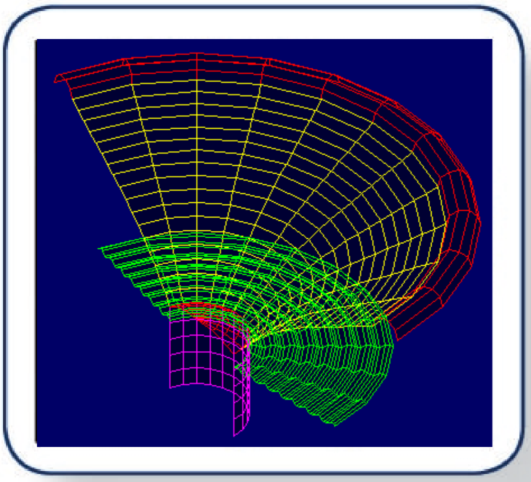
Manual



FINECone™

Acoustic Finite Element Dome/Cone Simulation Program

Contents

Overview

For a quick introduction to FINECone please refer to the demo video(s) where Peter Larsen introduces the various feature of FINECone and takes you through a design example (6m47s):

https://youtu.be/nlaGb67RPwc

https://www.youtube.com/watch?v=EJ7rDMYz9UE

1.1 Display

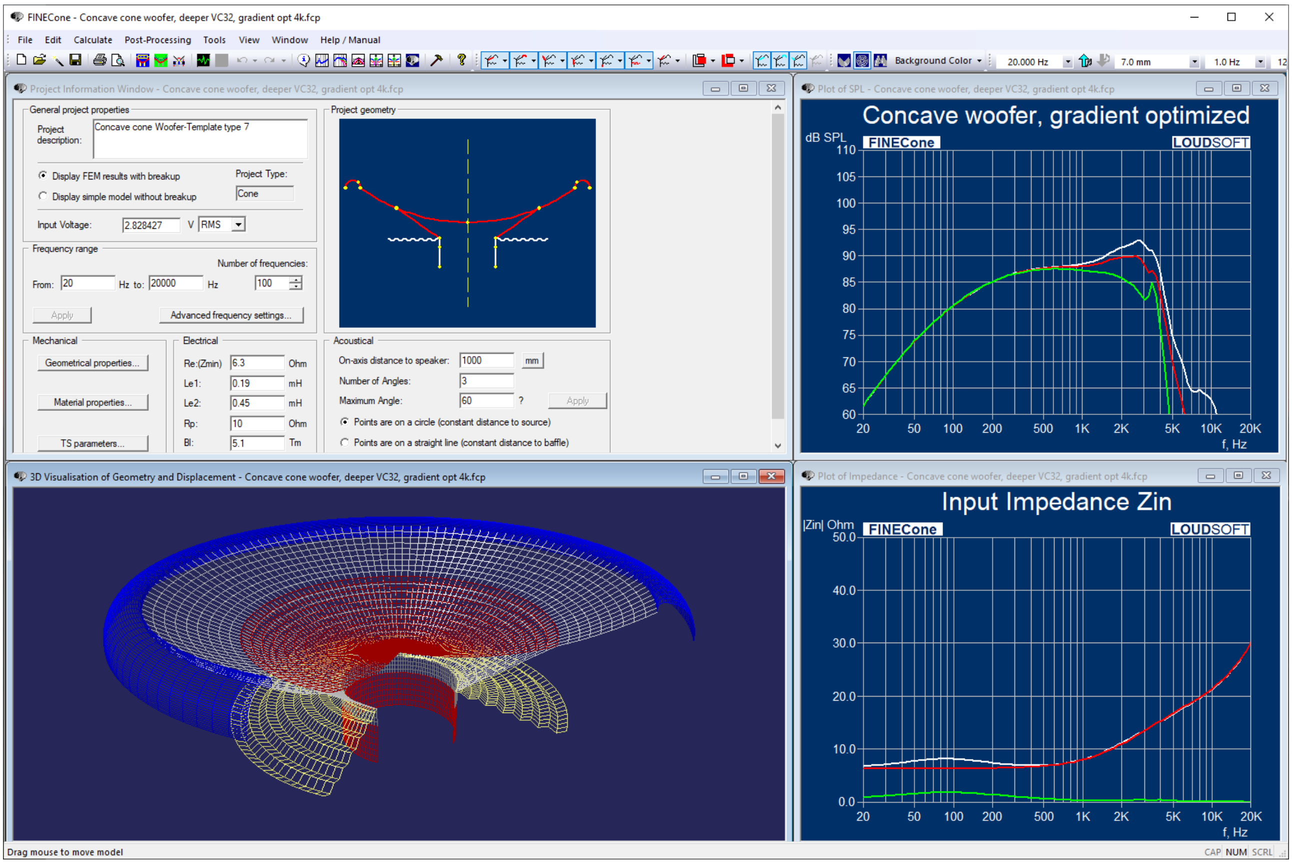


Figure 1 - Main Display

The main display Fig. 1 is here shown tiled (section 1.10.2). The blue background in the frequency response and impedance windows can be set by right clicking in one of those windows and selecting Plot Layout.

Then deselect the 3 squares and select Whole Page from the Plot Scale drop-down menu fig.3.

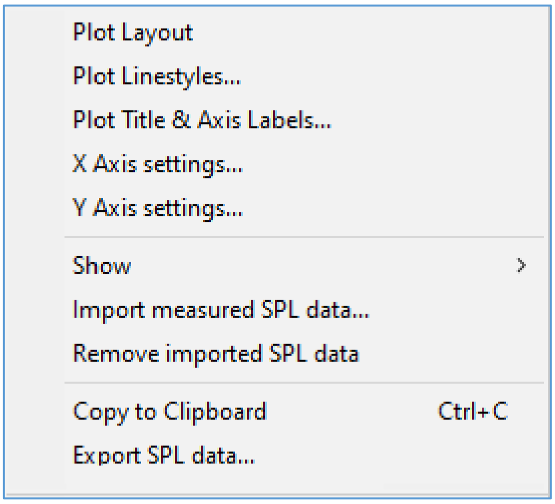


Figure 2 - Right Click menu (Freq window)

Export SPL data is in the standard FSIM format, which imports well into other FINE software. See also appendix

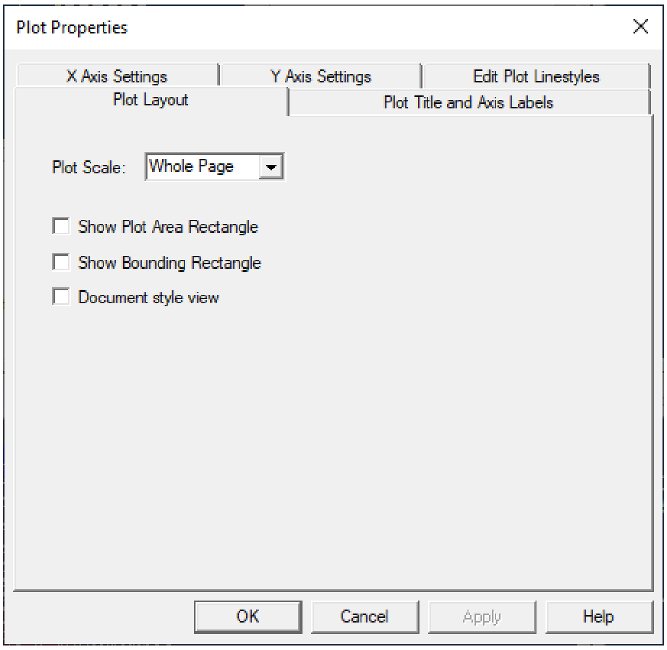


Figure 3 - Plot Layout for Blue background

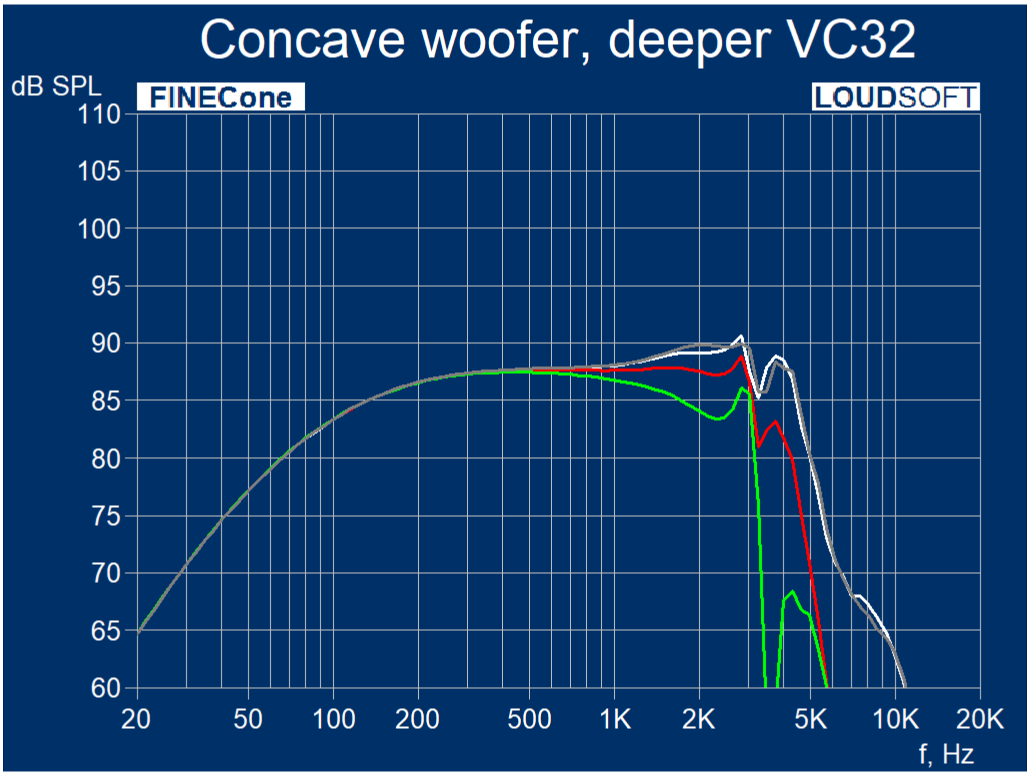


Figure 4 - Frequency response: Copy to Clipboard

1.2 Geometry Modeler

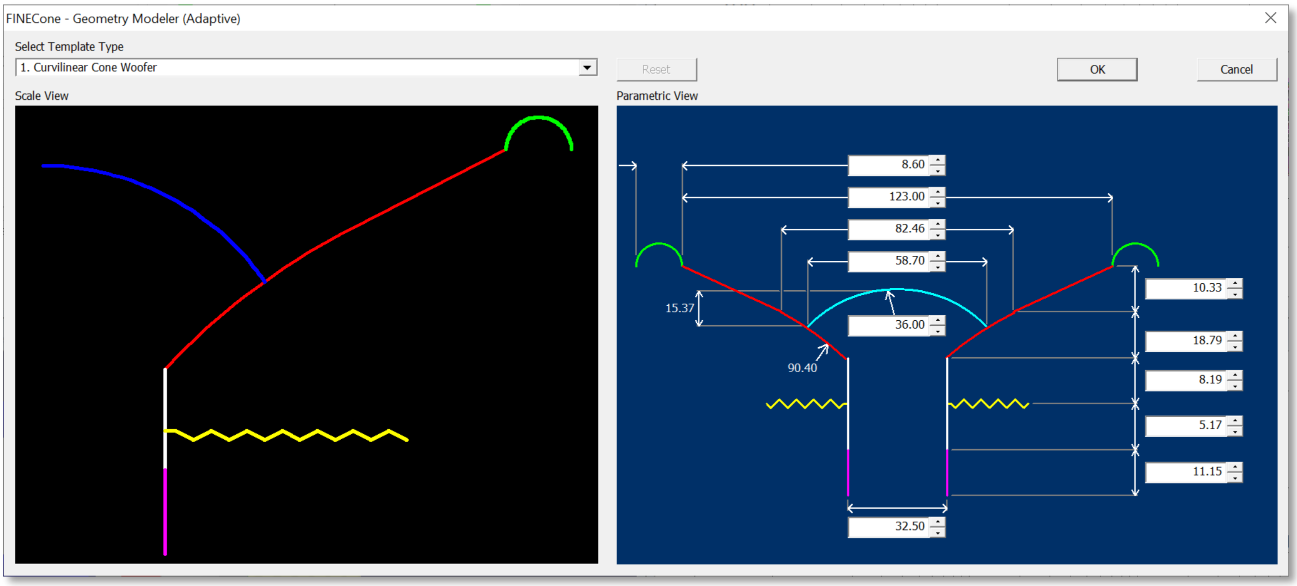


Figure 5 - Geometry Modeler

The **Geometry Modeler** (Tools/Geometry Modeler) makes it much easier to create the geometry for FINECone. From the drop-down menu at left in Fig.5 you simply select the template type, which is closest to the speaker geometry you want to simulate.

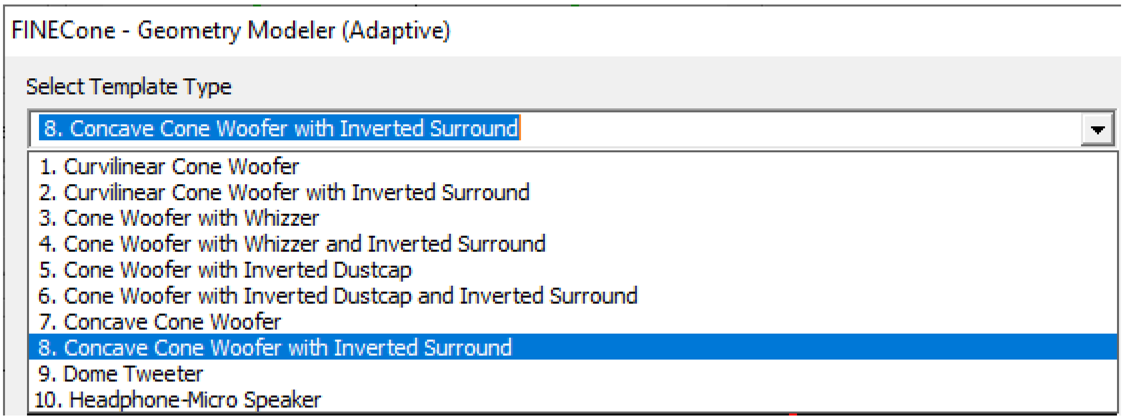


Figure 6 - Select Template from drop-down menu

Figure 5 shows the Curvilinear Cone Woofer template1, displaying the CAD file (DXF format) at left. You can just **change** all the **dimensions** to fit your design in the **right window**. You must save your chosen geometry as a DXF file.

The geometry DXF only defines the **geometry**. The material parameters like thickness, stiffness (E-module) and damping will be defined later.

Next the software automatically proposes a new project using your new geometry Figure 5. Be sure to press [Yes], as that will setup an **entire** project model with default materials and ranges etc.

THIS will save you much time getting started with FEM simulations!

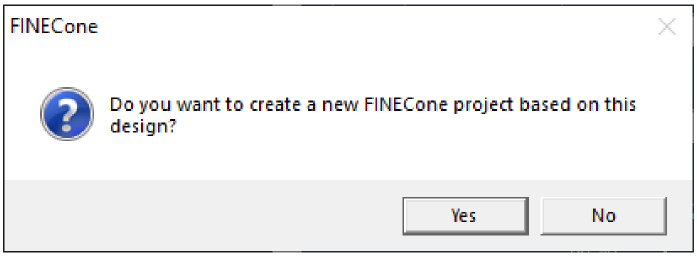


Figure 7 - Auto-creating your FINECone project

1.3 Geometry / Shape Optimizer / *New*

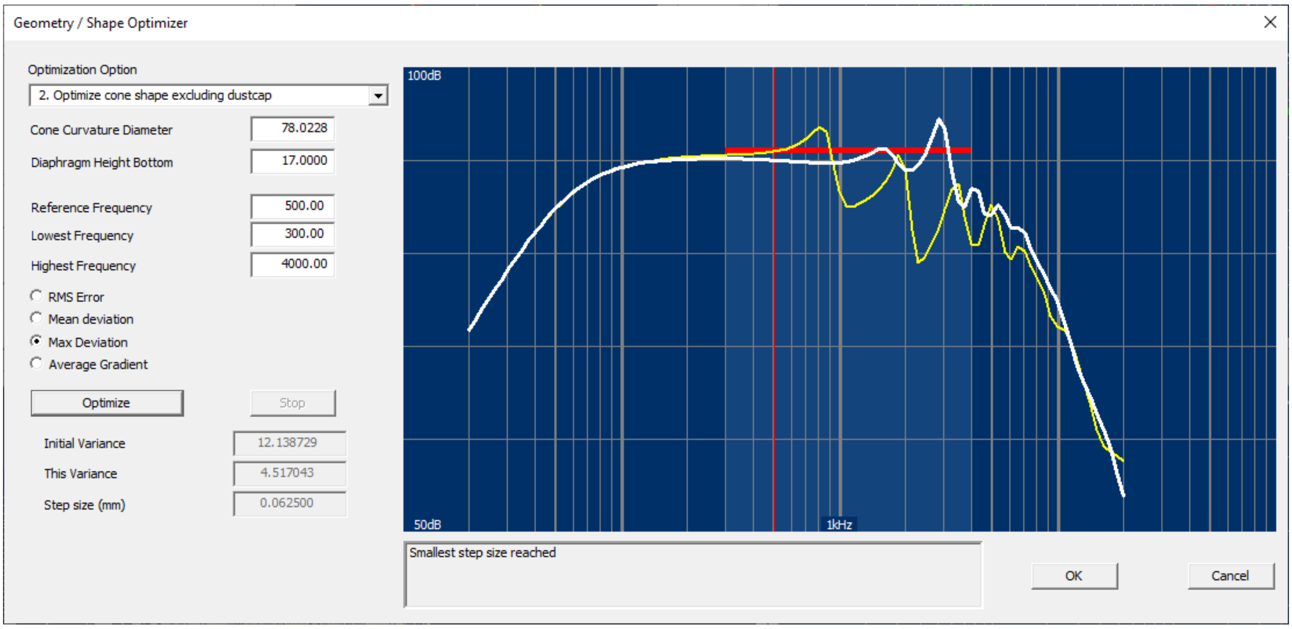


Figure 8 - Geometry / Shape Optimizer

The Geometry / Shape Optimizer (Tools/Geometry Shape Optimizer) is used to optimize cone and surround geometry automatically. Each optimize step takes less than 1 second, so FINECone can run 100's of iterations in minutes.

If the dust cap is significantly larger than the VC diameter, it is a good idea to optimize the cone + surround first.

When the main (outer) dimensions are set in the Geometry Modeler, you may start the Geometry / Shape Optimizer for finding the best cone geometry + surround thickness.

Before optimization the initial Variance is stated lower left in Fig. 8. The optimized variance is smaller. The step size of the geometry changes is also shown.

1.3.1 Optimization Options

It is recommended first to optimize the geometry, and then select the cone material because good geometry is more important than the material.

First use the Geometry Modeler to define the outer dimensions like OD and available cone height. It is recommended starting by optimizing the cone shape (Option 1 or 2) Fig.9. Following you should optimize the surround thickness.

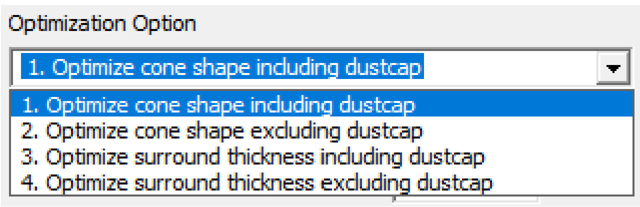


Figure 9 - Optimization Options

Before starting the optimization, you must set the red target line Fig. 10. Set the reference frequency somewhere in the mid-band (default is 500Hz). Next set the lowest frequency and the highest frequency indicated by the thick red line. Do not set the highest frequency too high. It may be better to flatten the mid-frequencies before being ambitious about the high frequencies.

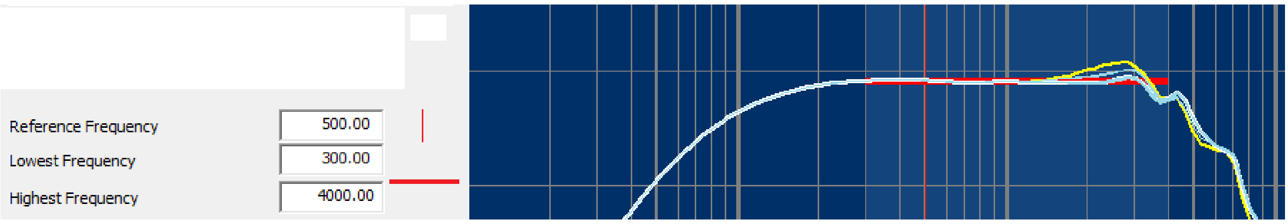


Figure 10 – Define Target response

- **See the Shape Optimization of a Concave Woofer example section 2.1 and**
- **Shape Optimization of a Shallow cone woofer section 2.2**

1.3.2 Optimization Modes

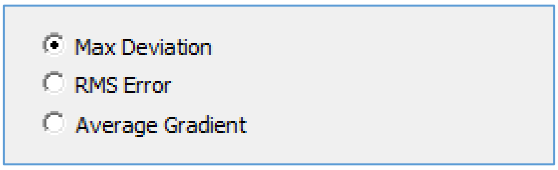


Figure 11 - Various Optimization modes

The default optimization mode is Max Deviation, which simply looks for peaks and dips in the frequency response.

