

MULTISIM DEMO 9.4: AC PARAMETER SWEEPS IN A RADIO TUNER CIRCUIT

How do you select frequencies in a radio? You use a variable filter which we generally call a tuner. The following Multisim demo is the simulation of a basic FM tuner. FM tuners are somewhat more complicated than the one presented so this is really more of a general simplified example of another real-world application of filters.

As you can see in Figure 9.4.1, the Antenna picks up a signal from the surrounding medium and that signal, a small voltage, is run through a band-pass filter (or tuner) composed of L1 and C1. The output of this tuner is fed into an amplifier

The FM band extends from 87.5 MHz to 108.0 MHz in the United States, so we want a tuner which can pass frequencies in this band relatively easily.

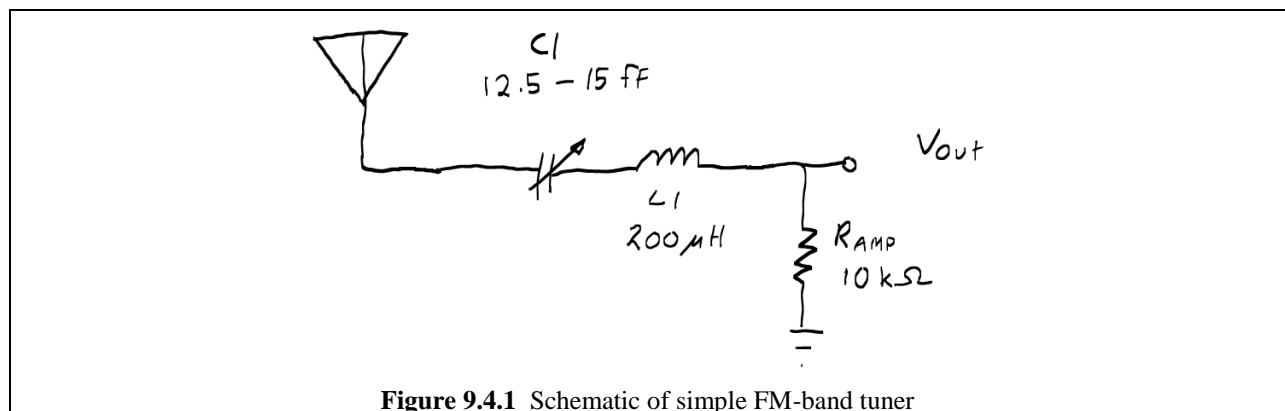


Figure 9.4.1 Schematic of simple FM-band tuner

When we go and construct this circuit in Multisim, we represent the antenna with an AC_VOLTAGE component. What should we do for the variable capacitor? Multisim has variable capacitors, but they will be of little use to us for this simulation. Unfortunately you cannot attach the Bode Plotter instrument and change a variable capacitor real time (Go ahead I dare you to try it...it doesn't work); You need to stop the simulation, adjust the capacitor, and then start the simulation again. It's just annoying, and because of that we might as well just invest the time in a Parameter Sweep/AC Analysis. We'll make our "variable capacitor" a normal fixed capacitor which we vary in the Parameter Sweep Analysis. When everything is put together your Multisim schematic should look like that shown in Fig. 9.4.2.

