LspCAD 6

2025-05-02

Change history

2025-05-02

- LspCAD 6.68, Updated diffraction and near boundary model

2025-04-19

- Major “been a real real long time coming” update.

2007-06-30

- Ch 9.1.7
  Text about extended data format added.
- Ch 4.5
  Info about Windows Vista added.
- Ch 9.2.3.10
  Info about BiQuad pole-shifter added.

2006-06-08

- Ch 9.1.2, 10.1
  Text about optimizer changed, main difference is that the optimize with respect to feature is removed as it only caused problems for the users.
- Ch 6.2.2
  Optimize checkbox in group properties dialog explained.

2006-05-01

- Ch 11.3
  Added description of ASIO access method in crossover emulator.
- Ch 9.3.8.2
  Added description of Ke parameter in dynamic loudspeaker model.

2005-10-12

- Ch 9.2.3.11 page 38
  Added description of Digital phase linear FIR filter
- Ch 11.1 page 64
  Added description of crossover emulator feature **Causal filter margin** and also description of exported impulse responses.
- Ch 6.2.1 page 14
  Removed comment that sets are not stored
- Ch 9.4.3 page 50
  Some changes in text

1 Introduction

LspCAD is a software targeted at the design of loudspeaker boxes and crossovers.

With LspCAD it is possible to model loudspeaker boxes, passive crossovers, active analog and digital crossovers.
A nice feature is that it is possible to model everything at the same time, i.e. there is no need to do box modeling first and crossover modeling later, this makes prototyping of new projects easier.

Crossovers and boxes are created from a tray of components, the components are picked from the tray and dropped on the schema. This allows for a larger degree of freedom for the user, on the downside is the problem that the design procedure may become more tedious. To alleviate this potential problem, the concept of network sets is implemented. Moreover, a number of filter templates are available. Another concept that simplifies designs is the group concept where one can simply group a number of components and later with a simple mouse click change on one set of component values to another set and see the results of the changes instantly.

The boxes are designed with building blocks such as ports, cabinet volumes and driver units. For well-known box types a special wizard functionality helps the user to pick an optimal alignment for the given driver.

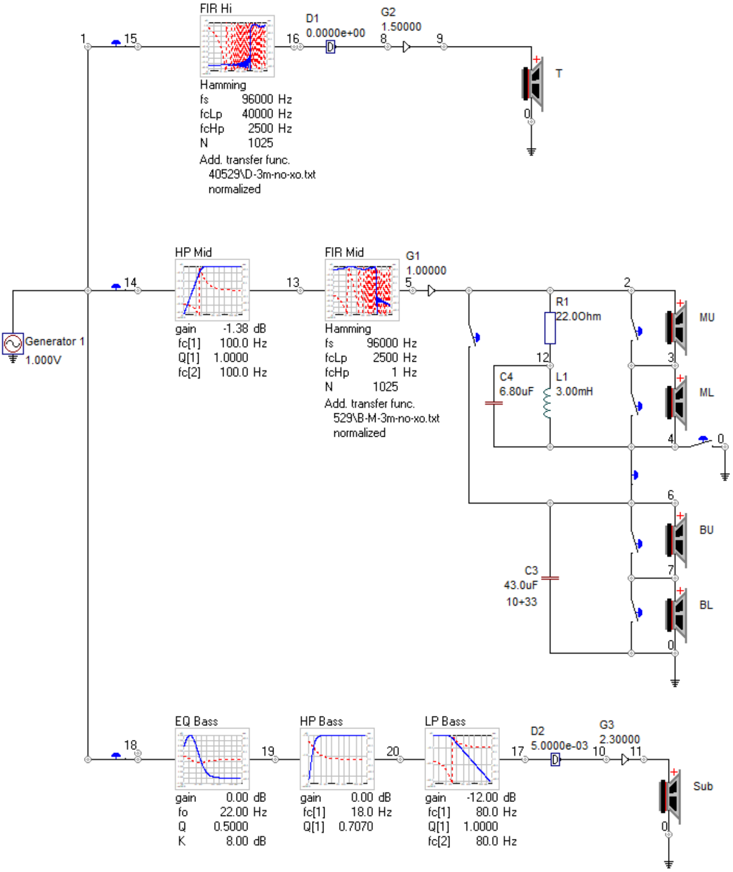
Crossovers that can be implemented are, besides analog passive and active filters, also digital filters with unit delays and sum units. A paradigm that makes the initial prototype modeling simpler is the parameter mode that allows the user to modify parameters such as crossover frequency and Q value rather than having to fiddle with component values. Components for well-known commercial digital filters such Behringer, dbxPro and XTA is implemented and also optimized. Additionally, it is possible to model phase linear loudspeaker systems and also to use impulse responses exported from LspCAD 6 in external convolvers such as BruteFIR or commercially available digital crossovers such as MiniDSP.

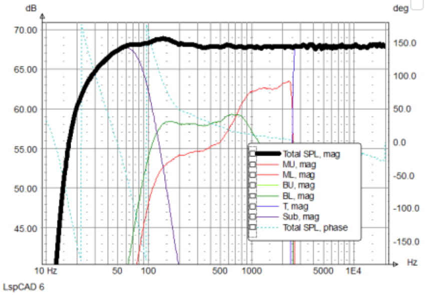
To be able to achieve a given target frequency response a powerful crossover optimizer can be used. This optimizer also gives the option to put constraints on things such as component values and resulting load impedance, the unique feature of the load impedance optimization is that it actually can increase the load impedance while maintaining the desired frequency response. Another feature is that one may lock the crossover frequency during the optimization.

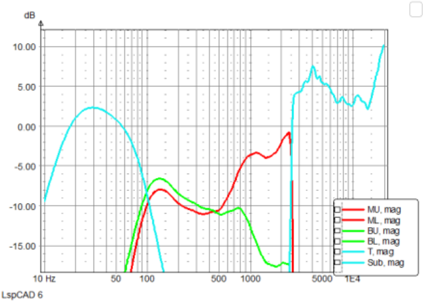
The tolerance analysis helps to give a view of how the resulting frequency response of a crossover deviates from the original response once we add the possibility that components might deviate from their specified values

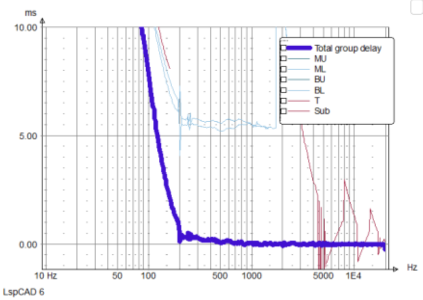
The popular real time update feature in LspCAD gives the user a "What you see is what you get" feeling as the graphs are updated each time one change e.g. a component value. Similar to earlier releases it is possible to store intermediate work.

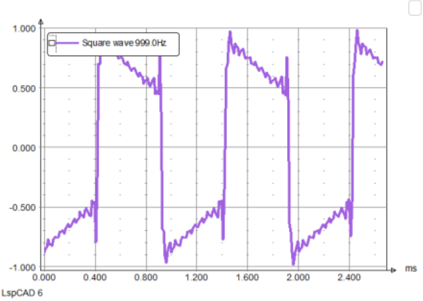
In release 6 the number of open projects is only limited by memory. The methodology differs slightly from earlier releases, rather than storing the current status in a memory location one can in release 6 see each construction as a sheet of paper that one scribble notes on, once one is done with a construction step one can either get a new sheet of paper (**New**) or go to the copying machine (**Clone**).
The schema below (with the resulting response further down) shows the crossover for the authors own Earendel loudspeaker system that use both passive analog components as well as digital IIR and FIR filters based on MiniDSP.




dB
deg
70.00
65.00
60.00
55.00
50.00
45.00
150.0
100.0
50.0
0.0
-50.0
-100.0
-150.0
Hz
10 Hz
50
100
500
1000
5000
1E4
Total SPL, mag
MU, mag
ML, mag
BU, mag
BL, mag
T, mag
Sub, mag
Total SPL, phase
LspCAD 6


dB
10.00
5.00
0.00
-5.00
-10.00
-15.00
10 Hz
50
100
500
1000
5000
1E4
MU, mag
ML, mag
BU, mag
BL, mag
T, mag
Sub, mag
LspCAD 6


ms
10.00
5.00
0.00
10 Hz
50
100
500
1000
5000
1E4
Hz
Total group delay
MU
ML
BU
BL
T
Sub
LspCAD 6


1.000
0.500
0.000
-0.500
-1.000
0.000
0.400
0.800
1.200
1.600
2.000
2.400
ms
Square wave 999.0Hz
LspCAD 6

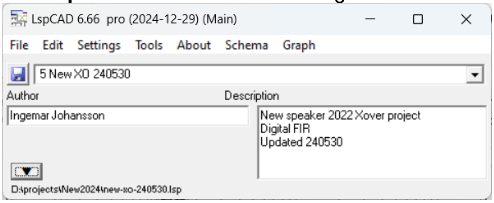
2 Examples and tutorials

Examples and tutorials are collected in a separate document, it may be well spent time to glance through that document as it describes how the software is used in a real situation.

3 Various stuff

3.1 Entering text

It is possible to enter text in various windows. An odd feature is that, for the text change to take effect one must click with the mouse button in another field. For example if one has entered a text in the **Author** field, then one must click with the mouse button in the **Description** field to make the change take effect.



5 New XO 240530

Author: Ingemar Johansson

Description: New speaker 2022 Xover project
Digital FIR
Updated 240530

D:\projects\New2024\new-xo-240530.lsp

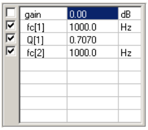
3.2 Entering/changing values

Numerical values are presented by three means

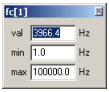
Property list

The properties in the property list can be modified in a number of ways.

1. Left-click on the parameter that you wish to modify, then click on the arrow left-right keys on the keyboard.
2. Left-double-click on a parameter, then enter a new value or use the arrow up-down to modify, to exit, click somewhere else.
3. First left-click then right-click on parameter. You now have the option to set the value and also upper and lower limits for the optimizer



□	gain	0.00	dB
☑	fc[1]	1000.0	Hz
☑	Q[1]	0.7070	
☑	fc[2]	1000.0	Hz

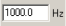


fc[1]

val 3966.4 Hz
min 1.0 Hz
max 100000.0 Hz

Single numerical input field.

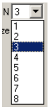
Left-click inside the box and enter the new value or use the up/down arrow keys to modify the values, the value increase is bigger if the Ctrl button is also held down.



1000.0 Hz

Dropdown list

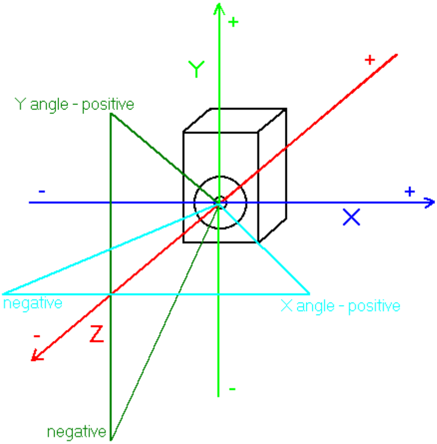
Used for properties such as filter order et.c. Left-click and select the desired value.



3
1
2
3
4
5
6
7
8

3.3 Coordinate system

As the drivers and other radiating elements in LspCAD can be placed at different locations relative to one anther and also made to radiate in other directions than perpendicular to the baffle that faces the listener each other we need to have a coordinate system.
The coordinate system is shown in the figure to the right.



Normally one would set the reference point exactly where one driver is located. The most convenient location is where the treble unit is located. Therefore the location of the treble unit is set to (dx,dy,dz) = (0,0,0).
The bass unit may be located with the center 150mm below the treble unit, additionally, due to the cone shape it is slightly behind the treble unit (25mm) .
The location of the bass unit is thus (dx,dy,dz) = (0,-150,25). Note that a positive dz is away from the listener.

3.4 Folders

Something can be noticed about the subfolders in the LspCAD folders

.\examples\	Contains examples that can be loaded
.\lib\	Contains PSD files that are useful for e.g thermal modeling purposes.

In recent windows versions, data is not stored in the typical C:\program-files\ folder. Instead it is stored in a virtual folder such as
C:\Users\USERNAME\AppData\Local\VirtualStore\Program Files (x86)\IJData\LspCAD\

3.5 Environment variables

A few environment variables can be set to make it easier to work with LspCAD

- LSPCAD6TEMP : Set path to folder to temporary files.
- LSPCAD6WORKINGDIR : Set path to LspCAD 6 working directory
- LSPCAD6_NO_AUDIO : Set parameter to 1 to disable instantiation of audio drivers in cases that LSPCAD6 crashes due to (rare) issues with audio drivers...

3.6 Windows versions

LspCAD 6 runs on windows versions up to 11 and likely beyond that.

3.7 Anti-virus

LspCAD 6 may trigger a warning by anti-virus such as Windows defender.

3.8 Some do’s and don’ts

3.8.1 Many dialogs open

It may be tempting to have a lot of dialogs open at the same time, for instance it is perfectly possible to run the optimizer at the same time as the crossover emulator is running. Mind though that this can become quite CPU intensive. Therefore, it is highly recommended to work with one different tool separately.

3.8.2 Several voltage sources

If one look at the schemas given by the many examples that come with this software one can see that for instance for a two-way crossover there are two voltage sources, one could say that biamping is used. This increases computation speed especially when the optimizer is used as the technique of using two voltage sources will ensure that only the necessary parts of a crossover are recomputed.

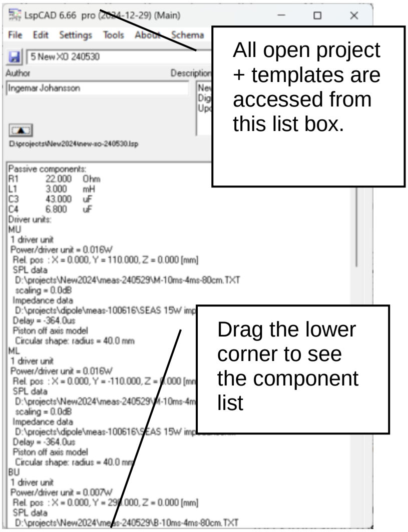
4 Main window

When LspCAD 6 is started two windows show up, one is the main window, the other is the schema window.

From the main window it is possible to do things such as open/save/close projects. Start the optimizer et.c.

In LspCAD one can have several projects active, the main control window gives the option to create new projects, clone the current project or simply delete a project. There is no limit to how many projects can be open (besides memory limitation).

Three special projects that cannot be deleted are the Scratch pad and the template sections, the template section contains ready to use (grouped) circuit elements of well-known circuits such as active filters and parallel resonance circuits, many of these elements are equipped with wizards that are very helpful at least in the initial design. There is also a user defined template project, here one can put personal templates.



5 New XO 240530

Author: Ingemar Johansson

D:\projects\New2024\new-xo-240530.lsp

```
Passive components:
R1        22.000    Ohm
L1        3.000     mH
C3        43.000    uF
C4        6.800     uF
Driver units:
MU
 1 driver unit
 Power/driver unit = 0.016W
  Rel. pos  : X = 0.000, Y = 110.000, Z = 0.000 [mm]
  SPL data
   D:\projects\New2024\meas-240529\M-10ms-4ms-80cm.TXT
   scaling = 0.0dB
  Impedance data
   D:\projects\dipole\meas-100616\SEAS 15W imp
  Delay = -364.0us
  Piston off axis model
   Circular shape: radius = 40.0 mm
ML
 1 driver unit
 Power/driver unit = 0.016W
  Rel. pos  : X = 0.000, Y = -110.000, Z = 0.000 [mm
  SPL data
   D:\projects\New2024\meas-240529\M-10ms-4m
   scaling = 0.0dB
  Impedance data
   D:\projects\dipole\meas-100616\SEAS 15W imp
  Delay = -364.0us
  Piston off axis model
   Circular shape: radius = 40.0 mm
BU
 1 driver unit
 Power/driver unit = 0.007W
  Rel. pos  : X = 0.000, Y = 290.000, Z = 0.000 [mm]
  SPL data
   D:\projects\New2024\meas-240529\B-10ms-4ms-80cm.TXT
```

All open project + templates are accessed from this list box.

Drag the lower corner to see the component list

The menu picks that are available are:

File … : Contains the well know save, open and close commands

Edit | Clone project: Possible to clone (make an exact copy) of the current project.

Settings : The window that pops up gives access to all project specific settings.

Optimizer : Open the optimizer window.

Emulator: Open the emulator window.

Tools | Snapshot : Open the snapshot window that makes it possible to take snapshots of the simulation results.

Tools | Tolerance analysis : Open the tolerance analysis window that makes it possible analyze the sensitivity to component variations.

Tools | justMLS: Start the justMLS measurement application.

Tools | LspCAD 5.25 : Start the good old LspCAD 5.25. LspCAD 5.25 was developed in the early 1990´s. Given that software does not age with beauty, as is the case with wine, there is a certain risk that LspCAD 5.25 won’t run on your PC.

5 Schema

The circuit schema of the project is drawn in the schema window.
There is no limitation to what one can simulate, this means that it is perfectly OK to simulate the low frequency part with a TS driver model while the simulation of the midrange and treble can be done with actual measured data. The schema is divided into 10 pages, each page can host its own circuits, and it is also possible to connect between schema pages with a dedicated connector component. Each page is 1600*1200 pixels wide (the grid size is 266*200), which means that quite large schemas fit into each page.

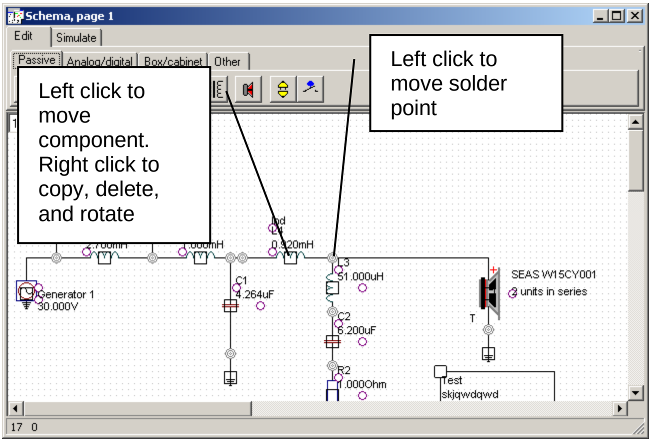
The schema window has two modes. The **Edit** mode and the **Simulate** mode.

In the edit mode the user can move, delete, duplicate, group and ungroup components. In this mode you create the layout of the project.

In the simulate mode one can change values, optimize and possibly also create and switch between network and driver sets.

5.1 Edit mode

In edit mode components are placed on a schema window according to the design goal, if the goal is a passive parallel topology two way filter the natural choice is two voltage generators, two driver units and a bunch of capacitors, inductors and resistors. Those who are familiar mechanical CAD software will get familiar with the schema editor in LspCAD pretty fast.



Left click to move component. Right click to copy, delete, and rotate

Left click to move solder point

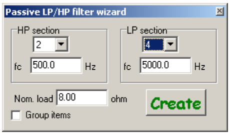
The components are picked from a tray and dropped on the schema. The leads of the components are soldered to one another.
In edit mode a component looks rather ugly, this is because it contains a number of hot spots. The round circles are the endpoints of the component connectors, red round circles also serve as locations for component text. The square is the hot spot used when one wish to move, rotate, copy or delete a component.

A component is selected by a click with the left mouse button, the right mouse button gives access to functions such as delete, group, ungroup or copy/paste. If one wish to select many components one can do in two ways
1. Select the first component, then hold down the shift key and select the other components.
2. Hold down the left mouse button and drag the mouse until all components of interest are included in the area, then release the left mouse button.
The selected components change color (the rectangular hotspots become red). To unselect one can simply click with the left mouse button outside the select rectangle.
Once the components are selected one can do a number of operations on them.
Left mouse button operations:

- **Move**: hold down the left mouse button when the cursor is inside the select rectangle and move the cursor, release the mouse button when you are ready

Right mouse button operations:

- **Rotate**: rotate the selected component (if possible, not all components can be rotated).
- **Copy**: the selected components are copied and can be pasted anywhere on the schema (or in another project).
- **Cut**: the selected components are cut and can be pasted anywhere on the schema (or in another project).
- **Paste**: paste a component (or several components) that has been cut or copied earlier..
- **Delete**: delete the selected components.
- **Group**: join the selected components in a group, the group concept will be explained in more detail below.
- **Ungroup**: the opposite of above, for "safety reasons" only one group at a time can be ungrouped.
- **Copy**: the selected components are copied and can be pasted anywhere on the schema (or in another project).
- **Insert | Passive HP/LP filter**: insert a passive highpass and/or lowpass filter of with a filter order up to 4. The component values are computed given a nominal load and desired cutoff frequencies. In the items are grouped (see **Group items** checkbox) the group will have wizard properties, which means that one can compute new values for new desired cutoff frequencies later.



HP section: 2
fc 500.0 Hz
LP section: 4
fc 5000.0 Hz
Nom. load 8.00 ohm
□ Group items

In some cases, components may disappear when one move things to return when one releases the mouse button, this is a normal feature and is part of the strive to make component move operations as smooth as possible.

5.1.1 The group concept:

Once components are joined in a group they are moved together, in one piece, also worth notice is that the grouped component leads cannot be moved, it is however possible to move the hot spots for the component descriptions and also move the entire group.

☞ **Note that Driver, TS driver, Generator, User note and Baffle components cannot be part of a group**.