The RMS is Root-Mean-Square, which is a standard method, known as the method, to calculate Harmonic Distortion.

Average Gradient is seeking to minimize the slope or gradient of the curve, meaning a smoother response, but not necessarily flat.

1.4 What is the purpose of **Simulation**?

In short simulation is a model of a speaker, which shows behaviour / break-up modes and responses of all moving components.

Simulation is used for:

1. Performance optimization of loudspeakers and components
2. Failure analysis
3. Frequency response dispersion / extension and SPL improvement
4. Cone/surround geometry optimization
5. Vibration analysis
6. Material evaluation and optimization
7. Individual frequency response of Cone, Surround and Dust cap / Dome
8. Cone shape surround thickness and dome experiments and much more.

You can do 100’s of simulations and optimizations in one day. Much faster that Trial and Error.

1.5 DXF geometry: How to change or make your own

If you said yes to automatic project creation, you don’t need DXF’s and may skip this section until later.

However, if you need to change a geometry or create a different geometry than given by the templates, here is how:

The left side Figure 12 shows a section view of a woofer. The same woofer is shown at right when prepared for FINECone analysis.

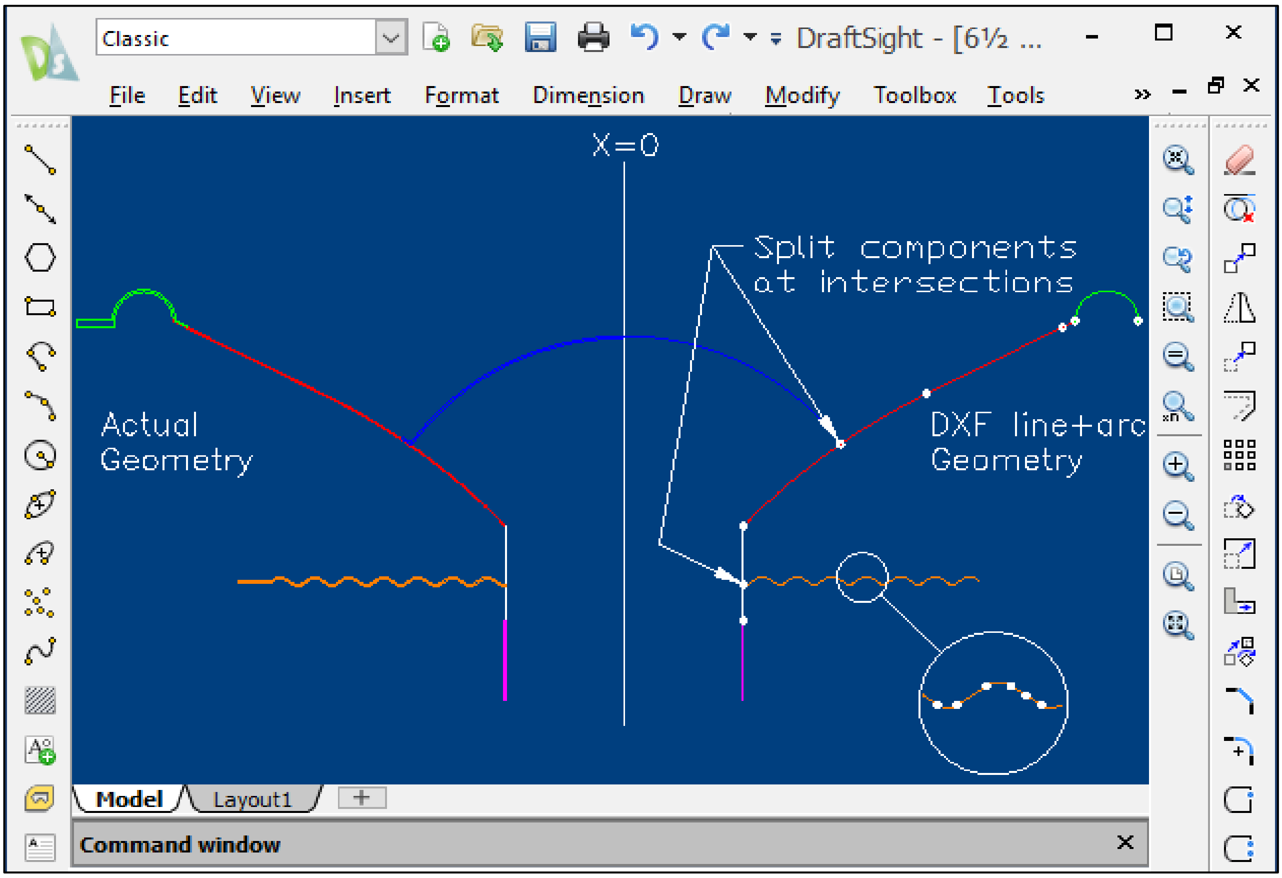


Figure 12 - DraftSight drawing - Left: Actual geometry - Right: DXF line + arc geometry

1) Changing an **existing** geometry:
    a. With a CAD program (DraftSight is recommended), open the file you want to change
    b. Read and observe the DXF Hints (section 1.51) and Figure 12
    c. Be sure to keep the changed component in its correct layer
    d. You can for example split a line in several (small) segments, which allows you to model a tapering cone thickness, or you can change radii or intersections.
    e. Glue can be modelled in a (small) segment by increasing mass and stiffness. Neck glue may be included in top of the VC former (Ex 6_5 Woofer Large Dust Cap)
    f. Note that surround and spider flanges are omitted, as they are not moving parts.
    g. DO NOT change the basic configuration in the DXF file, unless you have expert knowledge
2) Creating a **NEW** geometry.
    a. It is STRONGLY recommended starting from an existing DXF file (found in C:\Program Files (x86)\Loudsoft\FINECone\Project, because the layers etc. are already defined

b. Read and observe the DXF Hints (section 1.51) and Figure 12.

c. You can only define up to 7 components, see Figure 13.

Figure 12 shows an actual geometry at left. In the right side is shown how the corresponding DXF file should be made. Note that it must **only** use lines and arcs. We recommend DraftSight CAD software for making the DXF files, but most other CAD programs can be used.

The model must be axi-symmetric and only the right half is used. The symmetry axis is where X=0. Usually this is the midpoint of the dust cap.

Note that the surround and spider flanges should NOT be included in the DXF file, because these will not move, as they are glued to the chassis frame.

An example DXF file is shown in Figure 13. Each component like cone and dust cap etc. are placed in their **own layer**.

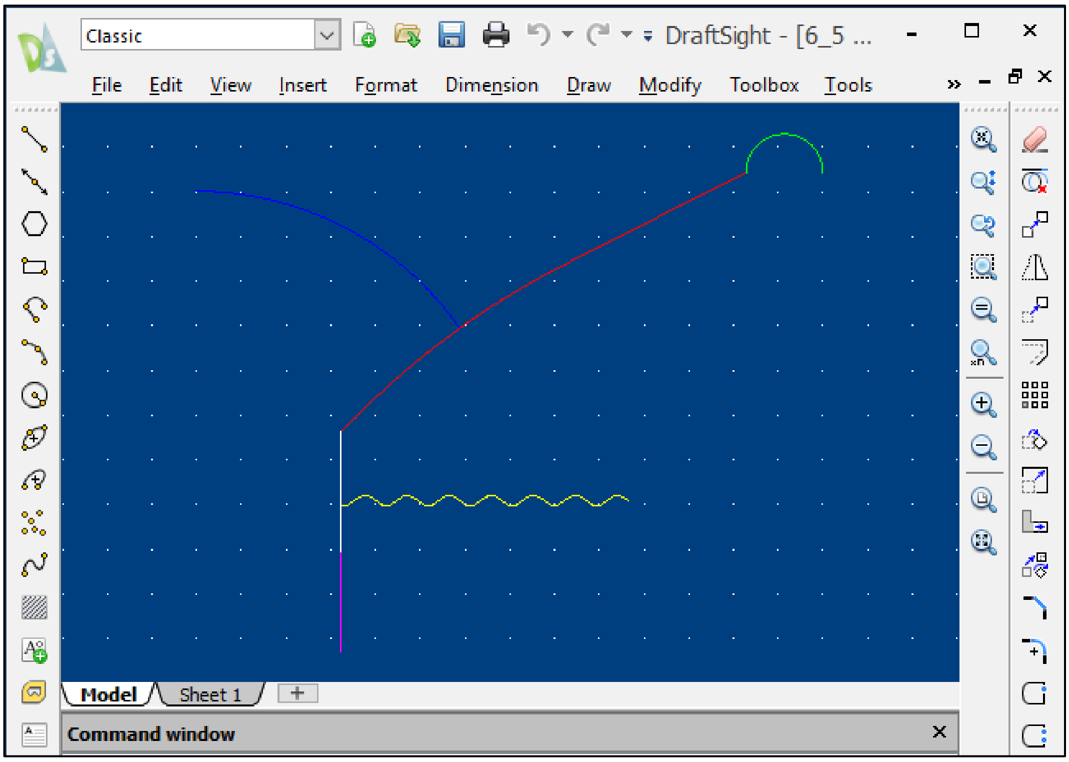


Figure 13 - FINECone reference DXF in DraftSight

1.5.1 DXF hints

1. Save as (AutoCAD) v12 DXF-format (ASCII) or use the generic DXF format (oldest).
2. DXF MUST be high accuracy: Better than 8 decimal places, 16 would be best.
3. ONLY normal LINES and ARCS are allowed. No double, poly-lines or blocks etc.
4. ALL intersections must meet in ONE point.

5. Use Snap in CAD. Example – Line from (x,y)10,21 to start of another arc: line 10,21(point to start on next arc) END (will indicate a square to lock to the ENDPoint)
6. Lines and Arcs to be broken at meeting points (=Intersections)
7. Place each component in its own layer: Cone, Surround, Dust-cap, VC former, VC and spider. The easiest way is to start with, for example the 6_5 Woofer Large Dust Cap.dxf example file and then modify this drawing. In that way the layers are already created.
8. No THICKNESS or DIMENSIONS, or text at all in DXF drawing. Thickness is only set in FEM Material properties, see Figure 47.
9. Use PURGE in AutoCAD to ensure removal of unwanted items.
10. Use Accurate solutions (and NOT FAST) to avoid errors when components do not move together in 3D animation. The settings are found in Tools/Program Options/Calculations.

The safest and easier way is to modify one of the FINECone example DXF files. In this way, the names of all the layers are defined by default settings and they will import easily into FINECone.

Note that we cannot have double lines in the FINECone DXF file and the DXF file is therefore a simplified representation, especially regarding the VC former in Figure 13. The voice coil is only one line for the winding (magenta) with the voice coil former starting as another line (white) up to the point where the spider is attached and one more line up to the cone.

Likewise, the dust cap and the flange, which are overlapping the VC former, are modelled as an arc + ONE line. (It is possible to model these small segments, see later. But you can get very far with a simplified model)

The same applies for the surround flange glued to the cone which is not included in the simple DXF drawing. The cone is thus drawn as one line (red) in the Diaphragm layer + two arcs. The full geometry is shown in Figure 15, where the split points are shown as small dots.

Remember that the thickness of the components is NOT set in the DXF file but later in Material Properties.

Pressing [Geometrical Properties] Figure 15 shows the input geometry from the chosen DXF file. Each component is shown with a different colour and here are all components imported directly. (Remember all layers will import when using the default layer names, see Figure 14)

IMPORTANT: Remember to split the cone in two or more segments where the dust cap is attached. This also applies where the spider is attached to the former and similar situations.

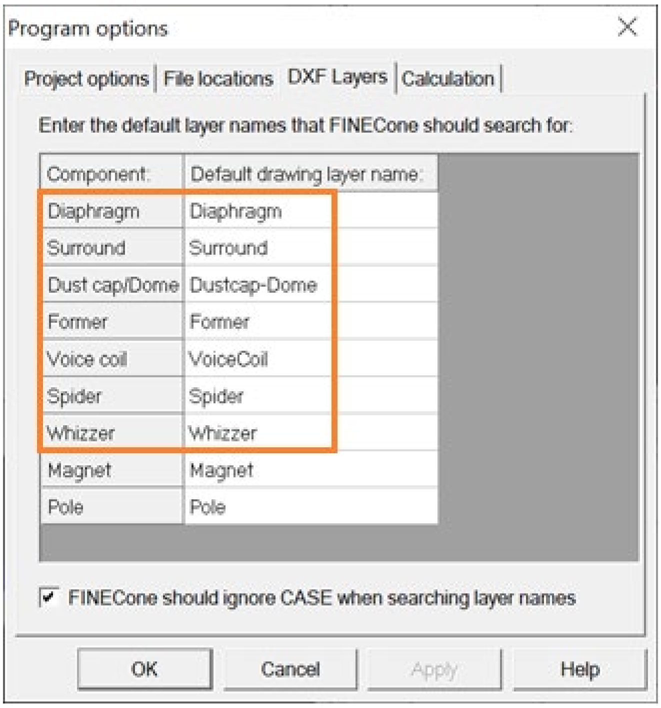


Figure 14 - DXF: Default Layer names (1-7 Main Acoustic components (in frame))

The default layer names are defined in Tools/Program options/DXF layers, Figure 14

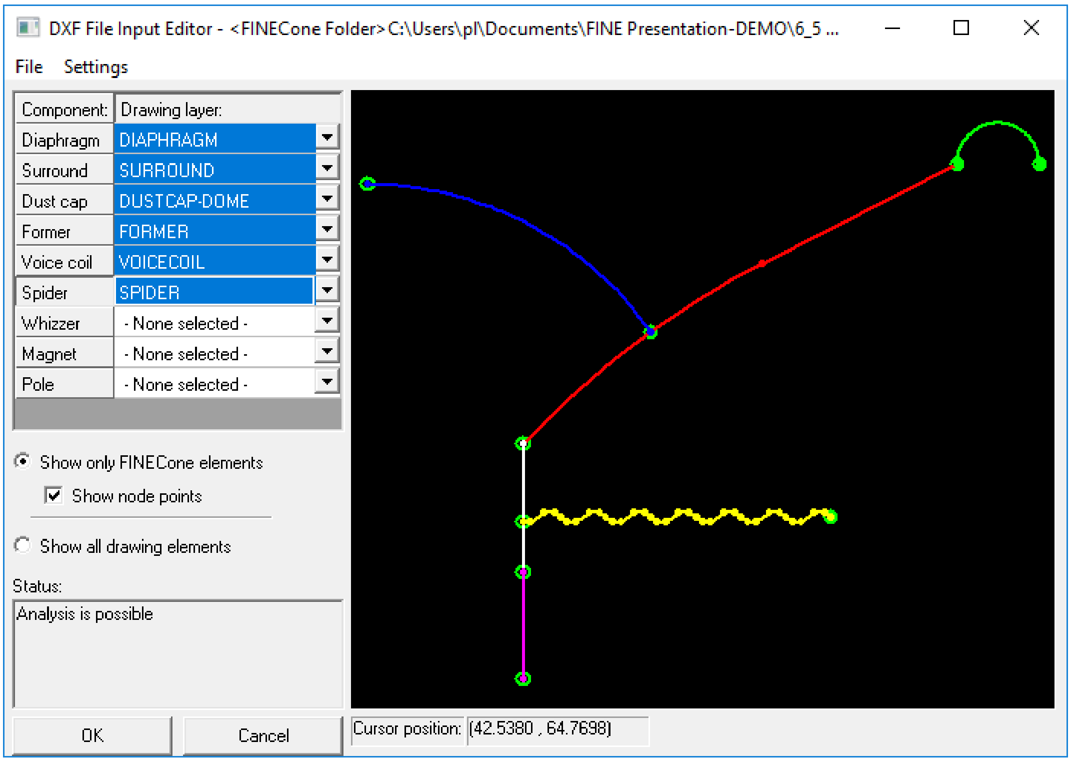


Figure 15 – Example FINECone Geometry

1.6 Project information window

1.6.1 Geometrical properties

Figure 16 shows the geometrical properties window where you can select which parts of the DXF belongs to which component of the driver. It is important to check that there are no red circles indicating errors in the connections. Also, be sure to check that all the different components are selected in the drop-down menus on the left.

When you get the message “Analysis is possible” in the Status window in the lower left the model is ready for FEM modelling.

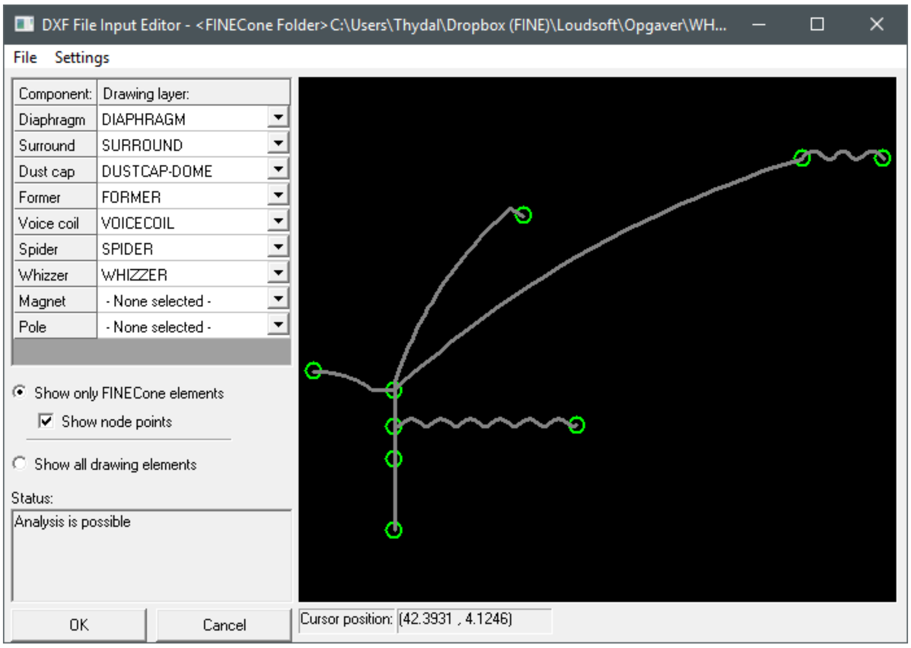


Figure 16 - The geometrical properties of a 6.5-inch woofer with a whizzer cone

1.6.2 Material properties

The Material Properties Window Figure 17 allows you to choose different materials from a list of predefined materials or input the user’s own values. The properties can be defined for the individual segments of the component or for multiple segments at a time by selecting multiple segments. This can be done by holding ‘Ctrl’ while clicking on the different segment numbers. Setting individual properties for the different segments allows for simulating a glue joint or tapering cone geometry.

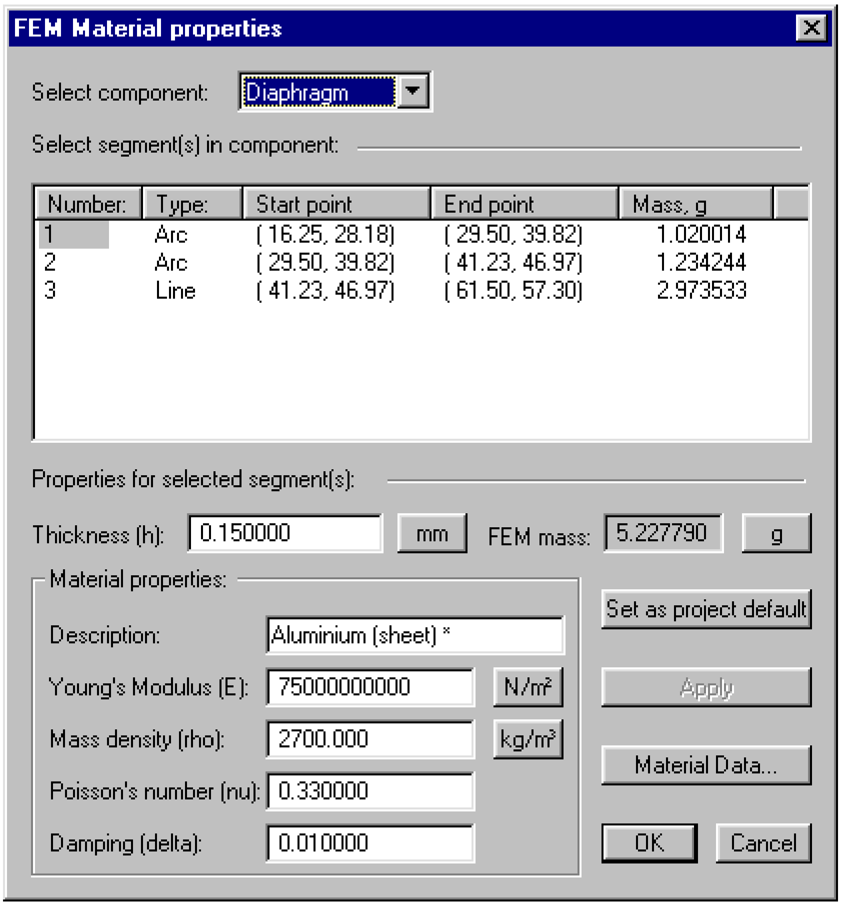


Figure 17 - Material properties can be defined for each individual segment in each component

The user can specify the following properties:

- Thickness (h)
- Young's Modulus (E) – The stiffness of the material in MPa or N/m^2
- Mass density (rho) – Defines the density of the material in kg/m^3
- Poisson's ratio (mu) – A measure of the compressibility of the material. Use the default value of 0.33 if the actual value is unknown.
- Damping (delta) – A factor specifying the internal damping (loss) of a material. Maximum damping is normally 1.00.