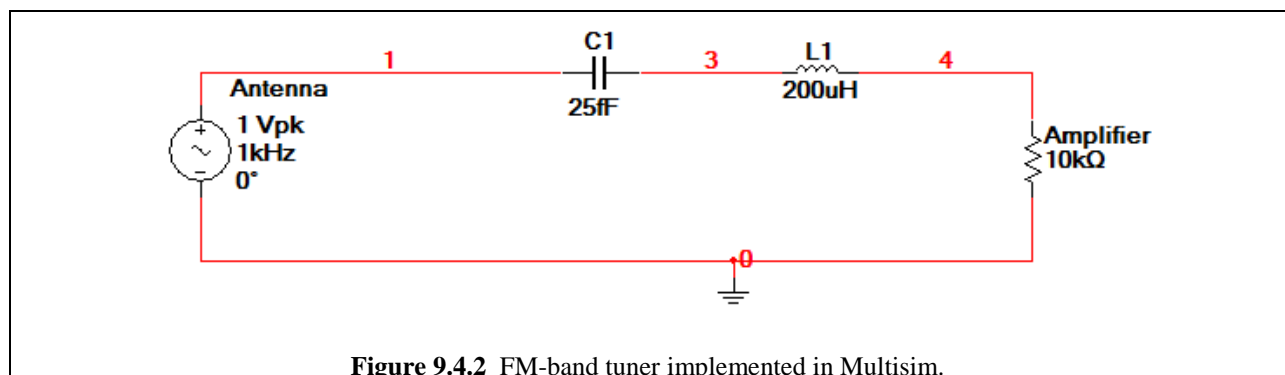



Figure 9.4.2 FM-band tuner implemented in Multisim.

So we need to do a parameter Sweep Simulation. To do one of these, go to Simulate>Analyses>Parameter Sweep, or just click on the down-arrow on the  button and select Parameter Sweep.

This will bring up the window shown in Figure 9.4.3 on the next page.

1. Go to the Analysis Tab
2. In the Sweep Parameter heading, make sure the Device Type is set to Capacitor
3. Make sure the name is “cc1”. There is only one capacitor in this simulation, so it should be “cc1” regardless.
4. Make sure the Parameter is set to capacitance (This is the parameter of the device you will be sweeping).
5. Under the Points to sweep heading, set Start to 12.5 pF
6. Set Stop to 15 pF
7. Set # of points to 3
8. Ensure that Increment is 1.25 pF (it should do the arithmetic automatically, but you never know)...
9. Under the More Options heading, make the Analysis an AC Analysis
10. Make sure that the “Group all traces on one plot” option is checked.
11. Click on Edit Analysis (this will bring up a simplified AC Analysis window)
12. Set it to the values shown in Figure 9.4.4.
13. Click OK
14. Click on the Output Tab of the Parameter Sweep Window, and insert an equation expressing the ratio of the output over the input. In the case of the schematic shown in Fig. 9.4.2, this would be $V(4)/V(1)$... (Your variables may be different since the circuit in Fig. 9.4.2 has skipped a node number 3.)
15. Click Simulate.

The plot which appears, should resemble what is shown in Fig. 9.4.5. Using the cursors in the Grapher window, we can find the maximums for the three tunings.

12.5 fF:	100.606 MHz	(Adult Contemporary)
13.75 fF:	96.0606 MHz	(The Greatest Hits of Yesterday and Today)
15 fF:	91.8182 MHz	(Public Radio).

If you want a different take on the plots, you can adjust the vertical scale to be in Decibels by:

1. Right clicking on the graph and selecting Properties.
2. Select the Left Axis tab.
3. Under the Scale field select: Decibels.

