SECTION 6 – CROSSOVER DESIGN
TABLE OF CONTENTS

SECTION 6 – CROSSOVER DESIGN

Speaker Workshop supports first through fourth order crossovers. There are several types of each order that will yield a slightly different result in sound reproduction. Butterworth Filters are maximally flat in their pass band. Chebyshev filters roll off faster than Butterworth Filters with the same slope at the expense of flatness in the pass band response; this is why they are not terribly popular as crossover filters. Bessel filters do not roll off as quickly as the other two and therefore one would generally need to use at least a 4th order filter. They have the best phase performance of the three.

Odd order crossovers are much more sensitive to listener position than even order crossovers. At any distance from the speaker, odd order crossovers cause at least one order of magnitude more response error than even order networks due to phase errors in multi-driver speakers; this leads to more difficulties with driver integration. Since it is often suggested that the listening position should be at least 3x the largest dimension of the loudspeaker to experience good driver integration, one would double this for odd order networks.

For even order crossovers, one would want to get the phase as consistent as possible between drivers. For odd order crossovers, they should be 90 degrees out of phase at the crossover frequency and everywhere else in the crossover region. (referred to as being in phase quadrature). With odd order crossovers, there is a vertical listening axis where the drivers are phase aligned and yet you get a +3dB peak in the response. This peak can occur above or below the listening axis depending upon how you set the drivers up. Some believe that odd order crossovers sound more natural due to a more even power response overall.

Speaker Workshop can also calculate in a baffle step, a Zobel network, and even the series resistance of the inductors if you choose to use this for even more accurate modeling. By default, SW adds components in sequential order on the amp side first. Impedance compensation should be placed closest to the speaker, next comes an L-pad or attenuation circuit, then the crossover, then and the amplifier.

SW assumes you've measured the tweeter and woofer at the same time on the same baffle. If not, you have to find the acoustic offset distance between the woofer and the tweeter and enter it on their driver properties page. If you don't, there is no way for SW to combine the woofer and tweeter curves to get meaningful results. Measuring the woofer and the tweeter with the gate starting at the exact same place, SW can calculate in the phase issues into the crossover. In addition, baffle diffraction will be addressed with the off axis measurements.

If you want to make a PCB for your crossover, an easy and free PCB program is ExpressPCB downloadable at
http://www.expresspcb.com/ExpressPCBHtm/Download.htm.

As power goes through the voice coil, it heats up and increases DC coil resistance. As this happens, the crossover network heats up as well and the Q of the crossover will increase leading to a hump in response based upon the amount and duration of power going through the system (John Kreskovsky has demonstrated this). One must choose whether or not to "heat up" the elements and design the crossover accordingly.

Series resistance affects the Q so if inaccurate, it will affect both ported and sealed box designs. You should measure the resistance of the probes themselves and subtract that from your results. Most multimeters are not that accurate at low resistance. Adding a known resistance to the path that can be accurately measured (i.e., higher values), you can then measure the resistance as the difference between the sum of the resistance of the probes plus the added resistance and the unknown minus the sum of the probes and of the added resistance. In most cases, you could use an estimate of .2 to .3 ohms. If you know the damping factor of the amplifier you can calculate the output impedance and add this to the wire resistance (see the AWG chart in the Appendix but remember that since this is AC current that you would double the length of the wire since you have two wires running together) and then the DCR of any inductors to be in series. To calculate the Output Impedance of the Amp, look at how it is specified. If it is specified at 8 ohms, you would divide 8 by the Damping Factor (DF). For example, if you have a DF of 200 and it is rated at 8 ohms, the Output Impedance would be 8/200 or .04 ohms.

If you don't know the Amplifier's Damping Factor, you can calculate it:
1. Use a 10 Watt Non Inductive Wire Wound resistor and keep the voltage below the resistors rating
2. Select a frequency and measure the voltage
3. Call this V1
4. Add a resistor in series and measure the voltage again. Do this quickly as the resistor will get hot.
5. Call this V2 and it should be less than V1.
   a. The amplifier's Impedance (I) is V2/R.
   b. The Output Impedance=(V1-V2)/I.
   c. The Damping Factor is the (Speaker Impedance at a given frequency)/(The amplifier output Impedance).
   Note that if the voltage increases with a load, it is indicative of bad internal wiring and these amplifiers should generally be avoided. This suggests a negative impedance and the voltage will rise with an increasing load.

You can compensate for speaker cables in the design of a speaker:
1. Create a new enclosure
2. Open the file
3. Properties/Chamber One/Enter the value of the cables and any inductor resistance in the series resistance box.
4. For networks, you can add a series resistor of the desired value from the source to the network or increase the DCR of a series inductor.

For Calculations

5. Use the measured impedance in the box for both drivers and the measured on-axis response for the tweeter (gated appropriately).

6. Use the spliced / merged frequency response of the woofer (merge the nearfield woofer and port then splice the gated response).

7. Ensure that the drivers are correctly set up. Open each driver (woofer and tweeter) and check the Properties to make sure that the Impedance and Frequency Response datasets are correctly mapped to the right datasets. Right Click/Properties/General: allows you to enter comments about the driver.

8. Examine Parameters tab: assure that the T/S parameters are entered for completeness though this plays no role in crossover calculation.

9. Examine Data tab: assure that the appropriate Impedance and Frequency Data sets are attached.

10. If there is no data set attached to the Impedance, click "?" next to Impedance and choose the appropriate .zma file.

11. If there is no data set attached to the frequency box, click "?" next to Frequency and select the appropriate .frd file (use the combined driver/baffle response for the woofer).

12. Hit OK and repeat for each driver.

13. Examine the frequency response and impedance charts of the drivers. Prior to using data to build a crossover, open the Measurement.In.l file for each driver and assure that the time of the pulse is exactly the same for each driver. You should check this each time you do a new acoustic measurement.

Driver Impedance (chart)
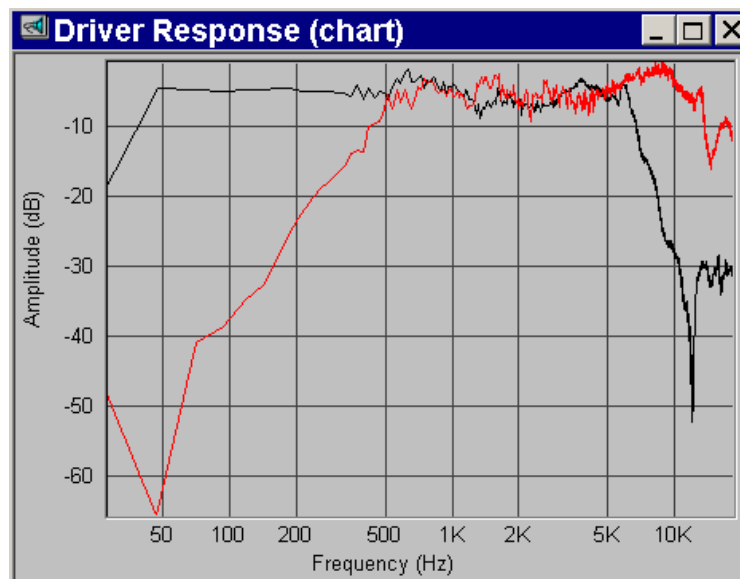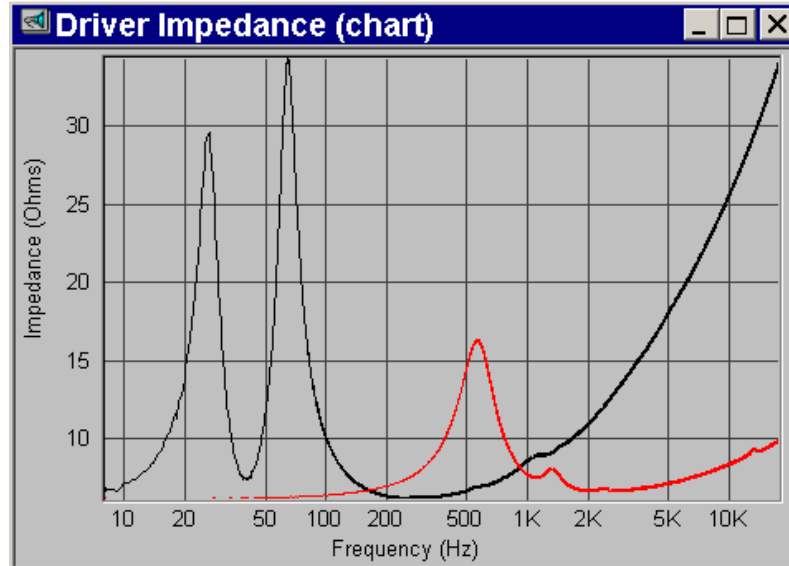
14. Choose a crossover frequency and decide upon a crossover type and order (see appendix for different types and orders).

    a. A general rule is to keep the crossover frequency 2-3x the resonant peak (Fs) of the tweeter though a more important consideration is where the tweeter excursion begins to cause distortion. You can run into problems using the 2-3x Fs method.
    b. See also the section on choosing a crossover point in the Appendix.
    c. A higher crossover frequency will decrease time and group delay however it may strain the ability of the woofer or go beyond the frequency at which it beams. This will cause off axis drop out which is probably a more important consideration than Group Delay unless the Group Delay is quite marked. I have attempted a spreadsheet that appears to work fairly well

    "calculate crossover range1.xls" but use it at your own risk! Another option would be to actually measure the distortion of the drivers using Non Linear Distortion testing and looking to see at what frequency unacceptable (to you) levels of distortion begin. This may be the best (but most tedious) method of determination. Remember to allow yourself some room between the crossover point and the distortion point as the crossover filters the amount of signal down gradually and does not suddenly stop the signal from getting to the driver. The spreadsheet above will be of assistance in estimating how much signal will get to the driver at a particular frequency given a chosen crossover point.
    d. Keep the crossover frequency within the range where both the tweeter and the woofer have a level frequency response.
    e. Try to find a place where the phase is not rotating in either driver (noted earlier).

f. Attempt to place midrange/woofer crossover frequency as close to the geometric mean of the impedance dips in the box as possible. The geometric mean = SQRT (Freq 1*Freq 2) and the following is a spreadsheet to assist in calculating this

"geometric mean.xls"

:   .

g. Attempt to avoid beaming as much as possible in the final design. Beaming is where the driver becomes almost completely directional, losing it's off axis response. This occurs at roughly 13650/diameter of the driver in inches. See the two spreadsheets below:

Note: Utilizing drivers much above this point will result in speakers with poor power response and tonal balance changes as one changes position in the room. Differences in dust caps, phase plugs, cone and surround shape, etc will cause this point to vary significantly from driver to driver. The lower the order of the filter for the tweeter, the more it can assist in off axis response at lower frequencies below the crossover (however the less protected the tweeter is from potentially damaging acoustical power).

Vance Dickason defines two 'widely accepted criteria' which suggest that crossovers should be limited to no higher than where the off axis response drops to either -3dB or -6dB at 45 degrees off axis. He further suggests the -6dB criterion is generally acceptable for most applications. Higher order acoustic slopes in general have better off axis response at higher frequencies. Further convoluting the issue is that the changes in acoustic slopes will cause phase differences and affect how the drivers sum off axis. Throw in mixed order slopes, under or overlapping crossover frequencies, etc. and the picture quickly becomes occluded. Hence the 'rule of thumb' that one should attempt to cross over a driver below the frequency at which the wavelength is larger than the diameter of the driver.

John Kreskovsky has contributed the following Excel spreadsheet to assist in calculating and visualizing the predicted beaming from a driver.

"My Off axisA.xls"

The following is a less graphic more numeric way of evaluating this adapted from John's spreadsheet:

"off axis my version
final1.xls"

15. Build the networks:
    a. Menu/Resource/New/Network
    b. Name each network (high pass, bandpass, low pass or woofer, midrange, tweeter)
    c. Open each network to assure that only the source is currently there.
    d. Click "OK".

Note: Don't combine the drivers into the same network at this point, as SW won't optimize a network using more than one driver. If you do, you will be unable to utilize the SW network optimizer. For MTM configurations, see the MTM chapter (MTM Configurations)

16. To Populate the networks:
    a. Select the network by right clicking on it and click on the corresponding Driver icon or Right Click in the window and Insert/Select your network.
    b. To populate the Woofer network select everything in the Woofer network (draw a box around them) then select Edit / Copy (or press Ctrl+C) and copy all of the network components to the clipboard.
    c. Go to the Woofer network and select Edit / Paste (or press Ctrl+V) to bring them in.
    d. Delete any extra Source components (highlight it and select Edit / Delete or hit the Del key).
    e. To move groups of components, the Network Chart has a "Rubber Band" feature which works as follows:
        i. Left click in a blank area and while holding down the button, drag and expand to capture the desired components you want to move.
        ii. Drag to position.
        iii. It is recommended to do this individually for each network prior to combining the networks later on.
        iv. Doing this will assist the Optimizer Function.