Groups are treated as components, which means that they can be grouped as well.
The groups play an important role when we start with the simulations and wish to try out several different combinations of crossover values once the crossover topology is fixed.
Once a group is created a rectangle is visible on the schema that displays the size of the group, the solder points of the grouped components also become smaller.

The group component is, if it is used correctly, a wonderful companion in the design and simulation work. The simplest application is to gather components into a single entity so that they are easier to move around.
The most interesting feature with a group is that one can use the same filter topology but with different components. Imagine the situation with a low pass filter with a cutoff frequency of 1000Hz. We can use the group to store the component values for a 1000Hz filter, then we can modify the component values so that we get a 1500Hz filter. With the two settings stored we can very easily switch between the settings. This is a very handy feature especially when we use the emulator.

5.1.1.1 Wizards

For well-known groups wizards are implemented, one such example is the 2nd order active low pass filter that is available in the template section. Another example is a bass reflex box where one can immediately see the Helmholtz resonance. The wizards are sort of semi-intelligent “gizmos” that knows what the involved circuit is about and can take proper action if one wishes to alter e.g. the cutoff frequency. Many of the groups in the Template project have wizards. Note that if the group is ungrouped the wizard properties are gone.

5.1.2 The templates

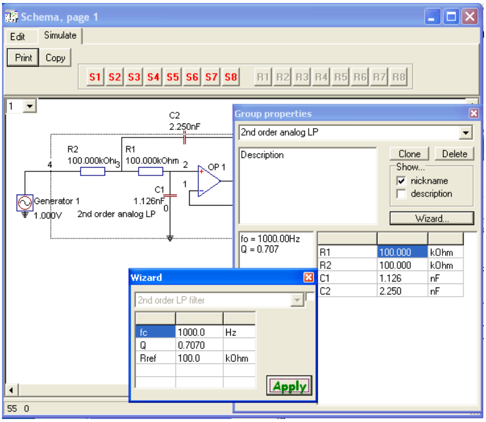
The templates are mentioned briefly in previous sections. The templates are loaded automatically when LspCAD is started and is accessed as a separate project (filename:*LspCAD_templates.lsp*) in the project dropdown listbox in the main window The template section is meaningful only in the edit mode.

☞ **Do not place your own defined templates in the template project as the templates may be updated with software updates, user defined templates can be placed in the *User defined templates* project**

The templates are divided into a few pages

5.1.2.1 Template page 1 (active templates)

This template page contains a number of good to have active filter blocks. One of them is the 2nd order analog LP filter. If one copies this template into a new project and switch to simulate mode and later right-click in the group frame (no on a specific component) a dialog will appear. In this dialog a click on the **Wizard** button makes it possible to compute appropriate values based on desired fo and Q.



Group properties: 2nd order analog LP

Description

Show...
☑ nickname
☐ description

fo = 1000.00Hz
Q = 0.707

R1	100.000	kOhm
R2	100.000	kOhm
C1	1.126	nF
C2	2.250	nF

Wizard: 2nd order LP filter

fc	1000.0	Hz
Q	0.7070	
Rref	100.0	kOhm

5.1.2.2 Template page 2 (passive templates)

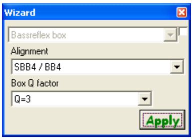
This template page contains passive templates such as allpass filters and resonance circuits. These components also contain the wizard properties described above.

5.1.2.3 Template page 3 (Loudspeaker boxes)

This page contains templates for the most common loudspeaker types using dynamic loudspeaker units. Note that both the group with the loudspeaker box components as well as loudspeaker unit component must be picked and copied to a new project. Once the components are placed in a project and simulate mode is entered. When one right clicks inside the component borders a dialog shows up that looks different depending on mode.

5.1.2.3.1 Simple mode

In the simple mode the dialog looks like in the previous examples, also in the schema one can see the different parts that constitute a box (box, port, load and radiation component). If one clicks on the **Wizard** button the programs will compute the appropriate box properties for a given, desired alignment and loudspeaker unit (feature not available for Bandpass box II).


Wizard

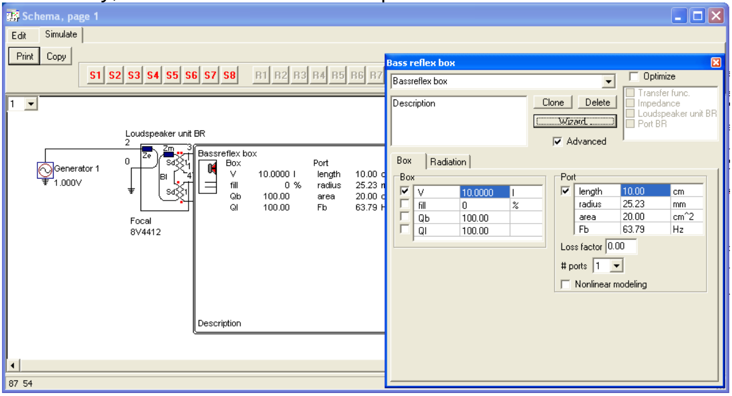
Bassreflex box

Alignment: SBB4 / BB4

Box Q factor: Q=3

5.1.2.3.2 Advanced mode (recommended)

Technically speaking the advanced mode is not more advanced than the simple mode, the difference lies in that in the advanced mode all the box properties are collected in one dialog which makes it very simple to alter values and instantly see the effects of the changes. Additionally, in the schema the different parts that constitute a box are hidden.



Bassreflex box

Box			Port	
V	10.0000 l		length	10.00 c
fill	0 %		radius	25.23 r
Qb	100.00		area	20.00 c
Ql	100.00		Fb	63.79 H

Bass reflex box

Bassreflex box

☑ Advanced

☐ Optimize

☐ Transfer func.
☐ Impedance
☐ Loudspeaker unit BR
☐ Port BR

Box

☑	V	10.0000	l
☐	fill	0	%
☐	Qb	100.00	
☐	Ql	100.00	

Port

☑	length	10.00	cm
	radius	25.23	mm
	area	20.00	cm^2
	Fb	63.79	Hz

Loss factor 0.00

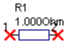
ports 1

☐ Nonlinear modeling

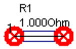
In the **Box** tab components such as boxes, ports and passive radiators are collected. In the Radiation tab one can access the properties for the radiation of the parts that face free air. For the bandpass boxes **Rear box** and **Rear port** denote the box and port that is located behind the loudspeaker unit while **Front box** and **Front port** denote the box and port that is located in front of the loudspeaker unit.

5.2 Simulate mode

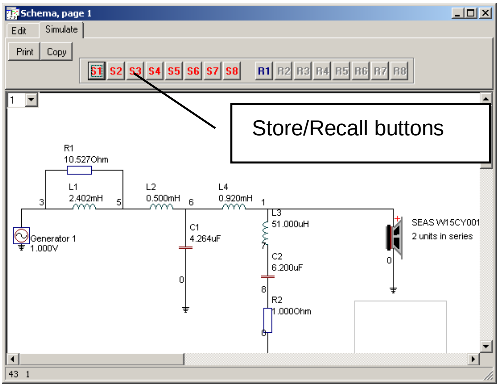
In simulate mode the component hotspots are hidden, also the group boundaries become invisible (it is possible to force them to become visible). Hopefully all components form a valid circuit schema. If you see a red cross at an endpoint of a component this means that the component is not connected anywhere, most likely this is an error that needs to be corrected.



Another error that might occur is that e.g. a resistor is short circuited, while this is often OK in a real-life situation it won't work in this case, this error is marked up with a large red cross + a circle.



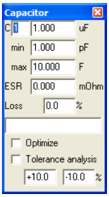
The figure above one can see that node numbers are attached close to each solder point. In most cases these node numbers are not needed and can be hidden (see ch. 7 settings).



In general component settings are accessible via either the left or the right mouse button, note that one need to click on the component to get access to the settings.

Left mouse button operations:
The left mouse button gives quick access to the most important property of the component. For a resistor this is the resistor value while for a buffer amplifier the gain value is the most important.
Normally a small edit box will appear where one may alter the value; the edit box will disappear when one clicks on the close button in upper right corner or click on another component.
If one clicks on a driver unit, the driver unit settings dialog will pop up.
All edit boxes and dialogs will be described in more detail later in this manual.
If one clicks with the mouse outside a component nothing happens unless the click point is inside the boundaries of a group, then a small list box will popup where one can select the network set.

Right mouse button operations:
With a right mouse button click on a component one gets access to more advanced component settings, for a capacitor one can set e.g. the loss factor and the ESR. Advanced group settings include the options to create and delete network sets. Also, it is possible to switch on optimization of individual components here.



C 1 1.000 uF
min 1.000 pF
max 10.000 F
ESR 0.000 mOhm
Loss 0.0 %

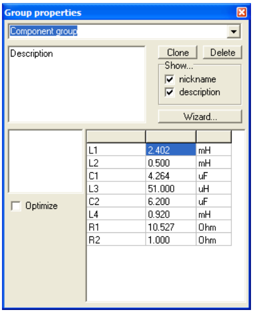
☐ Optimize
☐ Tolerance analysis
+10.0 -10.0 %

5.2.1 Store/Recall buttons

With the store/recall buttons it is possible to store different stages in a development project, this is an alternative to the group concept (ch 5.1.1).

5.2.2 Changing group properties

In one right click inside a group in simulate mode a group property dialog appears. In this dialog it is possible to clone and delete component sets in the group and also to enter a nickname and a description for the group.
An additional feature is that it is possible to control the optimization of all the grouped components, in short this means that even though a component within a group is enabled to be optimized, it will still not be optimized if the **Optimize** checkbox in the group is not checked.
This is a very important feature that makes it possible to turn on and off the optimization of entire filter sections.

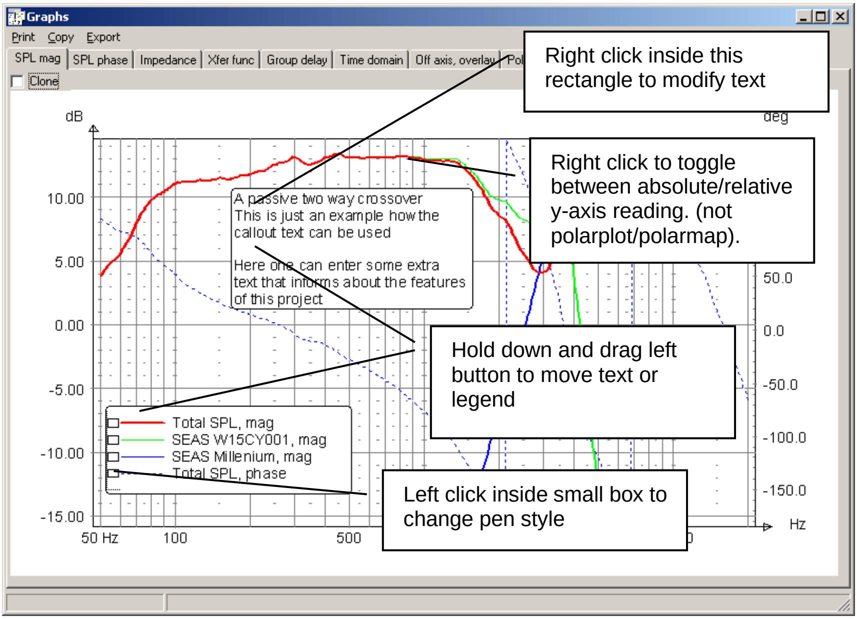


L1	2.402	mH
L2	0.500	mH
C1	4.264	uF
L3	51.000	uH
C2	6.200	uF
L4	0.920	mH
R1	10.527	Ohm
R2	1.000	Ohm

6 Graphs

The graphs in LspCAD are collected in a window with a bunch of tabs. Each tab hides its own graph. A graph may e.g. be a frequency response. To make reading easier a legend box is also included. The callout text can be used to add extra text in the graph to describe some extra features. By default, the callout box is visible as a gray box in the upper right corner. This box will change to fit the text entered. In is possible from the settings menu to hide the legend and the callout text completely if this is desired.

The graphs can be cloned (or undocked) to make it possible to view several graphs at the same time, this may be handy if one e.g. wishes to see both the sound pressure level and the impedance at the same time, note however that having many graphs visible at the same time will slow down computations a bit.
All graphs are updated in real time with the exception of the polar maps, as the latter are very CPU intensive. It is also recommended not to have the polar maps visible unless one need to as the redrawing of these is also quite heavy.



A passive two way crossover
This is just an example how the callout text can be used

Here one can enter some extra text that informs about the features of this project

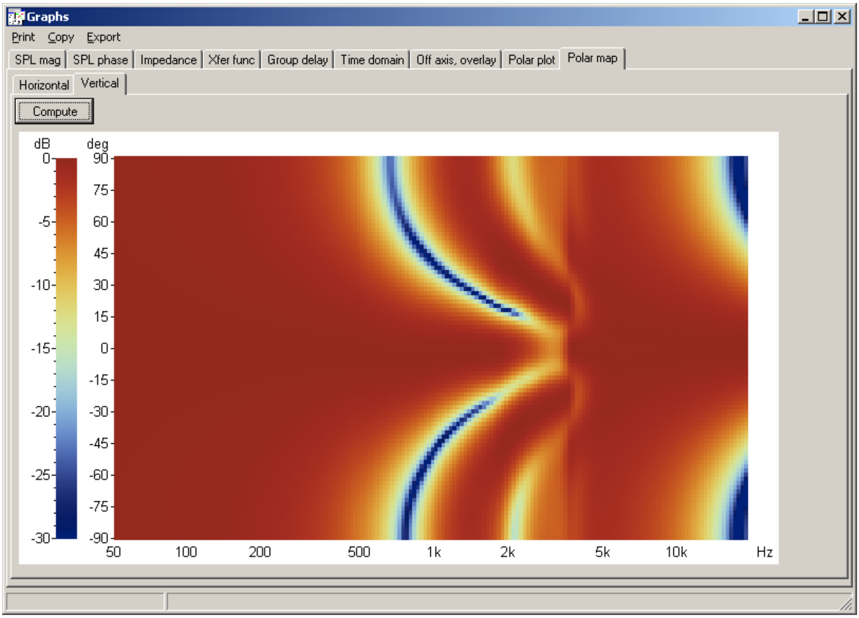
Right click inside this rectangle to modify text

Right click to toggle between absolute/relative y-axis reading. (not polarplot/polarmap).

Hold down and drag left button to move text or legend

Left click inside small box to change pen style

The figure below shows the polar map. Note that one need to click on the Compute button to make a computation.



6.1 Menu picks

6.1.1 Print

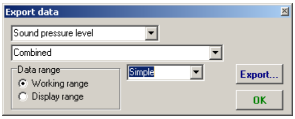
The graphs can be printed on any printer, click on Print in the menu to print out the graph that is visible for the moment. Note that the print scale may be changed in the Settings dialog (see ch. 7).

6.1.2 Copy

The visible graph is copied to the Windows clipboard

6.1.3 Export

The graph data can be exported, one can select to export either the displayed range or the working range (see Settings). The simple export format is just an export format with data arranged in columns while the Matlab format is used by eg. The Klippel system.



Export data

Sound pressure level

Combined

Data range

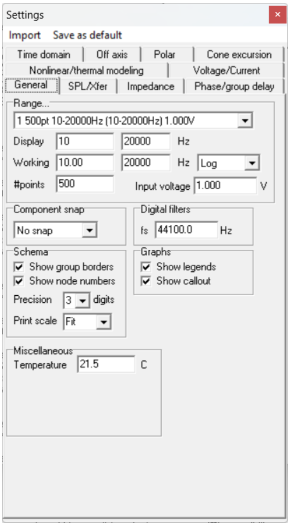
● Working range

○ Display range

Simple

7 Settings

All settings in LspCAD 6 are collected in a Settings dialog that is accessible from the main menu. This dialog contains a number of tabs and also a menu. In the menu one can select to ***Import*** settings from another LspCAD 6 project and also to ***Save as default*** the present settings.



Range...

1 500pt 10-20000Hz (10-20000Hz) 1.000V

Display 10 20000 Hz

Working 10.00 20000 Hz Log

#points 500 Input voltage 1.000 V

Component snap

No snap

Digital filters

fs 44100.0 Hz

Schema

☑ Show group borders

☑ Show node numbers

Precision 3 digits

Print scale Fit

Graphs

☑ Show legends

☑ Show callout

Miscellaneous

Temperature 21.5 C

General tab:

Range: Settings for display and working range and input voltage to the simulations and the number of simulation points. There are 8 presets that one can modify. The frequency steps can be

- Log : Logarithmic
- Lin: Linear
- Log-Lin (The 1st 3/4s of the steps are log spacing, the rest are lin. spacing.

Component snap: Makes it possible to snap the passive component values to E24 or E12 standard component values.

Schema: Affects the properties of the schema. **Show group borders** checked will make it possible to see the borders of component groups. Have **Show node** numbers checked if you want to see the node numbers in the schema.

The print scale can be set to a number of different values, if set to “Fit” the scale is adjusted to fit the entire paper.

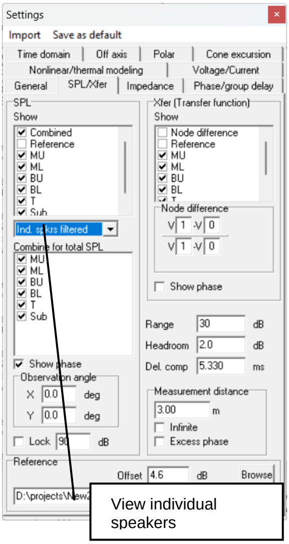
Digital filters: The sample rate of digital filters, this number is used if indicated by the settings for the digital filters.

Graphs: Uncheck **Show legends** if they are not needed in the graphs. Similar applies to callouts.

Miscellaneous: Temperature sets the ambient temperature for the simulation of loudspeaker boxes.

SPL/Xfer:

SPL:: Controls the appearance of the SPL and transfer function graphs. In the **Show** box one can select which simulated responses should be seen in the graph.
Two entities, Combined and Reference are always present in this box. When Combined is checked the combined SPL is displayed in the graph, this combined SPL is given by the settings in the **Combine for total SPL** box. If **Reference** is checked an imported reference SPL curve is displayed.
Individual spkrs. Makes it possible to view the SPL of individual speakers in case there are several in a Driver unit.
Show phase makes it possible to see the phase of the combined SPL.
In a separate subgroup one can specify the **Observation angle** if other angles that straight on axis is desirable. A positive angle in the X direction means "mic to the right of the speaker" while a positive Y angle means "mic above speaker" (see ch 3.3).



SPL
Show
☑ Combined
□ Reference
☑ MU
☑ ML
☑ BU
☑ BL
☑ T
☑ Sub
Ind. spkrs filtered
Combine for total SPL
☑ MU
☑ ML
☑ BU
☑ BL
☑ T
☑ Sub
☑ Show phase
Observation angle
X 0.0 deg
Y 0.0 deg
□ Lock 90 dB

Xfer (Transfer function)
Show
□ Node difference
□ Reference
☑ MU
☑ ML
☑ BU
☑ BL
☑ T
Node difference
V 1 -V 0
V 1 -V 0
□ Show phase
Range 30 dB
Headroom 2.0 dB
Del. comp 5.330 ms
Measurement distance
3.00 m
□ Infinite
□ Excess phase

Reference
Offset 4.6 dB
D:\projects\New2

Xfer (Transfer function): In the **Show box** one selects which simulated transfer functions should be visible in the Xfer graph. The Node difference feature makes it possible to probe the node difference voltage and to divide this figure by the node difference voltage of a reference.
The derived transfer function is given by the ratio of node voltage differences, in the example displayed above the voltage at node 5 is divided by the voltage at node 3.

Range: The display range of the SPL/Xfer graphs

Headroom: Makes it possible to add a few dB headroom to the graphs for better readability.

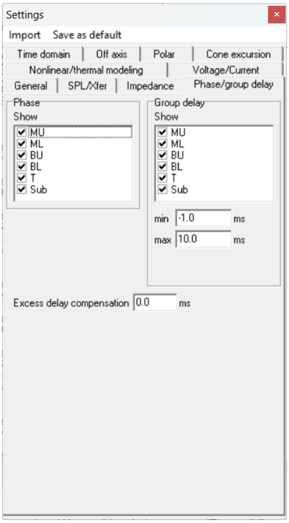
Del. comp: Makes it possible to subtract delay caused by for instance phase linear FIR filters.

Measurement distance: Simulated measurement distance (if not **Infinite**), also one can select to simulate the phase with the time of flight to the "measurement mic" added (**Excess phase**).

Reference: The reference is imported as an ASCII data file with values arranged column wise. The first (leftmost) column is the frequency points while the second is the dB values. The reference can be displayed in either the SPL or the Xfer graph.

Phase/groupdelay:

Controls the appearance of the phase and group delay graphs. For the group delay graph one can also set the min and max displayed value.
Excess delay compensation is used to remove delay caused by phase linear FIR filters



Phase: ☑ MU ☑ ML ☑ BU ☑ BL ☑ T ☑ Sub

Group delay: ☑ MU ☑ ML ☑ BU ☑ BL ☑ T ☑ Sub

min -1.0 ms

max 10.0 ms

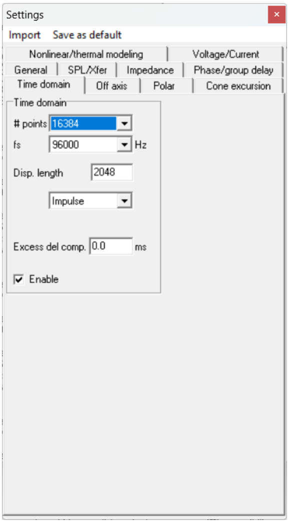
Excess delay compensation 0.0 ms

Time domain:

Set the properties of the time domain analysis.
#points: The number of analysis points
fs: The sample rate
Disp. Length: The displayed number of samples

It is possible to display the time domain response in four ways:
Impulse
Step
Energy-time
Square, in this case one can also set the frequency of the square wave. This frequency is however rounded according to the resolution of the inverse FFT operation that is used for the time domain response calculation.