Note: The damping model used in FINECone is frequency dependent (automatic), which ensures very good results compared to actual measurements.

1.6.3 Material database

As mentioned earlier it is possible to select predefined materials from a material database. This database can be accessed by clicking the button “Material Data…” in the Material Properties window, see Figure 9. This opens a window with a list of many standard materials that can be used for the initial design. Some of the materials are supplied by Dr. Kurt Mueller (www.kurtmueller.com) and have a DKM in their description.

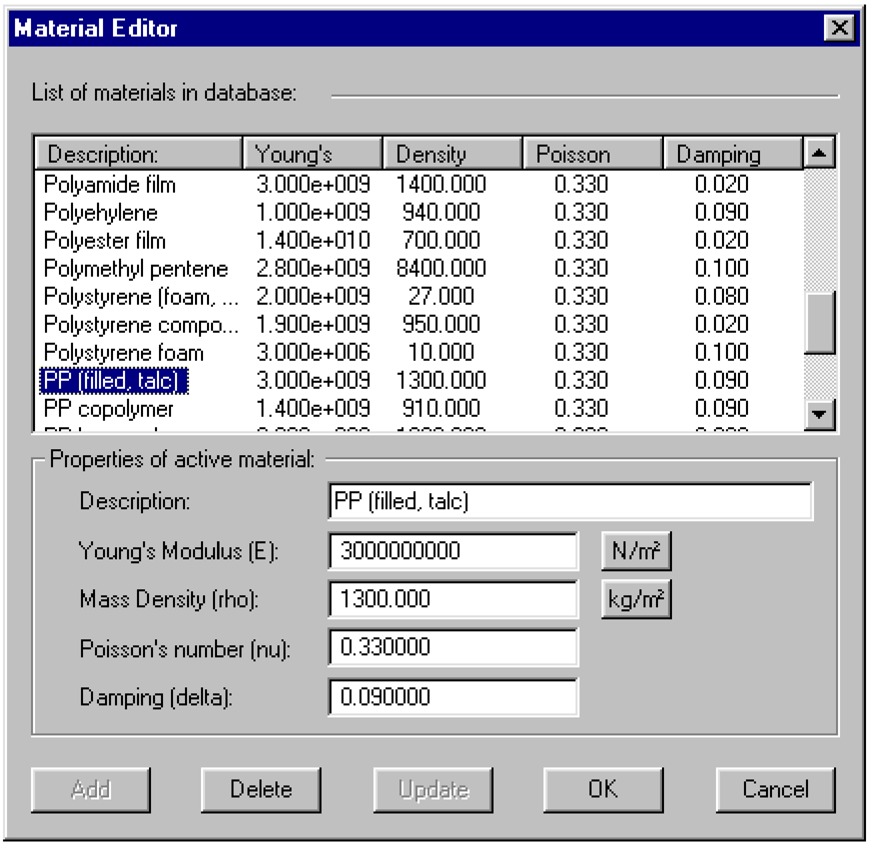


Figure 18 - Material database with several predefined, common materials

The user can also add new materials by typing the name of the material in the Description field, defining the material properties and then clicking the Add-button in the lower left corner of the window.

1.6.4 Display (simple) model without break-up

Pressing this button (Project Information window) bypasses the FEM calculation, and shows the components modelled as simple and ideal masses and compliances (inverse stiffness). This mode is normally only used to quickly simulate a response without break-up. The ‘TS Parameters can be used to extract the Thiele-Small parameters from the FEM model, see next.

1.6.5 TS Parameters (Lumped elements)

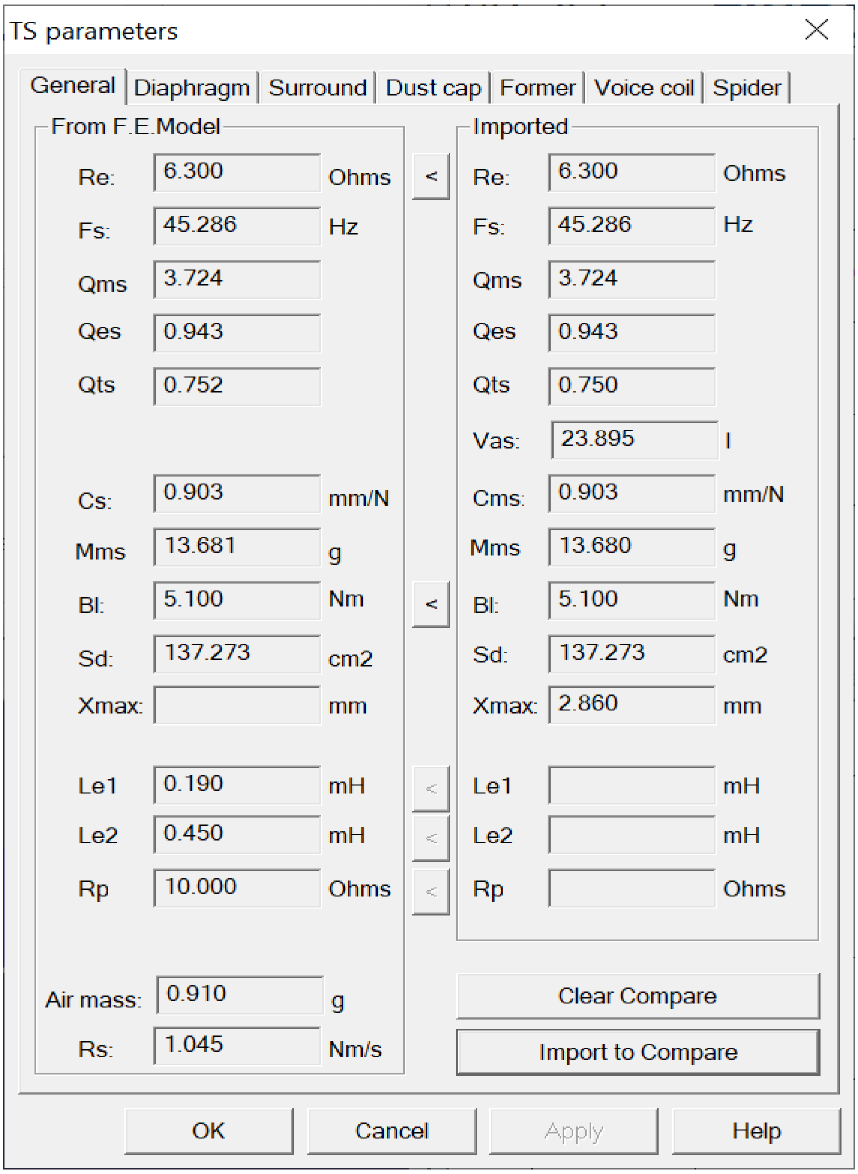


Figure 19 - Thiele-Small (TS) parameters. The right column is imported from FINEMotor or FINE R+D.

The FINECone FEM (Finite Element Model) is very advanced and detailed. In contrast the TS Parameters are only using simplified lumped elements. Therefore, the TS parameter button in Fig. 19. will show the actual TS parameters **extracted** from the FEM model.

For comparison you may import TS parameters as FM3 from FINEMotor, or as TXT from FINE R+D or other programs.

You can insert Re, BL, le1, le2 and Rp in FEM calculations by pressing < for each parameter.

Note: The FEM model often uses Zmin instead of Re, as that matches the impedance curve best. You may therefore use the low DCR for Re (instead of Zmin) when comparing to other TS parameters.

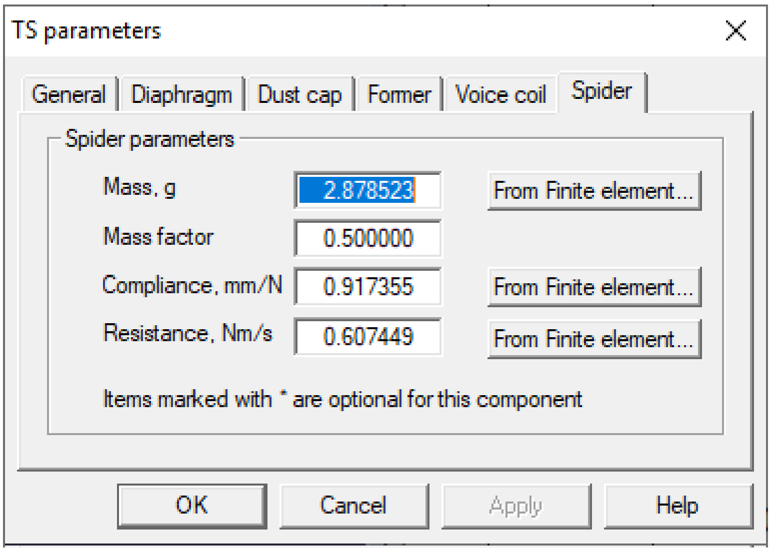


Figure 20 - Spider pars calculated, with surround excluded. Spider stiffness ~700MPa ~1mm/N

If you need to calculate the actual spider stiffness, then select the Spider tab in TS parameters Figure 20. HOWEVER, you must first exclude the surround, see Figure 21. (This is to avoid having both spider and surround stiffness in parallel). A spider stiffness ~700MPa is close to 1mm/N.

Similarly, you may exclude the spider for calculating the surround stiffness.

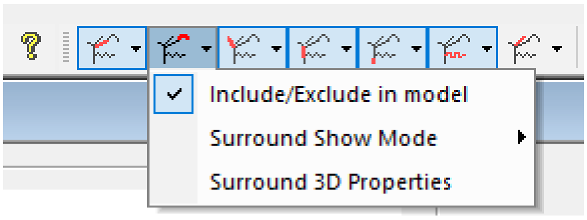


Figure 21 - Exclude Surround

1.6.6 Electrical properties

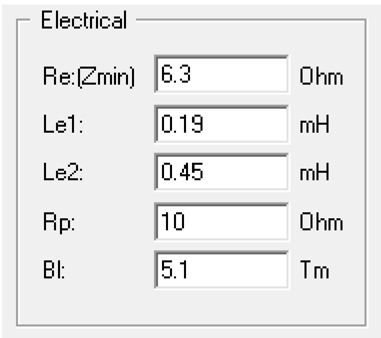


Figure 22 - The electrical parameters with Re and Bl imported (from FINEMotor)

In the Electrical Properties window Figure 22, the user can input basic loudspeaker parameters Re and BL. These can also be inserted from TS parameters, see section 1.6.5, and may include Le1, Le2 and Rp for simulating the loudspeaker impedance.

Note: The FEM model often uses Zmin instead of Re, as that matches the impedance curve better

- Re – The DC resistance of the voice coil. This can also be set Z_{min} (minimum impedance over Fs) to better simulate the actual impedance and thereby the actual SPL.
- Le1 – A serial inductor emulating part of the voice coil inductance.
- Le2 – A second serial inductor paralleled with a resistor Rp.
- Rp – A resistor emulating part of the voice coil resistance.
- Bl – The force factor of the motor system.

For comparison, the user can also choose to import a measured impedance curve by right clicking the Impedance curve, and selecting: [Import measured Impedance], Figure 23.

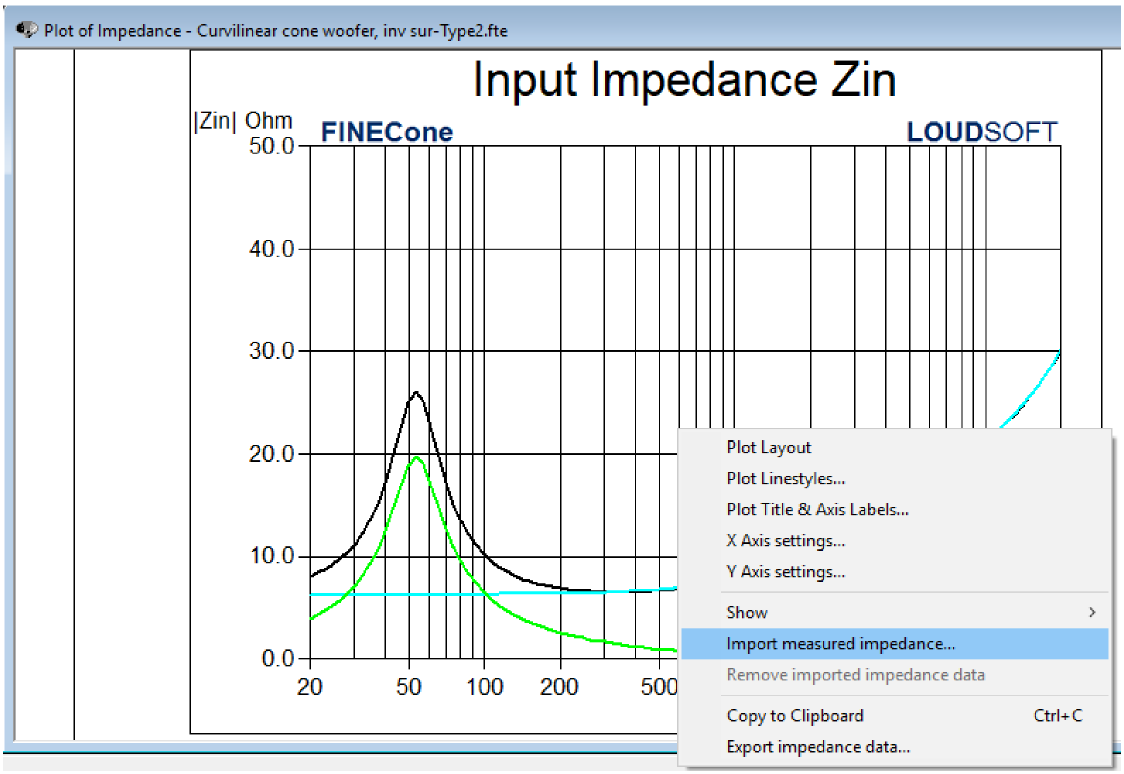


Figure 23 - Import measured Impedance curve

Note: It is important to match the simulated impedance curve with the measured impedance curve reasonably well to get a simulated frequency response that matches the real world.

1.6.7 Frequency Range

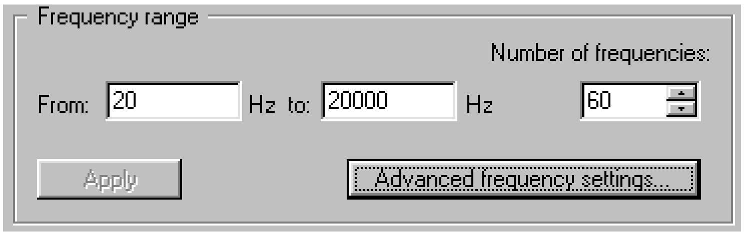


Figure 24 - The frequency range and how many sections the frequency range is divided into

The default frequency range is from 20 to 20000 Hz. The calculated frequency points are determined by the number of frequencies Figure 24. These are by default logarithmically spaced over the selected frequency band. Use 100 or more points to obtain a detailed response.

(Alternatively, you may select the Option "Fast Solution of Differential Equations" in Tools/Options/Calculation if using a slow PC. The accuracy is still quite good except for the highest frequencies).

The frequency range can be extended to start at a few Hz and extending beyond 100 kHz for ultra-sonic simulations.

The advanced frequency settings allow you to select all kinds of linear and logarithmic ranges Figure 25.

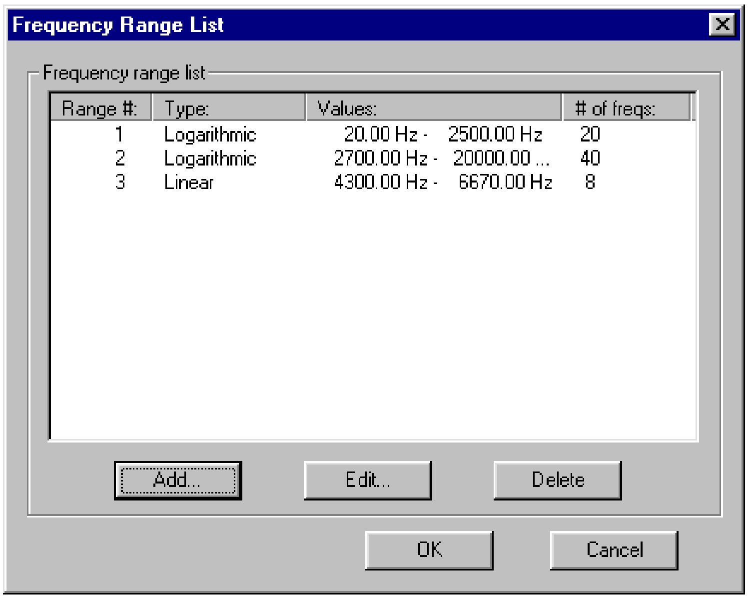


Figure 25 - List of pre-set frequency ranges

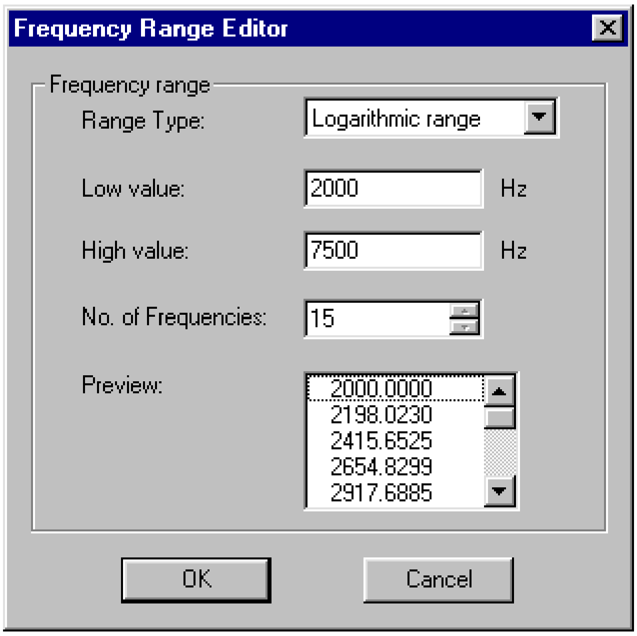


Figure 26 - Window allowing you to pre-set new frequency ranges

1.7 Impedance window

The impedance window in Figure 26 shows three simulated curves. The green curve represents the mechanical impedance of the driver, which is mostly dominated by the total suspension. The blue curve represents the electrical impedance of the driver, which is determined by attributes of the voice coil. The black curve is the summation of the green and the teal curve, and it represents the total impedance of the system.

Having the mechanical and electrical impedance separated allows for easier tuning of the some of the parameters when matching the simulated impedance with the measured. In FINECone it is possible to import a measured impedance curve into the impedance window and use that as reference when determining the parameters of the simulated driver. This is done by right-clicking on the impedance window and selecting 'Import measured impedance', see also section 1.6.6.

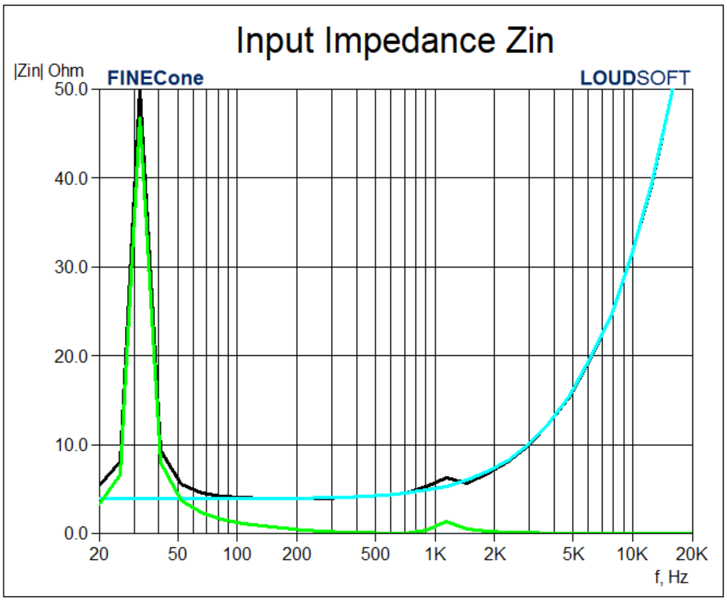


Figure 27 - Impedance window showing the impedance of a 10-inch subwoofer

Right-clicking on the impedance window also brings up the option to change the plot layout and axis settings to better fit what the user wants to see.