17. Do this separately for each driver, i.e., each driver will have it's own network drawing.
18. Connect the driver and the nodes to get it to look like this.
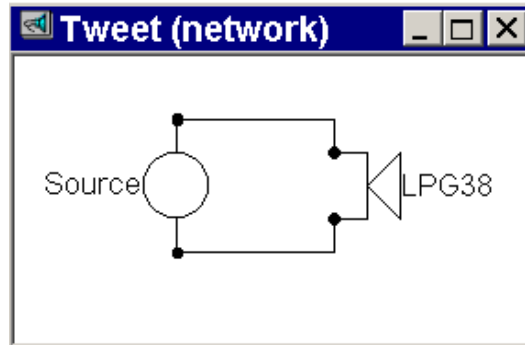
a. To connect two nodes just left click the driver (it highlights) and then drag it by holding down the button to the other device terminal (the cursor changes to a large + sign)

b. If there is already a connection, hold the shift key, otherwise the program will substitute the old connection with the new one.

c. To move both components - press the mouse and drag a rubber band box on the screen until it surrounds both components - release the mouse and they both highlight - then click a component and drag them both.

d. Click OK when you are done.

19. If symbols do not seem to orient correctly:
    a. Open the Network Chart
    b. Right Click the chart area/Properties/Components
    c. Select the component whose orientation you wish to change from the list on the left.
    d. Use the Orientation button on the right to change the layout.
    e. Select Okay

20. Right Click/Insert/Stock Crossover

21. Select type (high pass for tweeter, low pass for woofer), order, and crossover frequency target.

22. Select "OK" and the necessary components will be installed.
    a. *Doing it in this manner avoids the need to name each component for a particular crossover type.*

23. Menu/Network properties/click on Network Impedance so an impedance dataset gets built.

24. Look at the impedance chart. *You will likely want to compensate the woofer for its inductive rise.*
    a. Go to the Woofer network and select the (Insert / Impedance Compensation command (via right mouse button or from the menu).
    b. Also called a Zobel, this makes the driver impedance look more constant and resistive, particularly in the crossover region. In a three-way speaker, the midrange will sometimes require a Zobel for both the low pass and high pass regions.
    c. The dialog looks like:

Add Impedance Compensation

Input

○ Use Current input impedance
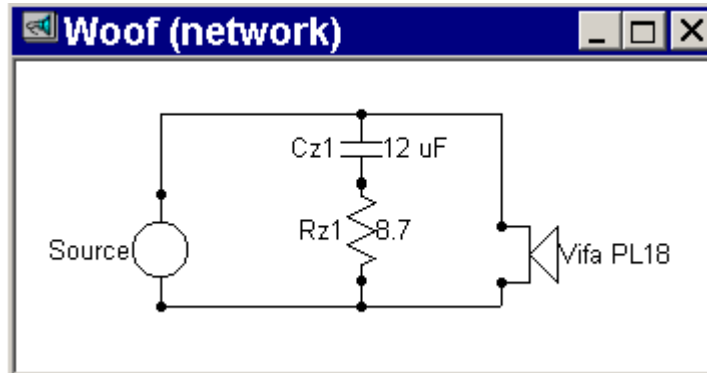● Use Driver impedance

Driver Vifa PL18

Compensation

Type Inductive rise

Frequency of interest 2.000 K Hz

OK

Cancel

d.   And the circuit it builds is:

Woof (network)

Cz1 ⎓ 12 uF

Rz1 ⟩ 8.7

Source

Vifa PL18

25. Click Calculate Response (right click or menu) to see the compensated impedance. The red chart is the original impedance and the black is the compensated impedance. Note the low resolution on the black chart

Woof.Impedance (dataset)
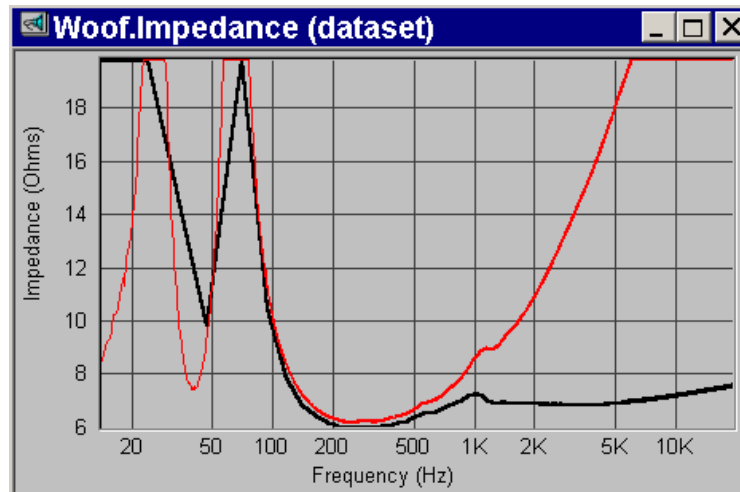
26. If a significant Impedance Peak is noted with the tweeter, a compensation network is called for in this case as well. The tweeter usually does not require one, especially if it is damped with fluid, but occasionally it requires one due to an inductive rise.

Tweet.Impedance (dataset)

Tweet (network)

Lz1 3.9 mH
Source
Cz1 21 uF
Rz1 12
LPG38
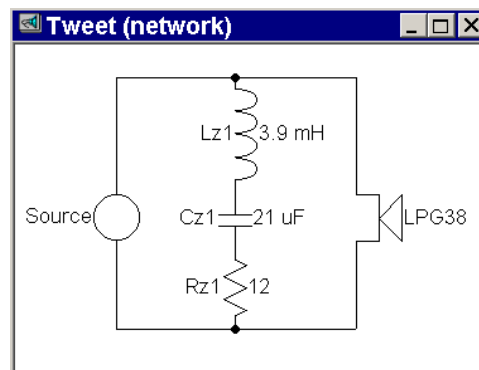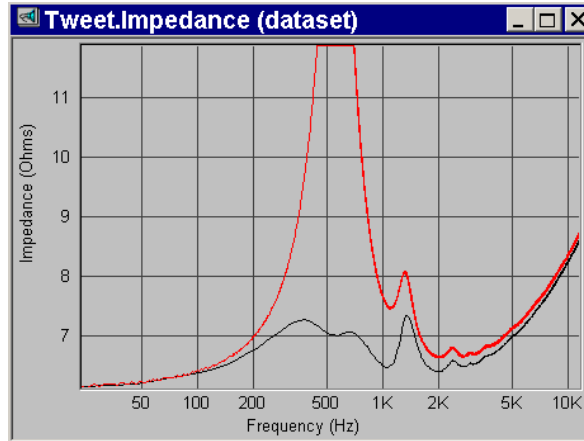
27. You may add Baffle Step Compensation here. The Baffle Step compensation will assist in leveling the decrease in Woofer on axis amplitude that occurs as a result of the sound going from a 2pi to a 4pi environment. This occurs at a frequencies based upon the distance from the driver to the end of the baffle. The Baffle Step is a 6dB slope when measured in a gated test without consideration of the surrounding reflecting surfaces. The pure baffle step is best measured as a farfield ground plane measurement (outdoors if possible) to avoid reflected sound effects. This slope is tempered down based upon room reflections and the placement of the woofer on the baffle with closer to the floor or side walls enhancing bass response; as a result, the more accurate baffle step for a particular environment is best measured in the room in which the speaker is to be used. Further explanation of this by John Murphy is provided in Appendix C.

    a. A spreadsheet program written by Svante Granqvist and freely downloadable at http://www.tolvan.com/edge can assist in

edge.exe

developing a compensation circuit.   . In calculating a BSC circuit with the Edge, set R1 to the woofer's Impedance and this will give you L and R2 values for the Circuit. L and R2 are wired in parallel with one another. This would then go in series with the positive input before the crossover in order to address any impact that it may have upon the tweeter as well. You may want

to modify f1 and f2 a little to decrease the BSC a little so that you get somewhat less than 6 dB as it is rare that you would need this in a real room and this may therefore muddy the bass somewhat. Another option would be to start out by dropping R2 by 1/3 to ½, which would also drop the baffle step somewhat. Try to come up with Inductor values that are available.

b. Another program that may be of assistance is Paul Verdone's Baffle Diffraction Simulator program, which may be freely downloaded at: http://www.pvconsultants.com/audio/frdgroup.htm. This program can simulate baffle steps and diffractions of various baffle set-ups (it is more comprehensive than that of "The Edge" (above) but it does not calculate a compensatory circuit as does "The Edge".

"Baffle Diffraction Simulator Beta6.07-2"   "Operation Manual Baffle Diffraction Sim"   "Wall parameters baby spreadsheet.xls

Baffle Step Compensation is really an estimate until you measure. Paul Verdone has developed a spreadsheet that assists this process in combination with several tools from the FRD Consortium.

    i. Use the Baffle Diffraction Simulator to calculate the baffle signature.

    ii. Output the .frd file and save

    iii. Load the .frd file into the Frequency Response Combiner Program

    iv. Extend the Baffle Response in the Normalizer with the low set to "Baffle gain to zero"

    v. Extend the Baffle Response in the Normalizer with the top set to "Extend at Value".

    vi. Go into the Generator and select a high pass shelf with gain at 6 dB and adjust dB at 6 dB.

    vii. Find the frequency that best matches the baffle diffraction curve imported from the BDS.

    viii. Load in the Wall Parameters Baby Spreadsheet

    ix. Enter the Baffle Width, Height, Listening distance, Speaker Depth, Speaker Back to Wall and the High pass shelf frequency just determined above (the frequency that creates curves that sum to zero).

    x. Return to the Frequency Response Combiner.

    xi. You can now see the two high and low pass shelves.

    xii. Summing these shelves will display the general dip caused by the time delay between the baffle and the rear wall.

xiii. Combining the two will show you the actual baffle signature with the rear wall reinforcement canceling the step.

xiv. When you combine these two, you will see the baffle step bump and post bump dip combined which is what you really want to correct for with Baffle Step Compensation.

xv. You can then use the FRC notch and peak parametric equalizer to flatten this curve. Look at the Q's, Frequencies, and gains.

xvi. This will bring you within a dB or so of the actual measured baffle step.

Note: the baffle step will be reflected in your acoustic measurement however it will be reflective of the environment in which the frequency response file (.frd) was measured.

28. An alternate way of creating a baffle compensation is to:
   a. Create a new network
   b. pull down network menu and select create goal
   c. Select acoustic goal that you want
   d. Set the level as absolute (level is unimportant)
   e. Export it and edit it in notepad to add 3-6 dB during the baffle step
   f. Import it.
   g. Open the FR graph and Create Chart
   h. Under Chart Properties, Add the goal to the Chart
   i. Scale the goal or the FR so that they match the levels just below the hump caused by the baffle.
   j. Go back to the network
   k. Properties/Select network Frequency Response, Network Impedance, and Auto Overview
   l. Add your driver to the network and connect it to the source
   m. Insert Imp Comp, Inductive Rise and a standard XO
   n. You will now have 6 components in the network.
   o. Select Optimize
   p. Use this as your goal for the optimization of the woofer network.

29. A third way of doing this is to:
   a. Create a new driver
   b. Create a goal for the slope that you want.
   c. Create a network using this driver.
   d. Insert the BSC network.
   e. Alter the values to obtain the BSC shape that you want.
   f. Once you have the shape that you want, rename this file as your woofer network goal and use it for crossover optimization.

30. You may add an L-Pad to the system at this point. An L-Pad is a two-resistor network that reduces the signal level to a driver without changing the input impedance (assuming the driver is resistive). The L-Pad values are based on how much of a decrease in dB is required to level the sensitivity of the more sensitive driver (usually the tweeter) to that of the less sensitive driver

(usually the woofer). dB attenuation = Series R+Parallel R divided by Parallel R.

    a. Network/Insert/L-Pad
    b. Enter an amount (in dB) to decrease the level by. A value of 6dB will halve the sound output, for example.
    c. Start   If using a dataset or driver enter the start frequency to scan.
    d. End   If using a dataset or driver enter the end frequency to scan.
    e. Resistance Enter the resistance the Pad is produced for.
    f. Driver   Enter the driver whose impedance is scanned from Start to End to get the pad matching impedance.
    g. Dataset   Enter the dataset whose impedance is scanned from Start to End to get the pad matching impedance. Usually this is a created impedance from a smaller network.

31. Right Click the Network/Properties
    a. *General* tab lets you insert comments and select a few options.
    b. *Generate* allows for the generation of impedance or frequency response to review predicted response.
    c. *Components* tab allows you to modify any inserted components; you can also change where the text for the component is displayed by clicking one of the buttons with a resistor and yellow and white boxes. It is important to specify the DCR for an inductor as this will impact the response (a higher DCR will cause more attenuation of the volume of the circuit).

