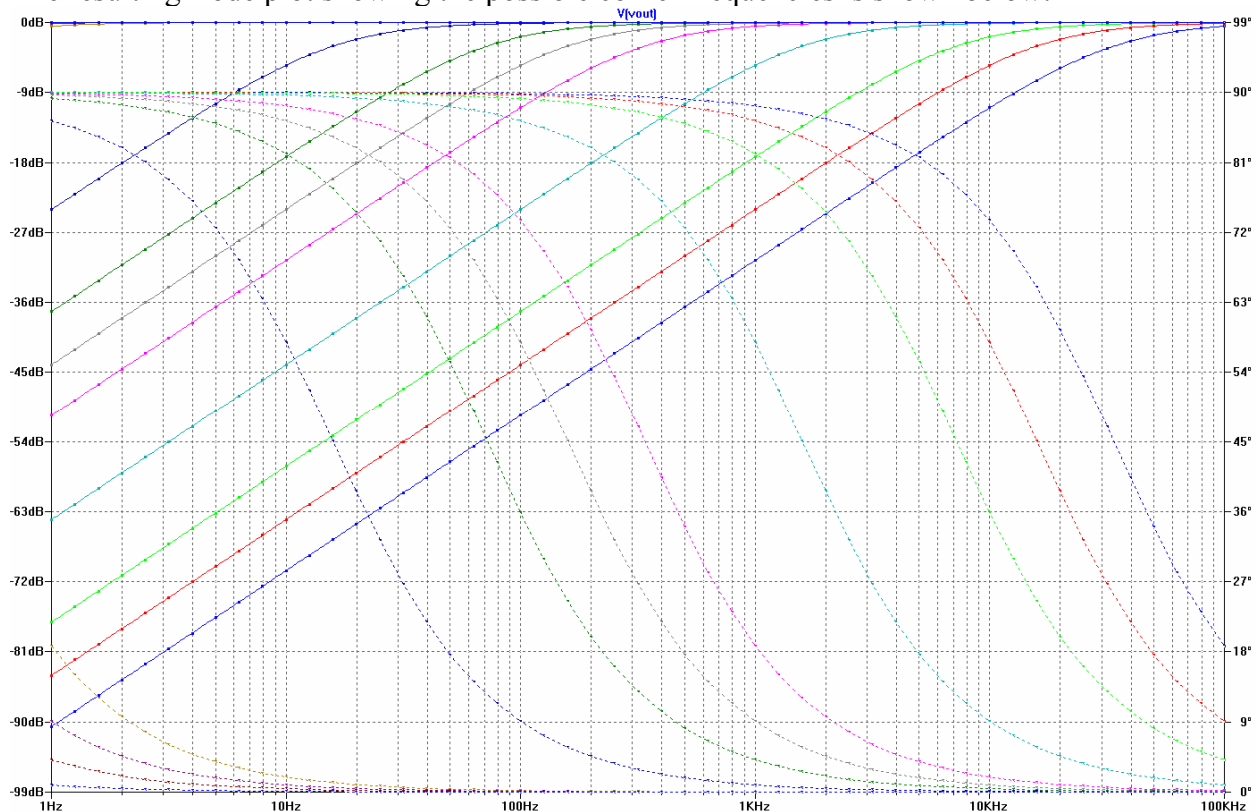


Figure 10. The result of sweeping both resistor and capacitor values simultaneously. Note that there are 12 possible transfer curves, which makes sense, based on the fact that we tested four capacitor values and three resistor values. Phase plots are indicated with dashed lines.

As is evident from the examples above, parameter sweeps can make short work of visualizing the effect of changing component values on circuit performance, and can aid in the design of circuits when detailed numerical analyses are not necessary.

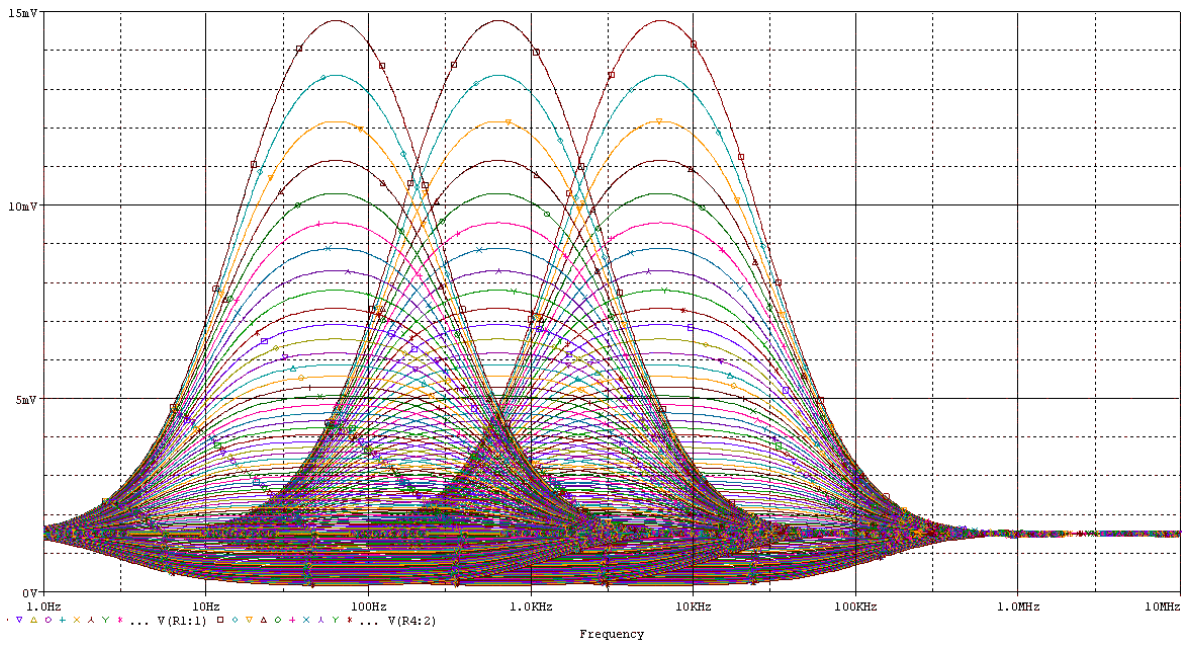


Figure 11. A more advanced parameter sweep showing the output of three summed bandpass filters.

References:

- [1] Switcher Cad III User's Guide, <http://ltspice.linear.com/software/scad3.pdf>