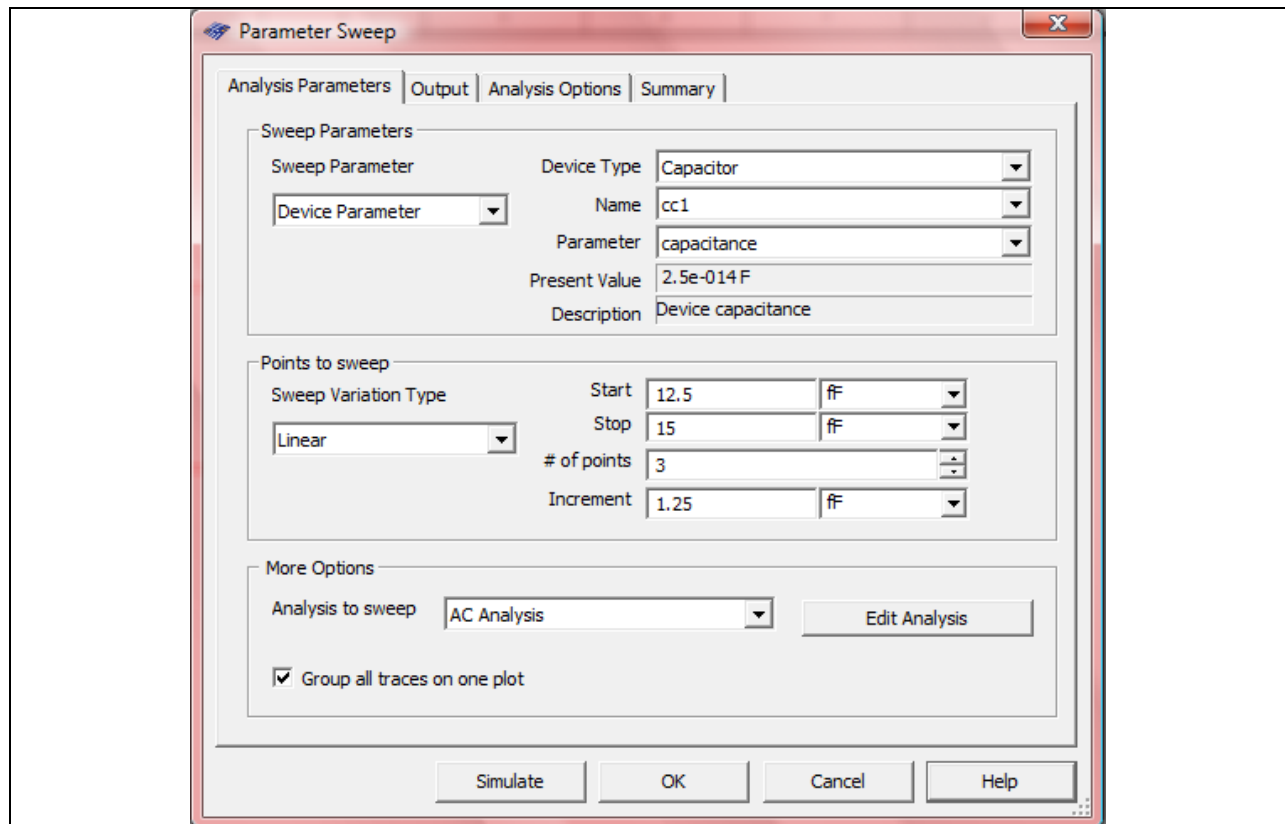


Figure 9.4.3 Setting up the Parameter sweep to do an AC Analysis on three different capacitor values.