Note: In the support forum, it was noted that there may be a problem with Network Properties/Apply. It would be better to use the Network Options Pop-up box. The default value for inductors in properties is mH and the Network Options and the Network itself seem to use Micro-Henries for anything under 0.999mH. Watch for this.

    a. *Driver* tab can add or delete drivers.
    b. Select "Individual Driver Response" to see the individual response of each driver.
    c. *Offset* tab allows you to set the relative offset of the drivers with a negative number meaning that the acoustic center is further from the listener. In general, the woofer would be –1 or –2 on a flat baffle.
    d. *Polarity* tab allows you to see what happens when you connect a driver with reverse polarity by selecting "Reverse Polarity".

# CROSSOVER OPTIMIZATION
(Section Index)

Note that Speaker Workshop is capable of simulating Series Crossovers however the Optimizer is not capable of optimizing for a Series Crossover so you will have to adjust values and then simulate the network. In drawing a series network, remember that you can only work with one driver at a time. You would have to do series crossovers manually, using calculate network response to see what you have and adjust. Then redo calculate freq. to arrive at the response you want. Slow but it works.

1. Create a target response for the low pass (note that the example below shows high pass, this would need to be changed:
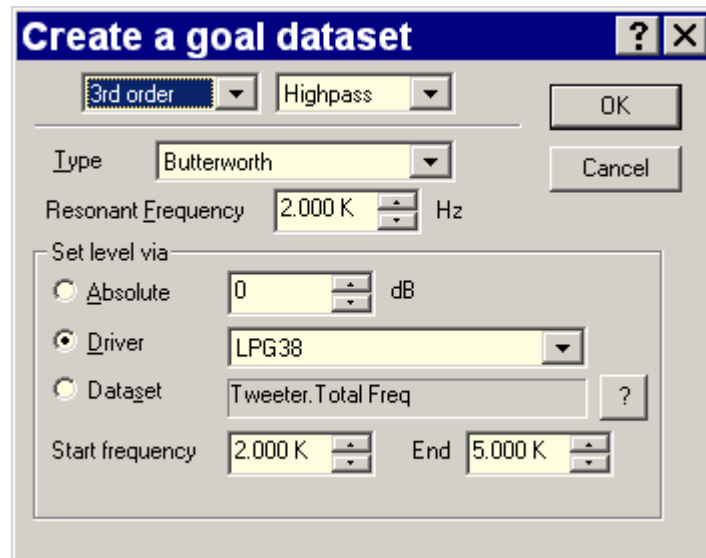
2. Click the woofer driver's network window.
3. Menu/Network/Create goal
4. *Note that this is not the actual crossover but rather the goal you will be targeting; it takes into account acoustical roll off of the driver as well as the filter's roll off.*
5. Choose Low Pass.
6. Create your Acoustic Response Goal.
7. Set crossover type.
8. Set desired slope (1st through 4th order).
9. Set Crossover Frequency (in the Resonant Frequency Box).
10. Click "Set Level Via"
11. Go to Driver/Properties/Data to assure that the frequency response and impedance data in the upper left are the ones that you want to use.
12. You can use either imported or measured data as long as the data is complete and is in the upper left.
13. Dataset/ "?" and select the woofer file.on axis.
14. If your desired data set is not in the upper left,
15. Select Data Set
16. Select the frequency file that you want to use in the Create Goal Dialogue

17. Click Okay
18. A chart called Your network name.goal will be created.
19. Open this chart and examine.
20. Set the lower usable frequency where the woofer curve starts.
21. Set the upper usable frequency of the woofer below where peaks on the curve are noted but target at least one octave above (2x frequency of) the crossover.
22. Create a target response for the high pass:
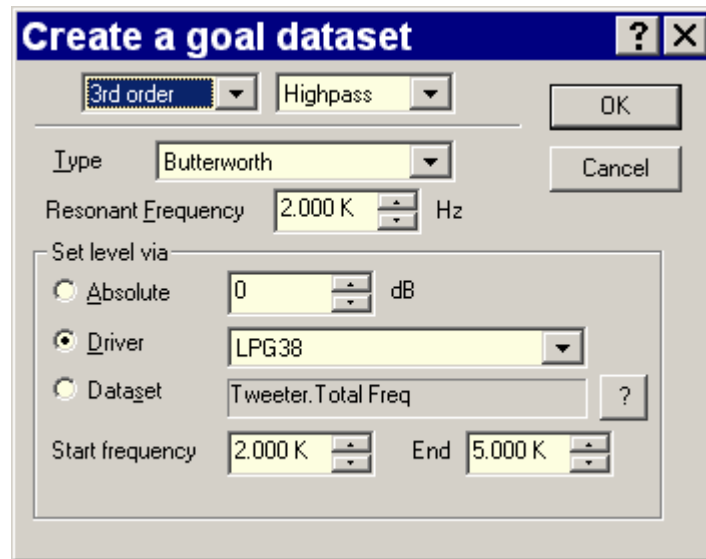
23. Click the tweeter or driver's network window.
24. Menu/Network/Create goal.
25. Choose High Pass.
26. Create your Goal.
27. This will be the same crossover point, type, and order as with the woofer unless you change it.
28. Click "Set Level Via"
29. Go to Driver/Properties/Data to assure that the frequency response and impedance data in the upper left are the ones that you want to use.
30. You can use either imported or measured data as long as the data is complete and is in the upper left.
31. Dataset/ "?" and select the tweeter file.on axis.
32. If your desired data set is not in the upper left,
33. Select Data Set
34. Select the frequency file that you want to use in the Create Goal Dialogue.
35. Click Okay
36. A chart called Your network name.goal will be created.
37. Open this chart and examine.
38. Set the lower usable frequency where the tweeter curve starts (target at least one octave (1/2 the frequency) below the crossover frequency.
39. Set the upper usable frequency of the tweeter below where peaks on the curve are noted.

OPTIMIZING THE NETWORK

Note: you will do this separately with each network to be optimized.

1. With the network open, Menu/Network/Optimize Network.
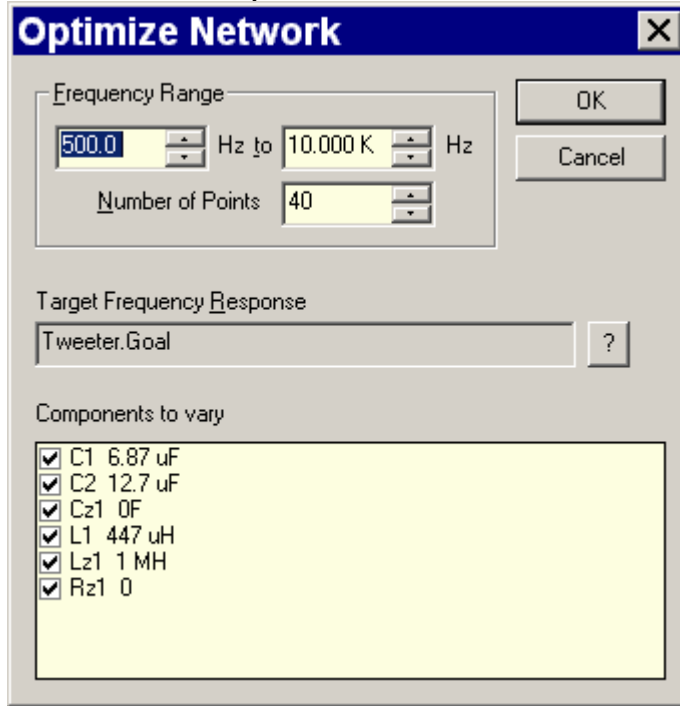
2. A new window opens.

3. *Frequency range:* choose a range of interest for the particular network (the usable frequency range for the driver is reasonable here).

> *Sometimes you may just want the response curve changed in a specific area. For example, if you added a response shaping network (a parallel resistor, cap and inductor) to remove a response peak. If the bothersome peak is from 6 kHz to 10 kHz, insert those values into the optimizer and it will calculate the component values to reduce the peak in this range. It may still affect other frequencies so you should include some flat response areas around the peak if possible. If it's difficult to see the dip, use a high value for your resistor (say 2,000 ohms), and then optimize manually by changing the value of the capacitor or the inductor until the dip is exactly where you need it. Then change the value of the resistor to something like 3 or 8 ohms.*

4. *Number of Points:* The more points you pick, the better the optimizing, but the slower it will go.   Around 400 points (range 100-1000) is generally adequate.

5. Click the box with *"?"* on it and choose the *network.goal* file (this is the target frequency response), where network is the name of the network you are optimizing.

6. Select the components you want to optimize, you may select all of them or hold certain components fixed , if you desire, and the optimizer will adjust other values around those components. Uncheck components you don't want optimized.

7. Occasionally SW will give odd results if there are too many components, in this case, optimize small areas of the network at a time. (Perhaps model an attenuation network

first. Just leave the two attenuation networks checked and *de*-select everything else - impedance compensation, parallel traps, etc...

8. Click okay on the optimizer. Then repeat this with the response shaping networks, then the actual crossover network, etc...).

> *If you keep getting odd values for certain components, the optimizer may be telling you that you can do away with that component or that you may need to either choose a different slope or change the network. If you have selected a higher order crossover, it is worthwhile to attempt lower order crossovers to see if the optimizer to achieve the desired slope with fewer components.*
>
> *For example, if you have a third order network and it keeps making the second capacitor 1000F, you will probably reach the target response with a second order network. Just right-click the 1000F cap and delete it. Then re-run the optimizer. Remember that you might get a 3rd order roll off with a second order slope due to acoustical roll off of the driver.*
>
> *The optimizer will pick components to match your goal.*

MTM CONFIGURATIONS

For both Impedance and Acoustic Measurements:

1. Take measurements of the individual woofers first.

    Note: During acoustic measurements, both woofers should be wired and active at the same time in order to obtain accurate combination effects in the measurement. If you do not do this, you will get box colorations that are not accurately calculated into the measurement. In addition, the woofer that is not being tested will act as a passive radiator, further altering the response.

2. Match pairs as closely as possible in frequency response and impedance.
3. Create a new driver file.
4. Repeat the measurements with the drivers connected as in the final design, i.e., series or parallel connections.
5. Perform both impedance and acoustic measurements for this "new" combined driver.
6. Use this "new" combined driver in the network and for crossover optimization.
7. Generate Frequency Response.
8. Generate Impedance Response.
9. Create a single driver and use the generated frequency response and impedance files.
10. Model the low pass section with a virtual single driver that has the properties created above.

Note: For MTM alignments, when a system is free of vertical axis tilt, it will have a maximum on the design axis. The problem, however, is that there is an interaction between the low frequency drivers as one moves away from the crossover point as their wavelengths extend and there is more interaction. In addition, as one's position changes in a vertical relationship to the drivers, one driver's output will be advanced in time vs. the other which will result in interference effects with a resultant change in the summed response of the drivers. At frequencies where the time displacement between the two drivers is equal to ½ wavelength, there will be a null. One can see this best with the Asymmetric Response Pattern Estimator (Freeware downloadable at http://www.pvconsultants.com/audio/radiation/arpe.htm) written by Ron Ennenga, John Kreskovsky, Kristian Ougaard & Paul Verdone. When the low and high frequency drivers are in phase quadrature, the vertical radiation at the crossover frequency will be the widest possible while a fairly narrow lobe will develop when the inter-driver phase shift is zero.

THREE-WAY DESIGNS

1. Create a goal
2. Create a network and a goal for the low pass crossover frequency.
3. Create a network and a goal for the high pass crossover frequency.
4. Optimize the lower crossover point.
5. Optimize the higher crossover point.
6. Attenuate the upper crossover point to the level of the lower crossover point.
7. Put the two networks together and run a predicted response in the optimizer.

Note: You may have to adjust the components in the individual networks to compensate for the band pass gain as the optimizer will only do one crossover point at a time.

IMPEDANCE AND RESPONSE SIMULATIONS

1. Make sure that all components (L-Pad, Zobel, Crossover, Drivers) are inserted.
2. Right Click Calculate Response and a new graph is generated showing the response of the driver and crossover response.
3. Set Chart Properties with the X axis range 200-20,000
4. Set Chart Properties with the Y-axis range 60-100 dB.
5. Set Major Grid Lines in increments of 5.
6. Right Click and select Calculate Impedance to calculate Impedance.

   If you want to add additional response shaping circuitry, you can individually add inductors, resistors, and capacitors. You can change their values by right clicking the network/Properties/Components.

   Note: Alternatively, you can double click on the individual component, which will bring up the network/properties dialog box. You can select the individual component in the network chart by left clicking on the component and then using the Network/Options pop up box to change the value . This feature automatically updates the response without having to do a calculate Frequency Response manually.

7. If you are changing a number of components at one time it is recommended that you follow the following steps.
   a. The boxes with yellow and blue rectangles display possible orientations of the components, their values and names. Choose the appropriate orientations.
   b. Change the value of a component by first selecting the item in the component list and then click the appropriate box and change it.
   c. To disconnect a component, click the component, click it's circle and unclick without dragging a line anywhere. All connections to this point will be undone.
   d. To connect added components, hold down the shift key while you connect them. Click and hold down the circle and drag it to where you want to connect, let up on your mouse once inside the desired connection point and then let up on the shift key.
   e. You can change a component's name by clicking in the appropriate box and changing it.
   f. Be certain to always return to the Network/Properties/Components and fill in the DCR of the inductor(s).
   g. Click OK when you are finished.