Excess del comp.: Allows to remove delay caused by e.g. phase linear FIR filters



points 16384

fs 96000 Hz

Disp. length 2048

Impulse

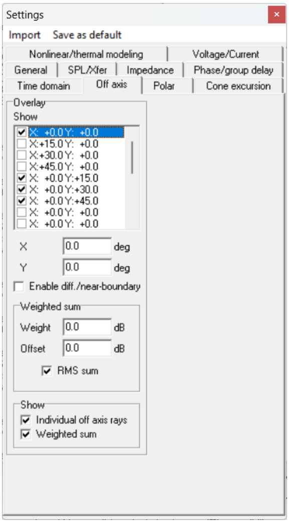
Excess del comp. 0.0 ms

☑ Enable

Off axis (Overlay):

Set the properties of the overlaid off axis simulations. Up to 20 off axis rays can be displayed. To set the properties for an off-axis entry one must first select a row in the list, after this one can set the **X** and **Y** directions.

Also is possible to compute a **Power response** based on a weighted sum of the off-axis rays, the power response is scaled up and down with the **Offset** setting.



Overlay
Show
☑ X: +0.0 Y: +0.0
□ X:+15.0 Y: +0.0
□ X:+30.0 Y: +0.0
□ X:+45.0 Y: +0.0
☑ X: +0.0 Y:+15.0
☑ X: +0.0 Y:+30.0
☑ X: +0.0 Y:+45.0
□ X: +0.0 Y: +0.0
□ X: +0.0 Y: +0.0

X 0.0 deg
Y 0.0 deg
□ Enable diff./near-boundary

Weighted sum
Weight 0.0 dB
Offset 0.0 dB
☑ RMS sum

Show
☑ Individual off axis rays
☑ Weighted sum

Polar:

Polar plot

Settings for the polar plot. Up to 10 different frequencies can be simulated either in the vertical or horizontal direction. If **Normalize** is checked the levels for each frequency is normalized with regard to the strongest value. Polar plot over the entire space around the loudspeaker is made possible if **+/-180** is checked.

Polar map

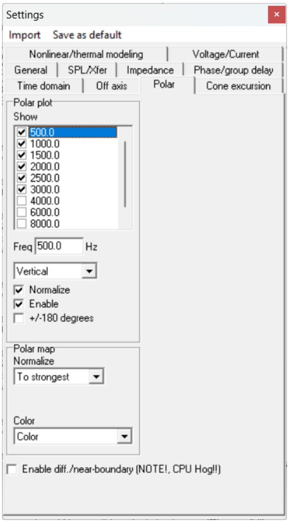
A polar map can be computed for both the horizontal or vertical direction. The polar map can be normalized by three means.

1 **No normalization**

2 **To 0 deg** i.e. the value at 0-degree angle. In this case one can also set an extra margin as other angle might give have a higher value than the value at 0 degree.

3 **To strongest**. Normalized to strongest value for each frequency.

It is also possible to choose between color or B/W.



Polar plot
Show
☑ 500.0
☑ 1000.0
☑ 1500.0
☑ 2000.0
☑ 2500.0
☑ 3000.0
□ 4000.0
□ 6000.0
□ 8000.0

Freq 500.0 Hz
Vertical
☑ Normalize
☑ Enable
□ +/-180 degrees

Polar map
Normalize
To strongest

Color
Color

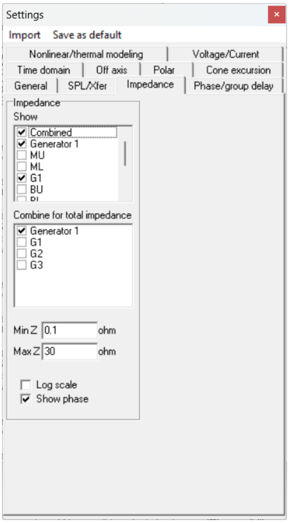
□ Enable diff./near-boundary (NOTE!, CPU Hog!!)

Enable diff./near-boundary:

Enables diffraction and near boundary near boundary reflection simulation for the polar modeling. Use this with care as it slows down computations considerably!

Impedance:

The Impedance tab follows the same principle as the SPL/Xfer tab, here is also added the possibility to specify a min and max displayed impedance value.



Impedance

Show

☑ Combined
☑ Generator 1
☐ MU
☐ ML
☑ G1
☐ BU

Combine for total impedance

☑ Generator 1
☐ G1
☐ G2
☐ G3

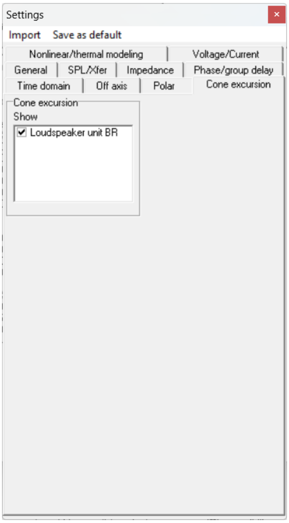
Min Z 0.1 ohm

Max Z 30 ohm

☐ Log scale
☑ Show phase

Cone excursion:

In some projects it may be desirable to simulate with several TS driver units, possibly also in different boxes. One example could be simulation of subwoofers and satellite systems in 5.1 cinema setups. In this tab it is possible to select what loudspeakers one wish to display the cone excursion for.

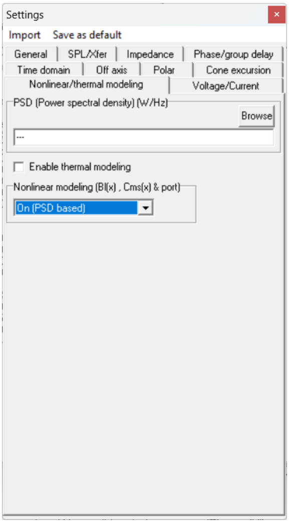


Cone excursion

Show

☑ Loudspeaker unit BR

Nonlinear/thermal modeling (also see ch 14):



PSD (Power spectral density) (W/Hz)

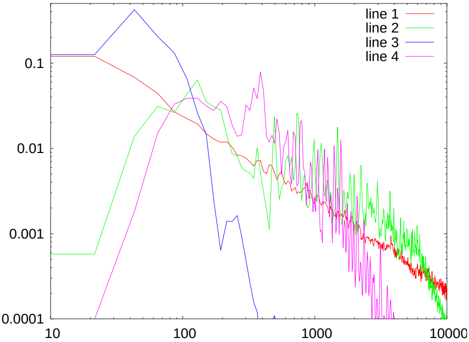
□ Enable thermal modeling

Nonlinear modeling (Bl(x) , Cms(x) & port)

On (PSD based)

This tab contains settings for modeling of voice coil heating in TS Driver and Driver components and the nonlinear properties of the force factor (Bl) and compliance (Cms) of TS Driver components. It can be wise to state that nonlinear modeling here refers to the modeling of the impact on the frequency response from non-linear properties i.e. estimation of things like THD and IMD is not handled in this software.

PSD (Power spectral density) (W/Hz): Import the PSD of some given music material, this is used for the voice coil heating simulation and also for the **Bl(x)** and **Cms(x)** simulations.
There are a few precomputed PSD's in the LspCAD lib folder that can be used with the voice coil heating simulations.
They all have the names
PSD-<title>.txt
an example is
PSD-Beethoven-no 5.txt
The PSD's are stored in such a way that in a simple project with just a voltage source and a 1 ohm resistor one would get a power dissipation exactly 1W regardless of which PSD one select. The distribution of the power for different frequencies differs though.



Various PSD's
1 : Pink noise
2 : Bruce Springsteen "Born in the USA"
3 : Krafwerk "Elektrokardiogramm 1"
3 : Beethoven "Symphony no 5" beginning.

Enable thermal modeling: Thermal modeling is a quite CPU intense part so it is possible to switch this off on a global base for the current project. If this box is unchecked thermal modeling is off regardless of the settings for the individual Driver and TS Driver components in the project.

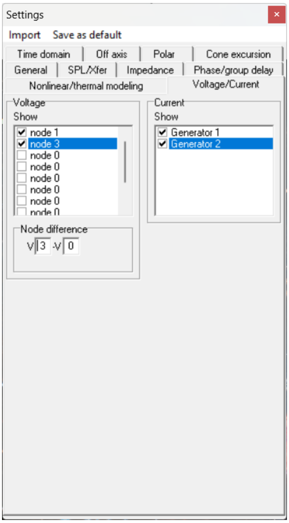
Nonlinear modeling (Bl(x) & Cms(x)): Besides the off position it is possible to simulate Bl(x) / Cms(x) in two different ways.

On (PSD based): The power spectral density of different music clips is taken into account. This is a better approximation of the impact on the frequency response due non linear effects.

On (Swept): The more classic approach. Here one can simulate with the assumption that the input signal is a swept sinewave with the same voltage for all frequencies

Voltage/Current :

This tab contains settings for Voltage and Current measurement. The Voltage is measured as the difference between nodes while current can be measured at the output of components such as voltage sources, OP amplifiers and buffer components.



Voltage

Show

☑ node 1
☑ node 3
□ node 0
□ node 0
□ node 0
□ node 0
□ node 0
□ node 0

Node difference

V 3 -V 0

Current

Show

☑ Generator 1
☑ Generator 2

8 Components

The following sections give information about the components (or building blocks) that make it possible to model crossovers and loudspeaker boxes in LspCAD. Additionally, the configuration dialogs that appear when one right-click on the component are described.

8.1 *Passive analog components*

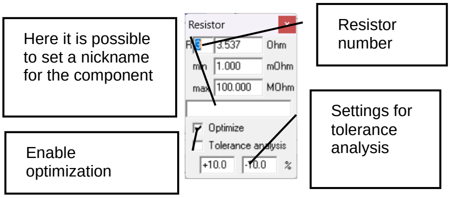
8.1.1 Wire component

The wire component is used to connect two end points that are not possible to connect by other means.

☞ **It is not possible to short-circuit other components with a wire as the results will then be unpredictable**

8.1.2 Resistor component

The resistor component does not have properties than the resistor value. If one right-clicks with the mouse on the resistor configuration dialog will popup. In this one can set, besides the value also the of the component.



Resistor

R 3 3.537 Ohm

min 1.000 mOhm

max 100.000 MOhm

☑ Optimize

□ Tolerance analysis

+10.0 -10.0 %

Here it is possible to set a nickname for the component

Resistor number

Settings for tolerance analysis

Enable optimization

The component numbering is automatic but can be modified, the program will ensure that no two resistors get the same number.

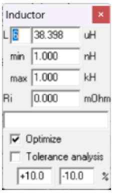
For optimization purposes one can set the min and max allowed values during the optimization process.

Tolerance analysis: It is possible to specify the upper and lower tolerance limits for the component. These values are then used in the dedicated tolerance analysis function.

☞ **If thermal modeling is enabled, one can see the power dissipated in the resistors in the component list**

8.1.3 Inductor component

Besides the inductor value an additional series resistance can be specified, similar to the resistor case a popup dialog will appear where one can set the options.



Inductor

L 6 38.398 uH

min 1.000 nH

max 1.000 kH

Ri 0.000 mOhm

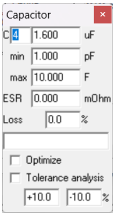
☑ Optimize

☐ Tolerance analysis

+10.0 -10.0 %

8.1.4 Capacitor component

The advanced settings for the capacitor are like the resistor settings with the addition of the possibility to set a loss factor and a series resistance.



Capacitor

C 4 1.600 uF

min 1.000 pF

max 10.000 F

ESR 0.000 mOhm

Loss 0.0 %

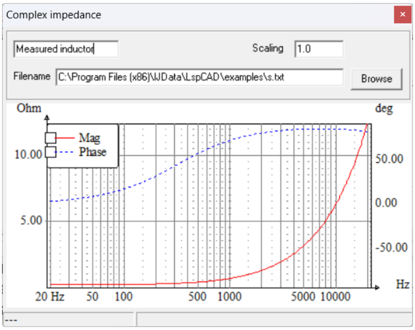
☐ Optimize

☐ Tolerance analysis

+10.0 -10.0 %

8.1.5 Complex impedance component

In certain cases, a model of e.g. an inductance does not reflect reality very well. The reason is that the simple $z(\omega) = ri + \omega L$ inductor model does not model stray capacitances and the skin effect that makes the simple model less suitable. For cases such as these it is possible to measure the impedance of the component and import the measurement using the Complex impedance component. The complex impedance component can of course be used for many different purposes.



Complex impedance

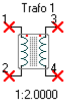
Measured inductor

Scaling 1.0

Filename C:\Program Files (x86)\IJData\LspCAD\examples\s.txt

8.1.6 Transformer component

The transformer component is an ideal transformer. The turns ratio can be modified. To make a less ideal transformer one need to add inductors, resistors and capacitors outside the transformer.



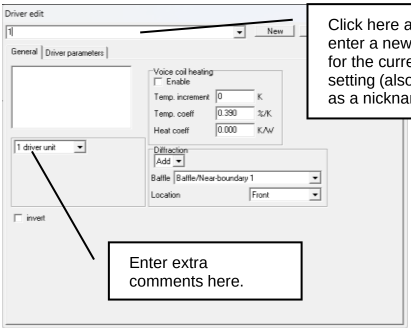
8.1.7 Driver unit (measured data)

With the driver unit component one can determine the properties of a driver with imported data. A driver unit can be either 1 or 2 actual loudspeaker units and the position of each loudspeaker can be set with respect to a reference point in an imagined coordinate system (see ch 3.3). The driver unit dialog consists of two main tabs, the **General** tab and the **Driver parameters** tab. It also contains functionality for the novel driver settings feature in LspCAD 6.

8.1.7.1 Driver settings

The driver components actually consist of the possibility to handle several different drivers' data and settings at the same time. The only properties that are common for all driver settings are the Voice coil heating and baffle diffraction settings, all the other properties belong to unique settings.

In the driver unit configuration one can create and delete driver settings with the **New**, **Clone** and **Delete** buttons.
New : Creates a new bare bone setting with default properties.
Clone: Make an exact copy of the current driver setting.
Delete: Delete the current setting, note that at least one setting must be available.



1

Voice coil heating
□ Enable
Temp. increment 0 K
Temp. coeff 0.390 %/K
Heat coeff 0.000 K/W

1 driver unit

Diffraction
Add
Baffle Baffle/Near-boundary 1
Location Front

□ invert

Click here and enter a new name for the current setting (also serves as a nickname)

Enter extra comments here.

In the topmost list box one can select between different driver settings and change the name of the current setting.

One application for this is if one wishes to try out driver unit A with a given frequency response and then compare it with a cheaper driver unit B with another frequency response but wishes to keep the crossover that feeds the driver unit the same, maybe one wish to offer the more expensive drivers for one market segment and the cheaper drivers for another market segment. In this case one simply creates two driver settings and set all the properties (SPL data etc) for each driver setting individually.
Another application is if one wishes to try out two different positions of the driver unit. In this case it is better to clone an existing driver setting and change the appropriate properties.

8.1.7.2 General tab

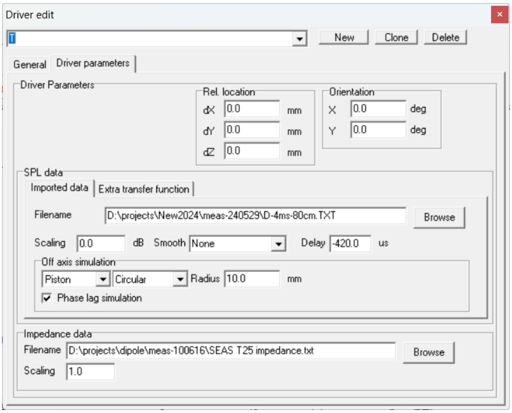
In the general tab one can select between one and two actual driver units and also the option to **Invert** the polarity. If two driver units are selected one has the option to connect them either in series or in parallel.

Voice coil heating and the impact of voice coil heating due to increased input power can be simulated.
It is possible to simulate either in manual mode (Heat coeff = 0) or in automatic mode (Heat coeff > 0).
In manual mode one set the temperature increment in the voice coil manually.
In automatic mode the input voltage and corresponding calculated power dissipated through the voice coil determines the heating of the voice coil.
The temperature coefficient depends on the voice coil wire material (0.39%/K for copper).
The input voltage is determined by the setting in the **Settings** dialog and by the transfer function between voltage source and driver unit.

☞ **Note that this function slows down computations considerably, it is therefore recommended that you turn off this function if it is not needed.**

Diffraction analysis. It is possible to either add or subtract the impact of baffle diffraction from imported data. A baffle must be selected from the dropdown list, this assumes that a baffle exist in the schema (see ch 8.3.1 about the **Baffle component**).The diffraction analysis allows to put the driver on the front, the left, the right or the rear side of the box. See section 8.3.1 for examples.

8.1.7.3 Driver parameters tab



T

dX 0.0 mm
dY 0.0 mm
dZ 0.0 mm

X 0.0 deg
Y 0.0 deg

Filename D:\projects\New2024\meas-240529\D-4ms-80cm.TXT

Scaling 0.0 dB Smooth None Delay -420.0 us

Piston Circular Radius 10.0 mm

☑ Phase lag simulation

Filename D:\projects\dipole\meas-100616\SEAS T25 impedance.txt

Scaling 1.0

I n the driver parameters tab one can set parameters such as relative location, orientation, imported data etc.
For the case of two actual driver units one can set the properties for each driver unit individually. This tab contains a number of main groups.

Rel location.
The relative location of each individual driver is set in these fields.

Orientation
The X and Y axis orientation is set here, see ch. 3 for a description of the orientation angles.

SPL data
In the case of two driver units one has the option to check an “As drv 1” , this means that the same properties as for driver 1 is within this group.
The **SPL data** field contains an **Imported data** tab and an **Extra transfer function** tab.
Imported data:

This tab contains all the functionality needed to import measured data and modify it in various ways for the use in the simulations.
Scaling: Set the scaling for the imported data
Smooth: Select between different smoothing of imported data, might be hand of the measured data contains a lot of ripples, which may be the case with e.g. horns.
Delay: An additional delay might be applied
Off axis simulation can be done in three ways:

Piston: Simple flat piston model, fairly good for direct radiation loudspeakers. This piston can also be set to either circular or rectangular
Imp. txt: A more detailed effort, needed for e.g. simulation of CD horns. In this case one import a special file that describes the off axis (horizontal/vertical) response at different frequencies. LspCAD will interpolate between the data points, so it is not needed to specify all angles/frequencies, the accuracy is of course proportional to the number of angles /frequencies specified.
An off axis data file looks like:

```
F:<frequency>
H: <angle:dB> <angle:dB> <angle:dB> <angle:dB> …….
V: <angle:dB> <angle:dB> <angle:dB> <angle:dB> …….
```

The first row describes the frequency, the second row the set of angles and dB deviation from on axis SPL in the horizontal plane, the third row is the same for the vertical plane. An example is given in the file “an off axis model.txt” located in the examples folder.
Imp. Praxis: Data from Praxis (.pxd) files can be imported. Here it is highly recommended that the .pxd file has been stored with log spaced frequencies.

Extended format: The extended format is a special text file format that contains several measurements, the first measurement is an on axis measurement, the following measurements are off axis measurements from different off axis angles. Both vertical and horizontal off axis measurements are possible to specify in the same file. The measurements are separated by special delimiters.
The extended off axis format has the syntax:

[data]
## EXTENDED DATA ## OFF AXIS <X,Y>
[data]
## EXTENDED DATA ## OFF AXIS <X,Y>
[data]
## EXTENDED DATA ## OFF AXIS <X,Y>
[data]
...

The **[data]** element has the default syntax
<freq> <mag> <phase>
<freq> <mag> <phase>
<freq> <mag> <phase>
freq in [Hz]
mag in [dB]
phase in [deg]

The first [data] element is the on axis measurement, this means that there is no special delimiter before it.

The
## EXTENDED DATA ## OFF AXIS <X,Y>
delimiter identifies that off axis data for a given X or Y direction follows

e.g
## EXTENDED DATA ## OFF AXIS 30,0
Means off axis left 30 degrees in X direction

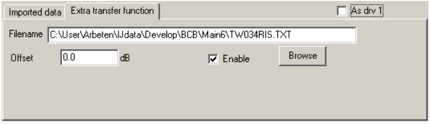
## EXTENDED DATA ## OFF AXIS 0,-45
[data]
Means off axis down 45 degrees in Y direction

Note that it is possible to enter off axis data for both positive and negative directions and also for X and Y directions in the same file. If negative axis measurements are missing, the data for the corresponding positive axis is used, also if e.g. Y axis measurements are missing, the corresponding X axis measurements are used.

One should not, however enter the data for a both X and Y axis measurement such as
## EXTENDED DATA ## OFF AXIS 30,-45
as this case is not covered in the code.

Extra transfer function:

An extra transfer function can be imported, this can be used of one wish to add the effects of e.g. an external complex room simulation to the measured data. The imported extra transfer function can be offset.



☐ As drv 1

Filename C:\User\Arbeten\IJdata\Develop\BCB\Main6\TW034RIS.TXT

Offset 0.0 dB

☑ Enable

Impedance data

In the case of two driver units one has the option to check an “As drv 1”
The imported impedance data can be scaled with the **Scaling** option.