1.8 Sound Pressure Level (SPL) window

The Sound Pressure Level window shown in Figure 28 includes 4 curves where 3 are simulated and one is a measured response imported into FINECone. The three simulated curves include the on-axis and two off-axis responses. To import a measured curve or change the plot layout and/or axis settings you can right click on the window and select from the drop-down menu.

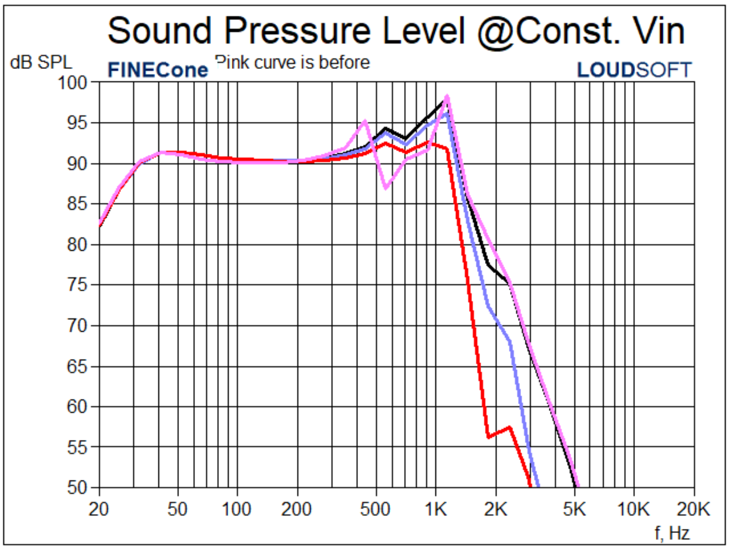


Figure 28 - The Sound Pressure Level window shows on-axis as well as two off-axis simulated curves of a subwoofer. A measured frequency response (Pink) has been imported by right-click.

1.9 Animation window / 3D Geometry + Displacement

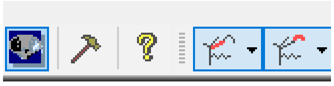


Figure 29 – Select 3D Geometry + Displacement

The animation window (3D Geometry + Displacement) Figure 30 is useful to visualise the behaviour of the parts of the loudspeaker at different frequencies. This can help pinpoint what component is breaking up and causing fluctuations in the frequency response.

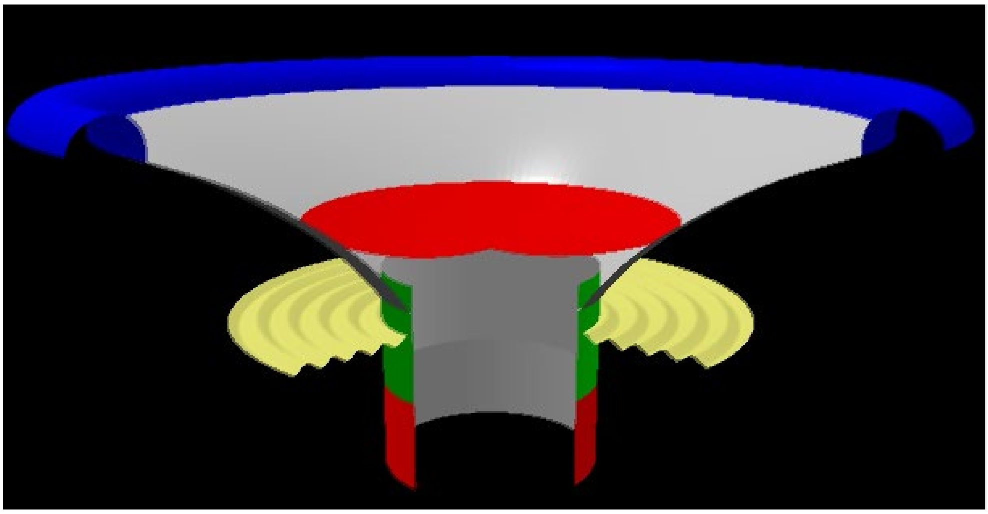


Figure 30 - The 3D animation window visualises the behaviour of the cone at different frequencies. Press to animate.

The button at right in Figure 29 starts and stops the animation. The first drop-down menu allows you to select the frequency you wish to examine. You can also increase or decrease the frequency using the arrows to the right of the drop-down menu. The second drop down menu from the left indicates the excursion of the voice coil. Exaggerating this excursion can help locate break ups at higher frequencies where the excursion normally is quite limited. The third drop down menu from the left is the animation speed. The last drop-down menu from the left is the number of frames calculated per oscillation. Higher numbers give a smother animation but is more demanding of the computer.

The buttons are only click-able when the animation window is selected.

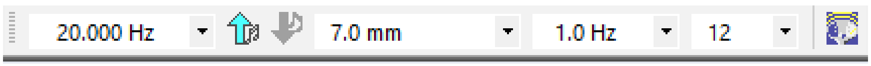


Figure 31 - Drop down menus for the animation window

The first button from the left in Figure 32 opens a window where you can select how many sections you wish to divide the model into and how many of these sections you wish displayed, i.e. to get a cross-sectional view instead of seeing the whole model. The second button from the left toggles the mesh of the model on and off. The third button from the left allows you to zoom and move the model around in the animation window Figure 33.

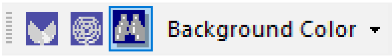


Figure 32 - Buttons for displaying the animation

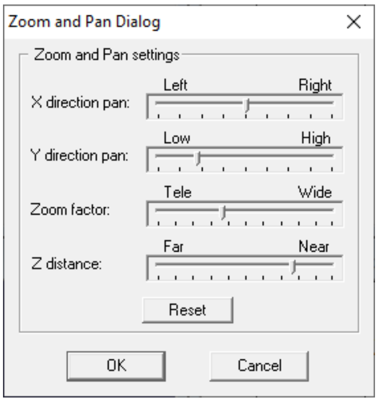


Figure 33 - Zoom and Pan settings

1.10 Toolbar

1.10.1 Import from FINEMotor

It is possible to import the Thiele-Small parameters from FINEMotor by clicking on , see also 1.6.6. This is highly useful when simulating a new driver which is not available physically. The magnet and pole properties can then be edited using these two buttons . However these are only for display.

1.10.2 Window selection

The different windows available in FINECone can be toggled on and off using the buttons shown in Figure 34. The first button from the left is the impedance window, the second button is the frequency response, the third is the directivity plot, the fourth is the 2D plot without displacement, the fifth is the 2D plot with displacement and the last button opens the animation window.

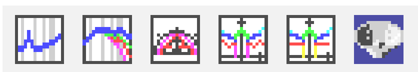


Figure 34 - The six buttons for the different windows in FINECone

1.10.3 Include/exclude mechanical parts

It is possible to include or exclude different parts of the model using the buttons shown in Figure 35. This can be useful if you for instance have measured the loudspeaker with the surround attached compared to another sample with the surround cut off. This is very useful to find the compliance of the spider (see also figure 21). Or if you have two identical drivers with and without a whizzer cone and wish to see the difference.

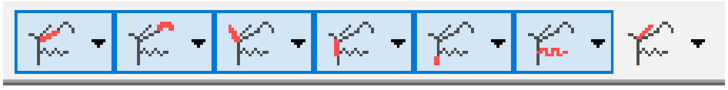


Figure 35 - Include or exclude different parts of the model

It is also possible to keep all the components included in the model, and then instead only exclude the SPL from each component. This can be extremely useful when trying to investigate the influence of the various parts of the driver, i.e. the contribution of the surround to the overall SPL. This can be done by using the buttons shown in Figure 36.

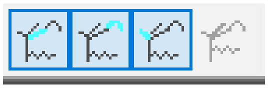


Figure 36 - Include or exclude the SPL contribution of different parts of the model in the frequency response

1.10.4 2D plot

FINECone can also show 2D plots of the model with and without displacement Figure 37. This method may be preferred to the 3D animation as it can make it easier to locate the breakup.

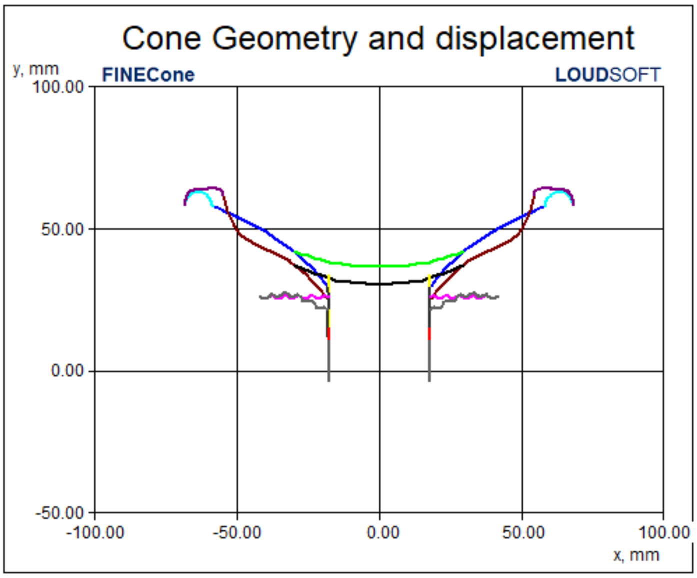


Figure 37 - 2D plot showing the model when at rest and when driven at 1863 Hz

1.8 How to calculate data for a NEW cone material

(Experimental Determination of Unknown Material Properties)

Here is described a method to approximate the mechanical properties of a new/unknown cone material. Other components may be done likewise.

Start by selecting a similar material from the database. Then input the actual thickness (of all cone segments). Now press Apply, and the FEM mass is calculated.

If the calculated mass is lower than the actual mass, then increase the Mass density (rho) until you get close to the actual mass (see blue arrow in Fig. 38).

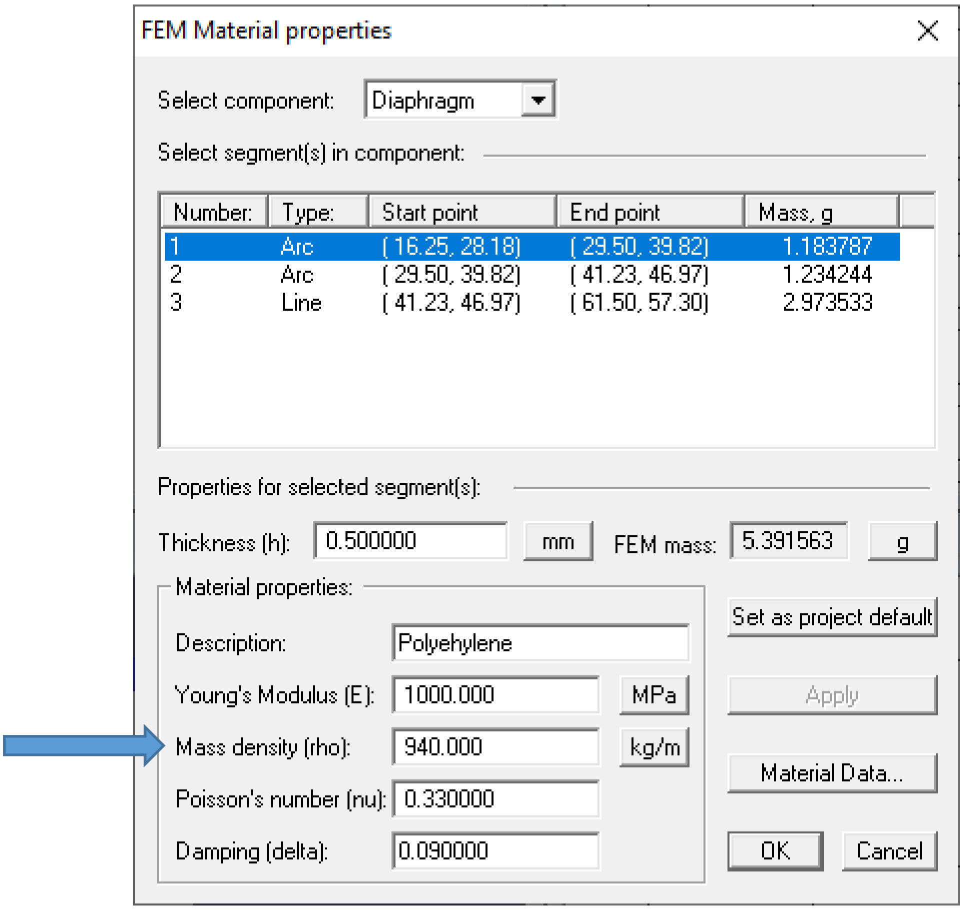


Figure 38 - Adjust Material specs for NEW material

Now OK and let FINECone calculate the frequency response. You may focus on the cone response by excluding dust cap and surround by these buttons. . Fig. 39 shows a typical response as the first try. The peak is lower in frequency compared to an imported measured curve (green). (Right-click on frequency response graph and select import)

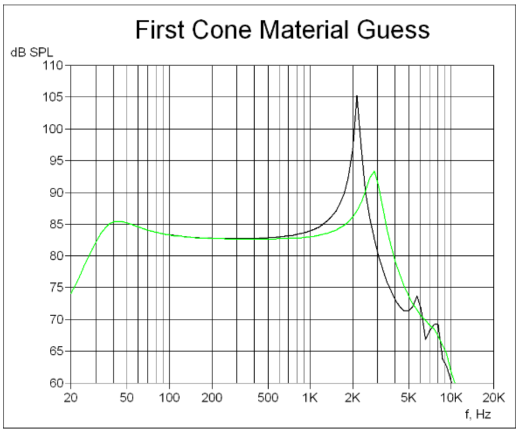


Figure 39 - 1st guess. Peak too low in frequency (green is measured response)

Since the peak is lower in frequency compared to the measured, we need to increase the stiffness in FINECone. That is done by increasing the value of Young's Modulus (E) (just above Mass density). Fig. 40 shows the peaks are now aligned when Young's Modulus was doubled.

In Fig. 41, damping was increased to match the peak of the measured. Save this in Material Data.

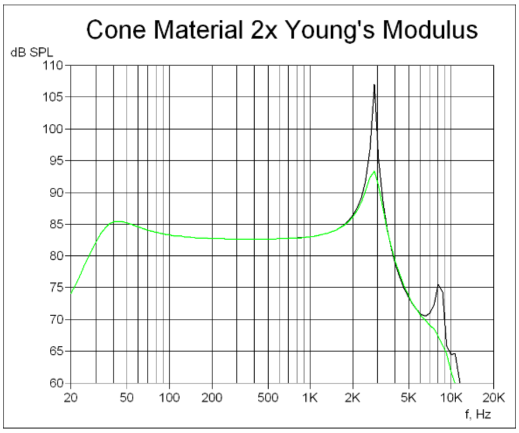


Figure 40 - 2nd guess. Peak now at the right freq.

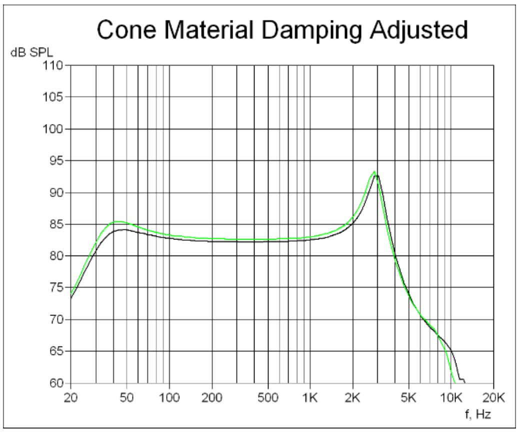


Figure 41 - Last approximation. Damping finally increased

2. Design Examples

2.1 Geometry / Shape Optimization of a Concave Woofer

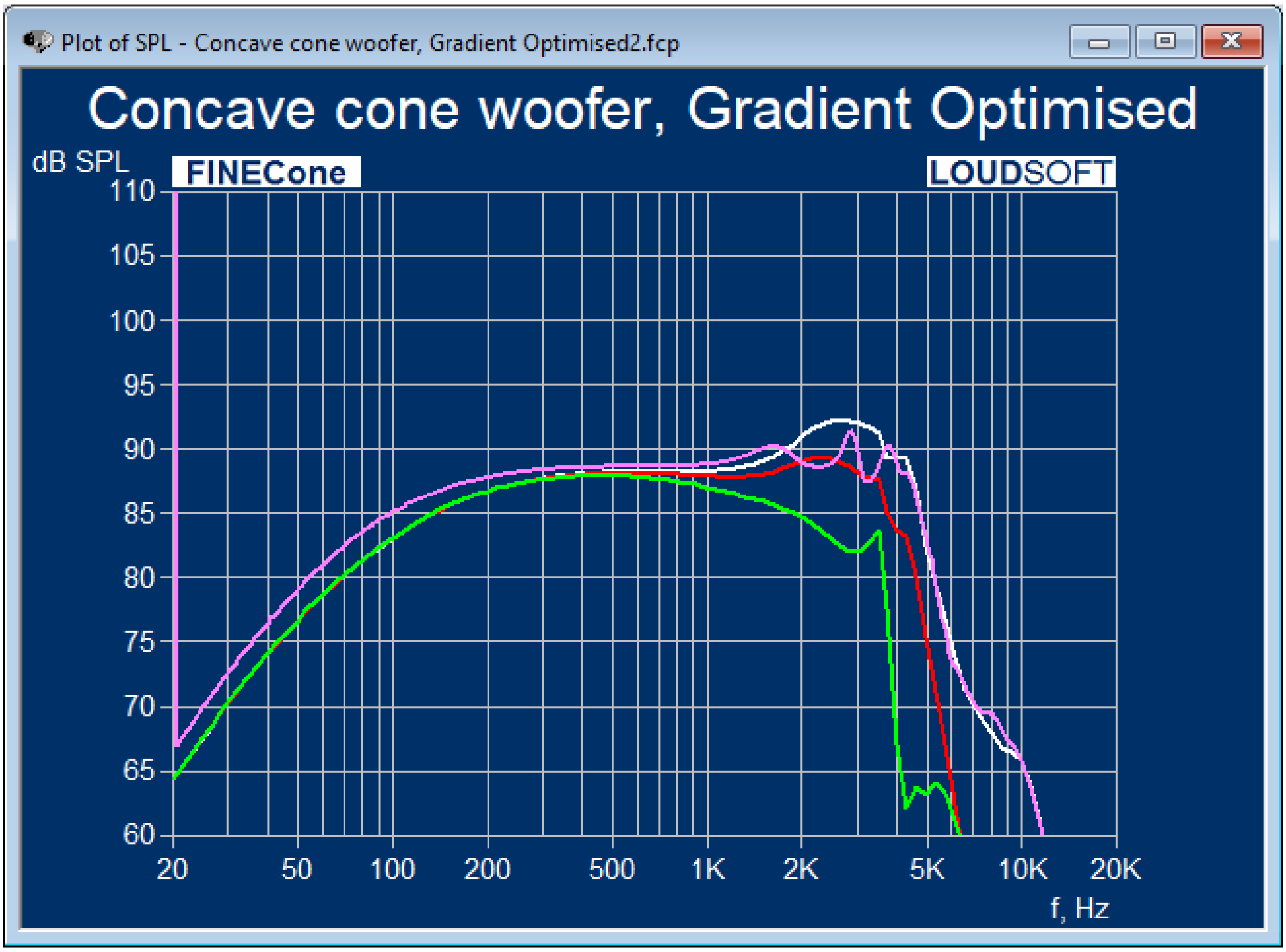


Figure 42 - Concave cone Woofer, Gradient Optimized

Fig. 42 shows the Shape optimized response of a 6.5in woofer with a concave cone and dust cap. Both have the same radius and forms a full concave speaker. This speaker configuration is known to have a large peak at mid frequencies.

The white curve was optimized with the Average Gradient option. That response is smoother, but not as flat as the other (pink) curve, which was optimized using the RMS option. However, the white curve is slowly rising without large peaks and dips and may therefore be easier to handle in a simple crossover circuit.

The cone material was standard N-paper (DKM) plus a normal rubber surround. After optimisation you may select other cone and surround materials.

The final optimized geometry of the concave cone woofer is shown in Fig. 43. Note the VC former (white) is not vertical, but instead changing from 25.4 to 88.59mm after optimization.