GRAPHING THE FREQUENCY RESPONSE
(Section Index)

1. Open the Network/Menu/Network/Calculate Response (or right click the network and choose *Calculate response*).
2. A couple of graphs will be created
3. Open the graph named for the network. This shows the network's response
4. Right-click the graph and choose *add*. This feature just <u>places</u> other graphs on the one you have open. It doesn't *add* (*combine*) the curves together. It just allows you to see separate graphs on the same graph.
5. Choose that network's goal for the goal to be added (if it's a high-pass network, choose the high-pass network goal).
6. It is recommended that you set the responses to different colors for each data set on the chart while you are working in Properties.
7. You can now see the network's response against the goal.
8. You can right-click the network and choose Calculate response as you change components to see if you are getting close to your goal.
9. Continue to do this with each driver until you are satisfied that you are close to your goal.

CROSSOVER DESIGN WITH EXTERNAL DATA

(Section Index)

If you don't measure the drivers yourself, you can use company supplied data or data measured by other people. This data must either be in .FRD and .ZMA formats or you must convert them to this. If you only have the graphic response, you can use SPL Tracer and then the Frequency Response Combiner (freeware pieces of software from the FRD Consortium at FRD Consortium on the Internet).

The image may be in any graphics format.

1. Copy the picture to Windows Clipboard.
2. Open SPL Tracer/File/Import Graph/From Clipboard and the graph will then appear.
3. Alignment/Register Low Frequency/Enter a frequency value and align the blue line with that frequency on the graph.
4. Click on the Graph once when the alignment is properly set.
5. Repeat this process for the high frequency.
6. Register the High and Low Amplitudes as well for .FRD files.
7. Register High and Low Impedance for .ZMA files.
   a. You must also choose your scale with base 10 being logarithmic, base 1 being linear, and base 2 being a doubling scale (1,2,4,8,16,etc.).
8. Trace/Start SPL trace (or Impedance trace if doing .ZMA)
9. To Trace with the Mouse
   a. Line up the white line with the graph on the far left and click the Mouse.
   b. When the field advances, it is ready for the next data point.
   c. Align the white line with the mouse again and click.
   d. Repeat until the entire graph is traced.

Note: Though the Mouse must capture the first point, all subsequent points may be captured by using the arrow keys and the "enter" key.

10. Trace/Stop Trace/Yes saves the file and then name the file when prompted.

Note: the data from manufacturers is often measured on an infinite baffle. The following will help you to interpret what is meant by that from AES2-1984 (r2003):

Standard Baffle for LF Driver Measurements

The preferred baffle is of the dimensions given in Fig. B1 and Table B1. The dimensions for the 200-mm (8-in) size are identical to those of the standard baffle in IEC Standard, Publication 268-5 (1972). The dimensions for the larger sizes have been increased approximately in proportion to the loudspeaker diameter. Smaller drivers may be tested on larger baffles, as stated in 4.4.

Manufacturers are encouraged to modify the basic IEC baffle construction to allow front mounting of drivers.
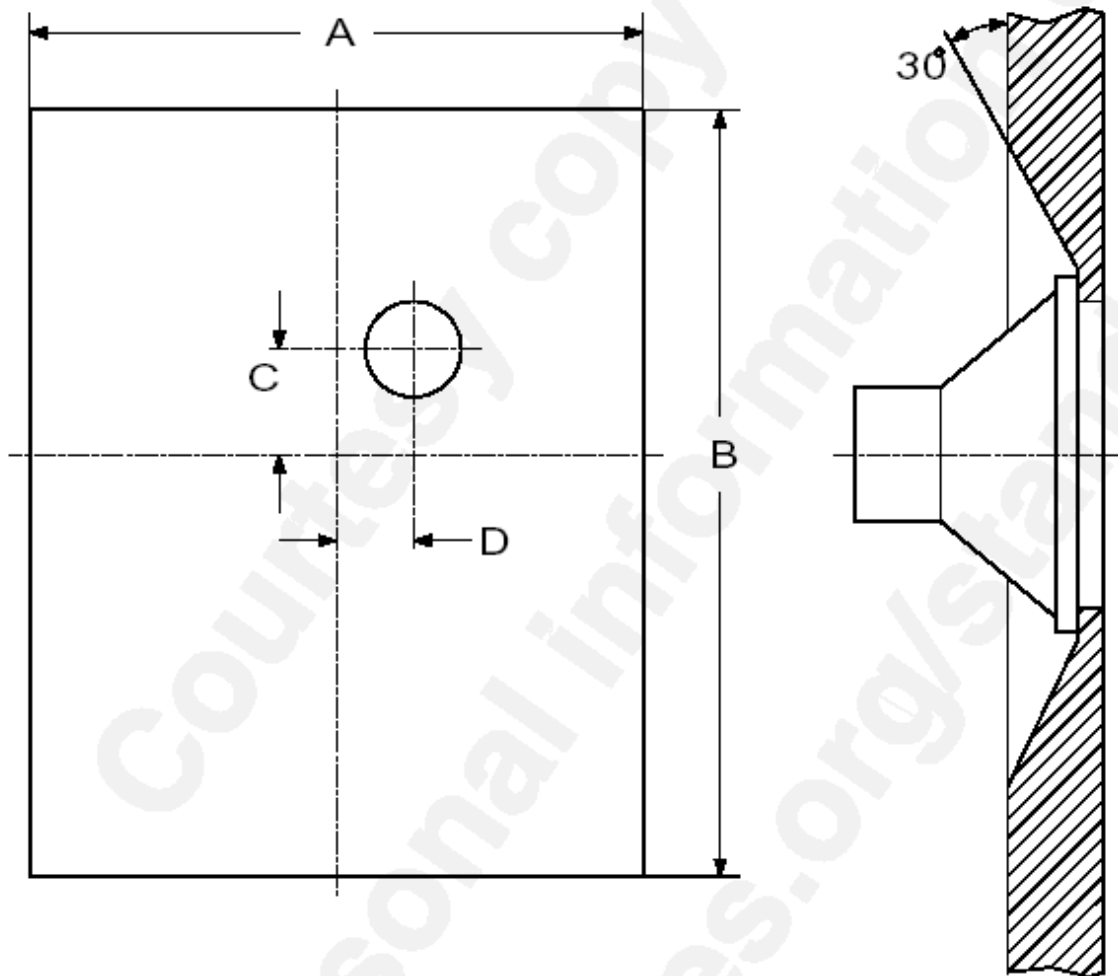
Fig. B1. Standard baffle dimensions for low-frequency driver measurements.

Table B 1. Preferred baffle dimensions for lf loudspeaker

Nominal Loud-speaker Size[1]	Baffle Dimensions[2]			
	A	B	C	D
200 mm (8 in)	1350 mm	1650 mm	225 mm	150 mm
250 mm (10 in)	1690 mm	2065 mm	280 mm	190 mm
315 mm (12 in)	2025 mm	2475 mm	340 mm	225 mm
400 mm (15 in)	2530 mm	3090 mm	420 mm	280 mm

---

[1] The nominal loudspeaker size is defined as the outer diameter of the frame, and the metric equivalent is given in the nearest preferred number according to IEC Standard, Publication 268-14 (1980).
[2] Baffle dimensions have been calculated from the IEC dimensions by multiplying them by the ratio of the nominal size in inches to 8 inches, and rounding the resulting dimensions to the nearest 5 mm.

USING THE BAFFLE DIFFRACTION SIMULATOR FROM FRD CONSORTIUM

(From http://www.pvconsultants.com/audio/diffraction/downloadbds.htm)

Baffle Diffraction Simulator

1. Select Add Basic Frequency Support
2. Select Add Advanced Support
3. Set the Sample Setting to Quadruple
4. Set the Source Setting to Combined
5. Set the Edge Setting:
    a. Sharp – No Edge modifications such as square corners and edges
    b. Complex - For rounded Edges
    c. Chamfer – For Chamfered Edges
6. Set the Edge Size parameter to match the amount of chamfer or round-over used
7. Set the Shape to Quadrangle assuming a Rectangular Box
8. Set the Type to Separate
9. Chart type should be set to Sum Down
10. Driver Parameter Data is located on the right side of the screen
11. Check "True" for each box included in the simulation
a. Box 2 is designated for the tweeter
12. Enter the size and placement of each driver
13. Input the cabinet dimensions in the boxes located on the left side of the screen
14. Shape Logic should be turned off in order to allow more cabinet dimension flexibility
15. Adjust the baffle tilt if you have an angled baffle
16. Set the Axis Distance to the distance you would like to simulate. 1 meter would be approximately 39.36996 inches.
17. Save Profile to save this data
18. Save the Baffle Diffraction Response for each driver analyzed
    a. Go to Save Response and Select the driver number with the drop down box
    b. Click Save Response
    c. Repeat for each driver
19. To Preview the Baffle Diffraction Responses, Scroll down to the Individual Driver Response plot
20. If you are using manufacturer data, measurements are commonly taken using an IEC baffle. This baffle is described above. You can start out with this simulation and modify it to your own box, if so desired. You can also use the Frequency Response Combiner to calculate the driver response without the box effect.

IMPORTING DATA AND WORKING WITH IT

In SW, if you want to import data to idealize a crossover:

1. Resource/Import will bring up a file window. You need both .FRD and .ZMA files for each driver.
2. By selecting each, they will appear in the project tree.
3. After you have imported all of the driver data files, Resource/New/Driver to add a driver to the project tree and a driver data window to the project work space. The driver parameters listed are used to simulate enclosures.
4. Within the driver data window, right click/Properties/Driver Properties/Data.
5. Click "?" next to the frequency response and Impedance response dialogue boxes and select the corresponding data files.
6. Click Okay when you are done.
7. A new graph within the driver data window will display frequency and impedance responses.
8. Repeat for each driver.
9. Enter Thiele Small Parameters

Building the network

10. Resource/New/Network and type in a name for the crossover network
11. Insert each of the drivers.
12. Check the box for individual response
13. Invert Driver polarity should be checked if this is appropriate to your circuit.
14. You can enter an offset value which is the offset of the drivers acoustic center compared to the baffle or to a given reference point. A positive value means that the driver is closer to the listening position while a negative offset value indicates that the driver is recessed into the baffle. If you are using data taken from SW measurements, you would not need to enter this data as it is already integrated into the data obtained **however** it is important to note that relative offset between drivers is only accurate if the measurement point of the microphone is unchanged between measurements of each driver.
    a. By making the offset more positive, you are simulating changes in vertical listening axis. Making the woofer more positive simulates a lower listening position while making the tweeter more positive simulates a higher listening position (closer to the tweeter).
15. Right Click within the network window and select "Insert" to insert components. You can also select pre-defined circuits here if you prefer but this takes away from some of the flexibility of the design process.
16. A dialogue box pops up in which you can enter the name and value of each component after you select it.
17. The component will then appear in the network window.
18. Place the mouse over the component and click, hold, and drag the mouse to move the component.

19. To re-orient the component to the circuit, double click the component and select the desired orientation from the diagrams in the bottom right of the window. Note: you can also change component values in this window, if you so desire.
20. To connect components, click once on the component that you want to make the connection from
21. Place the mouse over the component node and click, hold, and drag the mouse to the node of the component to which it is to be attached. Then let go.
    a. If you are connecting multiple wires to the same node, you have to hold down the SHIFT key while making subsequent connections.
21. Now connect the filter to the Source, which represents the amplifier.

Network Simulation
22. Right Click within the network window
23. Select Properties
24. A Network Properties window will have three check boxes
    a. Network Frequency Response – if checked, the network simulator will generate a response for the combined output of all of the drivers together
    b. Network Impedance – if checked, this causes the network simulator to generate an impedance response plot for the crossover network
    c. Auto Overview Chart – if checked, this will cause the network simulator to generate a chart, which includes the overall network frequency response along with the responses of each individual driver all on one chart.
25. Select Calculate Response
26. New datasets and charts will appear in the project tree to represent the new data that you generated based upon which boxes were checked.
27. To view the Network Response, double click on the Project Tree and a graph will appear in the project workspace.
28. To modify the graph appearance, double click within the graph window and a Chart Properties dialogue box will appear.
29. Select Data Sets
    a. You can use this to add or remove data from view and to choose the style and color of the graph.
30. Select the following tabs to choose the limits and resolution of the frequency (X axis), Magnitude (Y axis) and the Phase (Y2 axis)
31. After evaluating your response, you can experiment with various components in the network by Right Clicking on the Network or Graph window/Edit/Undo/Redo.