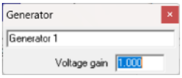
8.2 Active analog/digital components

The active components simulate all kinds of analog or digital networks.

☞ **Note that only the Voltage source, the OP amplifier and the Buffer amplifier can directly drive a passive component. In many cases one may not need a buffer amplifier but to ensure correct modeling always it is highly recommended to do so anyway.**

8.2.1 Voltage source

The voltage source is a zero output impedance generator with an output voltage that is given by the voltage setting in the settings dialog multiplied with a factor.



Generator

Generator 1

Voltage gain 1.000

8.2.2 OP amplifier component

The ideal OP amplifier has two inputs (inverting and non-inverting) and one output.

☞ **Note that the non-inverting input cannot be connected directly to ground, use a 1kOhm resistor in between the non-inverting input and GND**.

8.2.3 Transfer function module G(s) G(z)

The transfer function module gives the option to specify the transfer functions for a number of well-known active analog and digital filters such as HP/LP- FIR-filters, notch filters etc.

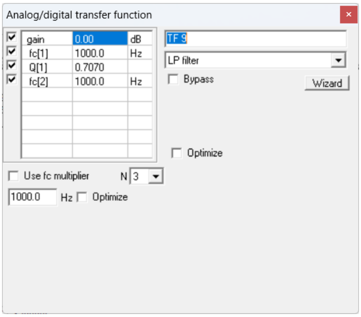
☞ **This module can not directly drive a passive component**

The transfer function module can model a great number of different things.

8.2.3.1 LP/HP filters.

Up to 8th order filters can be modeled.
Important is that one can select which of the filter parameters (cutoff, Q, gain…) should be optimized (see checkboxes to the left of the parameter list)..

The neat thing here is that one can use a common frequency multiplier that can also be optimized
The figure to the left shows the case where the fc multiplier is used. In this case the ratio between the different cutoff frequencies is kept constant and multiplied by the fc multiplier value. This is handy if one wishes to model well known textbook filter alignments such as the Chebyshev alignment.

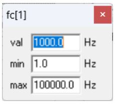

Analog/digital transfer function

☑	gain	0.00	dB
☑	fc[1]	1000.0	Hz
☑	Q[1]	0.7070	
☑	fc[2]	1000.0	Hz

TF 9
LP filter
□ Bypass
Wizard
□ Optimize
□ Use fc multiplier N 3
1000.0 Hz □ Optimize

The parameters in the parameter list can be modified in a number of ways.

1. Left-click on the parameter that you wish to modify, then click on the arrow left-right keys on the keyboard.
2. Left-double-click on a parameter, then enter a new value or use the arrow up-down to modify, to exit, click somewhere else.
3. First left-click then right-click on parameter. You now have the option to set the value and also upper and lower limits for the optimizer.


fc[1]

val	1000.0	Hz
min	1.0	Hz
max	100000.0	Hz

The Wizard button gives access to textbook based filter calculations for Butterworth/Chebyshev and Linkwitz filters up to 8th order.

8.2.3.2 Parametric Peak/Notch filter.

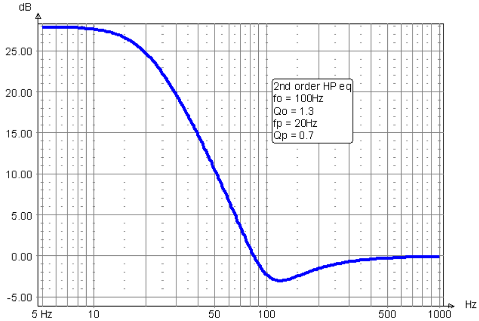
A one band parametric EQ with settings for gain, Q and fo

8.2.3.3 1st order treble/bass boost.

A first order treble bass/boost EQ

8.2.3.4 2nd order HP/LP equalizer,

Also know as Linkwitz transform, a popular means to extend the frequency range of closed loudspeaker boxes downwards.



8.2.3.5 1st order allpass

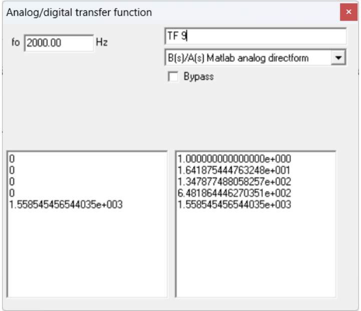
An allpass filter, the phase lag is 90 deg at the frequency given by property ‘f’.

8.2.3.6 H(s) = B(s)/A(s) Matlab analog direct form.

The direct form coefficients from Matlab™ (or Octave) are copied to the two input areas (B(s) to the left and A(s) to the right).

The following Matlab sequence gives the coefficients for a 4th order analog Butterworth filter with cutoff frequency 1Hz

```
>> format long e
>> [b,a] = butter(4,2*pi,'s'); % 2*pi =
6.28 rad/s = 1Hz
>> b'
ans =
                            0
                            0
                            0
                            0
    1.558545456544035e+003
>> a'
ans =
    1.000000000000000e+000
    1.641875444763248e+001
    1.347877488058257e+002
    6.481864446270351e+002
    1.558545456544035e+003
```


Analog/digital transfer function

fo 2000.00 Hz

TF 9

B(s)/A(s) Matlab analog directform

□ Bypass

```
0
0
0
0
1.558545456544035e+003
```

```
1.000000000000000e+000
1.641875444763248e+001
1.347877488058257e+002
6.481864446270351e+002
1.558545456544035e+003
```

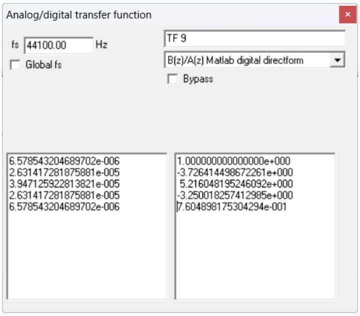
These coefficients are copied and pasted into LspCAD 6, after some editing (remove trailing blanks) it looks like the figure above.

The filter has a cutoff frequency of 1Hz. This is scaled with the **fo** property, if one set fo to 1000Hz it the cutoff will be moved to 1000Hz.

8.2.3.7 H(z) = B(z)/A/z) Matlab digital direct form.

Same as above but digital.
The following Matlab™ sequence gives the coefficients for a 4th order digital Butterworth filter with cutoff frequency 800Hz, fs = 48000Hz

```
>> format long e
>> [b,a] = butter(4,800/(48000/2));
>> b'
ans =
    6.578543204689702e-006
    2.631417281875881e-005
    3.947125922813821e-005
    2.631417281875881e-005
    6.578543204689702e-006
>> a'
ans =
    1.000000000000000e+000
   -3.726414498672261e+000
    5.216048195246092e+000
   -3.250018257412985e+000
    7.604898175304294e-001
```



Analog/digital transfer function

fs 44100.00 Hz

□ Global fs

TF 9

B(z)/A(z) Matlab digital directform

□ Bypass

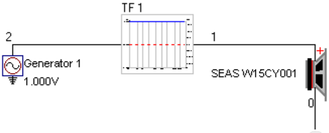
```
6.578543204689702e-006
2.631417281875881e-005
3.947125922813821e-005
2.631417281875881e-005
6.578543204689702e-006
```

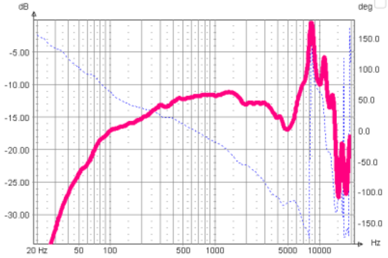
```
1.000000000000000e+000
-3.726414498672261e+000
 5.216048195246092e+000
-3.250018257412985e+000
7.604898175304294e-001
```

The coefficients are copied and pasted into LspCAD 6 and **fs** is set to 48000Hz. The result is as above. If **Global fs** checked the sample rate is determined by **fs** in the **Settings** dialog.

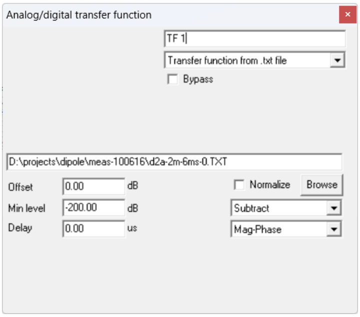
8.2.3.8 Transfer function from .txt file.

An imported transfer function is used. Here one can select to use only the magnitude part, the phase part or both. Also it is possible to either add or subtract the transfer function. One application is if one wishes to equalize the frequency response of a particular driver. This example deals with the equalization of a SEAS magnesium cone driver. Initially the frequency response looks rather peaky.





We open the configuration dialog and import the same frequency response that we use for the simulation (use the **Browse** button).



Analog/digital transfer function

TF 1

Transfer function from .txt file

□ Bypass

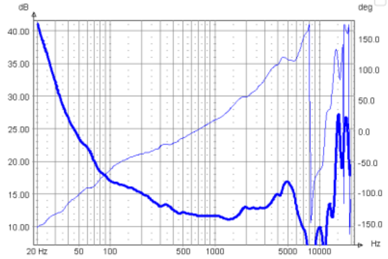
D:\projects\dipole\meas-100616\d2a-2m-6ms-0.TXT

Offset 0.00 dB □ Normalize Browse

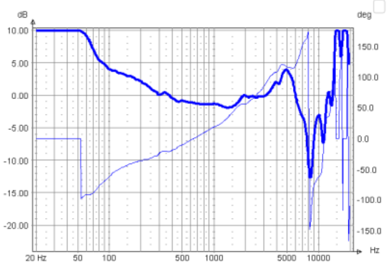
Min level -200.00 dB Subtract

Delay 0.00 us Mag-Phase

Select the Subtract option.



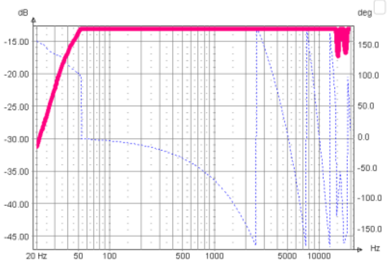
Looking at the transfer function we realize that it is not good to amplify too much. Also it would be handy if the gain in the pass band is 1. Therefore we set the **Offset** to 13dB and the **Min level** to
-10dB. This will make the transfer function nicer for a possible power amplifier.



This probably looks better; one problem however is that the delay is negative. This can be fixed by two means

1. Select the Mag only option
2. Add an extra delay

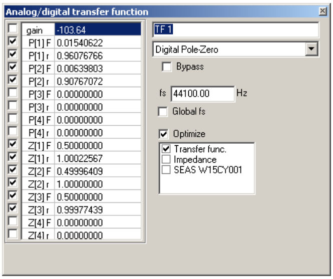
We select the 2nd option in this case and add an extra delay of 200us. This will give a transfer function that is realizable in e.g. the crossover emulator. The resulting equalized frequency response is like the figure below.



Inverting a transfer function may in principle lead to a transfer function that is non-causal i.e. it reacts to input signals before they actually reach the filter. This may cause problems if the inverted transfer function is used directly with the crossover emulator as its ability to handle non-causal transfer functions is quite limited. This however does not seem to cause any big problems if the inverting function is used in conjunction with high and low pass filters that limit the frequency range.

8.2.3.9 Digital Pole-Zero

Modeling of digital poles and zeros is made possible but means of the pole zero modeling feature.
A pole or zero is defined by an angular frequency ranging from 0 to 0.5 (0.5 is half the sample rate). And a distance from origo (radius). (e.g. P[1] F) and a radius (P[1] r).



Analog/digital transfer function

□	gain	-103.64
☑	P[1] F	0.01540622
☑	P[1] r	0.96076766
☑	P[2] F	0.00639803
☑	P[2] r	0.90767072
□	P[3] F	0.00000000
□	P[3] r	0.00000000
□	P[4] F	0.00000000
□	P[4] r	0.00000000
☑	Z[1] F	0.50000000
☑	Z[1] r	1.00022567
☑	Z[2] F	0.49996409
☑	Z[2] r	1.00000000
☑	Z[3] F	0.50000000
☑	Z[3] r	0.99977439
□	Z[4] F	0.00000000
□	Z[4] r	0.00000000

TF 1

Digital Pole-Zero

□ Bypass

fs 44100.00 Hz

□ Global fs

☑ Optimize

☑ Transfer func.
□ Impedance
□ SEAS W15CY001

In Matlab one performs the steps below to get the poles and zeros for a simple Butterworth filter.

```
>> format long e
>> [b,a] = butter(4,800/(48000/2));
>> [z,p,k] = tf2zpk(b,a);
>> angle(p)/(2*pi)
ans =
   0.01540622494703
  -0.01540622494703
   0.00639802562413
  -0.00639802562413
>> abs(p)
ans =
   0.96076766114578
   0.96076766114578
   0.90767071826573
   0.90767071826573
>> angle(z)/(2*pi)
ans =
   0.50000000000000
   0.49996408889787
  -0.49996408889787
   0.50000000000000
>> abs(z)
ans =
   1.00022566526762
   0.99999999630209
   0.99999999630209
   0.99977439303985
>> 20*log10(k)
ans =
   -1.036374053745645e+002
```

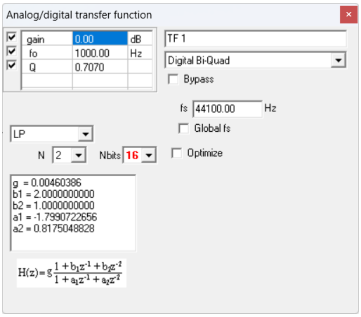
The poles and zeros are pasted in the appropriate lines (double click and enter the value). It is important to know that angles with negative sign are not entered into the list.
The locations of the poles and zeros can be optimized with the optimizer.

8.2.3.10 Digital BiQuad

The digital biquad filter is of the general form

$$H(z)=g\frac{1+b_1z^{-1}+b_2z^{-2}}{1+a_1z^{-1}+a_2z^{-2}}$$

This formulation is very popular in DSP implementations mainly because it is relatively robust to rounding errors that exist especially in a fixed-point DSP implementations. For higher order filters one can cascade several BiQuad filter links. Three types of filters can be implemented using BiQuad filters


Analog/digital transfer function

☑	gain	0.00	dB
☑	fo	1000.00	Hz
☑	Q	0.7070	

TF 1
Digital Bi-Quad
□ Bypass
fs 44100.00 Hz
□ Global fs
LP
N 2 Nbits 16
□ Optimize

```
g  = 0.00460386
b1 = 2.0000000000
b2 = 1.0000000000
a1 = -1.7990722656
a2 = 0.8175048828
```

$$H(z)=g\frac{1+b_1z^{-1}+b_2z^{-2}}{1+a_1z^{-1}+a_2z^{-2}}$$

LP/HP filters: 1st or 2nd order links

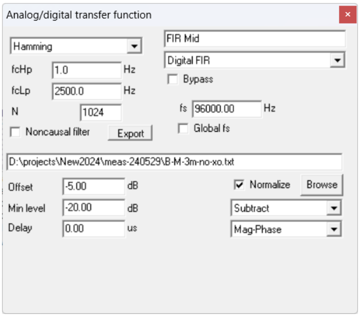
Shelving filters: These are used e.g. for baffle step compensation. It is possible to set an S parameter to control the slope in the transition region.

Parametric EQ: A parametric EQ with selectable Q value and gain/attenuation at the resonance frequency.

Parametric EQ: A BiQuad version of the 2nd order pole-shifter made famous by Siegfried Linkwitz and others.

8.2.3.11 Digital FIR

The digital FIR filter makes it possible to model phase linear HP and/or LP filters with varying filter orders between 129 and 4097 filter taps. The number of filter taps is always odd as this guarantees a filter delay that is a not a fraction of a sample. In order to increase stop-band attenuation in the filter frequency response one can select between a few different window options, for most cases **Bartlett** or **Hamming** windowing is sufficient.



Analog/digital transfer function

Hamming

fcHp 1.0 Hz

fcLp 2500.0 Hz

N 1024

□ Noncausal filter

FIR Mid

Digital FIR

□ Bypass

fs 96000.00 Hz

□ Global fs

D:\projects\New2024\meas-240529\B-M-3m-no-xo.txt

Offset -5.00 dB

Min level -20.00 dB

Delay 0.00 us

☑ Normalize

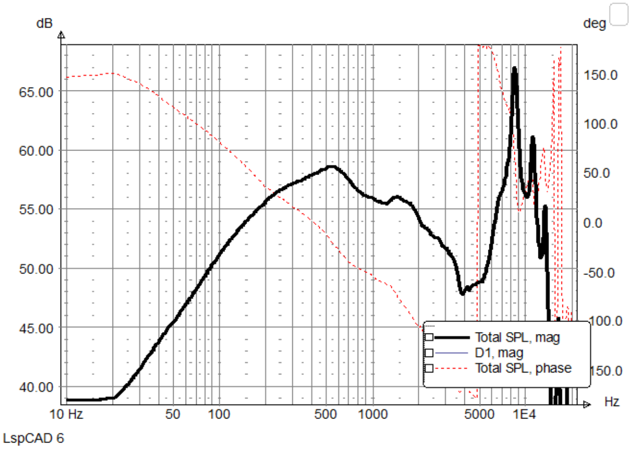
Subtract

Mag-Phase

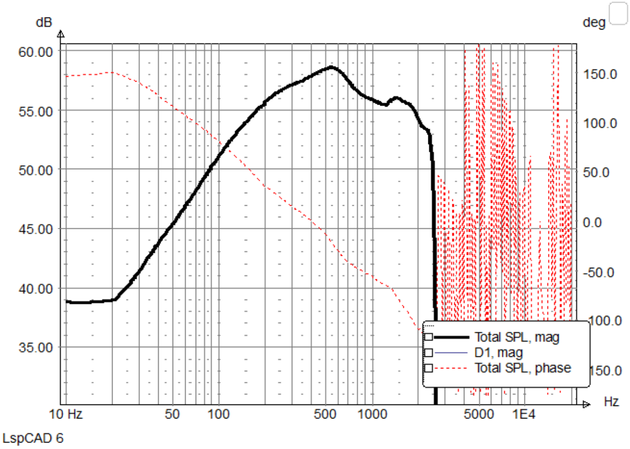
If is possible to simulate highpass (**fcHp**) and lowpass (**fcLp**) simultaneously, depending on setting one then gets either lowpass, highpass, bandpass or bandstop characteristics. The filter order (**N**) can be set to any value between 128 and 4096 filter taps (only odd filter orders). In a real life case an implementation of a phase linear filter requires a delay of half the filter order. It is however quite unpractical to have this extra delay in a modeling software as it makes it very difficult to read the phase plots. Therefore, a zero delay is used in the modeling if the **Noncausal filter** box is checked. The digital filter impulse response is exported to a text file via the **Export** button. The format is ASCII with one coefficient per line or 32 bit float for instance for the Mini DSP digital crossovers.

Besides the FIR filter function, it is possible to equalize the acoustic response of a loudspeaker for a phase and amplitude linear response. This is exemplified below and described better in the Earendel document (see references). The objective is to create a 1024 tap FIR filter with a high cutoff frequency of 2500Hz. The FIR filter should also equalize the acoustic response of a SEAS WY15C001 midrange unit in the passband.

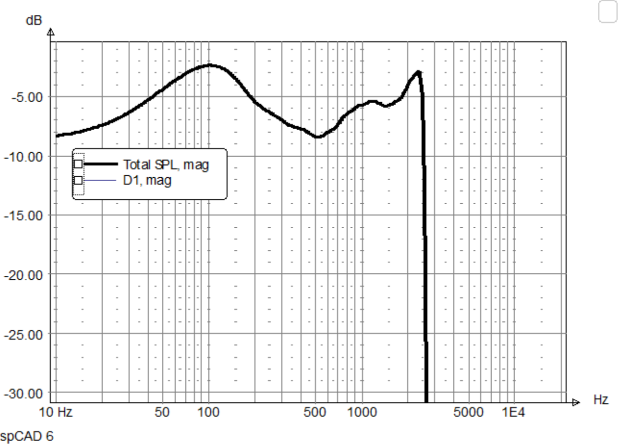
Without any filtering, the frequency response looks like the graph below.



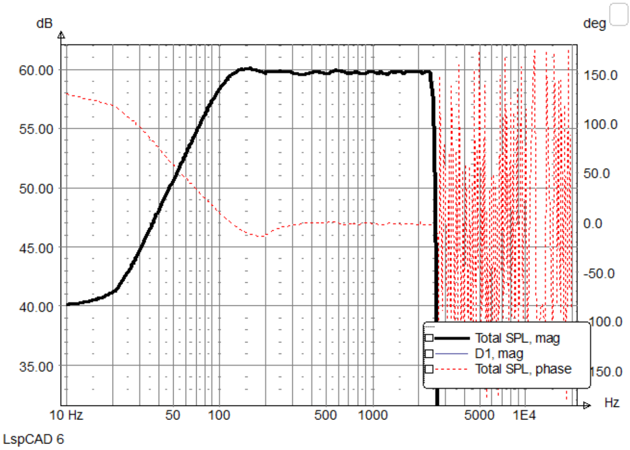
With a 2500Hz cutoff, we get the desired 2500Hz high pass filter response but the amplitude and frequency response is not flat in the pass band.


dB
deg
60.00
55.00
50.00
45.00
40.00
35.00
150.0
100.0
50.0
0.0
-50.0
-100.0
-150.0
10 Hz
50
100
500
1000
5000
1E4
Hz
□ Total SPL, mag
□ D1, mag
□ Total SPL, phase
LspCAD 6

Import the frequency response that is to be equalized, along with the 2500Hz cutoff, the transfer function becomes like below.
There are a few extra parameters that may need to be adjusted for optimal result:
Offset : Adjusts the pass band gain up or down
Min level : Depicts a limit on lowest off passband level to compensate.
Delay : Shifts the FIR filter taps forward or backward in time



With the 2500Hz FIR filter cutoff as well as the equalization for the acoustic response, the resulting response looks like the graph below.