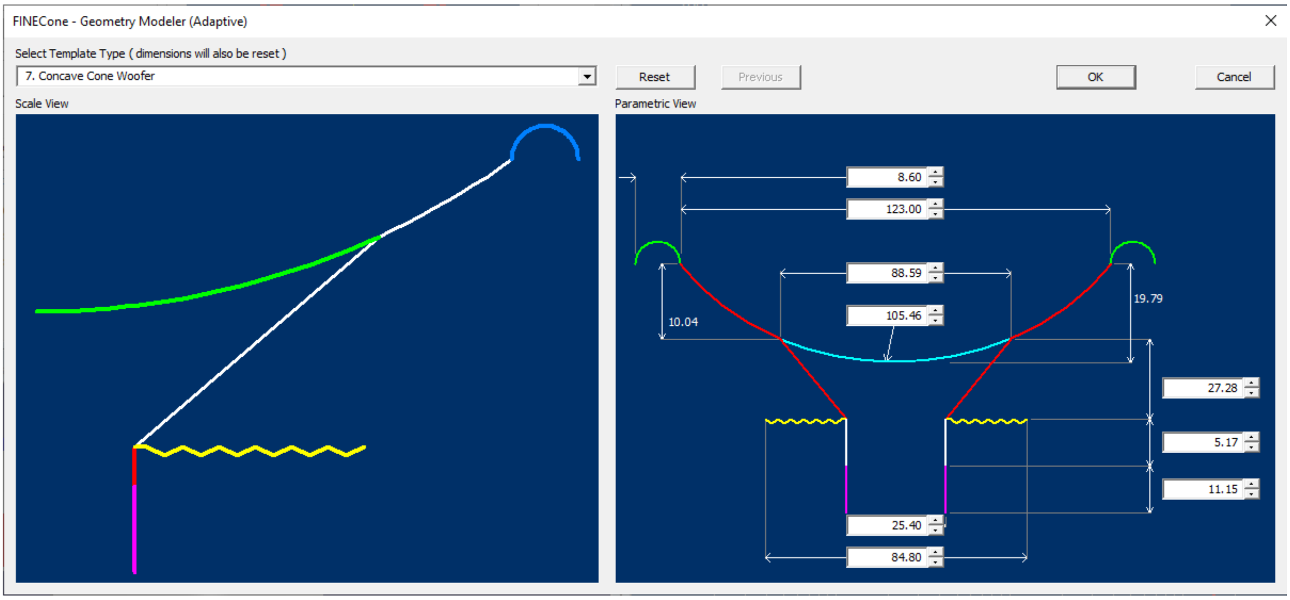


Figure 43 - Geometry of Optimized Concave Woofer

2.2 Shape Optimization of a Shallow cone woofer

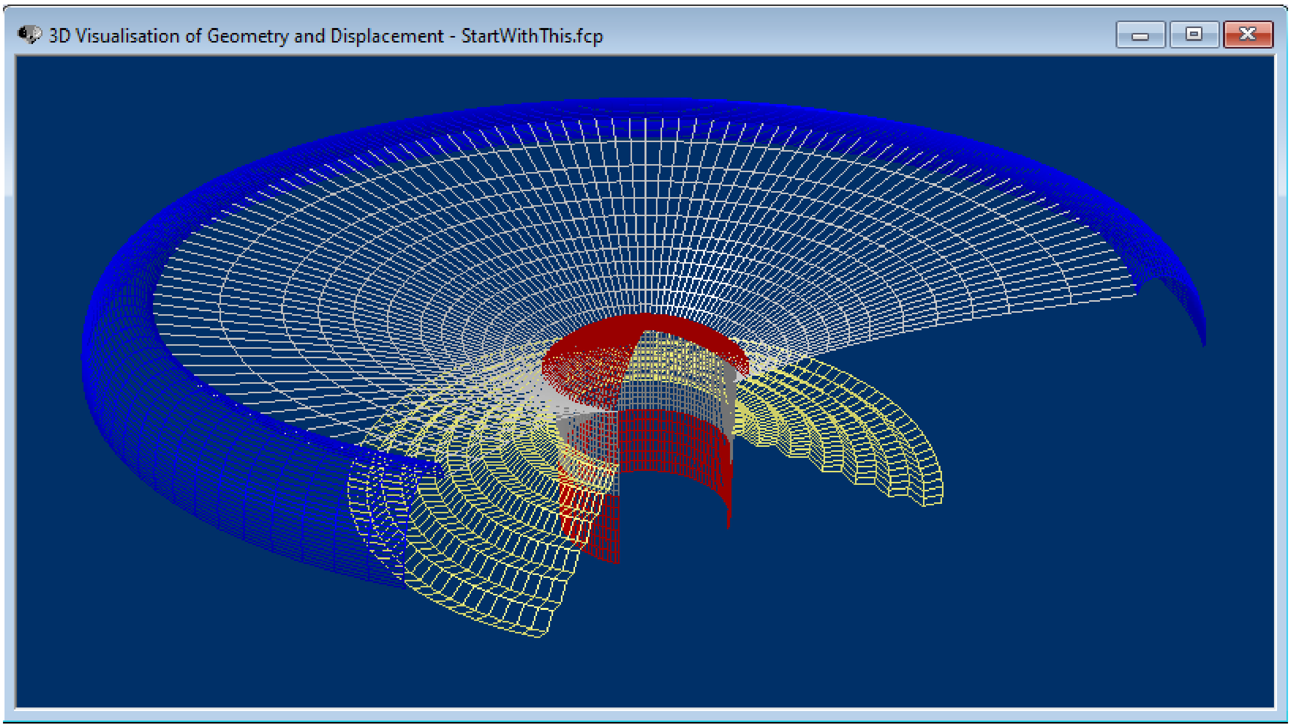


Figure 44 – Initial Shallow Curvilinear Cone Woofer

Next, we will optimize the geometry of a new shallow cone speaker shown in Fig. 44.

First an analysis is performed with inverse FINEBox to find suitable cone area Sd and TS parameters.

Following a large signal analysis of a suitable rubber surround was performed in FINESuspension, giving the surround dimensions for the required Xmax (75% symmetrical linearity)

The woofer geometry was then created with the Geometry Modeler in FINECone Fig. 45. The available OD and cone height in the frame was OD 140 and 19-20mm height

From many other FINECone simulations it is well known that a large dust cap can influence and disturb the cone + surround response considerably.

Therefore, a small dust cap was chosen initially. This can easily be changed later after optimization of the cone and surround.

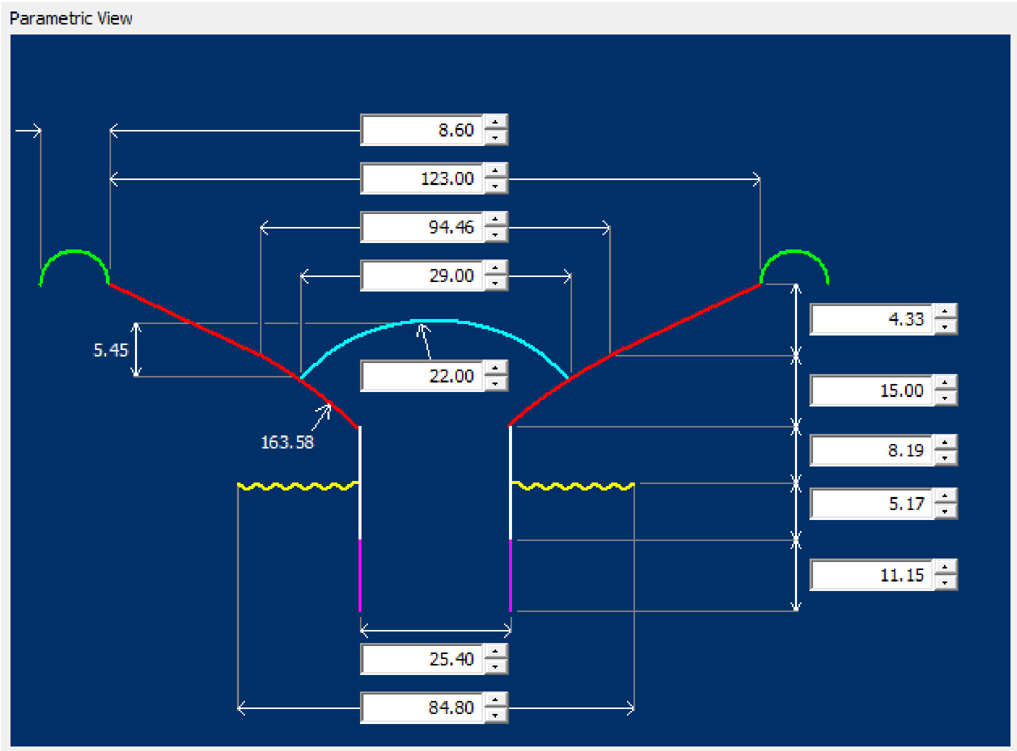


Figure 45 - Initial dimensions of Shallow cone + surround

After accepting and saving this geometry, the Geometry / Shape Optimization was opened Fig. 46. Using the default values 300-4k, reference 500Hz we selected RMS optimization for the cone shape +dust cap.

The simulation is based on default material parameters, N-paper cone (DKM) and thin standard rubber surround. The frequency response in Fig. 46 has optimized cone geometry using the default Max Deviation option and is flatter than the initial response.

The initial curve height of the new cone is changed from 15 to 19mm and the diameter where the cone becomes linear has decreased from 94.46 to 93.40.

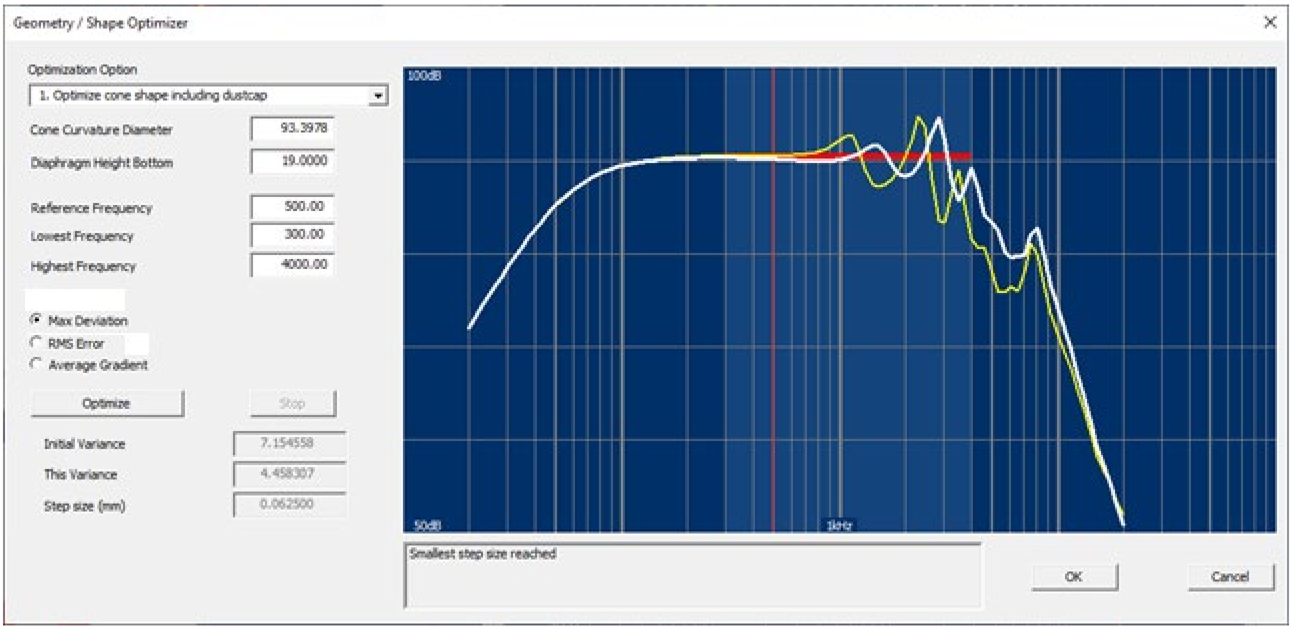


Figure 46 - Optimizing Cone shape + Dust Cap

Considering that a light surround can store less energy compared to the cone and has a large percentage of the total radiating area, we should investigate how the surround can be optimized.

Continuing the Max Deviation optimization now for the surround including dust cap we get the much flatter response (white) which is also considerably smoother Fig. 47.

A thin surround may be preferred but the (dis-)advantage is now more than surpassed by the advantage of the thicker surround. The surround linearity may then be perfected using FINESuspension for obtaining a symmetrical X-max.

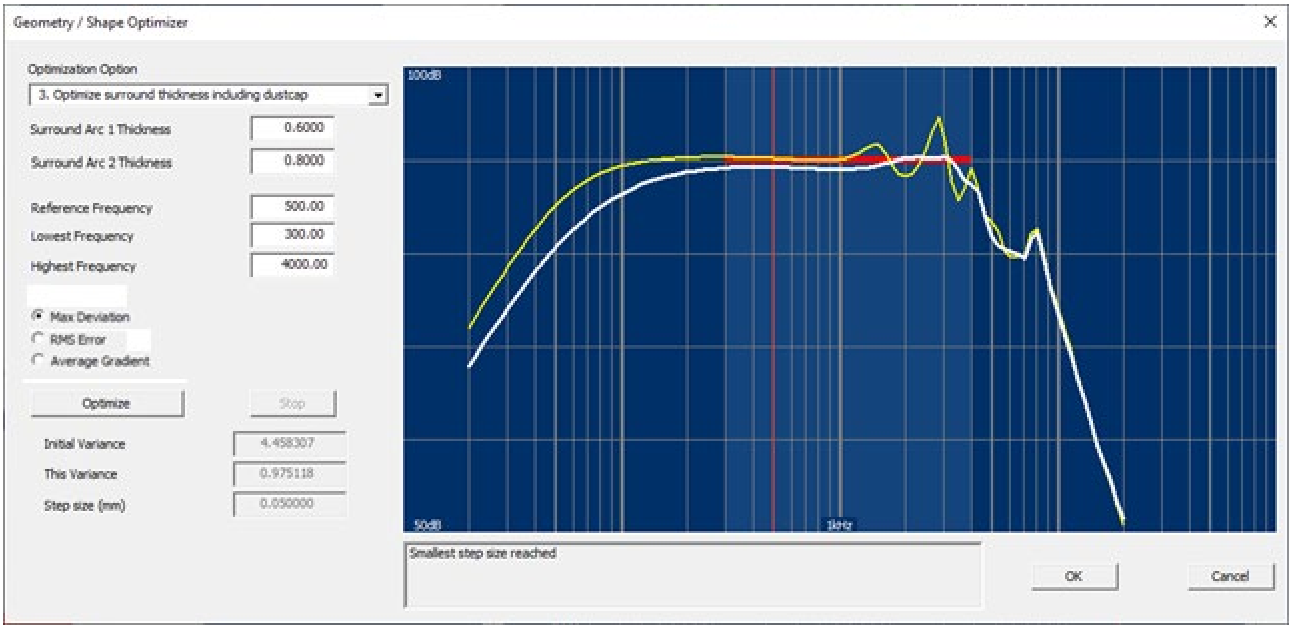


Figure 47 - Optimized cone + surround for the shallow woofer (white)

The final optimized shallow woofer response is shown in Fig. 48. The initial response with peaks and dips is shown in pink.

Using a standard PC the total cone and surround optimization took about 2.5 minutes, using ~137 iterations.

These 2 optimization examples are included in the install setup.

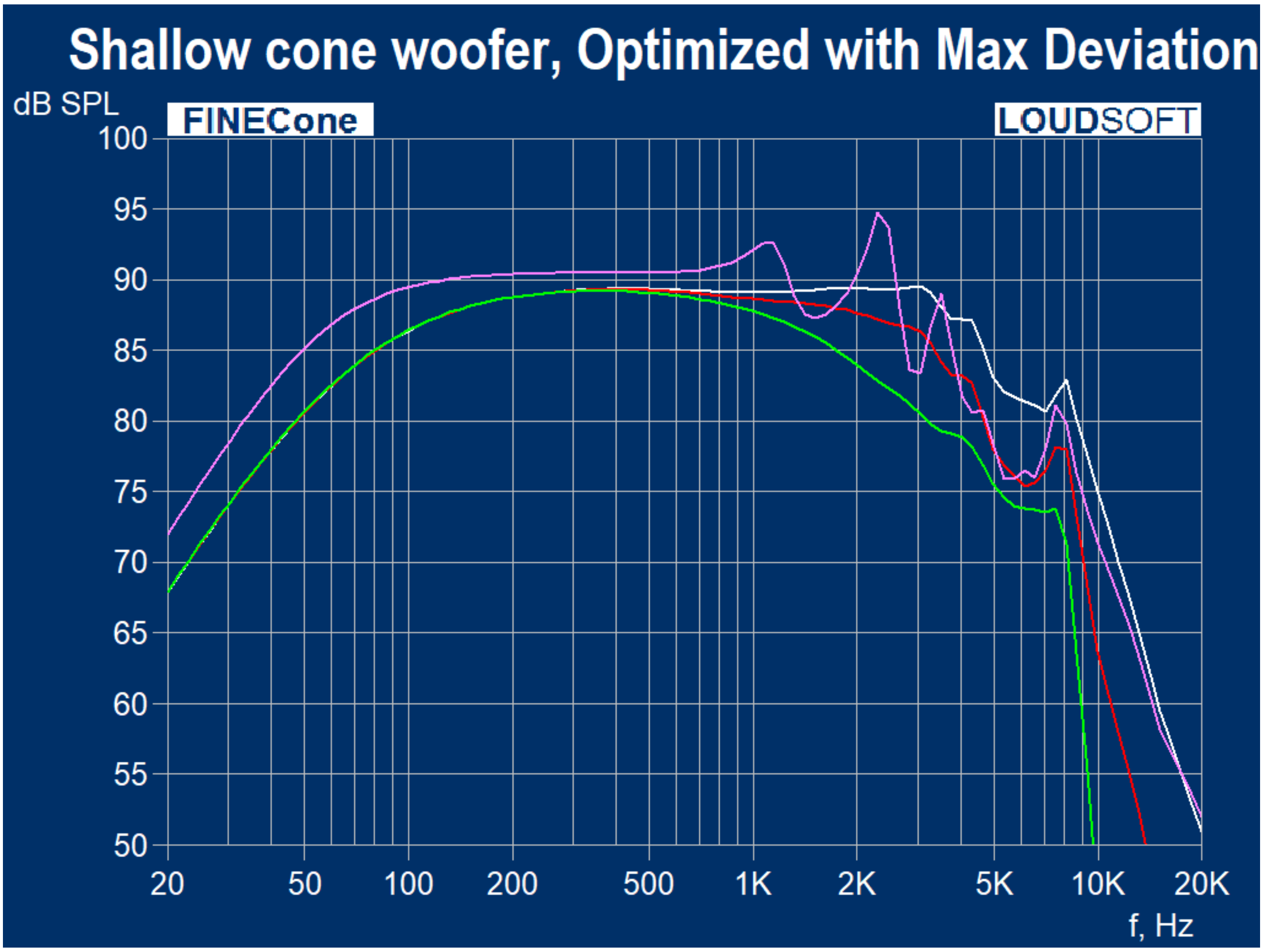


Figure 48 - Optimized Shallow woofer, Max Deviation Option 0_30_60deg. Before___(pink)

2.3 Simulation of an existing 6.5” woofer

The best simulation is made with a measured frequency and impedance response as reference (for comparison). When a good first simulation is found for one speaker you can use this first simulation and make changes to improve your speaker.

Step 1 - Measure the frequency response, impedance, and parameters.

First you need to measure the impedance and frequency response in a large baffle using FINE R+D (measures anechoic in normal rooms) or similar. Make sure the driver is properly recessed in the baffle, which must be large (IEC baffle or larger is recommended). Then export these as lab-files with phase (automatic in FINE R+D) or txt files with phase.

Then measure the TS parameters, preferably using the fixed mass method. We will later import the curves and data into FINECone.

Step 2 - Draw the geometry of the driver

We recommend using the **Geometry Modeler section 1.2** or creating your own DXF file, section 1.5.

We recommend the CAD software DraftSight for making DXF files if needed. The safer and easier way is modifying one of the FINECone example DXF files because the names of all layers are defined by default settings, and they will import easily into FINECone. See the DXF hints in section 1.5.1 how to make the DXF file. Please note that the DXF file is simplified, especially regarding the VC former (Figure 12 and Figure 13), so that the Voice Coil is represented by one line for the winding+ former (magenta), while the remaining VC former above the winding is another line (white) up to the point where the spider is attached and maybe one more line up to the cone.

Likewise, the dust cap and the flange, which is overlapping the VC former is modelled as an arc +ONE line.

The same applies for the surround flange on the cone. The actual cone is glued to the first roll of the spider. Since we cannot model double lines, we may split the cone into two arcs.

Note that we cannot have double lines in the FINECone DXF file

Step 3 - Example using the FINECone Wizard.

Press button to start the FINECone Wizard.

