
Subject: Crossover Doc Spcie models

Posted by [dbeardsl](#) on Thu, 14 Mar 2002 21:13:57 GMT

[View Forum Message](#) <> [Reply to Message](#)

Wayne, Thanks for the info. I'm probably not doing something right. I used your exact spice model in the crossover doc for the Delta 15 and the psd2002 with a simple first order butterworth. I put the current source across node 5 and 0 like you said, and measured the voltage at node 6 and 1. I get something close to yours, but not quite the same. Here is the spice model... Delta 15 and PSD 2002 with 1st order Butterworth! First order network L2 5 6 0.6mH C2 5 1 16uF! woofer virtual circuit (Eminence Delta 15)! voice coil reactance R3 6 7 6.9 L3 7 9 0.86mH! mechanical reactance (40Hz, Q=6.56) C5 9 0 400uF L5 9 0 40mH R5 9 0 65.6! tweeter virtual circuit (Eminence PSD2002)! voice coil reactance R4 1 10 6.6 L4 10 11 0.1mH! mechanical reactance (on H290) C6 11 12 20uF L6 11 12 2mH R6 11 12 100 C7 12 13 10uF L7 12 13 1mH R7 12 13 100 C8 13 0 5uF L8 13 0 0.5mH R8 13 0 100 R9 11 0 20 C9 11 0 50uF I 5 0 AC 1 And the link is the graph I get. Am I doing the graph wrong.. I expected it to be exactly like the one pictured, I tried different voltages, but of course I get the same thing, just a little up the scale.

<http://dsb.8m.com/pipics/spice.gif>

Subject: Re: Crossover Doc Spcie models

Posted by [Wayne Parham](#) on Thu, 14 Mar 2002 21:48:00 GMT

[View Forum Message](#) <> [Reply to Message](#)

I looked at your graph and it appears to be similar, except you are using a different scale. For the Frequency [X] axis, I use the scale factors, linear axis, min 0, max 20000, increment 5000, minor tics 4, major grid solid, minor grid dot. And for the amplitude [Y] axis, I use the factors linear axis, min -50, max 0, increment 10, minor tics 4, major grid solid, minor grid dot. Your tweeter curve looks the same as mine, and in the woofer curve, I can see the two peaks caused by your woofer cabinet. Good job!

Subject: My Cabinet?

Posted by [dbeardsl](#) on Fri, 15 Mar 2002 14:37:20 GMT

[View Forum Message](#) <> [Reply to Message](#)

cool, I'll mess with the scale. Though I'm not sure how you got your graph to show only negative values, is there some way to graph equations? like $(V[6] - V[5])$? I suppose I could use the real spice... I tried once and couldn't get the graph anywhere close to yours. Though I use it to lay out the circuits, then just grab the netlist which is the spice model. But How did a cabinet factor in here? I copied this straight out of the crossover doc. There isn't anything there to simulate a box... How would you simulate a box anyway?

Subject: Re: My Cabinet?

Posted by [Wayne Parham](#) on Fri, 15 Mar 2002 17:05:59 GMT

[View Forum Message](#) <> [Reply to Message](#)

Some of the models have a parallel resonance which is used to simulate the behaviour of the box. I put this in some of the models, but disabled it by commenting it out. You can uncomment this box resonance or make your own.