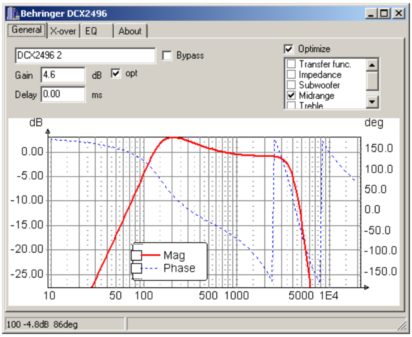


8.2.4 Behringer DCX2496 component

☞ **This module cannot directly drive a passive component**.

This component simulates the behavior of the Behringer UltraDrive Pro DXC2496 digital crossover/EQ. It has the same functionality as the device except for the dynamic equalizer. For deeper insight it is highly recommended to read the users manual for the DCX2496.

The component has one input and one output and is selected from the component tray (in the schema) in edit mode.


Behringer DCX2496

General | X-over | EQ | About

DCX2496 2 □ Bypass

Gain 4.6 dB ☑ opt

Delay 0.00 ms

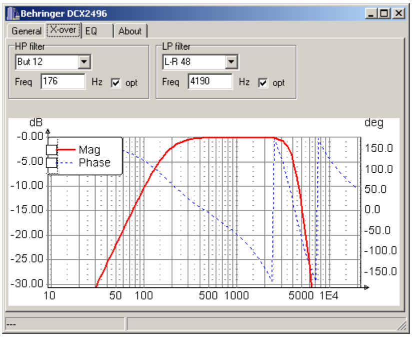
☑ Optimize

□ Transfer func.
□ Impedance
□ Subwoofer
☑ Midrange
□ Treble

100 -4.8dB 86deg

The configuration dialog that pops up when one clicks on the component block in simulate mode gives the possibility to set a few parameters.

General tab:
In this tab one can set the total **Gain** of the filter and also enable optimization of the gain. The Delay can also be modified. The graph shows the total transfer function including crossover and EQ part.


Behringer DCX2496

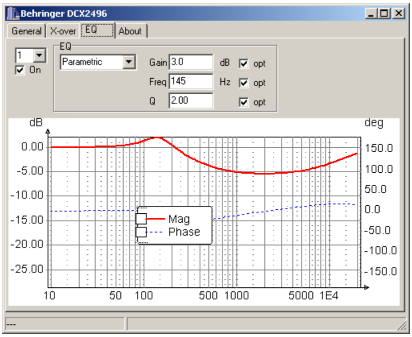
General | X-over | EQ | About

HP filter: But 12, Freq 176 Hz ☑ opt

LP filter: L-R 48, Freq 4190 Hz ☑ opt

X-over tab:
In this tab it is possible to select from the crossover slopes and alignments (high pass and low pass) that are available in the DCX2496. It is also possible to enable optimization of the cutoff frequencies.

EQ tab:
In this tab one adjusts up to 5 equalizer settings. The parametric EQ can be either parametric (peaking, bell shaped) or high/low shelving. In the shelving eq mode one can select the slope (6 or 12dB/octave). It is important to note here that each individual EQ must be enabled by checking the **on** button.


Behringer DCX2496

General | X-over | EQ | About

1 ☑ On

EQ: Parametric

Gain 3.0 dB ☑ opt

Freq 145 Hz ☑ opt

Q 2.00 ☑ opt

8.2.5 dbx Pro DriveRack™ 260 component

☞ **This module cannot directly drive a passive component**

This component simulates the behavior of the dbx Pro DriveRack 260 digital crossover/EQ. It has the same functionality as the device except for the dynamic equalizer. Unfortunately, the graphic EQ is not supported as. For deeper insight it is highly recommended to read the users manual of the DriveRack 260.

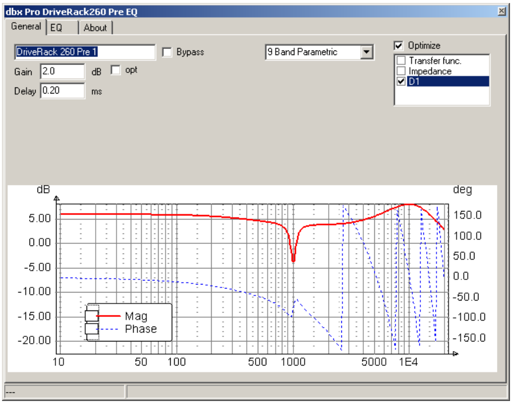
The component has one input and one output and is selected from the component tray (in the schema) in edit mode.

As the input and output parts differ this component, it is split into two parts

8.2.5.1 Pre EQ

The Pre EQ has two different types of equalizers, one 9 band parametric EQ and one 28 band graphic EQ.

The configuration dialog that pops up when one clicks on the component block in simulate mode gives the possibility to set a few parameters.



dbx Pro DriveRack260 Pre EQ

General | EQ | About

DriveRack 260 Pre 1 □ Bypass 9 Band Parametric ☑ Optimize

Gain 2.0 dB □ opt

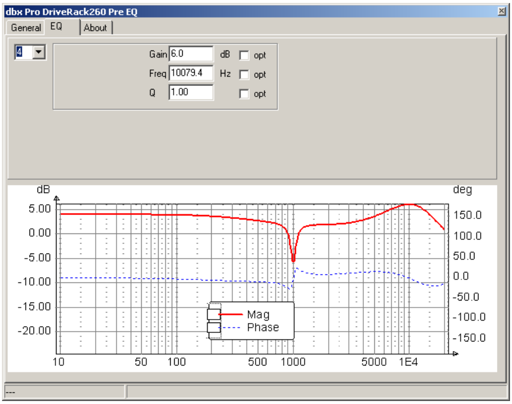
Delay 0.20 ms

□ Transfer func.
□ Impedance
☑ D1

General tab:

In this tab one can set the total Gain of the filter and also enable optimization of the gain. The Delay can also be modified. Moreover one can select between parametric and graphic EQ in this tab.

The graph shows the total transfer function including EQ part.



dbx Pro DriveRack260 Pre EQ

General | EQ | About

4

Gain 6.0 dB □ opt

Freq 10079.4 Hz □ opt

Q 1.00 □ opt

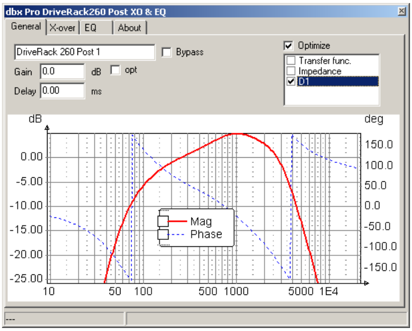
EQ tab, 9 band parametric EQ:

EQ number one is always a low shelving EQ, EQ band number 9 is always a high shelving EQ, the other bands are peaking (bell shaped). The opt boxes are used to enable optimization of the respective parameters.

8.2.5.2 Post Crossover/EQ

General tab:

In this tab one can set the total Gain of the filter and enable optimization of the gain. The Delay can also be modified. The graph shows the total transfer function including crossover and EQ part. The graph shows the total transfer function including crossover and EQ part.



dbx Pro DriveRack260 Post XO & EQ

DriveRack 260 Post 1 ☐ Bypass

Gain 0.0 dB ☐ opt

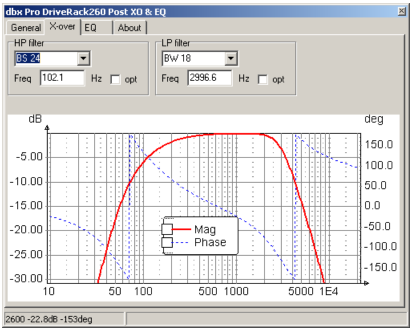
Delay 0.00 ms

☑ Optimize

☐ Transfer func.
☐ Impedance
☑ D1

X-over tab:

In this tab it is possible to select from the crossover slopes and alignments (high pass and low pass) that are available in the DriveRack 260. It is also possible to enable optimization of the cutoff frequencies.



dbx Pro DriveRack260 Post XO & EQ

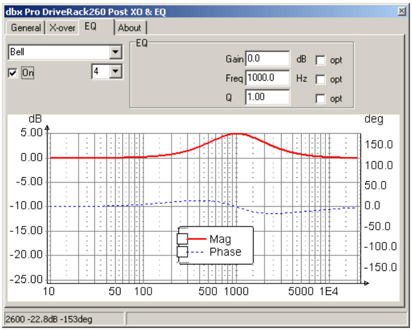
HP filter: BS 24, Freq 102.1 Hz ☐ opt

LP filter: BW 18, Freq 2996.6 Hz ☐ opt

2600 -22.8dB -153deg

EQ tab:

In this tab one adjusts up to 4 equalizer settings. The use of the 4 settings varies depending on which mode the equalizer is in (Bell, High shelf. Low shelf, High/Low shelf). The opt boxes are used to enable optimization of the respective parameters.



dbx Pro DriveRack260 Post XO & EQ

Bell

☑ On 4

EQ

Gain 0.0 dB ☐ opt

Freq 1000.0 Hz ☐ opt

Q 1.00 ☐ opt

2600 -22.8dB -153deg

8.2.6 XTA component

☞ **This module cannot directly drive a passive component**
The XTA component models the properties of the DP224 and the DP226 digital crossovers. The looks and feel of the LspCAD interface for this is quite a lot like the Behringer DCX2496.

8.2.7 Sum components

☞ **This module cannot directly drive a passive component**
The sum components exist with two inputs and three inputs. These components can be used if one wishes to implement digital filters.

8.2.8 Buffer amplifier

The buffer amplifier is simply put a gain stage with 0Ohm output impedance, with a click with the left mouse button one can set the gain. With the right mouse button one can also set the optimizer settings. A buffer amplified must be used as the last stage feeding a passive network (passive crossover, TS driver unit, driver unit…) if one uses active components. It is not necessary after an OP amplifier though.

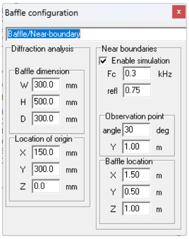
8.2.9 Delay unit

☞ **This module cannot directly drive a passive component**
A unit delay, if the delay symbol is clicked on with the left mouse button a configuration dialog pops up where one can set the delay, it is also possible to use a global setting for the delay for the case that the sample rate of digital filter is the same for most of the digital filters.

8.3 Box modeling components

8.3.1 Baffle and Near boundary component

The baffle component describes the geometry of the used baffle (for diffraction modeling) and the location of the baffle in a listening room. The baffle and near boundary component are shared by the driver units that model cabinet edge diffraction and the reflections from the nearest boundaries in a listening room (rear wall, floor and left wall).



Baffle configuration

Baffle/Near-boundary

Diffraction analysis

Baffle dimension
W 300.0 mm
H 500.0 mm
D 300.0 mm

Location of origin
X 150.0 mm
Y 300.0 mm
Z 0.0 mm

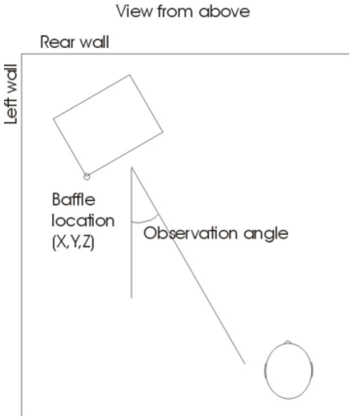
Near boundaries
☑ Enable simulation
Fc 0.3 kHz
refl 0.75

Observation point
angle 30 deg
Y 1.00 m

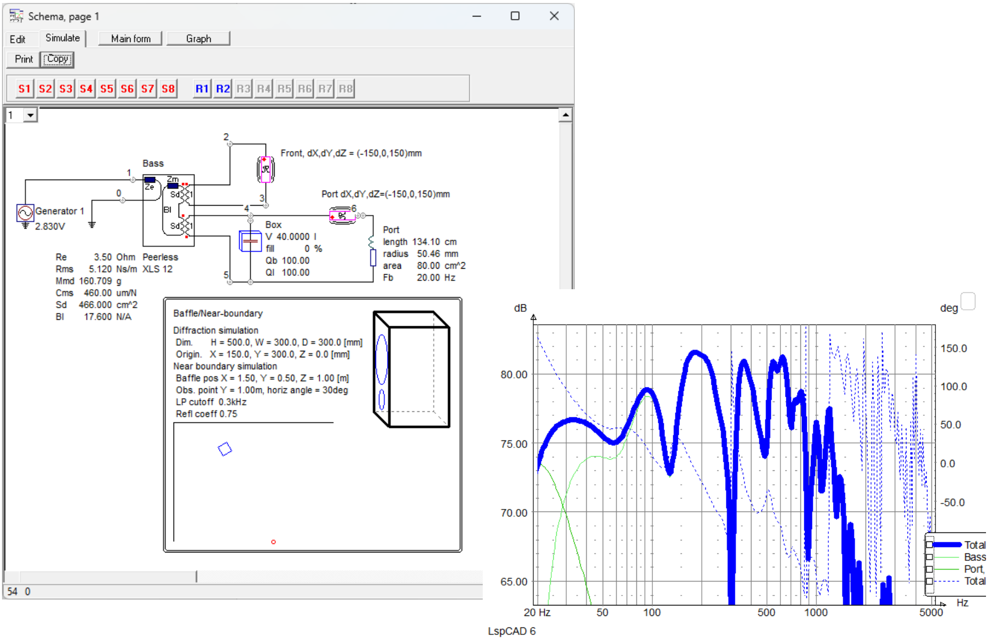
Baffle location
X 1.50 m
Y 0.50 m
Z 1.00 m

There may exist several baffle components in a simulation, e.g. one that describes the size of the subwoofer box and one that describes the midbass/treble box.

Baffle diffraction:
When the baffle symbol is clicked it is possible to set the Width, Height and Depth of the baffle. Also, it is possible to set a location of origin (distance from lower left corner). The drivers that use this baffle are then placed relative to this location of origin.
An example: If a bass reflex port refers to the baffle with the properties given in the figure above and has dX=0 and dY=200 it will be located at a position X=150+0=150mm and Y=600-200=400mm from the lower left corner of the baffle.
Near boundaries:
For simulation of the loudspeaker in a listening room. The baffle location is given as the distance from the lower left front corner of the baffle to the left wall, rear wall and floor. Additionally, one can specify the height of the listener (or the simulation mic.) and an angle between the normal to the rear wall that goes through the speaker and imagined straight line between the loudspeaker and the listener. The distance to the listener is given by the simulation distance in the settings dialog.
To reflect the fact that reflections at higher frequencies are more attenuated **Fc** is specified as the cutoff frequency of a 1st order LP filter for the reflections. Also, **refl** is a reflection coefficient that tells how much is reflected from the walls and floor.



The screenshots below show an example with a bassreflex box located in a listening room where the lower left corner of the frontside of the baffle is located 1.5m from the left wall, 0.5m from the floor and 1.0m from the rear wall. The driver unit as well as the port is placed on the side. The dX, dY and dZ for the driver (see ch. 8.3.5) and the port are adjusted to get the location correct on the baffle.



Front, dX,dY,dZ = (-150,0,150)mm

Port dX,dY,dZ=(-150,0,150)mm

Re 3.50 Ohm Peerless
Rms 5.120 Ns/m XLS 12
Mmd 160.709 g
Cms 460.00 um/N
Sd 466.000 cm^2
Bl 17.600 N/A

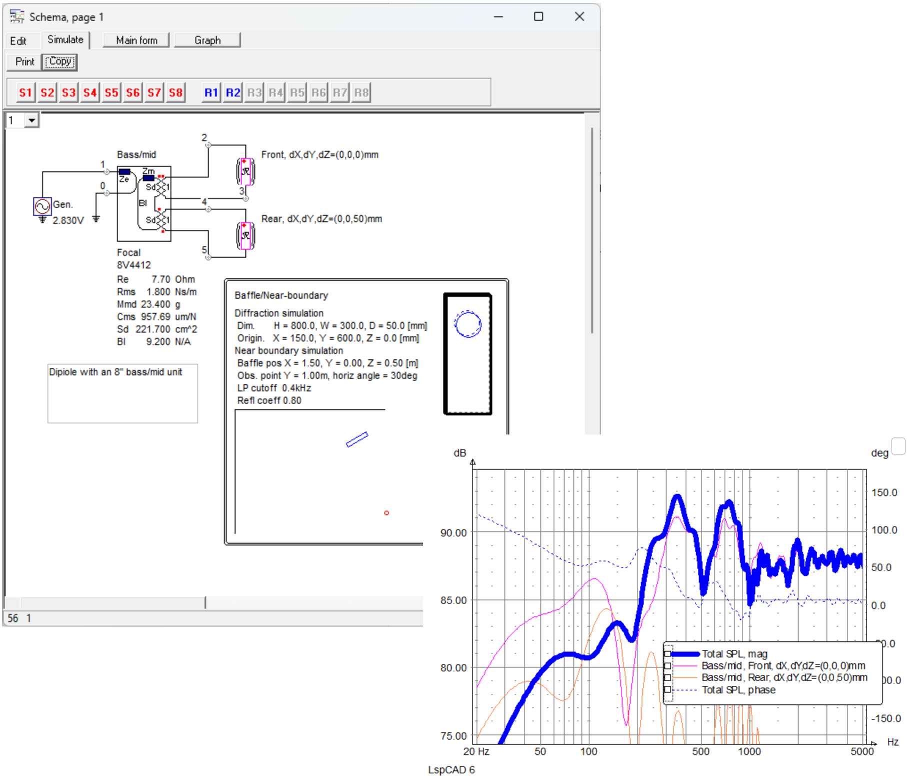
Box
V 40.0000 l
fill 0 %
Qb 100.00
Ql 100.00

Port
length 134.10 cm
radius 50.46 mm
area 80.00 cm^2
Fb 20.00 Hz

Baffle/Near-boundary

Diffraction simulation
Dim. H = 500.0, W = 300.0, D = 300.0 [mm]
Origin. X = 150.0, Y = 300.0, Z = 0.0 [mm]
Near boundary simulation
Baffle pos X = 1.50, Y = 0.50, Z = 1.00 [m]
Obs. point Y = 1.00m, horiz angle = 30deg
LP cutoff 0.3kHz
Refl coeff 0.75

Another example is a dipole loudspeaker with an 8” bass/mid loudspeaker unit mounted on a thin baffle. The rear radiation is set 50mm behind the front radiation, in addition the diffraction for the rear radiation is set to **Rear**.



Focal
8V4412
Re 7.70 Ohm
Rms 1.800 Ns/m
Mmd 23.400 g
Cms 957.69 um/N
Sd 221.700 cm^2
Bl 9.200 N/A

Dipiole with an 8" bass/mid unit

Baffle/Near-boundary
Diffraction simulation
Dim. H = 800.0, W = 300.0, D = 50.0 [mm]
Origin. X = 150.0, Y = 600.0, Z = 0.0 [mm]
Near boundary simulation
Baffle pos X = 1.50, Y = 0.00, Z = 0.50 [m]
Obs. point Y = 1.00m, horiz angle = 30deg
LP cutoff 0.4kHz
Refl coeff 0.80

Box component

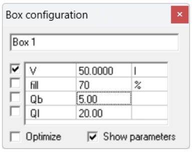
The box component describes the compliance and loss in a cavity of air in a box with a given volume. For a simple bass-reflex box one box component is connected to the backside of the diaphragm, a box load component is also needed to simulate the air load of the backside of the diaphragm. The properties that can be set are:

V: Box volume, i.e. the volume excluding the volume that drivers, passive radiators and ports occupy.

fill: Estimated fill of damping material in the box

Qb: Box Q-value due to damping material in the box

Ql: Box Q-value due to leakage from the box.



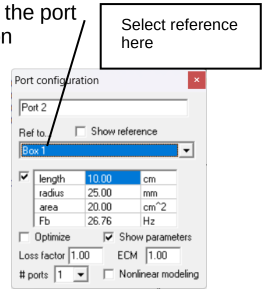
Box 1

☑	V	50.0000	l
□	fill	70	%
□	Qb	5.00	
□	Ql	20.00	

□ Optimize ☑ Show parameters

8.3.2 Port component

The port component has the properties, length, area and diameter, if the port radiates into free air it needs to be used in conjunction with a radiation component. It is possible to see the resulting port resonance frequency (Fb) if the port has a reference to a box. This reference is selected from the dropdown list box. For a bass reflex box the port and the box determine the port resonance, therefore the box must be referenced to in this case to make computation of the resonance possible.


Select reference here

Port configuration

Port 2

Ref to. □ Show reference

Box 1

☑		
length	10.00	cm
radius	25.00	mm
area	20.00	cm^2
Fb	26.76	Hz

□ Optimize ☑ Show parameters

Loss factor 1.00 ECM 1.00

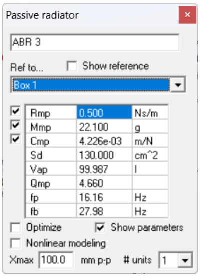
ports 1 □ Nonlinear modeling

The properties that can be set are:
length: Physical length of port
radius: Radius of port
area: Area of port
Fb: Port resonance, a box must be referenced for this property to be visible.
# ports: Number of identical ports.
Loss factor: The computation of the port loss factor is directly from [Beranek] . Experience has however shown that this loss factor is often too small, here it is possible to scale up the port loss up to a factor 10.0.
ECM: End correction multiplier, can be used to tune model to better match reality
Nonlinear modeling: Model port nonlinearities

☞ **Note that nonlinear modeling slows down computations considerably, it is therefore recommended that you turn off this function if it is not needed.**

8.3.3 Passive radiator component

A passive radiator or auxiliary bass radiator (ABR), has all the properties of an ordinary TS driver unit except for the absence of a magnet structure. Also, if a box is referenced (like is the case for the port component) it is possible to see the resonance of the ABR – box system (fb).