<u>Note that this is ignoring the effect of the box unless you make your measurements with drivers in the box.</u>

MINIMUM PHASE EXTRACTION
(WITH PAUL VERDONE'S FREQUENCY RESPONSE COMBINER)
(Section Index)

The Frequency Response Combiner (FRC) spreadsheet (from
http://www.pvconsultants.com/audio/utility/frc.htm) allows you to calculate the
minimum phase from an .frd file and it also allows you to "subtract out" the
manufacturer's measurement conditions and "add in" your target enclosure's effects.

To Calculate Minimum Phase:

1. Open the FRC spreadsheet
2. Select Loading so that Loading is displayed
3. Press Execute
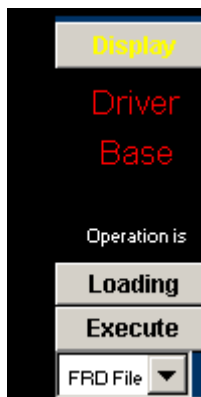
4. Select the file to load (you would generally use the baffled response of the driver here)
5. Select OK
6. The file will load and a frequency response graph will appear
7. You may have to scroll back up the page
8. Select the button Precise phase and group delay

9. Select Driver Base from the drop down menu just above the Precise Phase button

10. Select the Extract Minimum Phase button next to the drop down menu – this could take some time depending upon processor speed and the number of data points you have.
11. You can then look at the Phase response graph, which will be the minimum phase of your driver.

12. Now Click on the button that says "Loading" under Driver Base so that it switches to "Saving"
13. Select Execute and a dialogue appears. Select "Save FRD Standard File"
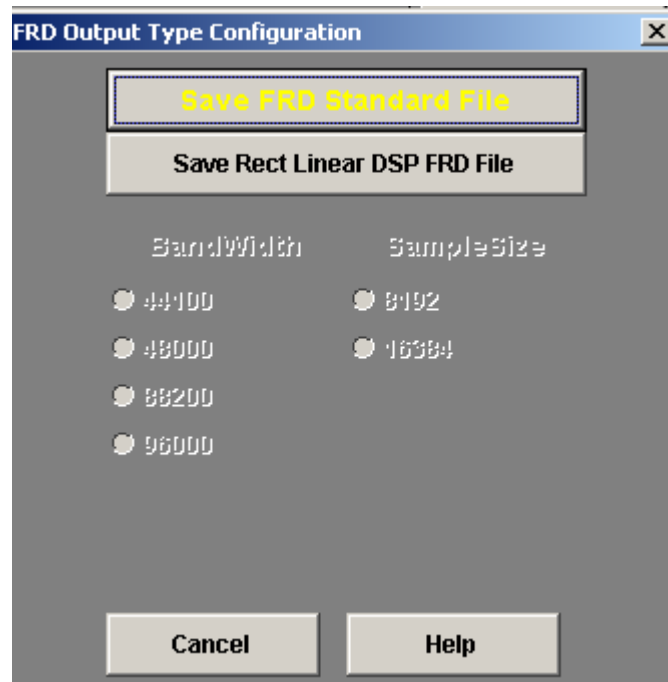
14. Select where you want it to save and save it under that file name.
15. Repeat this for each driver

To Import to Speaker Workshop,
In Speaker Workshop:

16. Open/New/Project
17. Resource/New/Folder
18. Create a New Folder for each data set
19. Resource/Import/Select the appropriate file (.zma, .frd)
20. Open. This imports that file to the projects base directory.
21. Move it into your chosen file by dragging and dropping it for all files of interest.

EXTRACTING A DRIVER RESPONSE FROM BAFFLED MEASUREMENTS
(WITH PAUL VERDONE'S FREQUENCY RESPONSE COMBINER)

The Frequency Response Combiner (FRC) spreadsheet allows you to calculate measurement conditions under which a supplied frequency response was obtained and then to project in your target enclosure's effects.

1. Open the FRC spreadsheet
2. Select Loading so that Loading is displayed
3. Press Execute
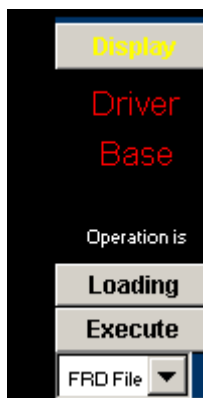
4. Select the files to load under "Driver Base" column
5. Select OK
6. Select "Load" under the "Box Measure" column and find the ".frd" file corresponding to the Enclosure response (either your measurement or that from the manufacturer).
7. Select "open"
8. The Baffle Measure Column is used to calculate in baffle effects. Use the FRD Consortium Program Baffle Diffraction Simulator to simulate the baffle response of the test baffle (knowing the dimensions of this) that the driver was measured on for the .frd file loaded in step 6. You would select 2pi from the drop down menu if the driver was measured on an infinite baffle.
9. The Box Target Column is used to identify enclosure effects on the frequency response. Using the FRD Consortium program, Unibox, you would enter the driver's Thiele Small Parameters and calculate a frequency response curve. Use 1 watt for your nominal power when doing this. Under the "Box Target" column, with it set to "Loading", press "Execute" and choose the Unibox generated frequency response to load the Unibox generated frequency response of your enclosure design.
10. Going back to the Baffle Diffraction Simulator (see above), you would model your baffle to develop a predicted frequency response curve from your baffle. This .frd file should be loaded under the Baffle Target Column.
11. Select the "Normalizer" from the menu on the right side of the screen.

i. Adjust the values for Samples to 150 for slower computers and to 300 for faster computers
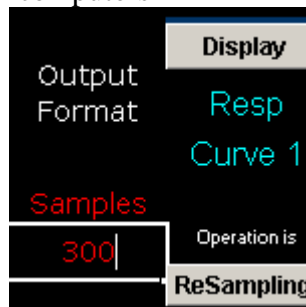
ii. Adjust "Start Freq" to 10

iii. Adjust "End Freq" to 40,000

iv. Extend the slopes of the responses
1. The "Extend Slope" mode extends the response along whatever slope it was following for the last few data points.
v. Choose "Extend Top Slope"

1. If the "Driver Base" information has a rising response at the top end then you'll have to select a different top end slope normalization setting. This would be done here:

2. You usually want a 2nd order (12 dB/octave) roll off in which you would choose
   a. "Aim Top at Point Slope"
   b. choose a frequency
   c. choose a magnitude level
   d. Changes can be made by toggling the following buttons (I have selected each of the options in the second row and you can change the numeric values below:

Select Top Slope	Select Top Slope	Select Top Slope	Select Top Slope	Select Top Slope	Select Top Slope
Select Low Value	Aim Low at Point	Aim Low at PointSlope	BaffleGain to Zero	Extend Low Slope	Extend Low Value
-12 db/oct	-12 db/oct	-12 db/oct	-12 db/oct	-12 db/oct	-12 db/oct
-36 db	10 Hz -36 db	10 Hz -36 db -12 db/oct			

   vi. Choose "Extend Low Slope"
   vii. Keep the default "extend Slope" settings for "Box Measure"
   viii. Keep the default "extend Slope" settings for "Box Target"
12. Set the "Baffle Measure""
   i. Extend Top Value"
   ii. "Extend Low Value"

13. Set "Baffle Target"
    i. "Extend Top Value"
    ii. "Extend Low Value"
14. Now click on the "Execute" button for each column that has loaded data including the "Baffle Measure" column if you haven't selected "2pi".
15. Once all of the data has been normalized, select the "Driver Correction" menu bar
16. Choose "Combine Response."
    i. This will generate a frequency response for the driver mounted in your test enclosure.
17. Generate phase information for this response
    i. Select "Precise Phase & Group Delay"
    ii. "Extract Min Phase."
        1. Wait while it determines the phase response for your driver/cabinet combination.
18. Select the "Save/Execute" button in the "Sum Response" column to save the response data.
19. The resulting file can be used to design a crossover for your project.

COMPARING THE CALCULATED RESPONSE TO YOUR TARGET RESPONSE

1. Right Click the graph
2. Click Add
3. Select the target response and it will be added to the graph.
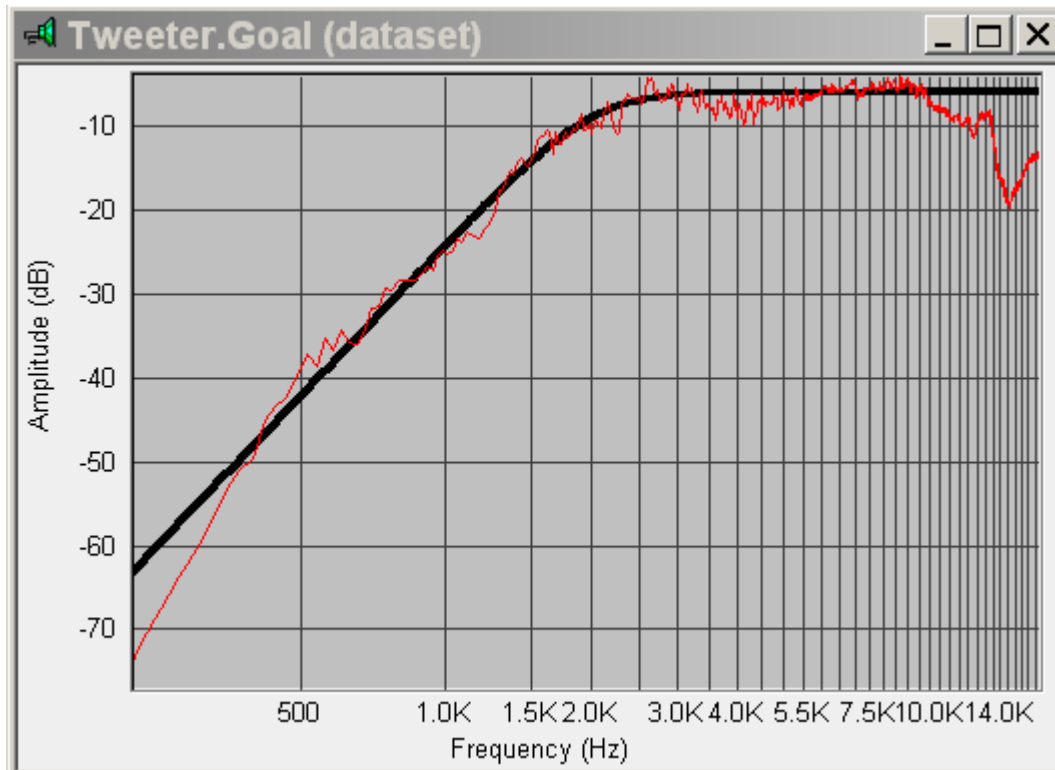
4. Adjust the component values until the simulation approaches the target response.
   Note: Using the network options pop-up menu will automatically update the frequency chart for the network. To use the network options feature:

5. Make the network chart active
6. Select the component to change by single left clicking on the component symbol
7. The network options pop-up will show the current value of the component
8. Modify this to the desired value
9. Click apply
   > *The component's value will be changed in the network and the network frequency chart will be update to reflect the change.*
10. Once you have closed in on the goal and things look pretty good, set the components to the nearest available values.
11. Recalculate the response
12. Add the network goal response to the network frequency response chart to compare it to the goal response.

Small differences will generally not have much impact. You might need to make some adjustments if they do.

13. When the response overlaps the goal, you are ready to combine (add) the networks together to get the total combined frequency response for the loudspeaker design.
14. Change the goal when the next network is being optimized.
15. Combine the Curves
16. Click on the woofer.total freq network frequency response graph.
17. Click the graph/Menu/Calculate/Combine. This will bring up a window from which you can choose a curve to combine.
18. Click the *network.driver.freq* file for the other network (tweeter or midrange). You can combine these in many ways
19. The dialogue box will give you the option to add, subtract, divide, etc., click add.
20. A dip near the crossover frequency suggests the need to reverse the polarity of the tweeter.
21. Fine tune components to get the desired response. Any changes to either network will require you to re-do network/calculate response and calculate/combine of the frequency charts.

Note: Using the network options pop-up menu will automatically update the frequency chart for the network. To use the network options feature:

22. Make the network chart active
23. Select the component to change by single left clicking on the component symbol
24. The network options pop-up will show the current value of the component
25. Modify this to the desired value
26. Click apply
27. The component's value will be changed in the network and the network frequency chart will be update to reflect the change.
28. If you optimized your crossover networks well there should be a fairly flat response one way and a dip or null of over -10 dB with opposite polarity of the tweeter or midrange.
29. For Even Acoustical Order Crossovers, one can check the phase by looking at the reverse null response. For even order crossovers, a reverse null that is wide, symmetric, and deep (20-30 dB) suggests good phase tracking through the crossover region. The lower the order of the crossover, the wider the null should be to cover the entire range.
30. If you need to re-do the polarity, you can open Network/Properties/Driver or double click on the driver symbol in the network chart and flip the polarity.
31. When this is done the polarity symbol for the driver in the network chart will change from "+" to "-".
32. If you have made any changes to the network you will need to repeat from step b.
33. If you want to compare individual network sections here, right-click the graph and click add. Then add the network responses or even the goals if you want. If you measured the actual system, you could add it here and compare the theoretical with the actual!