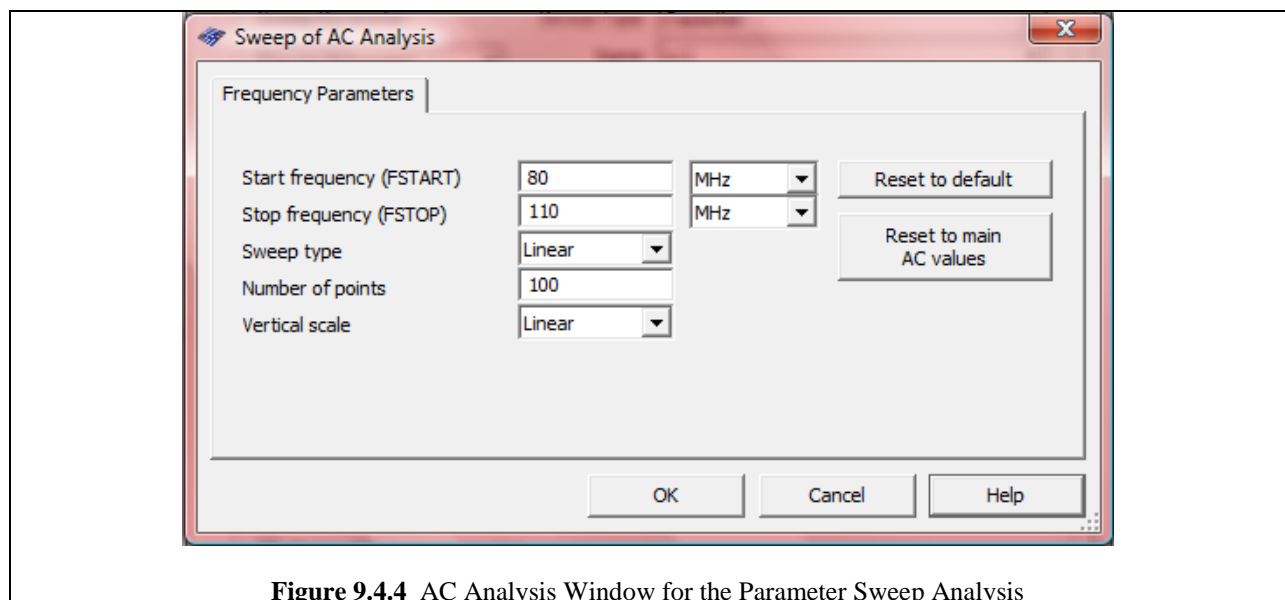


Figure 9.4.4 AC Analysis Window for the Parameter Sweep Analysis

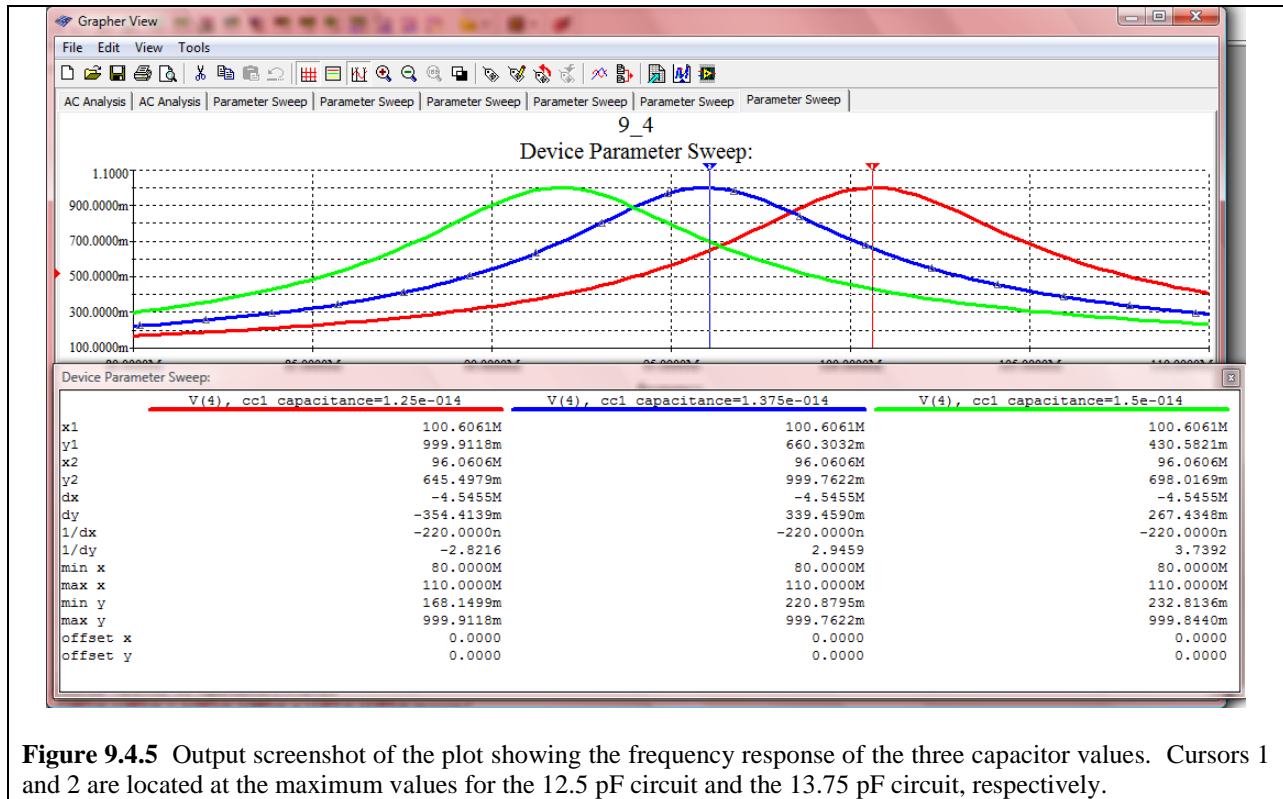


Figure 9.4.5 Output screenshot of the plot showing the frequency response of the three capacitor values. Cursors 1 and 2 are located at the maximum values for the 12.5 pF circuit and the 13.75 pF circuit, respectively.