Passive radiator

ABR 3

Ref to... □ Show reference

Box 1

☑	Rmp	0.500	Ns/m
☑	Mmp	22.100	g
☑	Cmp	4.226e-03	m/N
	Sd	130.000	cm^2
	Vap	99.987	l
	Qmp	4.660	
	fp	16.16	Hz
	fb	27.98	Hz

□ Optimize ☑ Show parameters

□ Nonlinear modeling

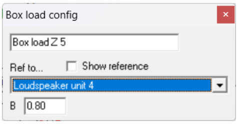
Xmax 100.0 mm p-p # units 1

The properties that can be set are:

Rmp: Loss resistance of suspension
Mmp: Moving mass
Cmp: Compliance of suspension.
Sd: Effective radiating area.
Vap: Equivalent volume.
Qmp: Loss factor.
fp: Resonance frequency of passive radiator in free air.
fb: Resonance of passive radiator in box.
# units: Number of identical units.
Of the above properties Rmp, Mmp and Cmp can be optimized.
Nonlinear modeling : Model nonlinearities

☞ **Note that nonlinear modeling slows down computations considerably, it is therefore recommended that you turn off this function if it is not needed.**

8.3.4 Box load component


Box load config

Box load Z 5

Ref to...

□ Show reference

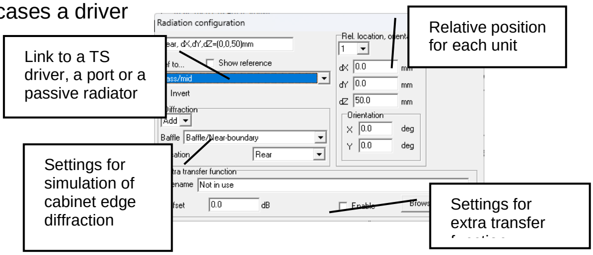
Loudspeaker unit 4

B 0.80

Each side of the diaphragm must be loaded by either a radiation component or a box load component, for the closed box a box load component is put in series with the box component on the backside of the diaphragm and a radiation component is put across the connectors of the front part of the diaphragm.
The boxload parameter **B** can be adjusted, this factor is from [Beranek]. For very small boxes this parameter may need to be increased to make the simulation reflect reality better.

8.3.5 Radiation component

To solve the problem that in some cases a driver component radiates into free air (e.g. closed box) and in other cases it is the interface between two cavities of a band pass box, a radiation component is linked to each element that actually radiates into free air (for instance TS drivers, ports and passive radiators that are visible).



Radiation configuration

ear, dX,dY,dZ=(0,0,50)mm

ef to... □ Show reference

ass/mid

Invert

Diffraction: Add

Baffle: Baffle/Near-boundary

ation: Rear

Rel. location, orient

1

dX 0.0 mm

dY 0.0 mm

dZ 50.0 mm

Orientation

X 0.0 deg

Y 0.0 deg

tra transfer function

ename: Not in use

fset 0.0 dB

□ Enable

Link to a TS driver, a port or a passive radiator

Relative position for each unit

Settings for simulation of cabinet edge diffraction

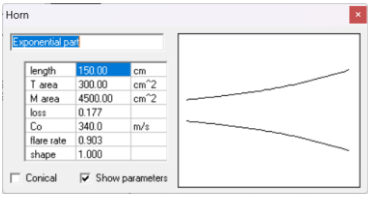
Settings for extra transfer

For the case of a bass reflex box we thus link a radiation component to the TS driver and one radiation component to the port. It is possible to see the reference on the schema if **Show reference** is checked.
Depending on how many identical units the component represents (e.g. two identical TS drivers), one can set the relative position for each unit.
For the diffraction analysis, it is possible to locate the radiation on the Front, Left, Right or Rear side of the baffle.

An extra transfer function can be imported, this might be a handy feature if one for instance wishes to include the effect of a large room acoustics simulation.

8.3.6 Horn/transmission line component

The Horn/transmission line component is a component that was first intended for the simulation of conical and exponential horns. With the addition of a loss factor and the possibility to set the speed of sound it is possible to simulate dipole subwoofers and transmission lines.



Exponential part

length	150.00	cm
T area	300.00	cm^2
M area	4500.00	cm^2
loss	0.177	
Co	340.0	m/s
flare rate	0.903	
shape	1.000	

☐ Conical ☑ Show parameters

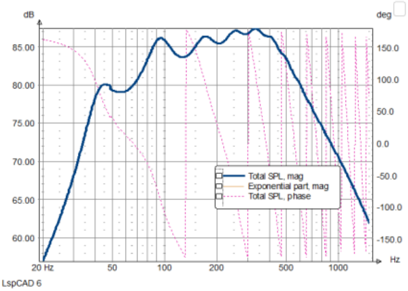
Length: Length of horn/t-line
T area: Throat area
M area: Mouth area
loss: Loss factor, expressed as a fraction of 2πf/c where f is the frequency and c is the speed of sound. For Horn modeling a loss factor of 0.1 to 0.2 seems to be adequate.
Co: Speed of sound.
Flare rate:
Shape: Shape = 1.0 gives an exponential flare while other values gives a hyperbolic shape.

The cross sectional area follows the equation

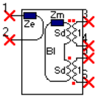
$$A(x) = Ao[\cosh(mx) + T\sinh(mx)]^2$$

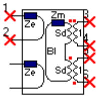
Where
A(x) : The area at distance x from the mouth
Ao : The throat area
M The flare rate
T The shape factor

An example simulation is shown in the graph below.



8.3.7 TS Driver unit (T/S parameter model)





The driver mech/acoustical model is a component that acts as an interface between the electrical and acoustical side of a loudspeaker unit. The model consists of 6 or 8 connectors.

Electrical side: The electrical side is either two connectors or four (dual VC).
Acoustic side: The acoustic side consists of two pairs of connectors, one for each side of the diaphragm.

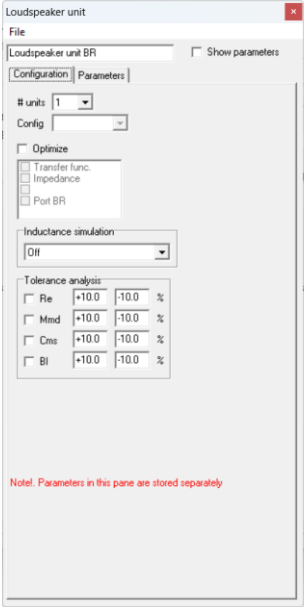
☞ **All parameters in this dialog are stored separately. In other words one must select File | Save in order to save changes.**
Note also that the unit data files stored in LspCAD 6 format cannot be opened in LspCAD 5.25

The configuration dialog for the driver mechanical model is quite complex.

8.3.7.1 Configuration tab

Tolerance analysis:
For tolerance analysis it is possible to set upper and lower tolerance analysis limits for Re, Mmd, Cms and Bl.

Inductance simulation:
There are three options here:
Off: No voice coil inductance simulation. Le, Reb and Leb has no meaning.
Load impedance only: Le, Reb and Leb only affect the load impedance (actually SPL is affected somewhat)
SPL and load impedance: SPL and load impedance are both affected.



Loudspeaker unit BR

□ Show parameters

units 1

Config

□ Optimize

□ Transfer func.
□ Impedance
□
□ Port BR

Inductance simulation: Off

Tolerance analysis

□ Re	+10.0	-10.0	%
□ Mmd	+10.0	-10.0	%
□ Cms	+10.0	-10.0	%
□ Bl	+10.0	-10.0	%

Note!. Parameters in this pane are stored separately

8.3.7.2 Parameters tab

T/S parameters tab :

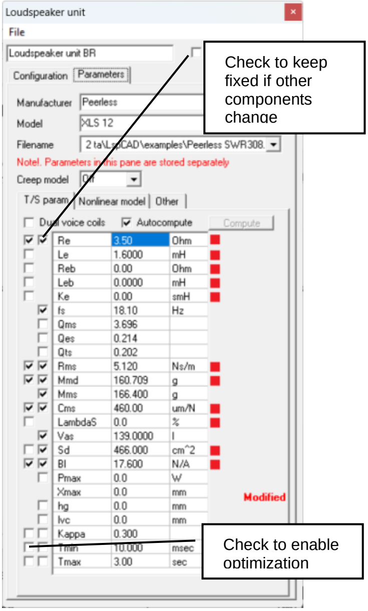
The T/S parameter (plus a few other things) are entered in a list. To the left of the parameter list are two columns with check-boxes.

In the leftmost column you select which parameters should be possible to optimize, note that only the parameters that are directly used in the LspCAD modeling are possible to optimize (also marked with **red**). In the other column one has the option to lock parameters that one wishes to keep fixed, i.e. they will remain the same although one alters a component that otherwise would have an impact of the locked parameter.

When one enters new parameters it is good to start from a new fresh parameter set (**File | New**).

After this one enters the parameters that one have. Important though is that one checks the lock box when the parameter is entered. If we for example have entered a value for the **Sd**, we then check the checkbox that is to the left of **Sd**. This procedure is repeated for each value that is entered, i.e. enter the value for a parameter, then lock it so that it won't change when other parameters are entered.

After a few parameters have been entered one will see that other parameters are automatically recomputed. If one don't wish this autocompute property one can uncheck the **Autocompute** checkbox. A manual crosscomputation of unlocked parameters can then be enforced with a click on the **Compute** button.



Loudspeaker unit

File

Loudspeaker unit BR □

Configuration | Parameters

Manufacturer: Peerless

Model: XLS 12

Filename: 2 ta\LspCAD\examples\Peerless SWR308.

Note!. Parameters in this pane are stored separately

Creep model: Off

T/S param | Nonlinear model | Other

□ Dual voice coils ☑ Autocompute Compute

☑	☑	Re	3.50	Ohm	🟥
□		Le	1.6000	mH	🟥
□		Reb	0.00	Ohm	🟥
□		Leb	0.0000	mH	🟥
□		Ke	0.00	smH	🟥
	☑	fs	18.10	Hz	
	□	Qms	3.696		
	□	Qes	0.214		
	□	Qts	0.202		
☑	☑	Rms	5.120	Ns/m	🟥
☑	☑	Mmd	160.709	g	🟥
	☑	Mms	166.400	g	
☑	☑	Cms	460.00	um/N	🟥
□		LambdaS	0.0	%	🟥
	☑	Vas	139.0000	l	
□	☑	Sd	466.000	cm^2	🟥
☑	☑	Bl	17.600	N/A	🟥
	□	Pmax	0.0	W	
		Xmax	0.0	mm	
	□	hg	0.0	mm	
	□	lvc	0.0	mm	
□	□	Kappa	0.300		
□	□	Tmin	10.000	msec	
□	□	Tmax	3.00	sec	

Modified

Check to keep fixed if other components change

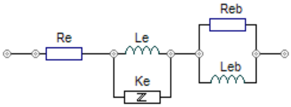
Check to enable optimization

The parameters that can be entered are:

Re	DC resistance	[Ω]
Le	Voice coil inductance	[mH]
Reb	Extended inductance model parameter	[Ω]
Leb	Extended inductance model parameter	[mH]
Ke	Extended inductance model parameter	[smH]
fs	Resonance frequency in free air	[Hz]
Qms	Mechanical Q-value	
Qes	Electrical Q-value	
Qts	Total Q-value	
Rms	The loss resistance of the suspension	[Ns/m]
Mmd	Moving mass excluding surrounding air	[g]
Mms	Moving mass including surrounding air	[g]
Cms	The mechanical compliance of the suspension	[m/N]
LambdaS	Cms creep factor	[%]
Vas	Equivalent volume	[l]
Sd	Cone area	[cm^2]
Bl	Force factor	[N/A]
Pmax	The maximum, thermal limited input power	[W]
Xmax	The maximum linear excursion, NOTE peak to peak value	[mm_{p-p}]
hg	Airgap height of the magnet	[mm]
lvc	The voice coil winding height	[mm].'
Kappa	Suspension creep model parameter	

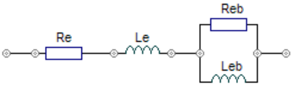
Tmin	Suspension creep model parameter	[msec]
Tmax	Suspension creep model parameter	[s]

Reb and Leb and Ke are specified if one wish to model the voicecoil inductance of the TS driver unit more accurately.
The equivalent electrical circuit for this is shown below.

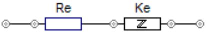


If Ke, Reb and Leb is set to zero, the voice coil self inductance is modeled only by the parameter Le. This is the classical way to model the voice coil inductance. This model does not represent real behavior very well, esp. for large subwoofers. For this purpose, one can do one of the following

1. Let Reb and Leb model the frequency dependent inductance (Ke=0). The model then looks like.



2. Let Le=0, Leb = 0, Reb = 0 and let Ke model the frequency dependent inductance, the model then looks like.



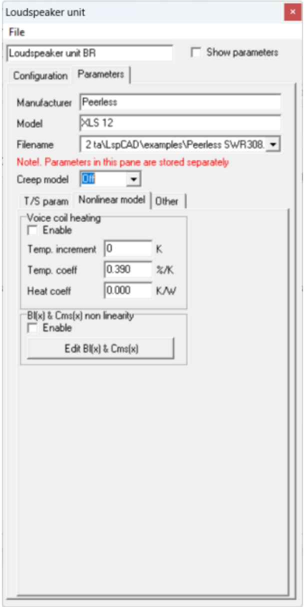
3. Use the three additional parameters Ke, Leb and Reb in combination

Improved suspension creep model:
The suspension creep model has been improved according to the paper "Modelling Viscoelasticity of Loudspeaker Suspensions using Retardation Spectra" by Tobias Ritter and Finn Agerkvist (AES 128, paper #8217).
For this purpose, a selector "Creep model" is available in the TSdriver pane
This can be set in 5 different modes
+ Off
+ Default (according to the implementation in older versions of LspCAD)
+ LOG
+ 3PC
+ 4PC
The three last modes are implemented according to the paper mentioned above.

In the LOG mode, suspension creep is controlled by lambda
In the 3PC mode, suspension creep is controlled by Kappa and Tmin
In the 4PC mode, suspension creep is controlled by Kappa, Tmin and Tmax

Nonlinear model tab:
Voice coil heating and the impact of it can be simulated. It is possible to simulate either in manual mode (Heat coeff = 0) or in automatic mode (Heat coeff > 0).
In manual mode one set the temperature increment in the voice coil manually.
In automatic mode the input voltage and corresponding calculated power dissipated through the voice coil determines the heating of the voice coil. The temperature coefficient depends of the voice coil wire material. (0.39%/K for copper)

☞ **Note that this function slows down computations considerably, it is therefore recommended that you turn off this function if it is not needed.**



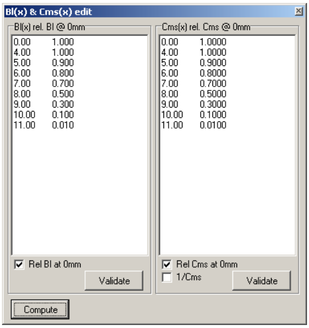
Loudspeaker unit BR
□ Show parameters

Manufacturer: Peerless
Model: XLS 12
Filename: 2 ta\LspCAD\examples\Peerless SWR308.
Note!. Parameters in this pane are stored separately
Creep model: Off

Voice coil heating
□ Enable
Temp. increment: 0 K
Temp. coeff: 0.390 %/K
Heat coeff: 0.000 K/W

Bl(x) & Cms(x) non linearity
□ Enable

Bl(x) and Cms(x) non linearity modeling
Non linear properties such as Bl and Cms can be modeled in LspCAD, to be more specific, what can be modeled is how the frequency response is affected when Bl and Cms changes due to excessive cone excursions. The Bl(x) and Cms(x) properties are entered in a dedicated input dialog that is accessed via the **Edit Bl(x) & Cms(x)** button.
In this dialog it is possible to enter as absolute values or relative to the values at the resting position (x=0). The 1st column is the deviation from x=0 (negative values are possible) and the 2nd column is the actual Bl or Cms values. In the example shown the values entered are relative to the values at x=0. Once the values are entered the **Validate** buttons must be clicked. To effect a computation the **Compute** button is clicked.



Bl(x) rel. Bl @ 0mm

0.00	1.000
4.00	1.000
5.00	0.900
6.00	0.800
7.00	0.700
8.00	0.500
9.00	0.300
10.00	0.100
11.00	0.010

☑ Rel Bl at 0mm

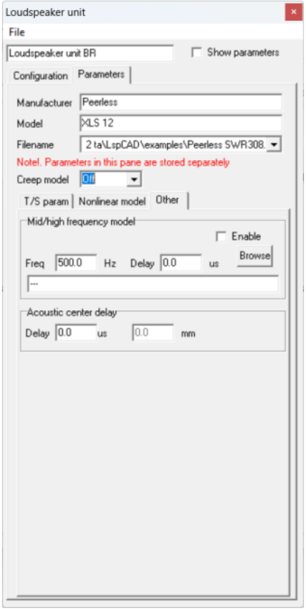
Cms(x) rel. Cms @ 0mm

0.00	1.0000
4.00	1.0000
5.00	0.9000
6.00	0.8000
7.00	0.7000
8.00	0.5000
9.00	0.3000
10.00	0.1000
11.00	0.0100

☑ Rel Cms at 0mm
□ 1/Cms

☞ **Note that this function slows down computations considerably, it is therefore recommended that you turn off this function if it is not needed.**

Other tab:
Mid/high frequency model: It is possible to import a frequency response that models the behavior at higher frequencies. This imported response is used for frequencies above the value given in the field named Freq.



Loudspeaker unit BR
□ Show parameters
Manufacturer: Peerless
Model: XLS 12
Filename: 2 ta\LspCAD\examples\Peerless SWR308.
Note!. Parameters in this pane are stored separately
Creep model: Off

Mid/high frequency model
□ Enable
Freq 500.0 Hz Delay 0.0 us

Acoustic center delay
Delay 0.0 us 0.0 mm

8.3.7.3 Menu

The menu contains a few selections. Besides the ordinary New/Save/Open there exist the possibility to import T/S parameters from Praxis, CLIO and Klippel (Matlab™ format). It is also possible to open or import LspCAD 5.25 and older unt files.

8.4 Other components

8.4.1 Connector component

The connector component is in normal cases used to connect between two schema sheets, one can also use the connectors to connect two points without the need for a wire. Each connector has a number, if two connectors share the same number, they are connected together.

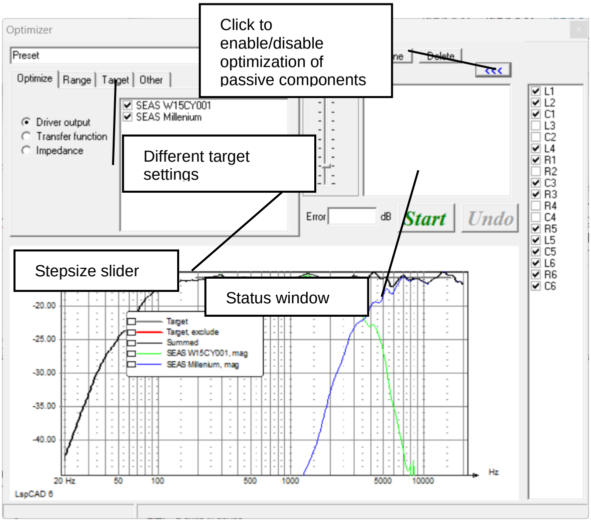
8.4.2 Switch component

The switch component is a toggling open/short circuit. For simplicity this component is implemented as a very high/low resistance.

8.4.3 User note component

The user note is a small window that can be placed anywhere on the schema. In this window one can type text information.

9 The optimizer


Click to enable/disable optimization of passive components

Different target settings

Stepsize slider

Status window

Optimize tab: ● Driver output ○ Transfer function ○ Impedance

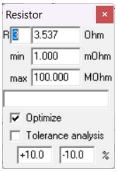
☑ SEAS W15CY001
☑ SEAS Millenium

☑ L1 ☑ L2 ☑ C1 □ L3 □ C2 ☑ L4 ☑ R1 □ R2 ☑ C3 ☑ R3 □ R4 □ C4 ☑ R5 ☑ L5 ☑ C5 ☑ L6 ☑ R6 ☑ C6

The optimizer is opened from the main menu. In the optimizer one can optimize the output response of a driver, a summed response, a transfer function between two points in the schema (see general settings dialog, SPL/Xfer tab), it is also possible to optimize the impedance to a given target. As targets one can select from either imported targets or a mixture of low/high pass slopes, notches etc. The optimize dialog contains a graph, a status window, a step size slider and a number of fields collected in a tab control.
A special feature is that it is possible to store away several different target settings by means of the **New**, **Clone** and **Delete** buttons The desired setting is recalled from the dropdown list box (also possible to change name for each setting). The optimizer starts with the **Start** button. The **Undo** button is a good companion in case the last optimization went FUBAR. Keep in mind though that there is only one undo-level, this means that only the last optimization can be undone.