Note: If you combined with "+" symbol showing in the network chart for the driver, the driver should be wired in-phase. If the flattest graph was combined with "-" symbol showing in the network chart for the driver, wire the driver out of phase.

34. Copy components from both low pass and high pass networks into the Combined Network.
35. Press Ctrl-C to copy an entire network.
36. Press Ctrl-V to paste an entire network.
37. Make sure that there is only one source in the Combined Network and that it is correctly wired.
38. Select Network/Properties/Drivers and make sure that "individual response" is selected.
39. Calculate the response for the combined network.
40. This will generate three new charts:
41. Combined *network.tweeter.frequency*
42. Combined *network.name.woofer.frequency*
43. Combined *network.name.total.frequency*.
44. Another option is to generate the combined low and high networks in a new network and then
    a. Calculate Frequency Response to see the full range response
    b. Generate the network impedance by Network/Properties/General
    c. Generate the individual response under Network/Properties/Driver.
45. Experiment with different components and networks until you get the one you want. Remember that you can generate many different topologies that can sum to flat yet they may sound differently. This is where tweaking comes in. It may be due to different impact of the baffle diffraction off axis vs. on axis or due to phase differences.
46. If there is a slight bump at the crossover, you could attempt to separate the high-pass and low-pass goals slightly, then recalculate.
47. At anytime you can right click the chart and use the *undo combine data* to return to the previous chart version.
48. Once you have used *undo* it will switch to *redo* so that you can flip back and forth between the charts.
49. Dips and peaks are most often going to be from phase errors if they were not seen in the driver acoustical response measurements.
50. The next step would be to build a test network, listen, and measure.

USING SPEAKER WORKSHOP IN ACTIVE CROSSOVER SPEAKER DESIGN

(this section was developed and written by Sergiusz Urbaniak, Berlin, Germany and has a few portions that need to be finalized)

(Section Index)

Software Requirements

1. Speaker Workshop
2. LTSPice/SwitcherCAD III, which can be downloaded from http://www.linear.com/company/software.jsp. Follow the LTSPICE link, register optionally and install the software.

Preparation

Unfortunately LTSpice doesn't come with a potentiometer built-in, which is very often needed, in active filters for setting gain, phase, etc. So before you can use pots in your circuit you have to perform the following steps:

Copy the following file to the <install_dir>/lib/**sym** directory (just drag&drop in the file explorer):

potentiometer.asy

Copy the following file to the <install_dir>/lib/**sub** directory:

potentiometer.sub

There are many other LTSpice library files, just have a look at the pretty active LTSpice Yahoo Group at http://de.groups.yahoo.com/group/LTspice.

Now let's start the program and create a new schematic

New Schematic

You will become an empty area where you can place different components and LTSpice directives. This toolbar is going to be used the most:
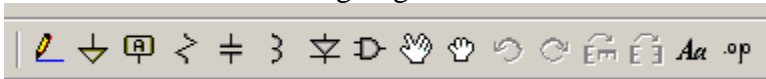
The first thing we have to include is a signal generator. First click on the component link:

Component

Then select from the list the voltage source:
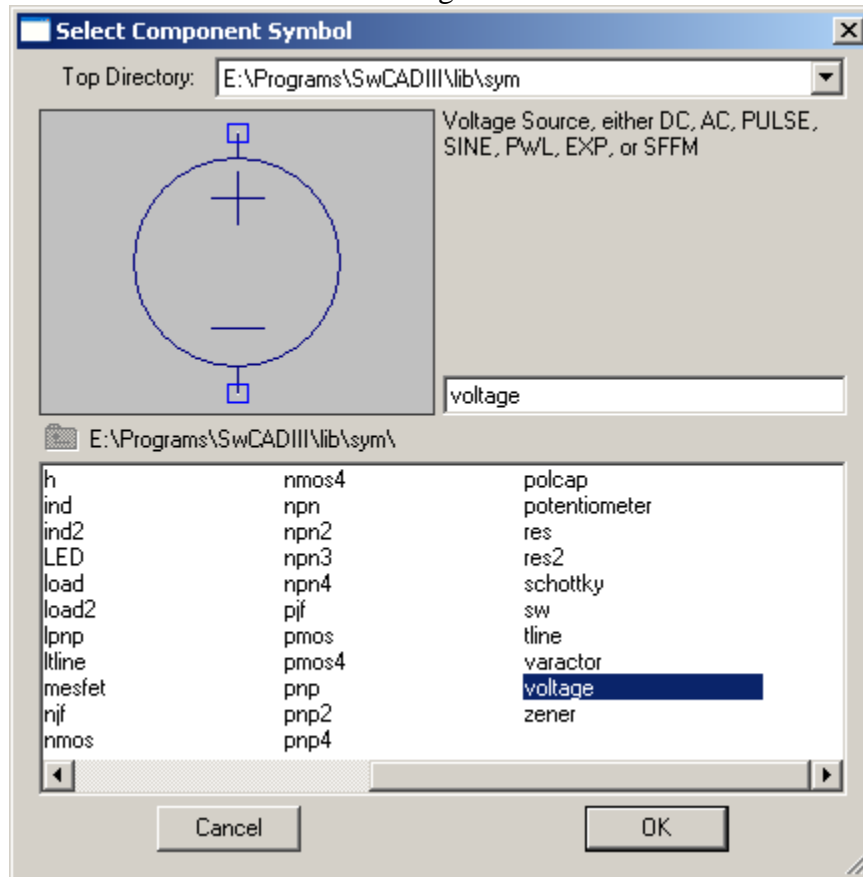
As you can see there are many components in this library, also our manually included potentiometer. There are also opamps, which you will need to include.

Place the voltage source anywhere on your empty circuit. First thing we'll do now is to set the minus pin to ground. Select the ground symbol:

And place it underneath the minus pin of the voltage source.
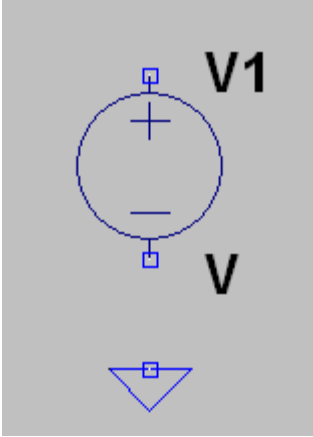
Now LTspice wants you optionally to place another ground symbol, we don't need that so simply click the ESC button. Now let's wire the ground pin with the minus pin of the voltage source using the wire symbol:

You will see a cross in the circuit pane, first left-click on the minus-pin of the voltage source, drag to the ground-symbol and left-click again. After that again press the ESC key since we don't want further wires for now.
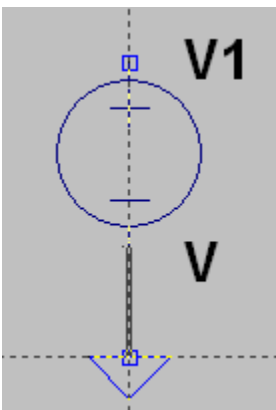
Voila, you made your first wire connection between two symbols. That way you can include resistors, caps, opamps, etc. and connect them with wires.

Now set up the voltage source. Right-click the voltage source symbol on the circuit pane to edit its properties. In the window you can enter a fixed voltage for this source, but that is not what we need. So click on the <u>Advanced</u> button and the following property window appears:
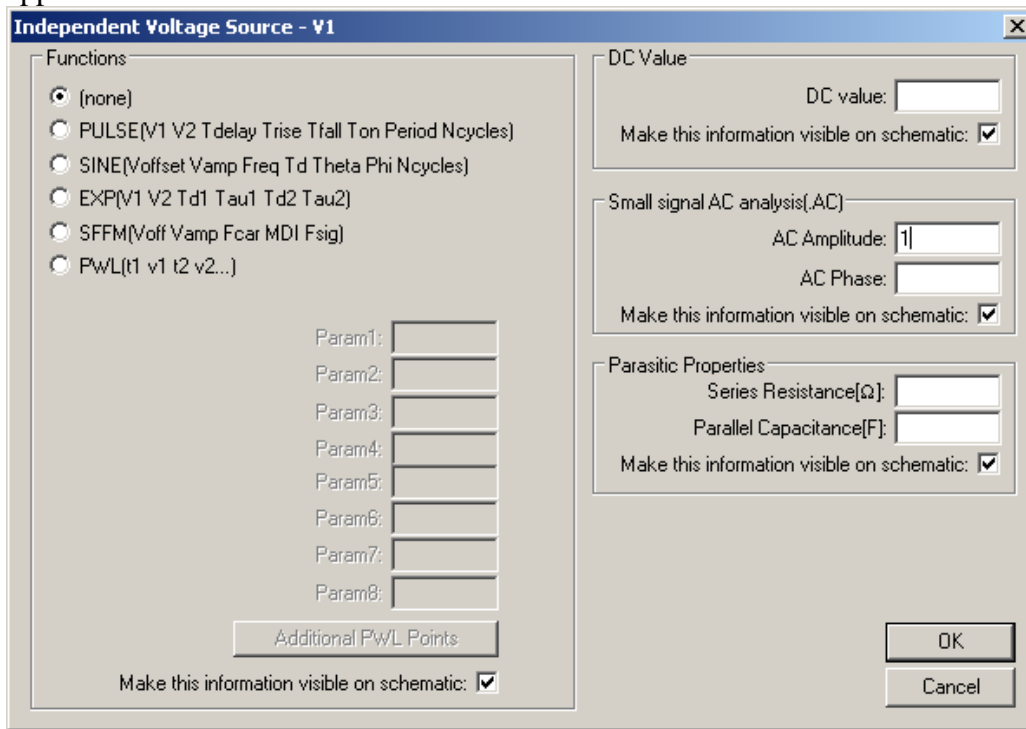
What we need is the "Small signal analysis". Just enter "1" for the AC amplitude.

The next thing we'll do is to tell the LTSpice engine how the analysis has to be done. Click on the SPICE directive symbol:
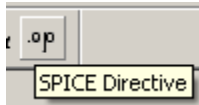
And enter simply ".ac".
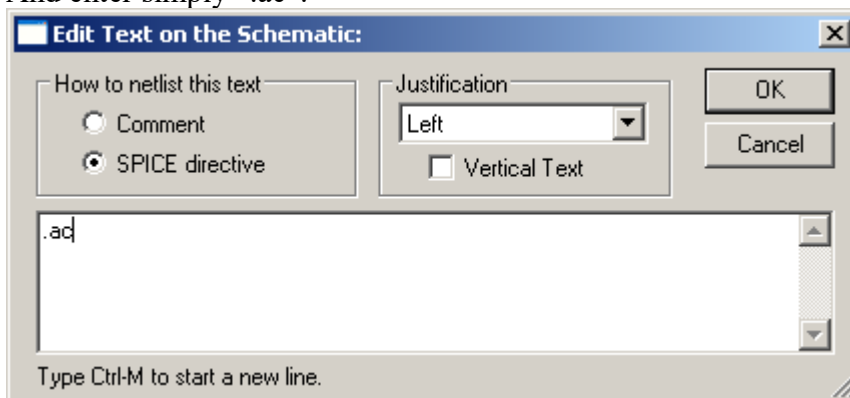
Now you can place this directive anywhere on your pane as any other component.
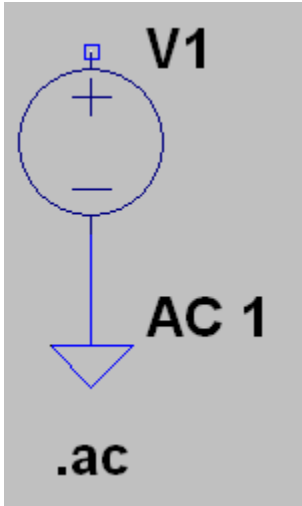
Now right-click the ".ac" directive on the circuit pane to enter the following properties:
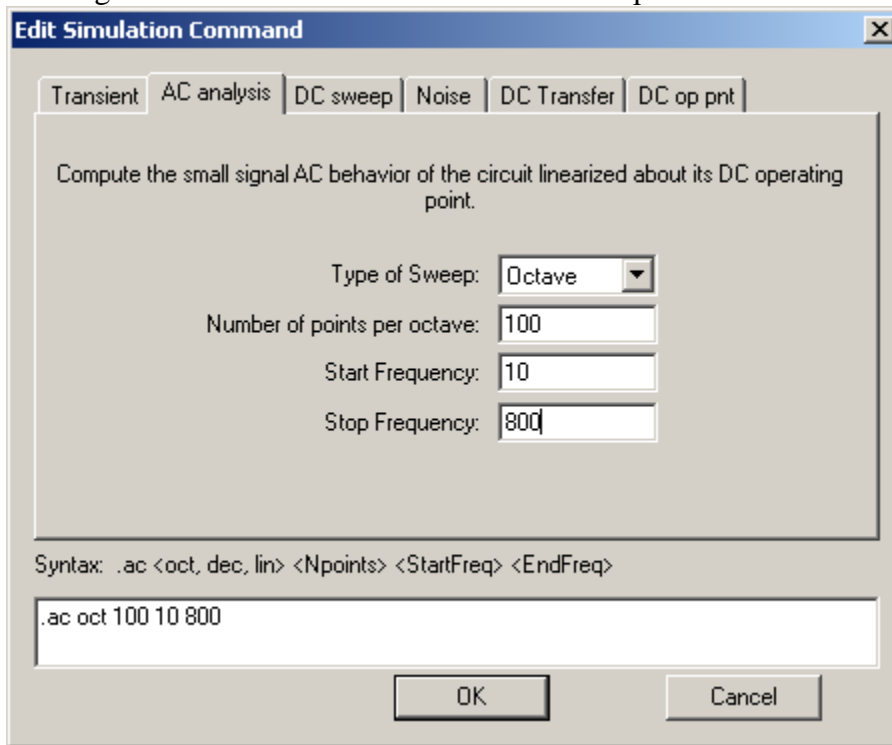
Take a little care for the "number of points" setting since it affects the file size of the exported file and Speaker workshop can get into trouble.