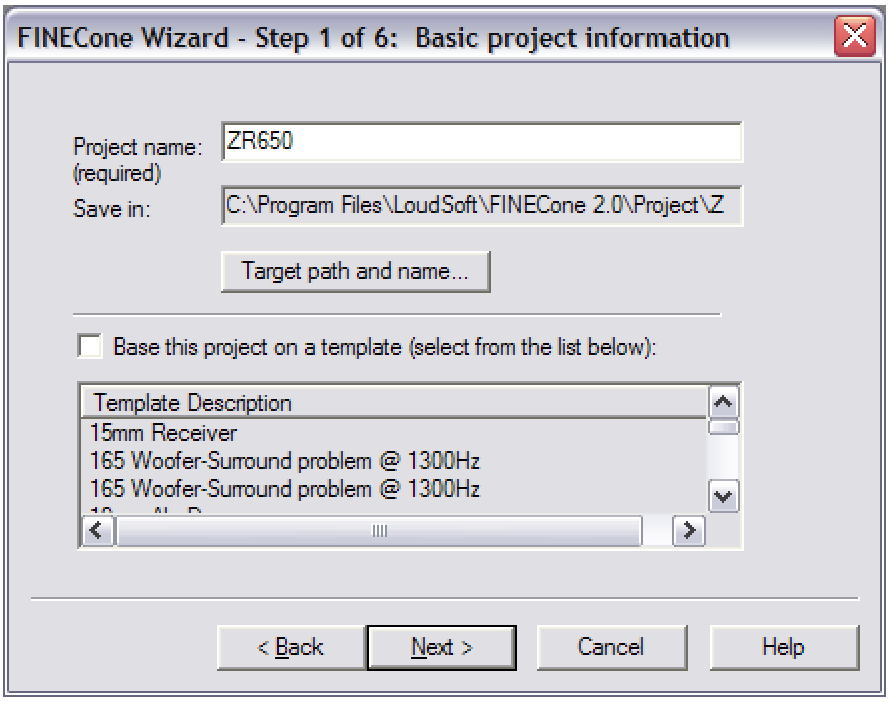


Figure 49. FINECone Wizard

If you cannot use the **Geometry Modeler section 1.2**, we strongly recommend selecting a template file (.FTE), or an earlier simulation as a start.

However, if the analysis is new, we therefore continue with next.

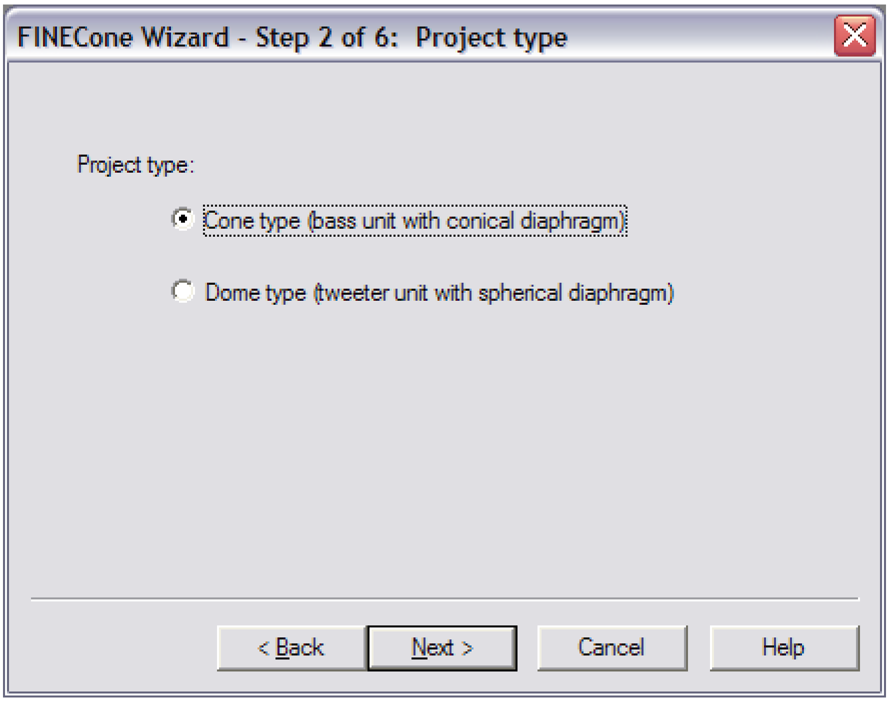


Figure 50. Choose Cone type (Project Type)

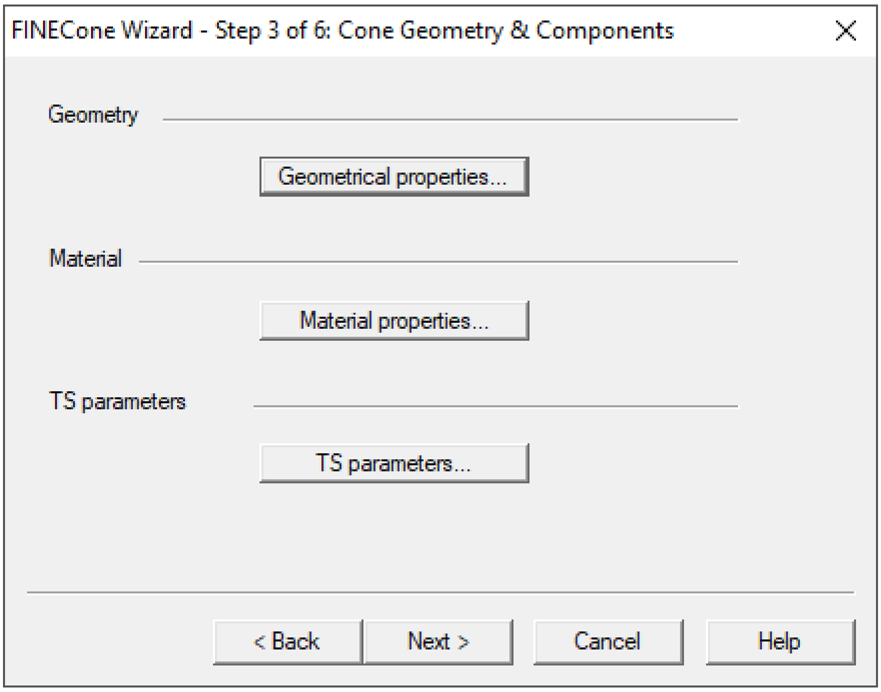


Figure 51. General FINECone Steps

The 3 buttons indicate the general procedure:

1. Define the Geometry,
2. Input material properties
3. Set other simulation parameters

Press 

button to input the geometry of the driver.

After opening the DXF file (Figure 52) we find that two rows are not selected because the names of those layers are not the same as the default names. We must choose dust cap and voice coil layers by finding the layer where the component is from the drop-down menu as shown in Figure 46.

Note that the DXF file is analysed as indicated by the green circles. A red circle would indicate that the lines were not properly attached.

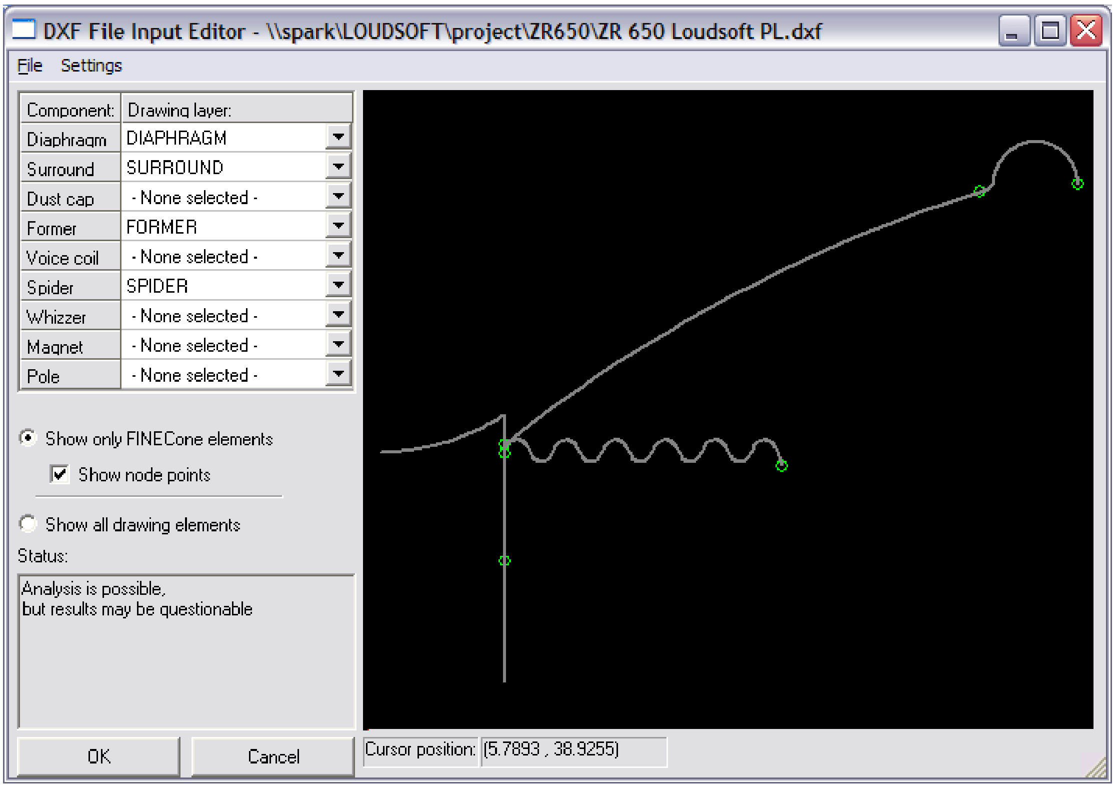


Figure 52. DXF Import -Input Editor

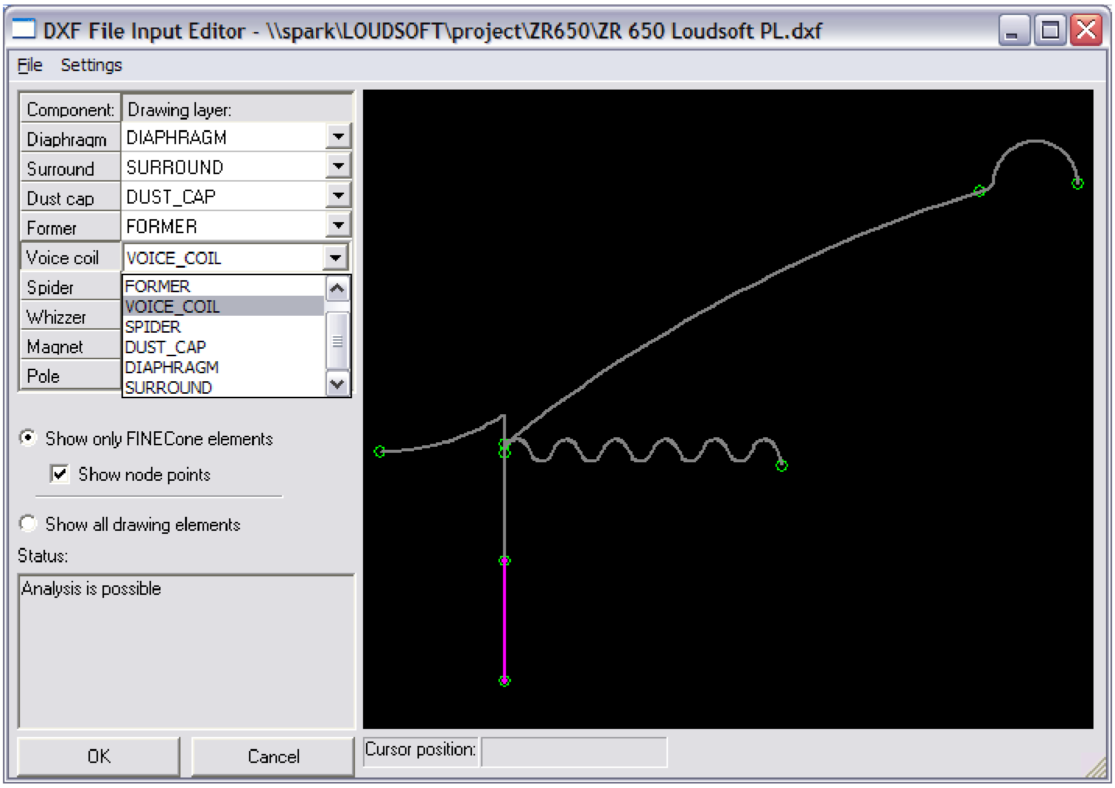


Figure 53. DXF layer - drop down menu

Press Material properties... button to input the materials of all components.

The preparation for this step is to decompose the driver into parts like those segments we have in our DXF file and measure the thickness and mass of them.

Choose the material of each segment of the diaphragm. We may select all the parts and choose the material for all of the parts if they are same. However, the safe way is to do it one by one as we may have different thicknesses for different parts. We should avoid making it the same by mistake.

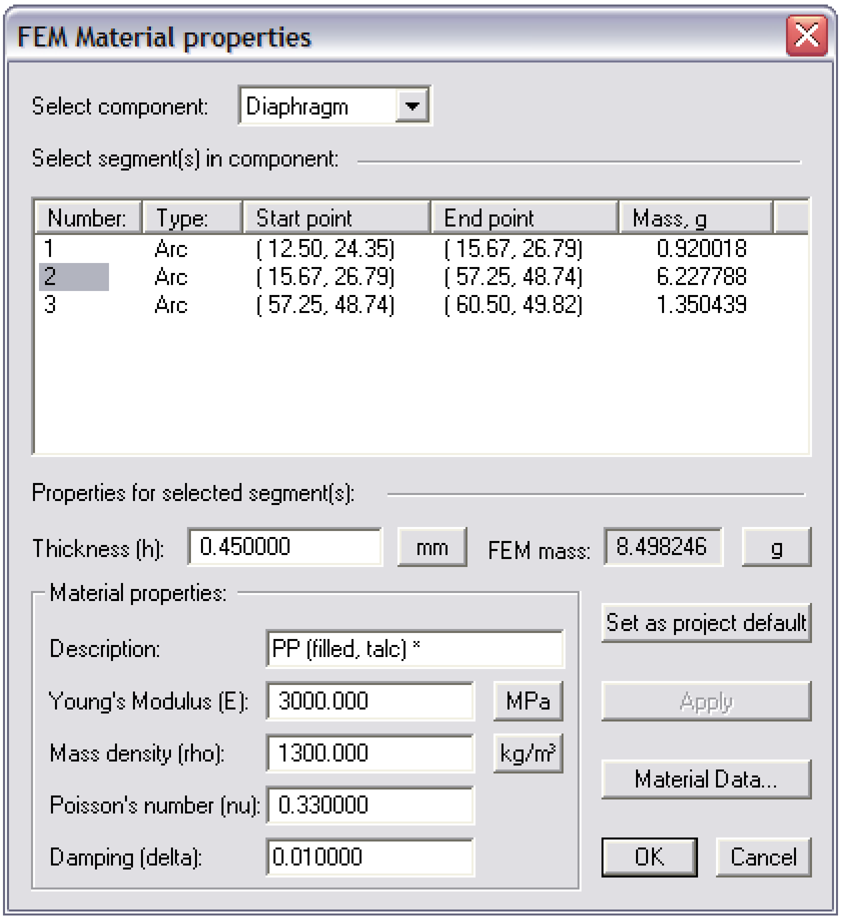


Figure 54 - The thickness of the various components is set in the FEM Material properties window

The cone thickness is 0.45mm, shown here as input for segment Number 2. Press “Material Data” to enter the database where we have selected PP (filled, talc) material for the cone.

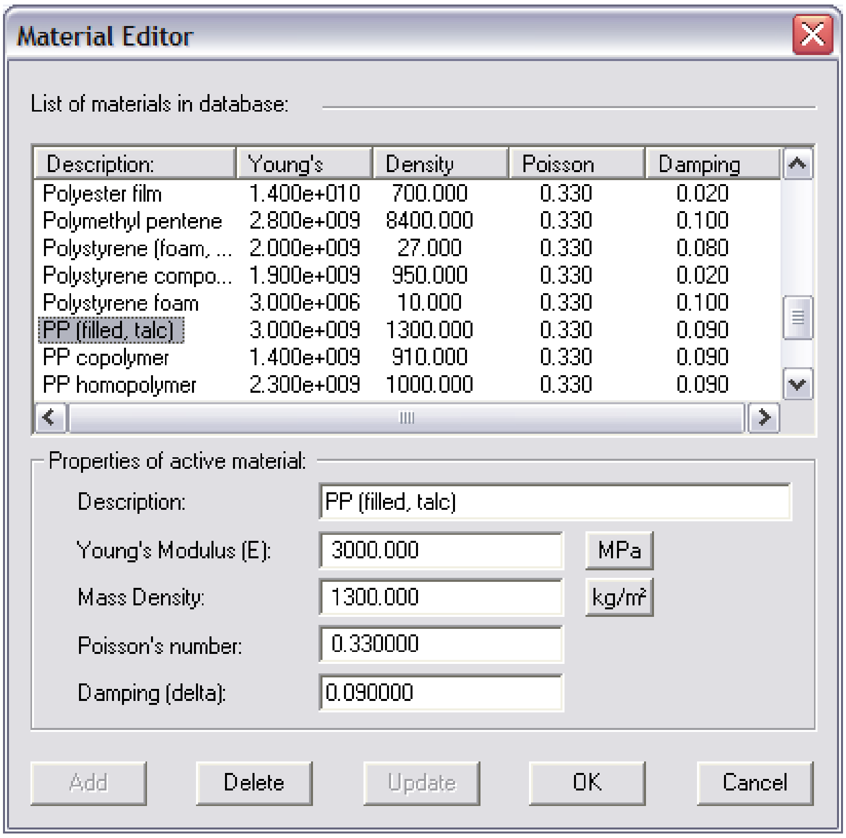


Figure 55 - The cone material is selected from one of the standard materials in the FINECone materials database

The influence of glue should be considered. The actual speaker has much glue between cone, VC former and spider. In our model this glue can be in three positions, on the inner part of the cone, on the upper part of the VC former, or on the inner part of the spider.

In this case, we choose to model the glue on the inner part of the cone, because we want to simulate the influence on the cone response. We do that by setting a larger thickness for that cone segment and change the density until the mass is same as the measured value.

This is done in Figure 49, where the first cone segment (1) is specified with 2mm thickness. We may later change the stiffness by adjusting Young’s Modulus

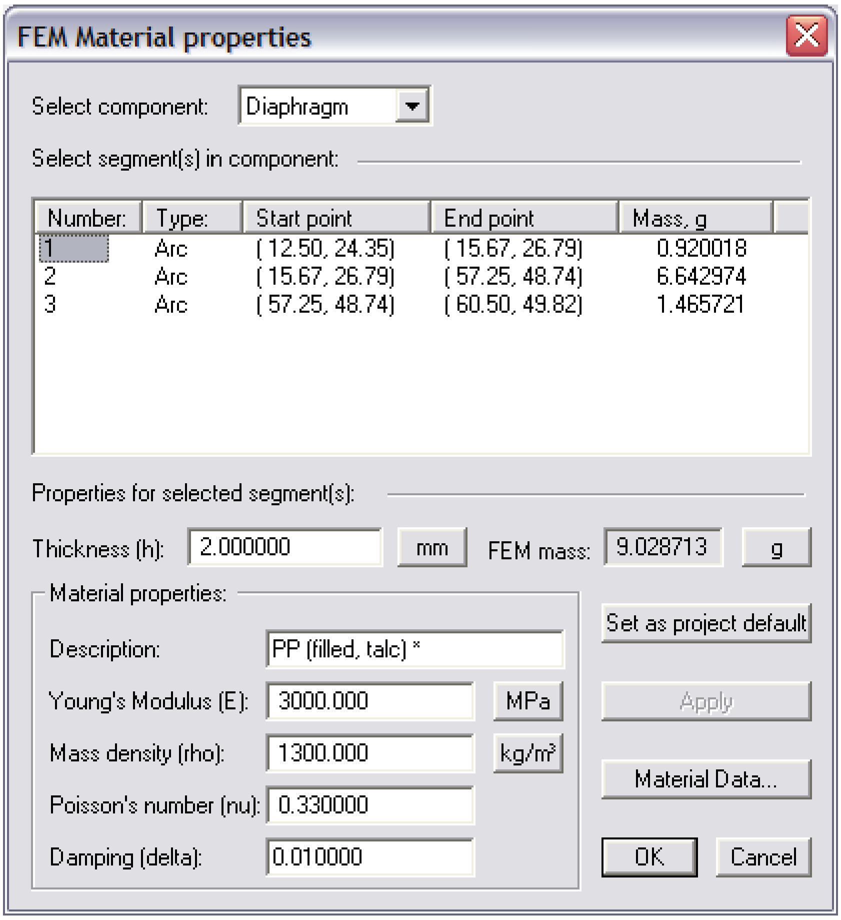


Figure 56 - Part of the cone is made thicker to account for the glue on the voice coil

The surround is 0.41mm rubber from the database.