9.1 Selecting components

The components that are to be selected for optimization are done from the settings of each component.
As an example, we wish to put a resistor in the optimization list. Right click on the resistor.
The advanced settings dialog that pops up gives the option to put the component on the optimization list.


Resistor

R 3 3.537 Ohm
min 1.000 mOhm
max 100.000 MOhm

☑ Optimize
□ Tolerance analysis
+10.0 -10.0 %

9.2 The optimize tab

Here one may select what one wishes to optimize, (see previous page for a view).

Driver output:

The most classical use is of course to optimize the response of a radiating element (or a sum of them). In this case select the Radiating elements button and check the radiating elements that you wish to optimize the response of. The responses are added, and the individual responses are also shown in the graph.

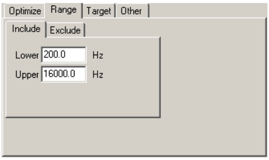
Transfer function:

It is also possible to optimize a transfer function between two points in the schema (the nodes are selected in the settings dialog).

Impedance:

The third use is to optimize the total impedance against a target, this is quite handy e.g. if one has measured the impedance of a driver in box and wishes to derive the true volume and loss factors of the box. Moreover, it is actually possible to derive the TS parameters with the measured free air impedance as target with this option.

9.3 The range tab



Lower 200.0 Hz

Upper 16000.0 Hz

The error that is computed in the optimizer and is a basis for the optimizer decision is computed over a frequency range that is determined in the range tab.

The range is defined in two ways.

Include : Defines the upper and lower frequency limits for the optimization.

Exclude: In some cases, it may be desirable to exclude a small region (e.g. a peak in the frequency response) this may be done by means of the exclude region.

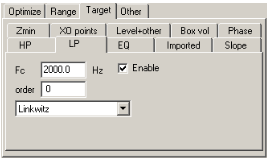
It is good practice to use a range that covers the frequency range, no more, no less. If the range is set too wide, it may be difficult for the optimizer to find a good solution. If the range is set too narrow the optimization may goof out. In general, the optimization range should cover frequency range that the optimized components can affect if they are changed.

9.4 The target tab

In the target tab one has the ability to create a number of different targets. The target responses are overlaid i.e. one may combine a low pass, a high pass and a notch filter response to form a complete target. While it is possible to do lots of optimization one must keep in mind that there are some physical barriers that cannot be crossed, so if your optimization goes wrong, chances are big that you are simply asking for too much from the given set of drivers and filter topology.

9.4.1 LP

A low pass target, one can select from different slopes such as Butterworth and Linkwitz-Riley of different orders, also one can use a variable slope.
The **Enable** box can be unchecked if one wishes to disable the low pass target.



Fc 2000.0 Hz ☑ Enable

order 0

Linkwitz

9.4.2 HP

A high pass target, same options as above.

9.4.3 Slope

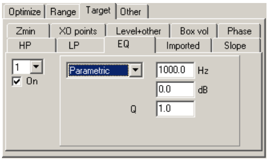
An overall positive or negative slope in the target response.

9.4.4 EQ

The EQ (there are two available) can be either a parametric or a low/high shelving EQ.

9.4.4.1 Parametric

The center frequency and the Q and gain at the center frequency are specified.



1 ☑ On

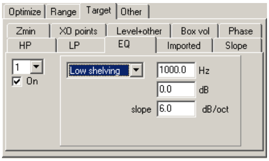
Parametric 1000.0 Hz

0.0 dB

Q 1.0

9.4.4.2 Low/high shelving

The transition frequency and the max/min gain is set also is possible to specify how fast the transition should go (**slope**).



1 ☑ On

Low shelving 1000.0 Hz

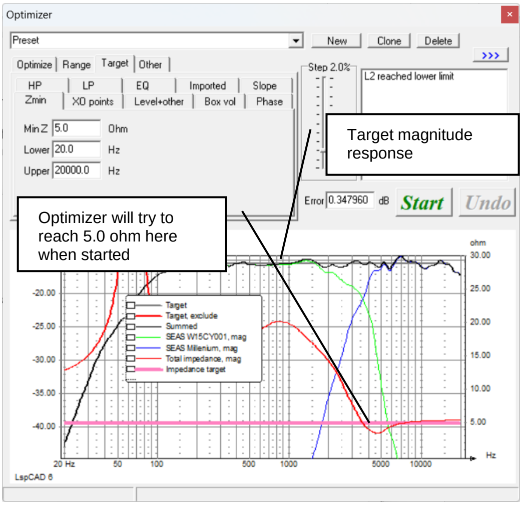
0.0 dB

slope 6.0 dB/oct

9.4.5 Z min

A minimum value of the allowed load impedance on the driving amplifier. If this is in use the optimizer is forced to meet this condition.
The impedance response is optimized in the range given by the **Lower** and **Upper** settings Given this fact it is possible that the optimizer will "goof out" if the min impedance is set too high, therefore this option should be used with care. A good practice is to set the impedance target moderately and increase it gradually and watch the magnitude response while the optimizer is running.

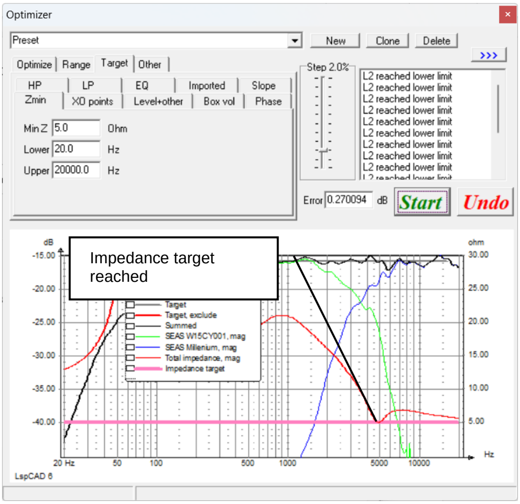
For this particular case, the Undo button may be very useful as the strive to reach both desired SPL target and a lower limit of the load impedance is often contradictory.



Min Z 5.0 Ohm
Lower 20.0 Hz
Upper 20000.0 Hz
Step 2.0%
L2 reached lower limit
Error 0.347960 dB

Target magnitude response

Optimizer will try to reach 5.0 ohm here when started

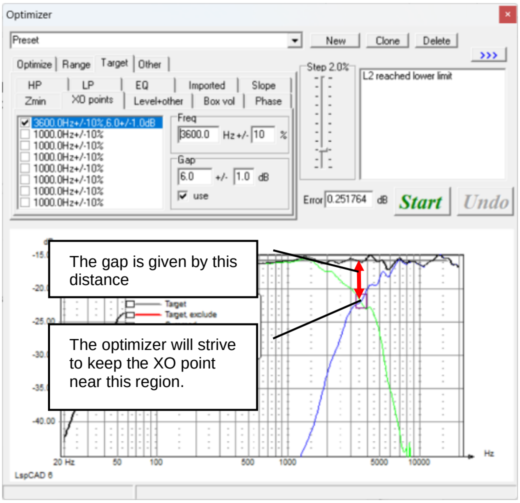


Min Z 5.0 Ohm
Lower 20.0 Hz
Upper 20000.0 Hz
Step 2.0%
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
Error 0.270094 dB

Impedance target reached

9.4.6 XO points

In this tab one has the option to lock the crossover points within specified limits and also a gap between the summed output and the levels of each individual driver at the crossover point. The crossover points are not completely locked to the specified values, instead the distortion metric i.e. the difference between the target and the total summed response is gradually increased when the crossover point leaves the specified limits. This feature is good to use to avoid that the crossover frequency drifts away during the optimization.
Note however that in some cases it may be good to allow the XO point to drift a little for a better response or a better impedance.



The gap is given by this distance

The optimizer will strive to keep the XO point near this region.

9.4.7 Imported

In this tab one can import an ASCII text file that describes the desired target. The text file should be arranged in frequency-SPL pairs, the first, leftmost column is the frequency and the second column is the magnitude in dB.
A typical text file could look like

```
20      0
2000    0
3000    -3
4000    -3
6000    0
20000   0
```

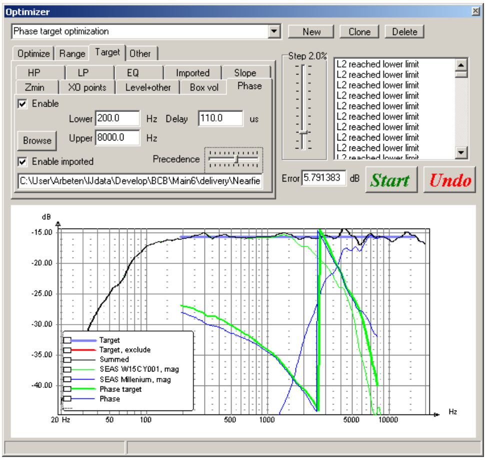
The above text file gives the target of a BBC dip.

9.4.8 Level + Other

This tab contains controls for things such as:
Punish negative interference, if this control is checked the optimizer will do its best to assure that drivers are summed in phase in the overlap regions.
Lock, if this is checked the target is locked to a specified mean value, otherwise it will float. This feature could be useful if one e.g. wish to squeeze an extra dB in efficiency at the cost of worse load impedance for the driving amplifier.

9.4.9 Phase

The phase target tab makes it possible to optimize against a target phase response. The phase response is optimized in the range given by the **Lower** and **Upper** settings. The **Delay** control makes is possible to add an excess phase while for more complex phase responses it is possible to import a target phase response. The imported file should be in ASCII format with three columns. The first (leftmost) column should be the frequency and the **third** is the phase in degrees.
With the **Precedence** control it is possible to set how great impact this phase optimization should have on the combined distortion metric.



Phase target optimization

Step 2.0%

☑ Enable

Lower 200.0 Hz Delay 110.0 us

Upper 8000.0 Hz

☑ Enable imported

Precedence

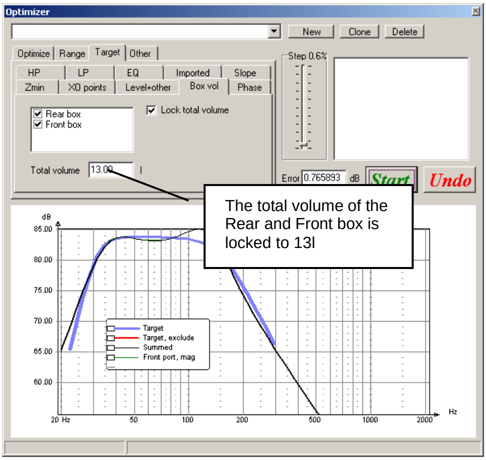
C:\User\Arbeten\IJdata\Develop\BCB\Main6\delivery\Nearfie

L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit
L2 reached lower limit

Error 5.791383 dB

9.4.10 Box vol

For e.g. band pass boxes it may be desirable to lock the total volume. Check the box volumes that should be added and set the total volume that should be kept during the optimization.



Step 0.6%

☑ Rear box
☑ Front box

☑ Lock total volume

Total volume 13.00 l

Error 0.765893 dB

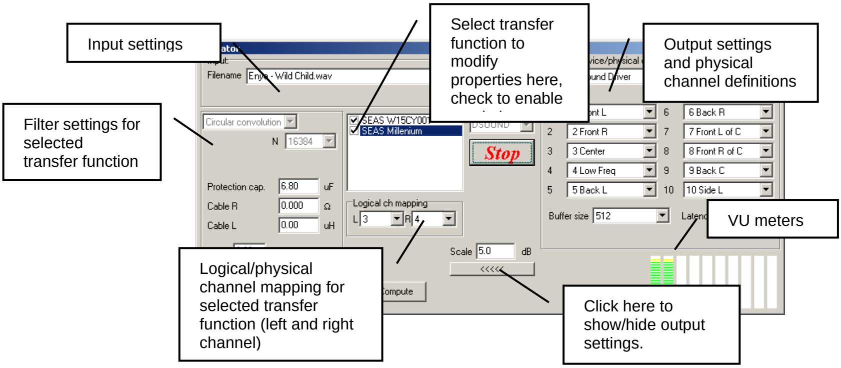
The total volume of the Rear and Front box is locked to 13l

9.5 *The other tab*

In the other tab one can for instance select the **Fast mode, which** will increase the speed about 5 times at the cost of lower accuracy.
Another option is the **Live update**, this option slows down the optimization a bit but makes it possible to see the other graphs change during the optimization.

10 The crossover emulator

The crossover emulator is a tool that makes it possible to emulate the transfer function of the different parts of the crossover.



Input settings

Select transfer function to modify properties here, check to enable

Output settings and physical channel definitions

Filter settings for selected transfer function

VU meters

Logical/physical channel mapping for selected transfer function (left and right channel)

Click here to show/hide output settings.

Filename Enya - Wild Child.wav

Circular convolution

N 16384

☑ SEAS W15CY001
☑ SEAS Millenium

Protection cap. 6.80 uF

Cable R 0.000 Ω

Cable L 0.00 uH

Logical ch mapping L 3 R 4

Scale 5.0 dB

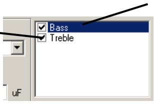
Buffer size 512

	...ont L	6	6 Back R
2	2 Front R	7	7 Front L of C
3	3 Center	8	8 Front R of C
4	4 Low Freq	9	9 Back C
5	5 Back L	10	10 Side L

This means that one can pre-evaluate the crossover and listen to how it sounds without having to build it.
As input one can use either pre-recorded WAV files or the soundcard input. The applications are many, one application is to determine the most appropriate crossover topology or crossover frequency for an existing loudspeaker prototype that needs an appropriate active or passive crossover.

10.1 Filter settings

2) Click on the checkboxes to enable/disable emulation of a transfer function.



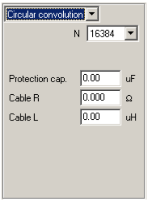
1) Click here to select what transfer function you wish to set the emulation properties (filter settings) for.

The emulation properties (also the logical channel mapping described later) can be set for each transfer function individually. To modify settings for e.g. the “Bass” network transfer function one simply click on the text “Bass” in the checklist box (see figure above).

Two types of digital filters are implemented in the crossover emulator. **IIR** filters and **Circular convolution** filters. Besides the settings for these two filters there are a few other settings available.

Protection cap,:
A “real” capacitor may be used between a power amplifier output and the tweeter in a crossover emulation situation to protect it from unwanted low frequency sounds such as power on transients and Windows notification sounds. If this is used one should enter the capacitor value in this field. It is also important that the impedance data is imported for the treble unit in the simulation.

Cable R and **Cable L**:
The series resistance and inductance of the cable between the power amplifier and the loudspeaker is entered here if it is deemed important. Like with the use of the protection capacitor it is important to import impedance data for the driver if this feature is used.



Circular convolution

N 16384

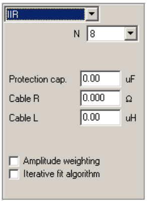
Protection cap. 0.00 uF

Cable R 0.000 Ω

Cable L 0.00 uH

Circular convolution:
The simplest way to filter, only problem is that it is more complex (read CPU demanding) than the IIR filter method. It is possible to select different filter lengths, a general rule is that for high pass filters (treble units) a short filter is sufficient while in order to represent the transfer function of a subwoofer well it may be needed to have a very long filter length. The **Emulator, target and achieved transfer function** graph is a good guidance to how to select the filter lengths as it displays the worst-case transfer function resulting from the selected filter lengths.

IIR:
The IIR filter method is less complex than the Circular convolution method, but it is a bit more complicated to work with, therefore the recommendation is to use the Circular convolution when possible.
The filter order can be set in the range 4-30. The optimal filter order is determined by looking at the achieved transfer function and testing different filter orders until the best match is achieved. In many cases the **Iterative fit algorithm** may be used to improve the matching.



IIR

N 8

Protection cap. 0.00 uF

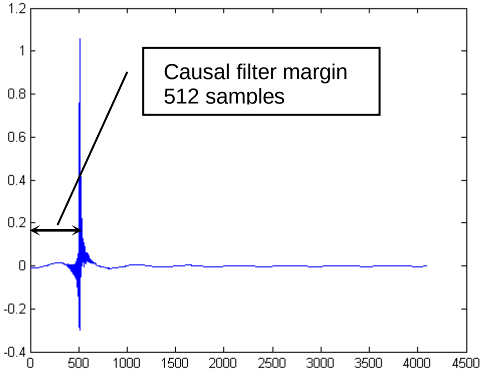
Cable R 0.000 Ω

Cable L 0.00 uH

☐ Amplitude weighting

☐ Iterative fit algorithm

Causal filter margin:
As there is a risk that filters become non-causal (like laughing before being tickled) it is necessary to add some extra delay to be able to run the filters in the crossover emulator. Phase linear FIR filters are normally non-causal and if they are implemented as such in the simulation it is necessary to add a **Causal filter margin** that corresponds to half the number of filter taps. As an example, if a Digital FIR filter with 2049 taps is simulated and we wish to emulate this filter a Causal filter margin of 1024 samples is needed.



Automatic export of impulse responses (only Circular convolution mode):
The impulse responses for each filter leg are exported automatically to the LspCAD 6 root folder, or in a path such as
C:\Users\USERNAME\AppData\Local\VirtualStore\Program Files (x86)\IJData\LspCAD\
on newer windows versions.

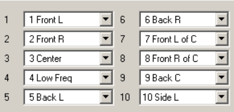
The file names are:
LspCAD_6_xo_emulator_1.txt : Corresponding to the 1st item in the list
LspCAD_6_xo_emulator_2.txt : Corresponding to the 2nd item in the list

and so on.
The length of the exported impulse response is determined by the length indicated for each filter leg.
The format of the files is ASCII engineering format with one coefficient per line. The impulse responses can be imported to a convolution engine such as BruteFIR.

10.2 Mapping to audiocard output

To bring some order to the use of multichannel audio cards the concept of logical and physical channels is used, this mapping applies to MME and DSOUND access methods, for ASIO the mapping is somewhat simplified (see ch. 10.3).

Physical channel mapping:

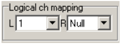


1	1 Front L	6	6 Back R
2	2 Front R	7	7 Front L of C
3	3 Center	8	8 Front R of C
4	4 Low Freq	9	9 Back C
5	5 Back L	10	10 Side L

The physical channel mapping is a means to enumerate the physical output channels of an audio card, for instance one can map logical ch 1 to the left loudspeaker output and logical ch. 2 to the right loudspeaker output. For a 5.1ch soundcard one can map the logical channel 3 to the center channel and so on. The physical channel mapping is set once and for all for a specific soundcard and is used for all projects.

Logical to physical channel mapping

To get any sound out from the emulator the filtered signal must be routed to the physical channels.



Logical ch mapping
L 1 R Null

The logical channel mapping is a way to direct the emulator output related to each transfer function to the physical channels. In the example to the right the filtered left channel is routed to logical channel 1 while the filtered right channel is not connected to any output. It is possible to connect several filtered channels to the same logical channels, for instance one may connect both the left and the right channel for one filtered output to the same logical channel if one has a limited number of physical channels available.

If for instance we have a two way crossover we can direct the emulator output of the low pass filter to logical channel 1 and the high pass filter output to logical channel 2, this way the low pass signal is directed to the left output of the audio card and the high pass output is directed to the right channel of the audio card.

With the example of the Bass and Treble filter given in the figures presented in the preceding pages we can map the logical channels like below if we only have a two channel audio card available

Bass L ->	1
Bass R ->	NULL
Treble L ->	2
Treble R ->	NULL

In other words.

The left channel processed through the Bass network is sent to logical channel 1, that is routed to Front L of the sound card.

The left channel processed through the Treble network is sent to logical channel 2, that is routed to Front R of the sound card.

In both cases the right input channel is not used (sent to NULL).

If you only have a two-channel soundcard but want to have both left and right input channels as input in the emulation, you should have a logical channel mapping like this.

Bass L ->	1
Bass R ->	1
Treble L ->	2
Treble R ->	2

With this setting the output levels will be increased app. 6dB so you might need to adjust the **Scale** in order to avoid clipping.
The output level can be monitored in the VU meters, it is recommended to avoid getting into the red area

The logical channel mapping, (or the mapping between emulator filtered output and logical channels) is project dependent. The motivation behind this distinction between logical and physical channels is for instance that with a four-channel audio card it is possible to emulate a stereo two-way system while with a four-way system one can only emulate a mono four way.
In the two-way case the logical channel mapping may look like.

Bass L ->	1
Bass R ->	2
Treble L ->	3
Treble R ->	4

In the four-way case the logical channel mapping may look like.

Bass L ->	1
Bass R ->	NULL
Low mid L ->	2
Low mid R ->	NULL
High mid L ->	3
High mid R ->	NULL
Treble L ->	4
Treble R ->	NULL

In both cases above the physical channel mapping stays the same, it may for instance look like

1 ->	1 Front L
2 ->	2 Front R
3 ->	3 Center
4 ->	4 Low Freq.

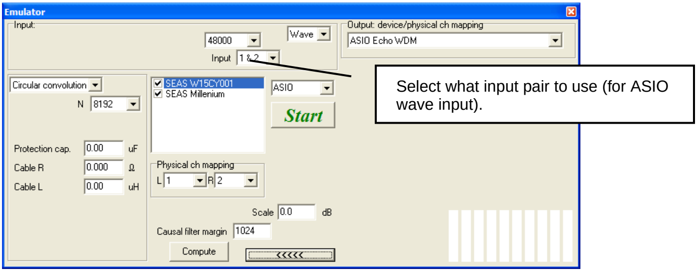
10.3 Soundcard (audio card) access methods

There are two ways to access the soundcard

DSOUND Requires DirectX ver 8 or higher. One may experiment with the **Buffer size** and **Latency** setting, often it is possible to go down as low as 50ms with this setting.