Now we set up the AC analysis with a frequency range from 10 Hz to 800 Hz (should be sufficient for the subwoofer, for active filters affecting the whole frequency range, enter "20k") with 100 points per octave.

Design a 2nd order Linkwitz Low pass filter

Let's design a subwoofer using an active filter. The subwoofer amplifier is going to be an old unused stereo amp. The active circuit can be first modelled in LTSpice, then you could make a PCB using the freeware edition of eagle (see www.cadsoft.de) or you can solder point-to-point.

Now the first thing we need is a concept for the circuit. There are many places on the internet where you can examine nice active filter circuits, i.e.:

- http://www.linkwitzlab.com/filters.htm (Mr. Linkwitz's site is a knowledge treasure, not only the filter part is interesting)
- http://sound.westhost.com/projects-3.htm (Rod Elliot ... very interesting)

Now let's see what Mr. Linkwitz suggests for a 2nd order 99 Hz low pass filter:
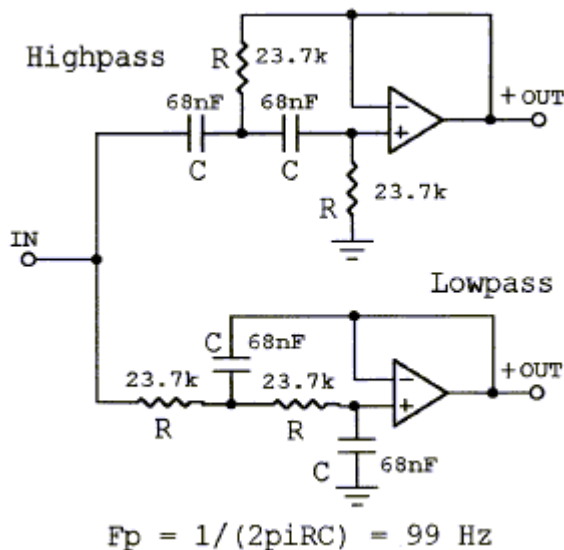
Let's redraw the Low pass section using resistors, caps and an "ideal" opamp. If you are too lazy to do this, just load the following file into LTSpice:

Lowpass.asc
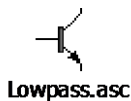
You should see the following schematic:
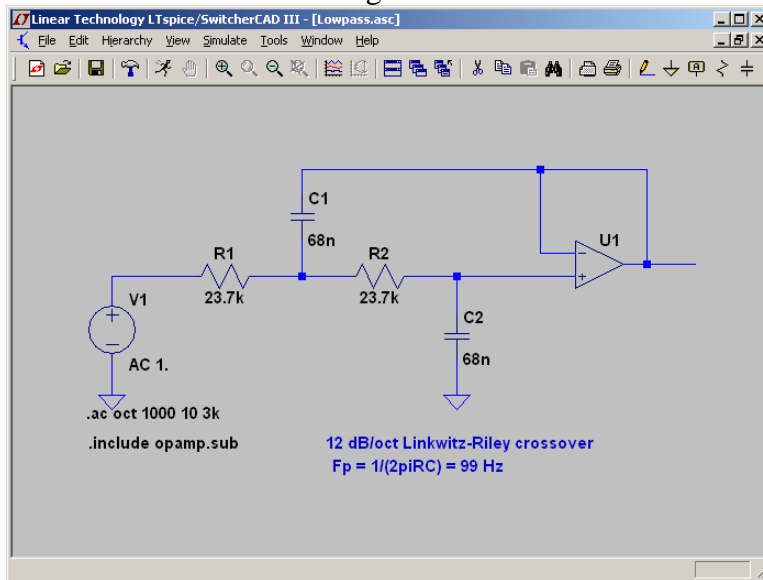
As you can see there is another LTSpice directive ".include opamp.sub" which tells the LTSpice engine to use the opamp library, otherwise the engine doesn't know how to handle this active circuit. What you can also see is that you can enter comments and texts, which illustrate your circuit. These texts are blue.
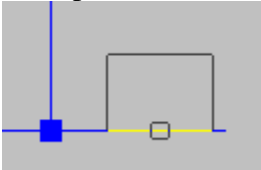
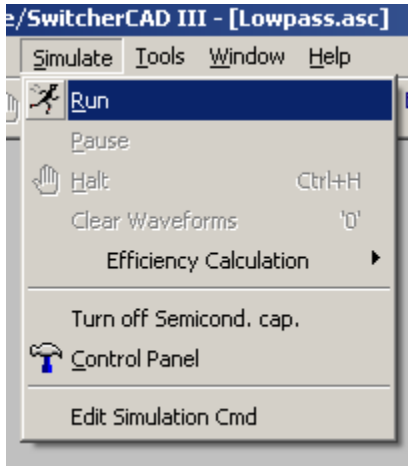Before we run the simulation let's label the wire at the output of the opamp by right-clicking the output wire and select "Label net":
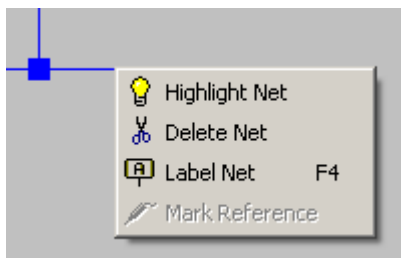
Just enter "out" in the text field and press Enter:
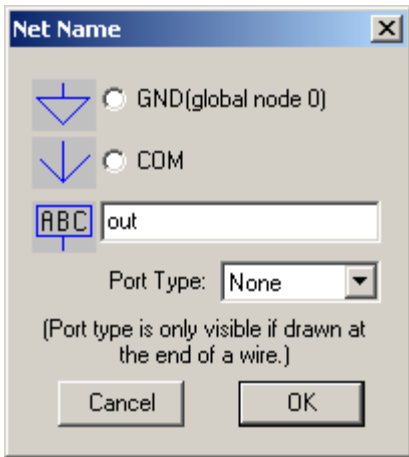
Now place the label <u>on the wire</u> to associate the label with the wire:

Now let's run the simulation:

You should see a window asking you which wire to show on the output window. Select our labelled net "V(out)":
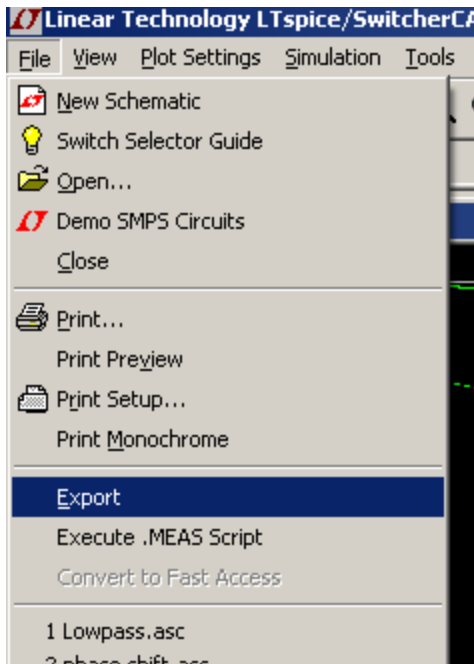
And now you have a wonderful transfer function and phase response of this filter:
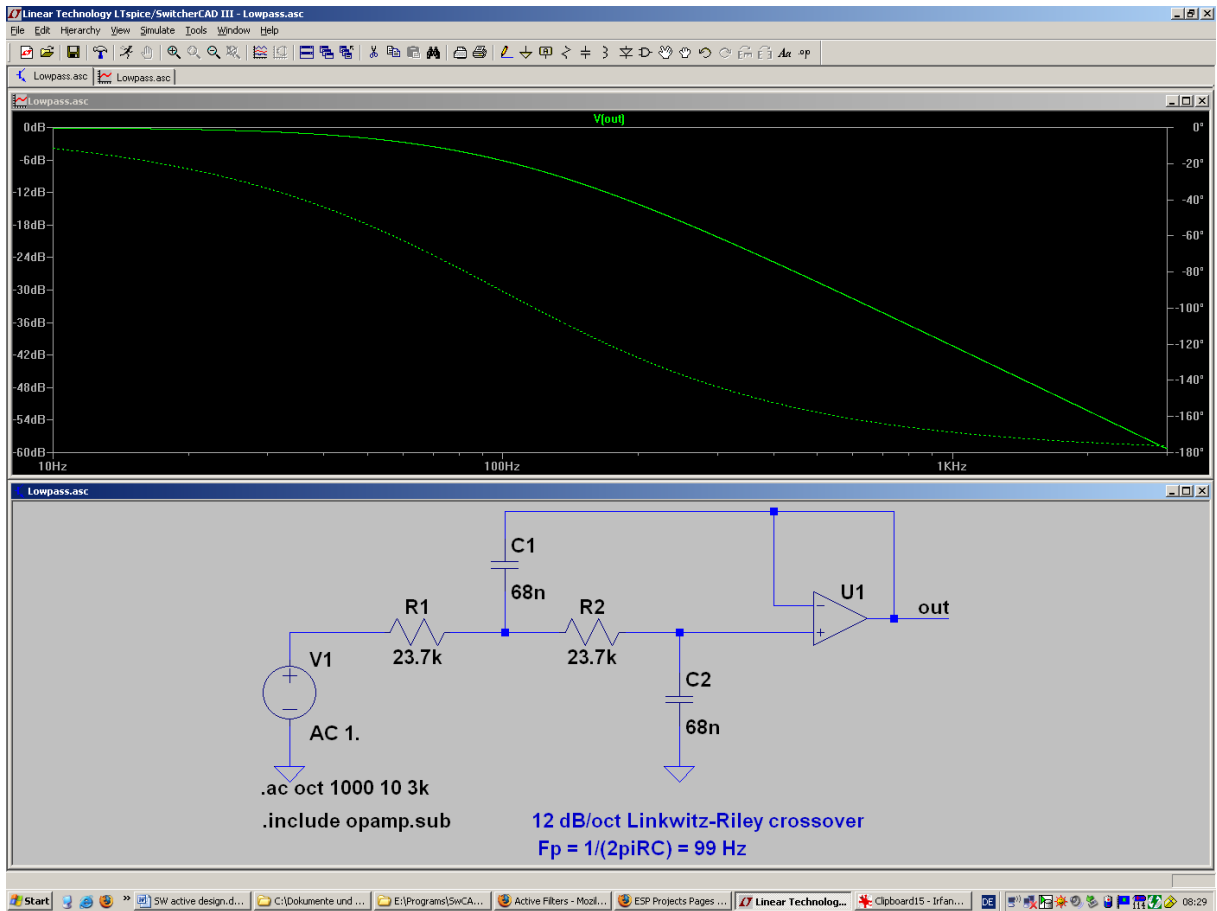
Export to Speaker Workshop

Now you are ready to export the filter transfer function to speaker workshop. First active the "Lowpass.asc" window graph representing our transfer function.

Then click on Export:

Again select our "V(out)" wire and save the file as "c:\export.txt".

Open the following conversion sheet:

conv2000.xls

Execute the "newconvert" macro. You should see the import.frd file in excel. Close excel without saving.