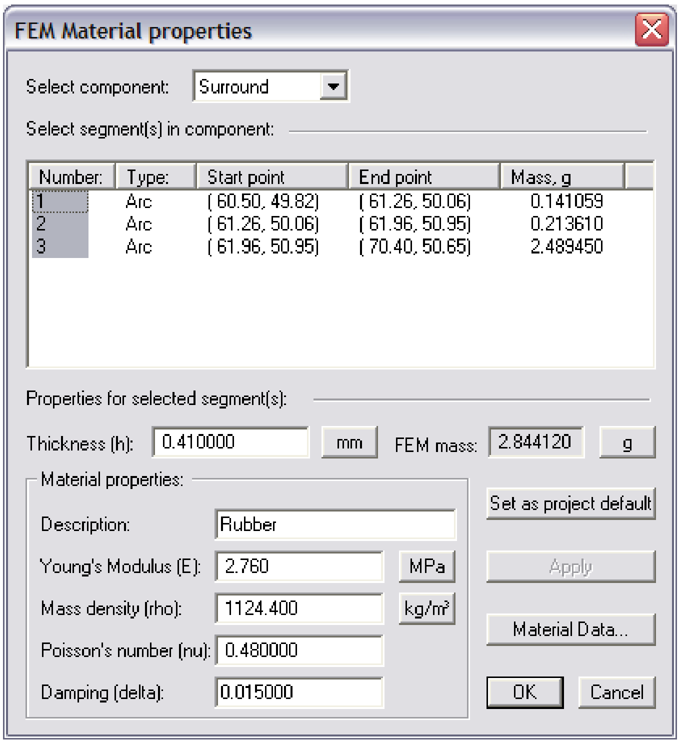


Figure 57 - The surround material is using the Rubber material which has the properties of the most common type of rubber used for surrounds

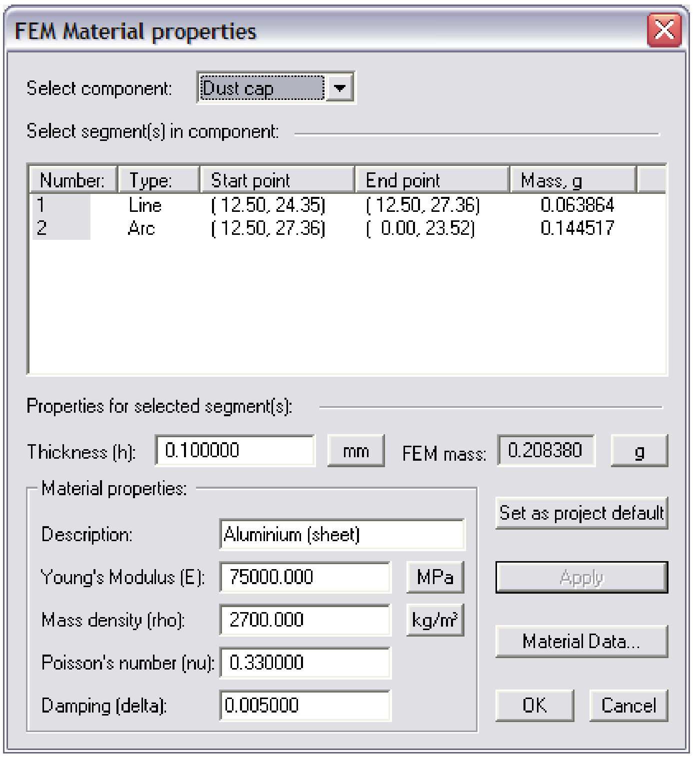


Figure 58 - For the dust cap aluminium is chosen

Note: Don't forget to press Apply*, whenever you have made a change.*

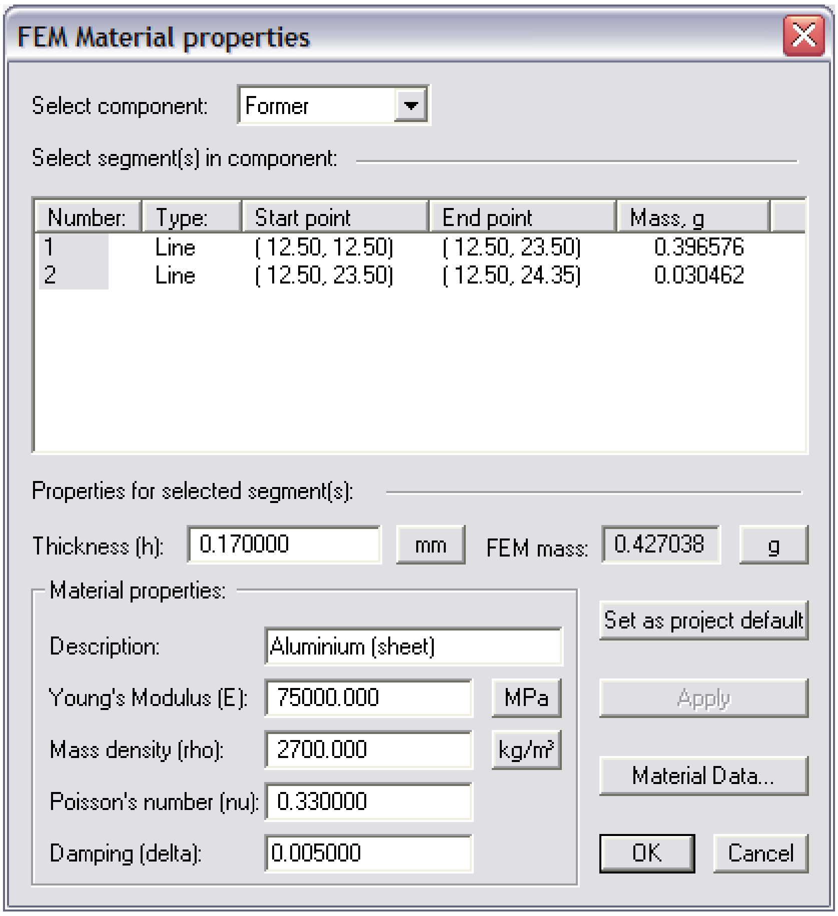


Figure 59 - Also for the voice coil former aluminium is used

The important parameter of the voice coil is the mass of the coil winding + former covered by it, see Figure 60. We then adjust the thickness to get the same mass as measured. The VC stiffness is not used.

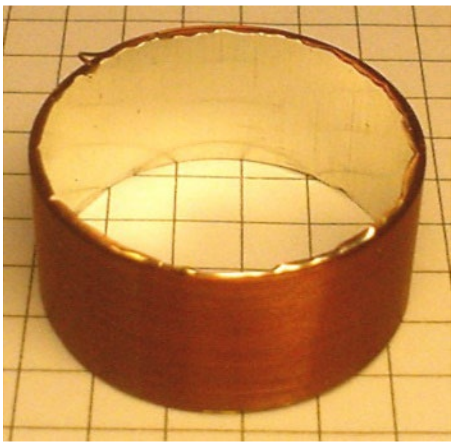


Figure 60 - The voice coil used in the woofer we are simulating

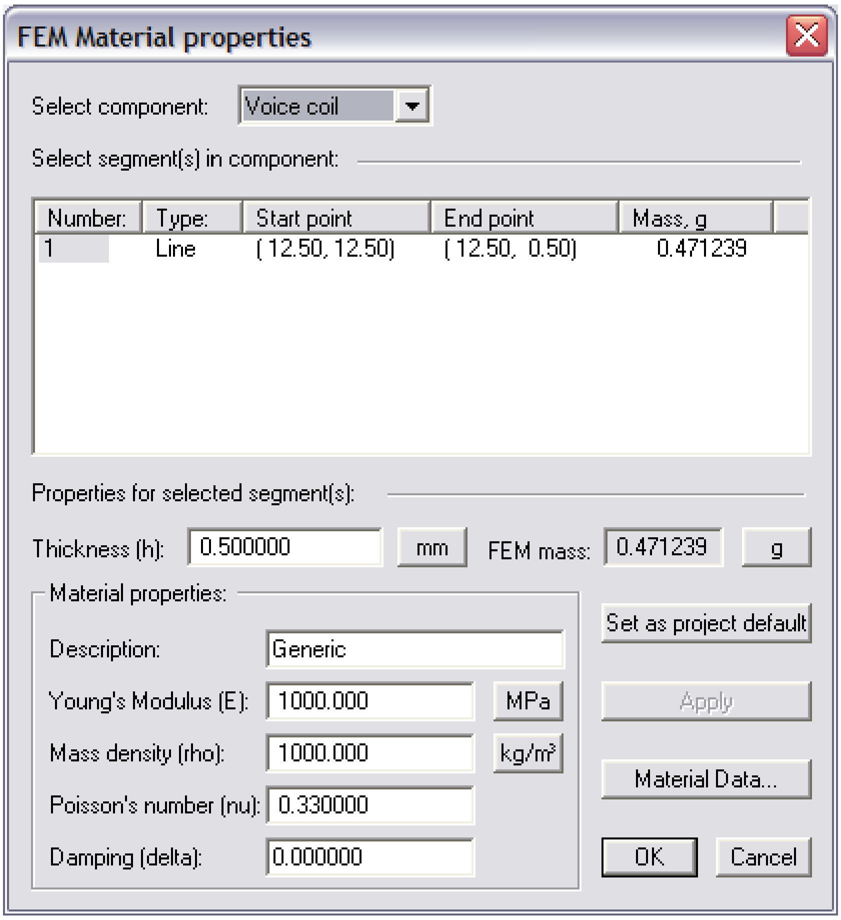


Figure 61 – For the voice coil we choose the 'Generic' material as we are going to tweak the parameters ourselves.

Since we do not know the accurate material of the spider, we may use the 'generic' material from the example file or for example "Spider/ Typical" found in Material data.

Remember to select all segments with CTRL+A.

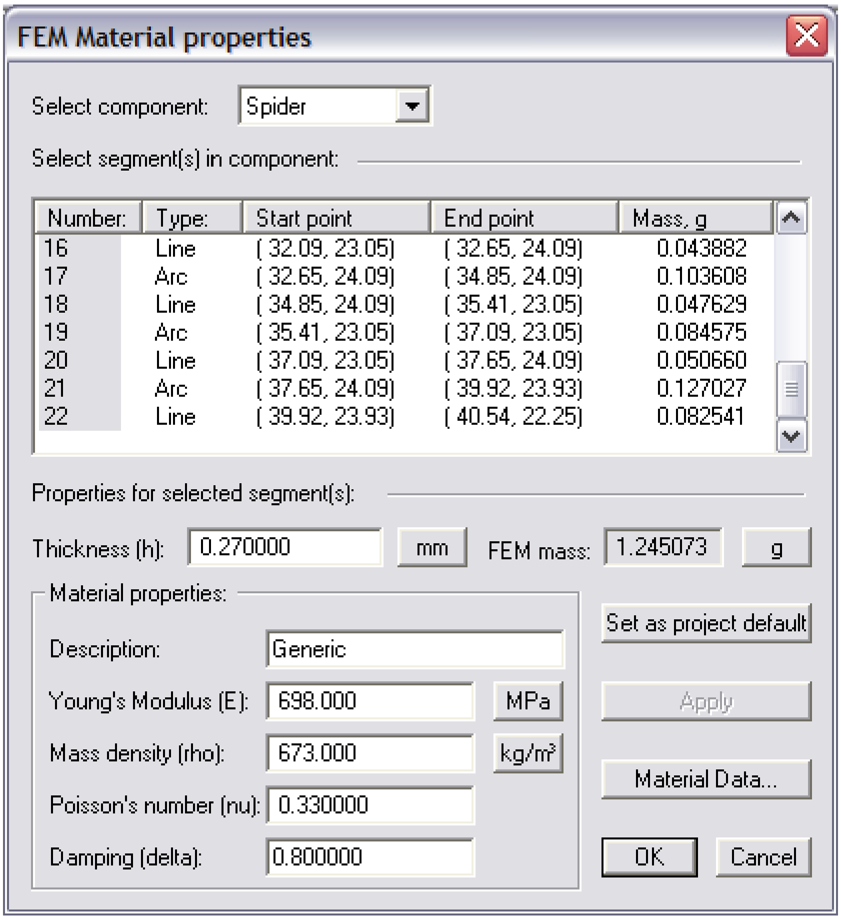


Figure 62 - For the spider the 'Generic' material or “Spider/ Typical” is used as we do not know the precise material of the spider

Note: The common error of setting materials is when we select more than one segment and set the materials for them together, we may forget they have different values in some parameters, e.g. thickness.

(If using a slow PC, you may use the ‘Fast solution of differential equations’ setting to get faster calculation. The setting can be found in Tools/Program Options/Calculation).

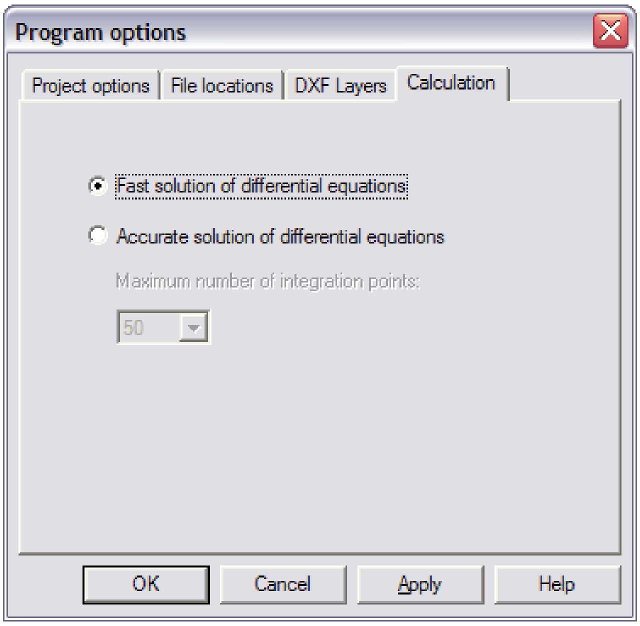


Figure 63 - Choose between quicker but less accurate and rough calculations or slower but more accurate simulations

Note: When we change the number of segments in the DXF file, the materials setting of the changed layer must be defined again. Values cannot be stored when there are different segments.

Press [TS parameters...] button to enter the lumped TS values, see section 1.6.5 and 1.6.6.

The other lumped parameters are not used in the accurate FEM calculations (except Re and BL). However, they can be found from the FEM calculations and used for comparison.

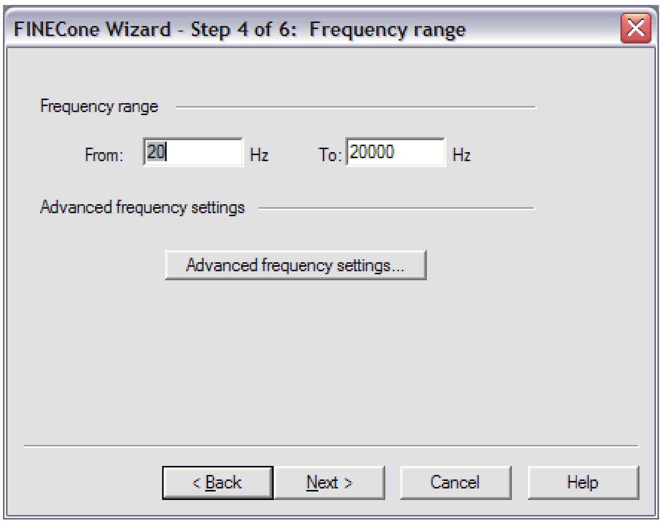


Figure 64 - The frequency range can be specified depending on the simulation

The TS parameters below are measured with FINE R+D:

Parameter	Value	Unit
Re	3.47	Ω
Fs	61.46	Hz
Qms	4.975	-
Qes	0.779	-
Qts	0.674	-
L1	0.176	mH
L2	0.39	mH
R2	4.059	Ω
Vas	12.583	L
Mms	13.597	g
Cms	493.179	m/N
Bl	4.836	Tm

Start the simulation using some of the values: Re and Bl from here in the next step. Use the value from L1, L2 and R2 for Le1, Le2 and Rp.

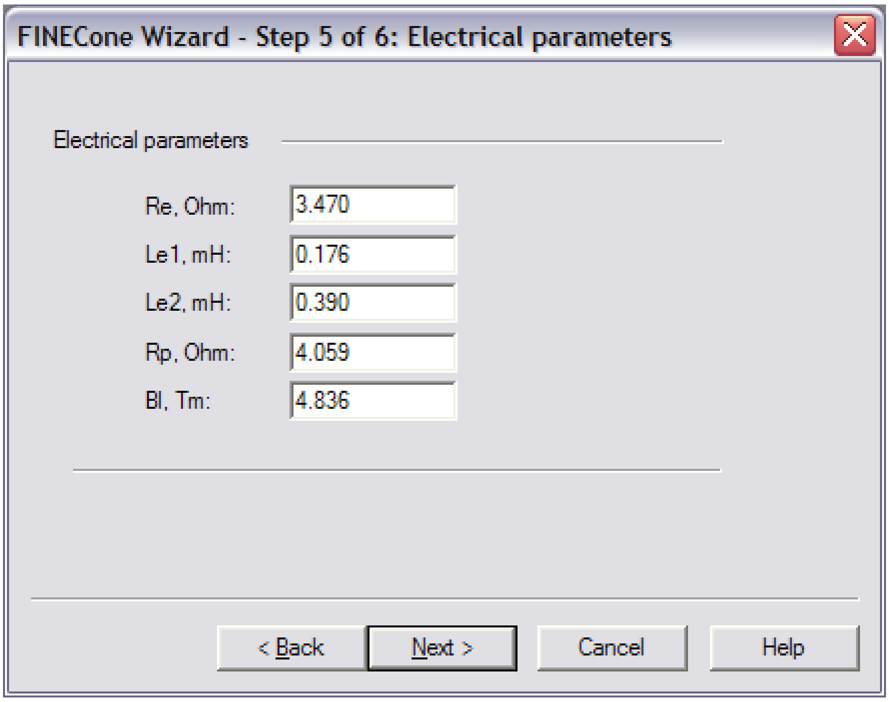


Figure 65 - The electric parameters found using FINE R+D are entered

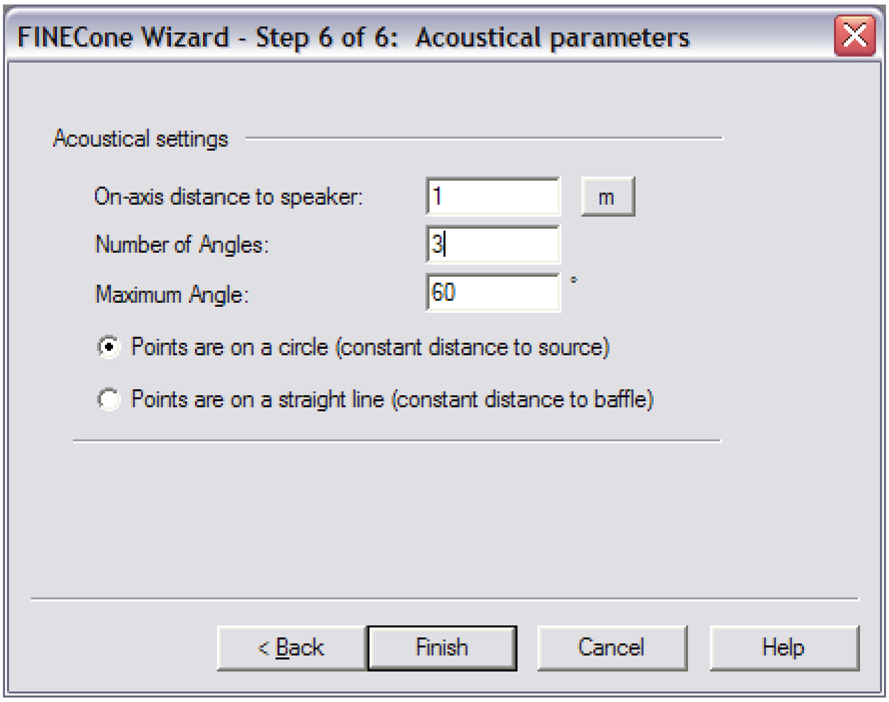


Figure 66 - Measuring points are specified as the last thing

3 angles mean 0, 30 and 60 degrees off axis responses.

The Finite Element (FEM) calculation is done automatically after Finish is pressed.