ASIO : Audio Synchronous Input Output, an audio I/O access method devised by Steinberg. This method provides for very low latency and is also quite stable, provided that the ASIO drivers supplied by the hardware vendors are stable. Unlike the DSOUND access method there exist no dedicated logical to physical mapping of the outputs, instead the logical channels are mapped directly to the physical channels of the same number, for instance, if the left channel of the midrange is put on logical channel 5, it is automatically put on output channel 5.

It is common that professional ASIO audio cards has more than two inputs. For this purpose, one can select what input pair should be used. Input pair 1&2 might be a microphone input and input pair 5&6 might be a digital input.


Emulator

Input: 48000, Wave, Input 1&2

Output: device/physical ch mapping: ASIO Echo WDM

Circular convolution, N 8192

☑ SEAS W15CY001
☑ SEAS Millenium

ASIO

Protection cap. 0.00 uF
Cable R 0.000 Ω
Cable L 0.00 uH

Physical ch mapping: L 1, R 2

Scale 0.0 dB

Causal filter margin 1024

Select what input pair to use (for ASIO wave input).

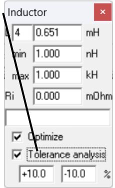
The number of input and output channels that can be accessed is equal to the number of channels available in the audio card (the number is however limited to 10 in LspCAD for practical reasons).

☞ **Don’t address more channels than are available on your audio card. For instance, if your audio card has 6 outputs, mapping one signal to output 7 does not have any meaning and can produce unexpected behavior.**

11 Tolerance analysis

The tolerance analysis tool is a good means to test a design under more normal production-based circumstances where component tolerances play a role. Besides doing tolerance analysis on resistors, capacitors and inductors one can also analyze how the production spread of a normal loudspeaker driver will affect the response. In the loudspeaker unit case one can set tolerance limits for Re, Mmd, Cms and Bl and see how it affects response. Once we have selected what components we wish to do tolerance analysis on it is time to start the iteration.

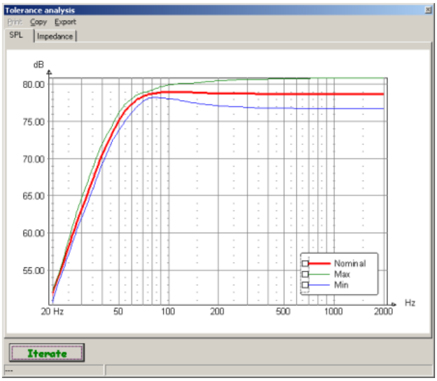
Check this box and set appropriate tolerance limits for the tolerance analysis



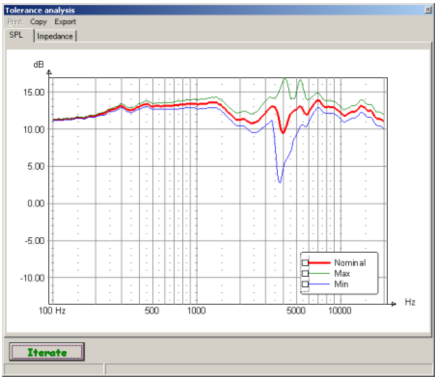
L 4 0.651 mH
min 1.000 nH
max 1.000 kH
Ri 0.000 mOhm
☑ Optimize
☑ Tolerance analysis
+10.0 -10.0 %

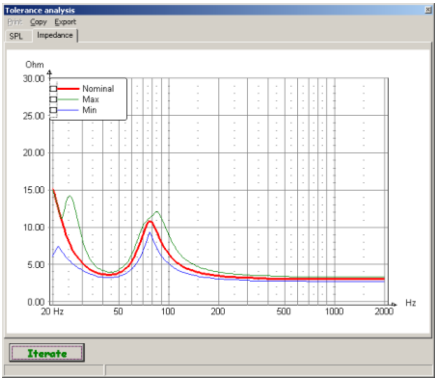
Below is the result of the tolerance analysis of a normal bass reflex box.

The min and max graphs tell that the response is inside the area between min and max.



It is possible to include many components in the tolerance analysis, below is the result from the tolerance analysis of a passive crossover with 10% tolerance limits on all components.



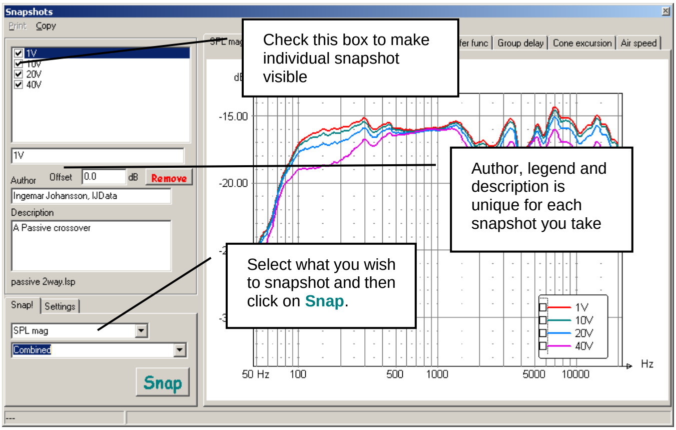


It is possible to export the tolerance analysis results for later use in e.g. a pass/fall test situation. The exported data is in ASCII format with four columns (freq, Nominal, Min and Max), below is an example.

```
LspCAD 6 tolerance analysis data
 SPL data
 release=5 99 18 20031007
       Freq     Nominal         Min          Max
     50.00         4.75        4.72         4.79
     51.77         5.14        5.10         5.17
     53.59         5.54        5.51         5.57
     55.49         5.71        5.68         5.74
     57.45         5.86        5.83         5.89
     59.48         6.17        6.15         6.19
     61.58         6.58        6.56         6.60
     63.75         7.01        7.00         7.03
.
.
.
  34130.36        14.34       13.41        15.23
  35335.91        14.33       13.40        15.22
  36584.04        14.32       13.40        15.21
  37876.25        14.32       13.39        15.20
  39214.11        14.31       13.39        15.19
  40599.22        14.31       13.39        15.19
  42033.26        14.30       13.38        15.18
  43517.96        14.30       13.38        15.17
  45055.09        14.29       13.38        15.17
  46646.52        14.29       13.37        15.16
  48294.16        14.28       13.37        15.15
  50000.00        14.28       13.37        15.15
```

12 Snapshot

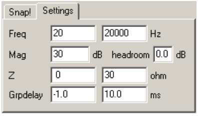
With the snapshot function it is possible to make snapshots from one or many projects. For each snapshot one can set some information and also an offset.
Below is an example where the snapshot visualizes the power compression for a two-way crossover.



Check this box to make individual snapshot visible

Author, legend and description is unique for each snapshot you take

Select what you wish to snapshot and then click on **Snap**.



Freq	20	20000	Hz			
Mag	30	dB	headroom	0.0	dB	
Z	0	30	ohm			
Grpdelay	-1.0	10.0	ms			

There are a few settings that one can use to customize the looks of the snapshots.
Freq: The displayed frequency range
Mag: The magnitude range for the SPL and transfer function graph + an optional headroom
Z: min and max displayed impedance values
Grpdelay: min and max displayed group delay.

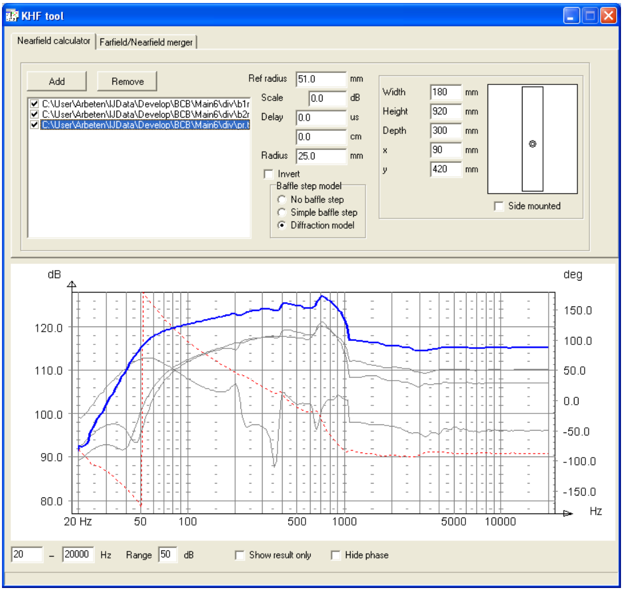
Menu:
Besides the print and copy (to clipboard) menu alternatives one can also save or open a snapshot file.

13 KHF Tool

The KHF tool (named after the famous acoustic designer Karl-Heinz Fink) is a tool that is used when one wish to combine e.g. Near and far field measurements in order to get a resulting response curve useful for loudspeaker simulation. Additionally, one can combine the responses from driver and port in a bass reflex box.

13.1 Nearfield calculator

The nearfield calculator is used to combine several near field measurements and also apply baffle diffraction effects. In the example below, three measurements are imported, two representing a near field measurement of the driver and one for the port in a bass reflex box. Measurements are added with the Add button, moreover the checked items are summed to form a total summed graph.
There are a few controls in this tab to allow modification of each item.



☑ C:\User\Arbeten\IJData\Develop\BCB\Main6\div\b1r
☑ C:\User\Arbeten\IJData\Develop\BCB\Main6\div\b2r
☑ C:\User\Arbeten\IJData\Develop\BCB\Main6\div\pr.t

Ref radius 51.0 mm
Scale 0.0 dB
Delay 0.0 us
0.0 cm
Radius 25.0 mm
□ Invert

Baffle step model
○ No baffle step
○ Simple baffle step
● Diffraction model

Width 180 mm
Height 920 mm
Depth 300 mm
x 90 mm
y 420 mm
□ Side mounted

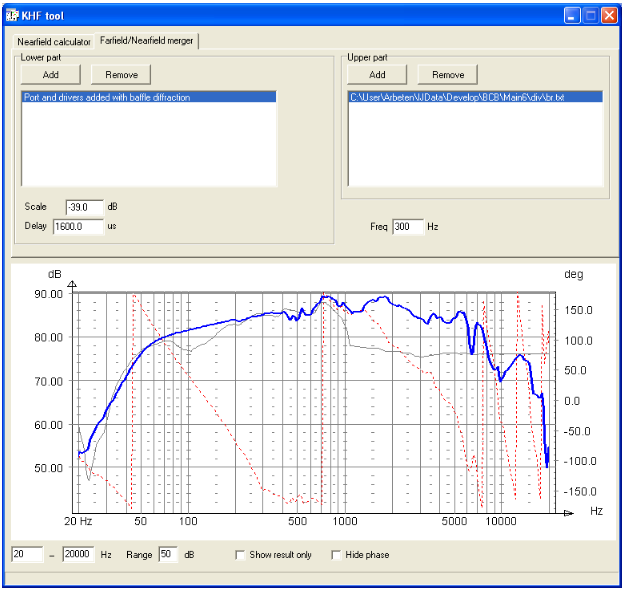
20 – 20000 Hz Range 50 dB □ Show result only □ Hide phase

Ref radius: A reference radius, typically one set this to the radius of the drive unit.
Scale: An arbitrary scale factor
Delay: An additional delay that can be applied to the item.
Radius: The radius of the item, for instance this handles the rescaling of the port measurement so that it matches the contribution of the drive unit. It is actually the ratio between **Ref radius** and **Radius** that determines the scaling.
Baffle step model: This applies the baffle step which is normally not seen in near field measurements. The **Simple baffle step** just applies a simple step at a given frequency while the **Diffraction model** has a few more options. Besides the possibility to enter the

Width, **Height**, **Depth**, **x** and **y** location one can also simulate the side-mounted placement of the drivers in a rectangular box that is popular with subwoofer units.
Once the result is as desired, one can right click with the mouse to either save the result to a text file for later use in e.g crossover simulations or copy the result to the internal clipboard for use in the Farfield/Nearfield merger.

13.2 Farfield/Nearfield merger

The farfield/nearfield merger is used for two measurements, a typical case is where one use the farfield measurements for the upper frequency range and the nearfield measurement for the lower frequency range. In order to make the magnitude and phase match between the lower and the upper part one can adjust he **Scale** and **Delay** of the lower part. Right click with the mouse to either save the result to a text file for later use in e.g crossover simulations or copy the result to the internal. In the example below the nearfield calculation from the previous page is pasted into the left box with a right click on the mouse.



Port and drivers added with baffle diffraction

Scale -39.0 dB

Delay 1600.0 us

C:\User\Arbeten\IJData\Develop\BCB\Main6\div\br.txt

Freq 300 Hz

20 – 20000 Hz Range 50 dB

14 Thermal and nonlinear modeling

This chapter describes in a little more detail how the thermal a nonlinear modeling works. Ideally the SPL output from a loudspeaker system should increase linearly with input power. This is however rarely the case, instead the output is compressed; a well-known term is power compression. In many cases this phenomenon is accompanied with nonlinear distortion. Several different causes of power compression exist in a loudspeaker system

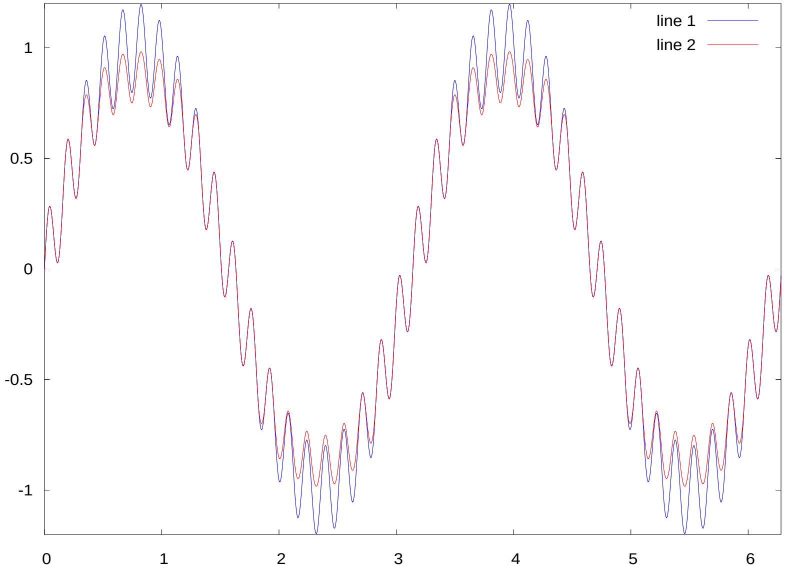
14.1 Brief presentation of the causes for power compression

14.1.1 Thermal modeling (Voice coil heating)

Voice coil heating: as the temperature increases due to increased power dissipation in the voice coil the DC resistance is also increased. For copper wire this increase is 0.39%/K at room temperature. The effect of this is a change in the system sensitivity and the frequency response but it is also possible that the DC resistance is modulated with the input signal if the frequency is very low, giving nonlinear distortion.

14.1.2 Bl (force factor) reduction:

As the voice coil leaves its resting position the force factor will reduce, and a shorter part of the voice coil is in the magnetic air gap. This phenomenon gives nonlinear distortion and will affect the frequency response, as the average Bl is decreased the system damping will decrease giving higher system Q value, the system efficiency will also decrease due to this. The figure tries to show what happens when one applies this nonlinearity to a signal that consists of a low frequency tone and a higher frequency tone, the figure displays the cone excursion, the tone at the lower frequency has a higher amplitude as the cone excursion increases with lower frequency. One can clearly see that both tones are affected by this. If one would make another experiment and test both tones individually one would see that only the low frequency tone would be affected while the higher frequency tone would be unaffected. The two-tone experiment is a very simple approximation of music signals (also IM distortion test) while the individual tone experiment is more like the swept distortion measurements.



14.1.3 Cms (compliance) reduction:

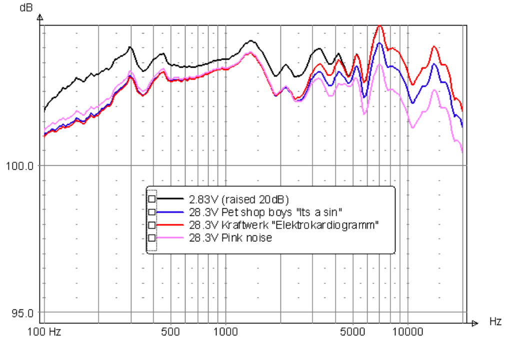
As the voice coil moves away from its resting position the force that seeks to move it back increases not linearly with distance but at a higher rate. This is also called progressive stiffness and is deliberately applied to ensure that the voice coil does not leave the magnetic gap when Bl decreases, a problem that can occur in ported systems.
As the average Cms is decreased the resonance frequency will move upwards.
A specific problem is if the Cms(x) curve is asymmetric, the effect of this is that a DC level will build up that will pull the voice coil out of its working range.

14.1.4 Port nonlinearity

At low levels a loudspeaker port acts like mass + a flow resistance in series. As the airflow increases though the flow resistance will start to increase (a few m/s) at even higher airflows the port will stop working as a port and will instead behave as a jet (port mass will decrease).

14.2 Thermal modeling

Thermal modeling in LspCAD 6 can be applied to both Driver and TS driver components. In order to get a fairly accurate idea of the behavior it is necessary to specify a PSD that is used for the modeling. PSD stands for power spectral density and describes how the power is distributed across the frequency range. There is a great difference on how the power is distributed for different music pieces. Below is a crossover simulation of a two-way MTM loudspeaker system where the midranges have a temperature coefficient of 3deg/W and the treble unit has a temperature coefficient of 20deg/W. One can see that the simulated response for the high-level cases depends heavily on the PSD (or input material).



Modeling like this there is some room for confusion. For instance the level 28.3V is what a true RMS voltmeter would read in this case means that over an 8 ohm load the power dissipation would be $(28.3)^2/8 = 100$W. One can in this case argue that the Kraftwerk piece will probably have a much higher risk of clipping the power amplifier output than the Pet Shop Boys piece as the bass content is much higher in the former case.

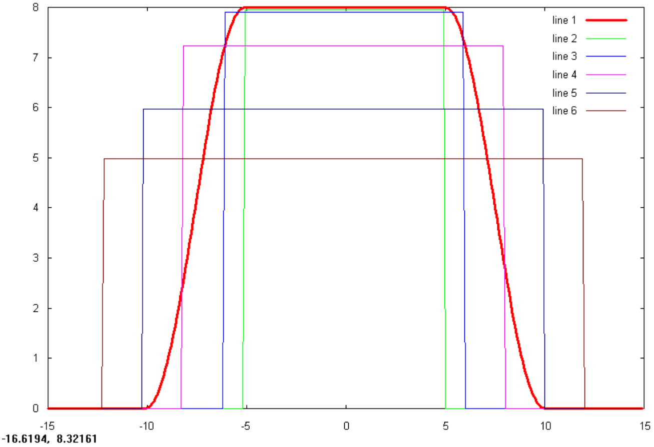
14.3 Nonlinear modeling

The nonlinear modeling can be done in two ways
Swept mode:
Sort of the classical method. This method simulates a frequency sweep with a given input voltage
PSD based:
This method assumes that the input is music with a given power spectral density.

The nonlinear modeling in LspCAD 6 is not intended to say what happens at extremely high levels, as this kind of modeling would require the solution of differential equations. The cone excursion and port air speed graphs are implemented such that the plots are grayed out when the simulation becomes unrealistic.

14.3.1 Force factor Bl(x)

The nonlinear modeling in LspCAD is an iterative approach where the cone excursion is computed, and the average force factor is computed from this. This average force factor is simply computed as the integral between the cone excursion extremes, one can refer this to a kind of rectangular equivalent Bl.



Line 1: Bl(x) curve entered in the
Line 2 Average Bl 10mm$_{p-p}$ cone excursion
Line 3 Average Bl 12mm$_{p-p}$ cone excursion
Line 4 Average Bl 14mm$_{p-p}$ cone excursion
Line 5 Average Bl 20mm$_{p-p}$ cone excursion
Line 6 Average Bl 24mm$_{p-p}$ cone excursion

This simulation is done in an iterative manner, as changes in Bl will affect the behavior of the rest of the system. This will slow down computation speed quite dramatically, a fact that could be good to consider when one for instance runs the optimizer as it is a good practice to turn off nonlinear modeling when optimizing unless one has reasons to incorporate the effects of nonlinear modeling.

14.3.2 Compliance Cms(x)

The modeling of nonlinear Cms is done in the same way as for Bl. An exception is passive radiators where a simpler model is used where one specify an Xmax value where Cmp is assummed to gradually decrease to zero between xmax*0.5 and xmax*2.0.

14.3.3 Port nonlinearity

In the case of port nonlinearity the port loss is scaled according to the equation
Rpp = Rpp*(airspeed/15.3)^2

Where airspeed is the port air speed in m/s

15 References

[Beranek] “Acoustics”, Leo Beranek, ISBN 0-88318-494-X

[Klippel] http://www.klippel.de

[D’Appolito] Testing Loudspeakers ,Joseph D’Appolito, ISBN: 1-882580-17-6

[Dickason] The Loudspeaker Design Cookbook, Vance Dickason, ISBN: 0-9624-191-7-6

[Ritter, Agerkvist] Modelling Viscoelasticity of Loudspeaker Suspensions using Retardation Spectra, AES 128, paper #8217

[Johansson] Earendel loudspeaker system https://www.ijdata.com/Earendel.pdf

[Urban. M] The Distributed Edge Dipole (DED) Model for Cabinet Diffraction Effects* J. Audio Eng. Soc., Vol. 52, No. 10, 2004 October