TODO: Currently the conversion sheet takes c:\export.txt and converts to c:\import.frd. I'll include a nice file selection. Also the closing of the .frd file should be done automatically, I can include this.
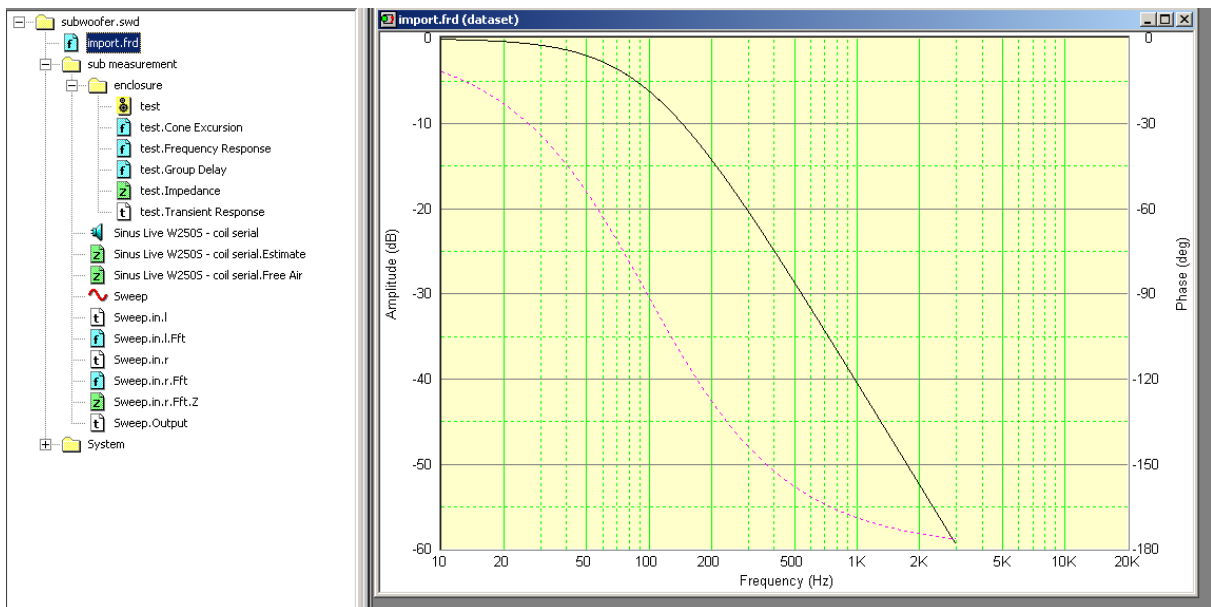
Now there should be two files on your C: root directory:
- export.txt
- import.frd

Open Speaker Workshop and open your subwoofer project:
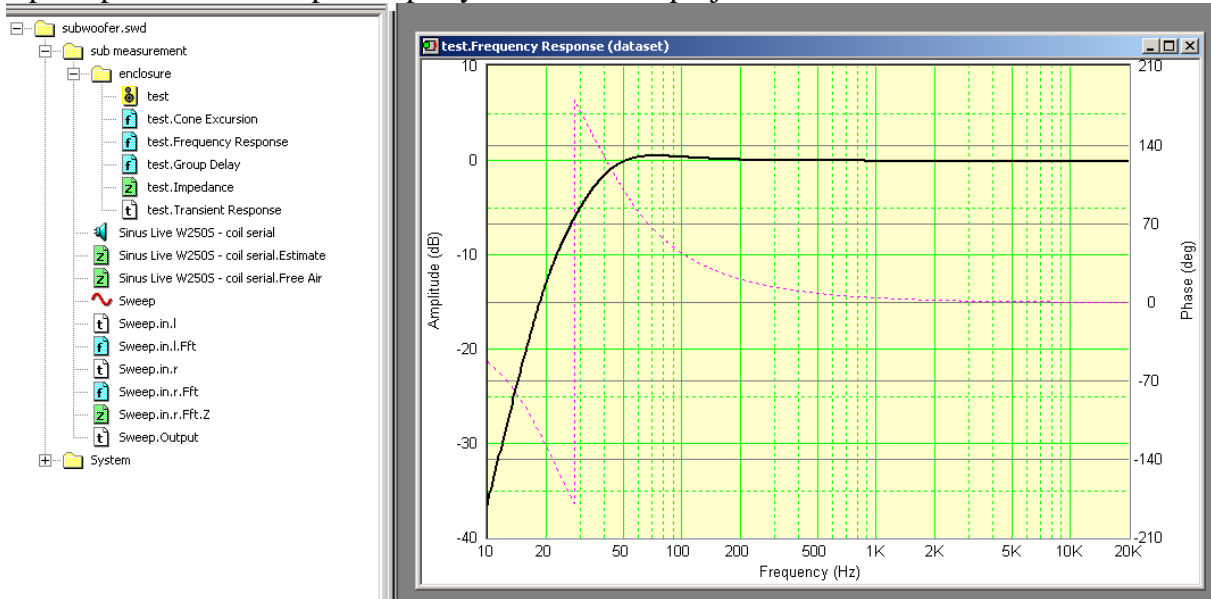
Now import the generated import FRD file and open it in SW. You should see our filter transfer function:


Now what you can do is to create a directory "Active Filter" and rename the file.

The next step is to take your subwoofer frequency response (either generated by the enclosure simulation or real measurement) and apply the filter transfer function by simply multiplying the response. We can do this because the active filter modifies the input signal to the amplifier and is not a passive circuit acting after the amplification. So the impedance of the speaker does not affect it, only by its natural response.

(TODO: someone with EE knowledge should write this explanation)

So let's multiply the subwoofer response and the active filter response:
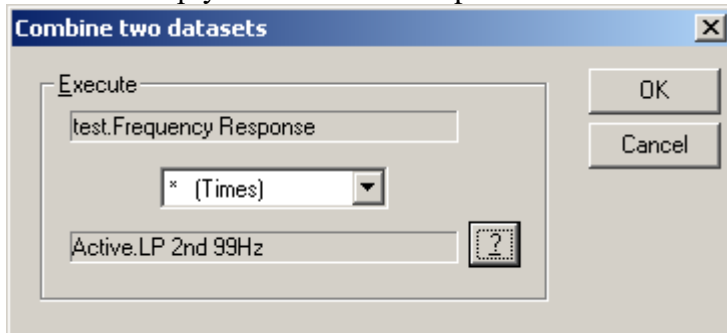
And voila, you have the predicted low pass response including the active filter:
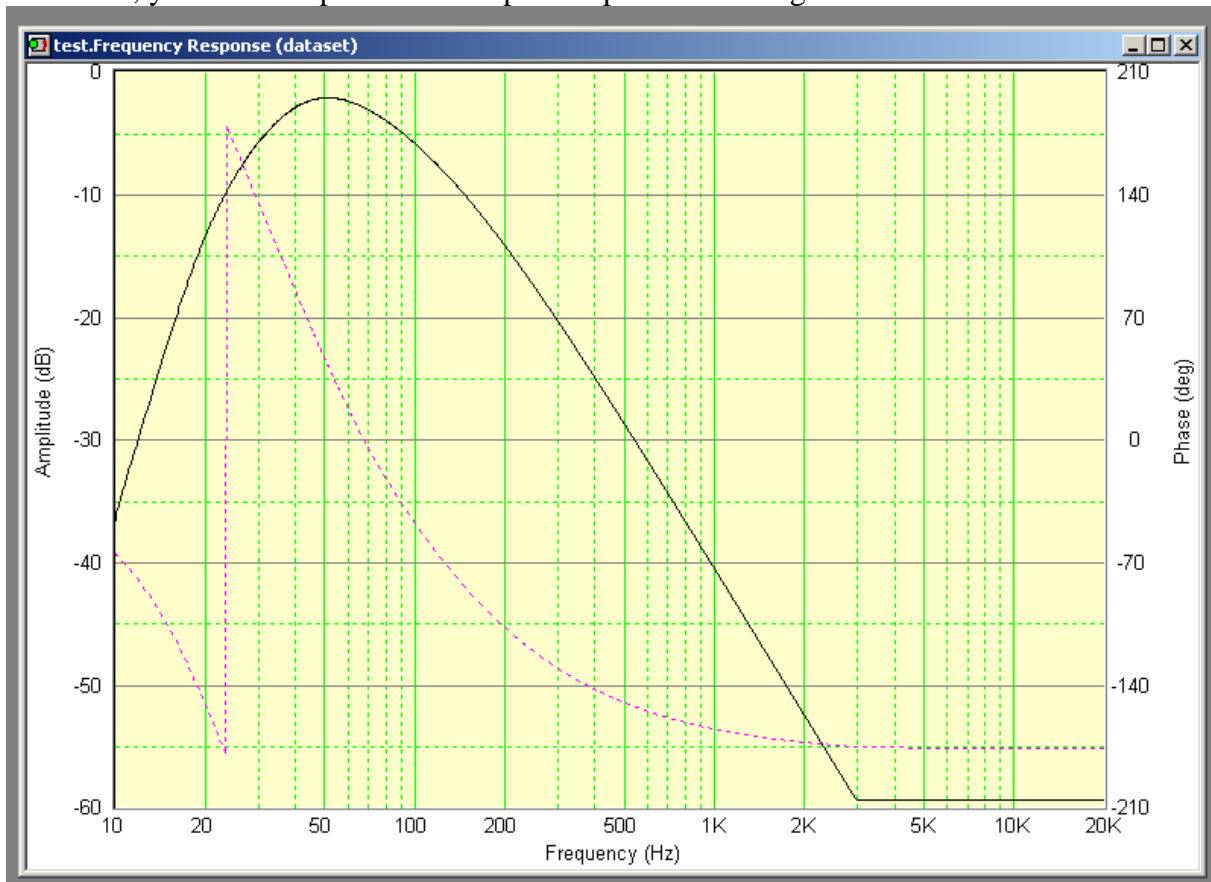
Please note that this transfer function is only valid until 800 Hz because we told the Spice engine to generate the filter function only to this frequency!

A last note worth mentioning is that this method also takes phase into account. If you compare the original natural subwoofer response with the active filter response and their phase curves you see the change:
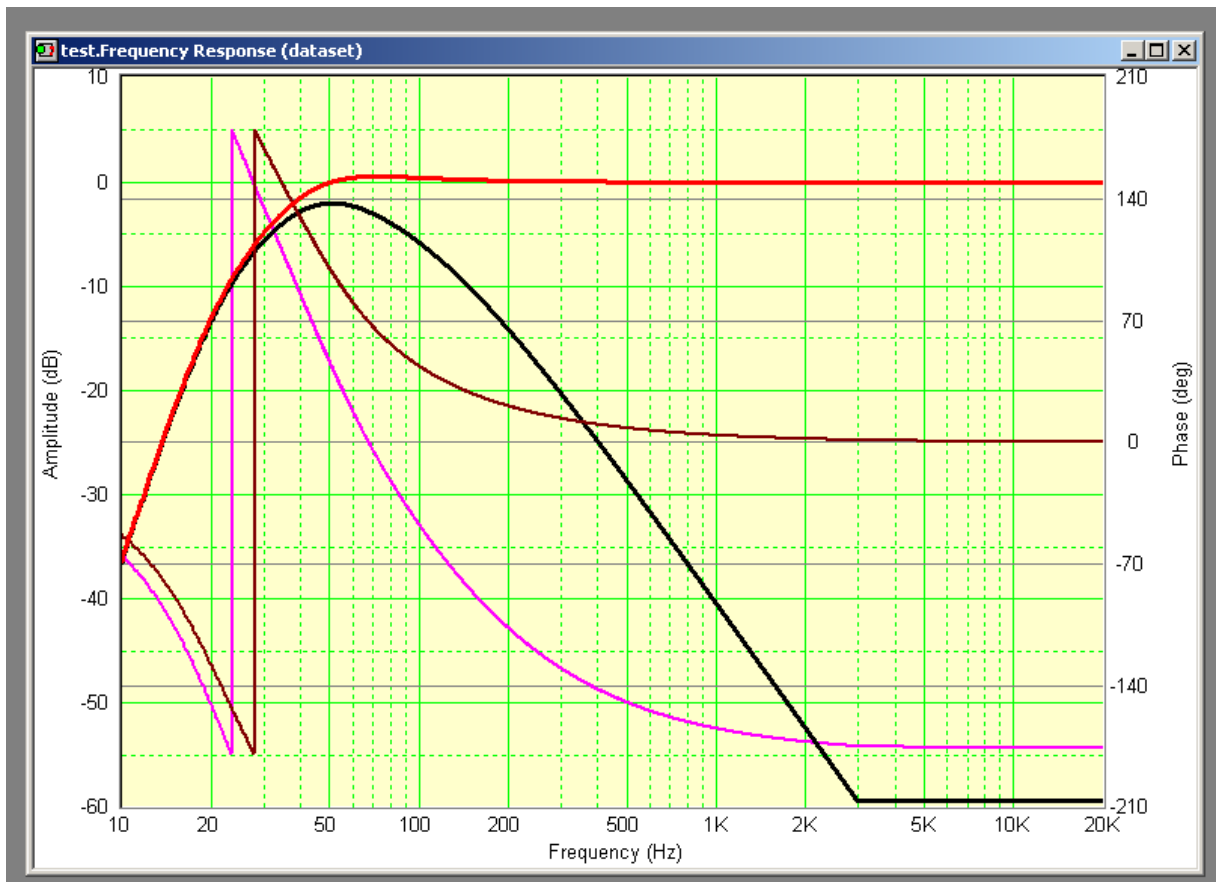
This way it is possible to accurately create active filters and to predict the response in combination with the natural speaker responses. This method can be applied also to mid and high frequencies and can be used to model an active loudspeaker setup.

TO DO: I can write another chapter with potentiometer settings and a bass boost circuit or Linkwitz transformation example.