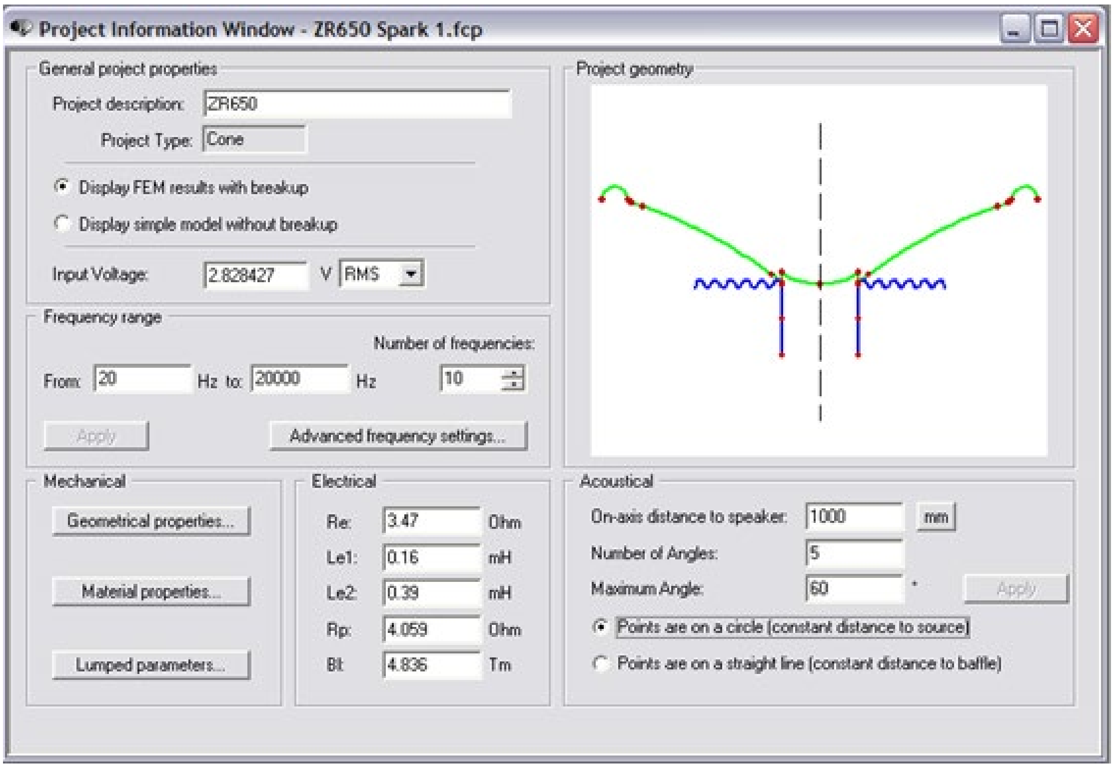


Figure 67 - This is the window you will see when you press finish if you have followed the instructions in the tutorial

Step 4 - Fit the impedance curve

Now is the time to look at the impedance curve we have calculated in FINECone.

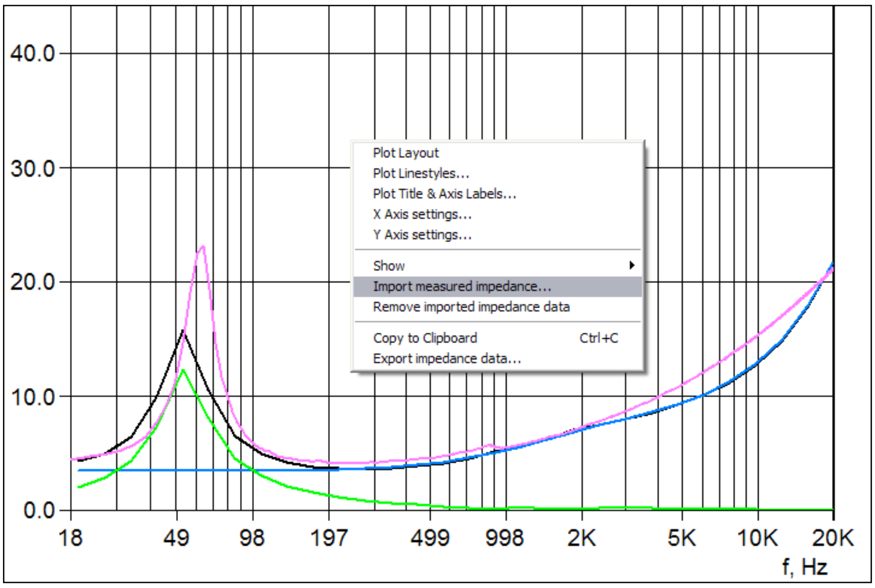


Figure 68 - The measured impedance of the driver is imported as a reference

The pink curve is the imported curve. It was imported from FINE R+D as a *.lab-file, but FINECone can also handle *.txt-files from other software.

The simulated impedance curve (black) is lower than the measured curve around 300Hz. It is because Zmin is a little larger than Re. So, we should increase the value of Re.

Then we change the values of Le1, Le2, and Rp, to get good agreement at frequencies up to 10 kHz.

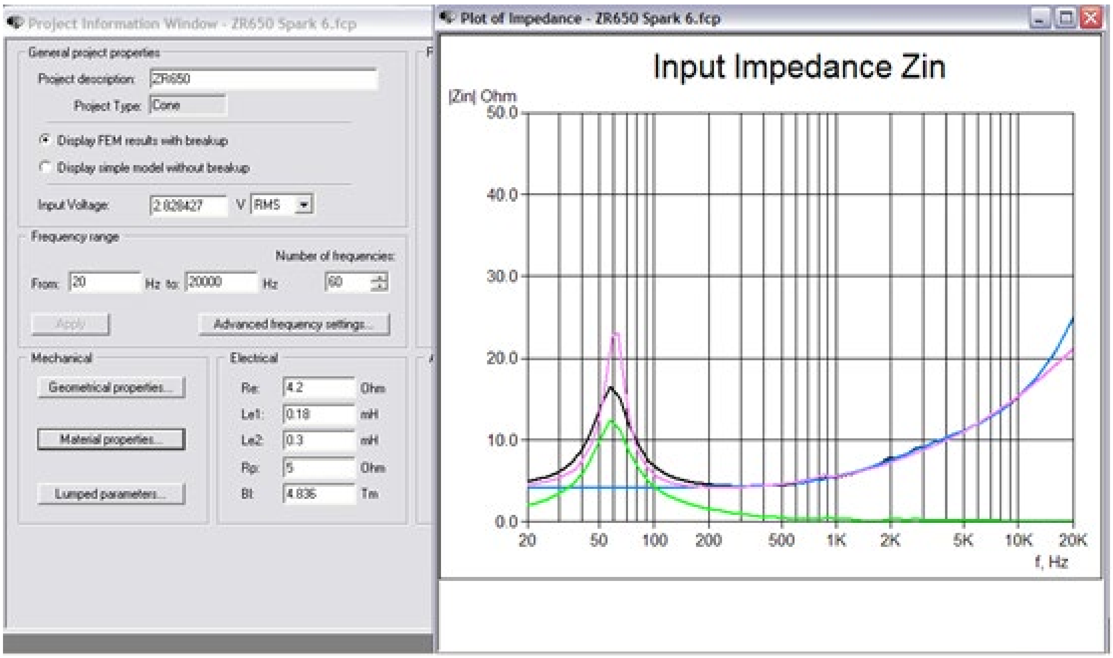


Figure 69 - After having imported the measured curve (the pink curve) the electrical parameters are tweaked to fit the simulated impedance curve

Step 5 - Fit the SPL curve

Finally, we work on the SPL curve

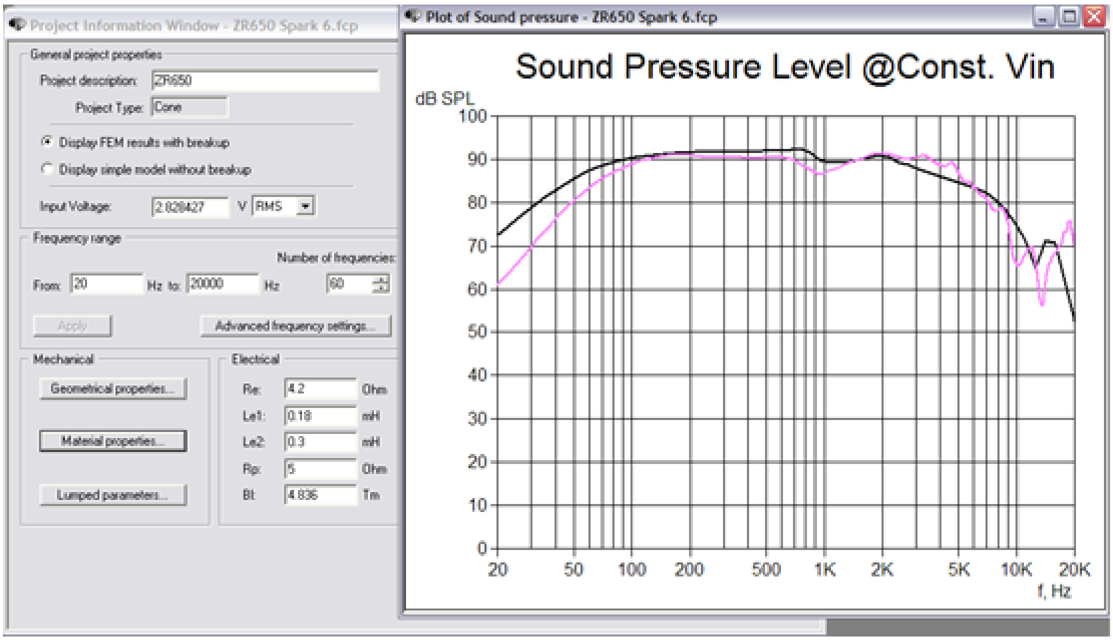
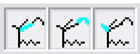


Figure 70 - The measured frequency response of the driver is imported into FINECone and is seen as the pink curve

At low frequencies, the black curve has more extension than the actual curve, because the black (simulated) curve is simulated assuming an infinite baffle, but the pink (real) curve is measured in a smaller finite baffle.

Using  buttons we can study the effect of the 3 main components: Cone / Surround / Dust Cap.

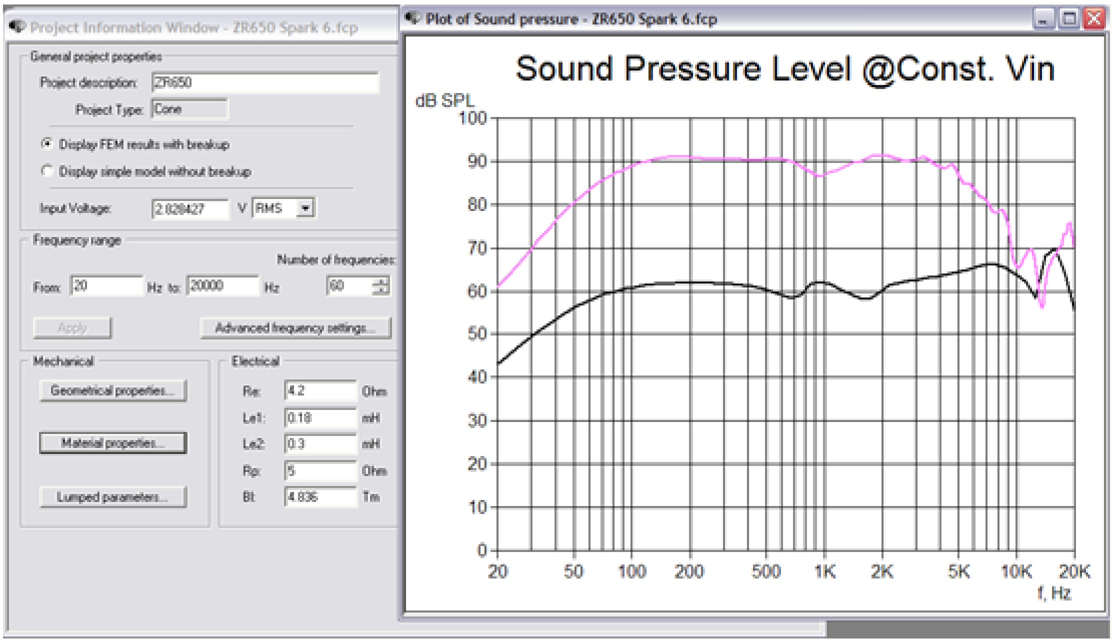


Figure 71 - Here the frequency response is shown for the dust cap alone

The dust cap only affects the very high frequencies.

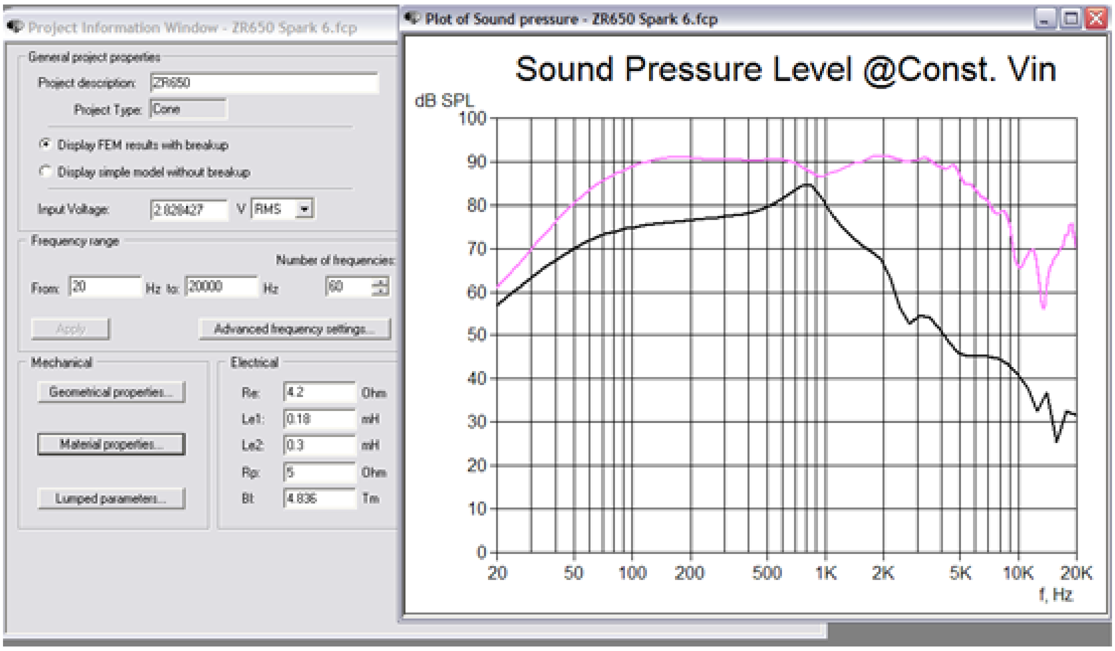


Figure 72 - The frequency response for the surround alone

The surround produces more SPL than the dust cap and has a peak around 800Hz close to the dip in the measured curve. So, we may change the surround parameters to get a better simulation.

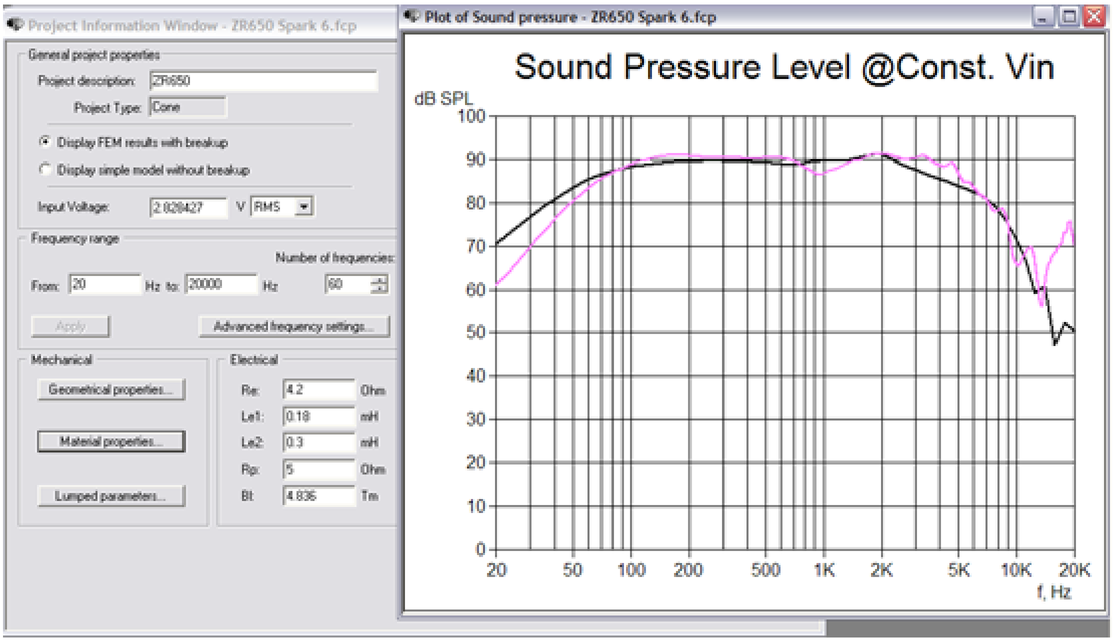


Figure 73 - The frequency response for the cone alone

The cone dominates at almost all frequencies. To get better agreement, we should first simulate the cone accurately.

If the SPL curve looks very smooth, it may be because the damping is too high. The rule is that we use less damping during the first simulations to be sure to see all the break-up details. After that, we will change the damping to the correct value.

Reducing the damping of the cone from 0.09 to 0.01, we get the following responses.

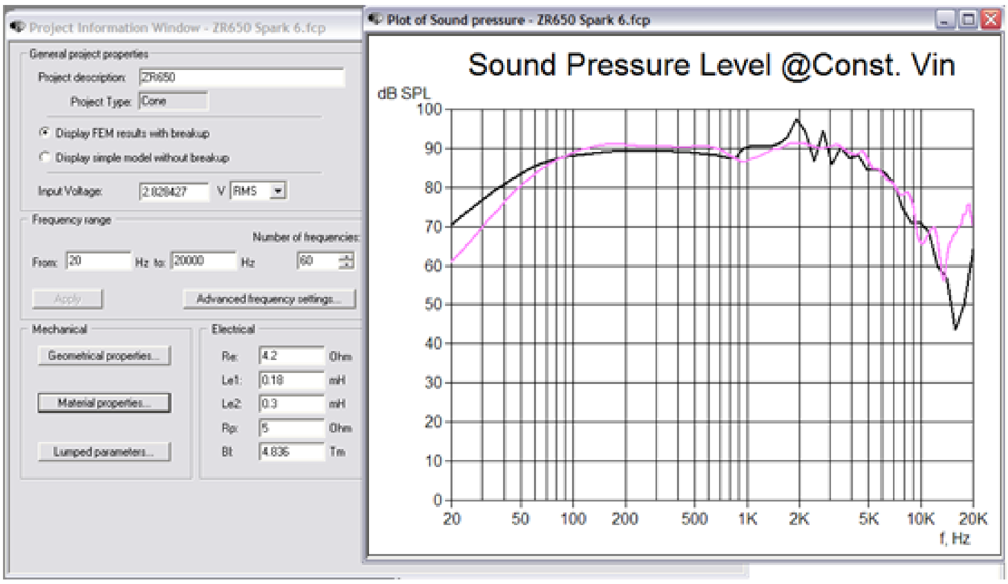


Figure 74 - The overall frequency response with the damping of the cone reduced

Firstly, let's find the reason for the disagreement around 1k Hz. Press button and set the *selected frequency* around 1k Hz.

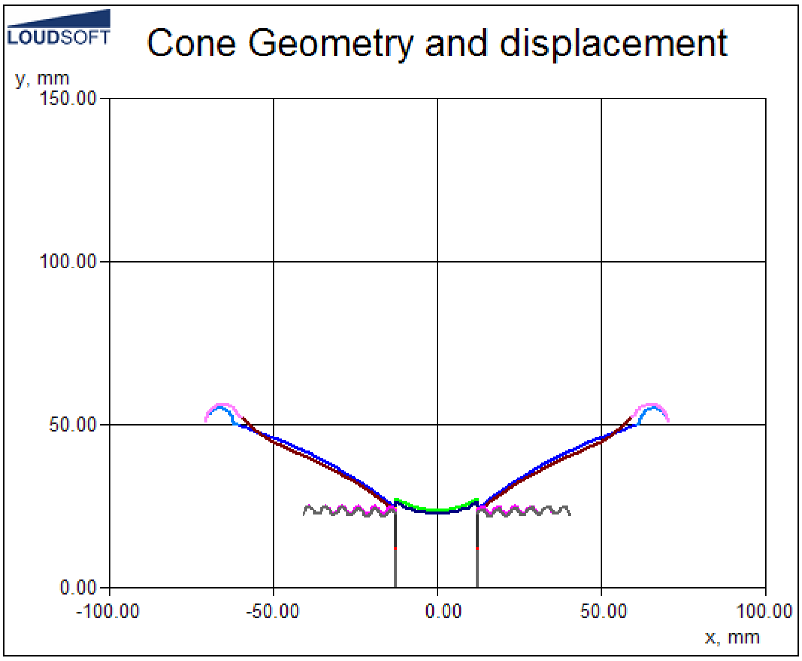


Figure 75 - A 2D plot of the driver at rest and when moving at the selected frequency.

It is very clear that the outer part of the cone is bending. So, let us open the cone material properties.

We will change the thickness of the outer part of the diaphragm back to 0.45 to get the correct stiffness but increase the density to 2300 to keep the mass, since this part is a combination of cone and surround flange. The glue also influences the stiffness, so we should change the Young's Modulus to move the peak/dip to its right position.

Finally, we find the correct damping of the cone, which is lower than the Material data

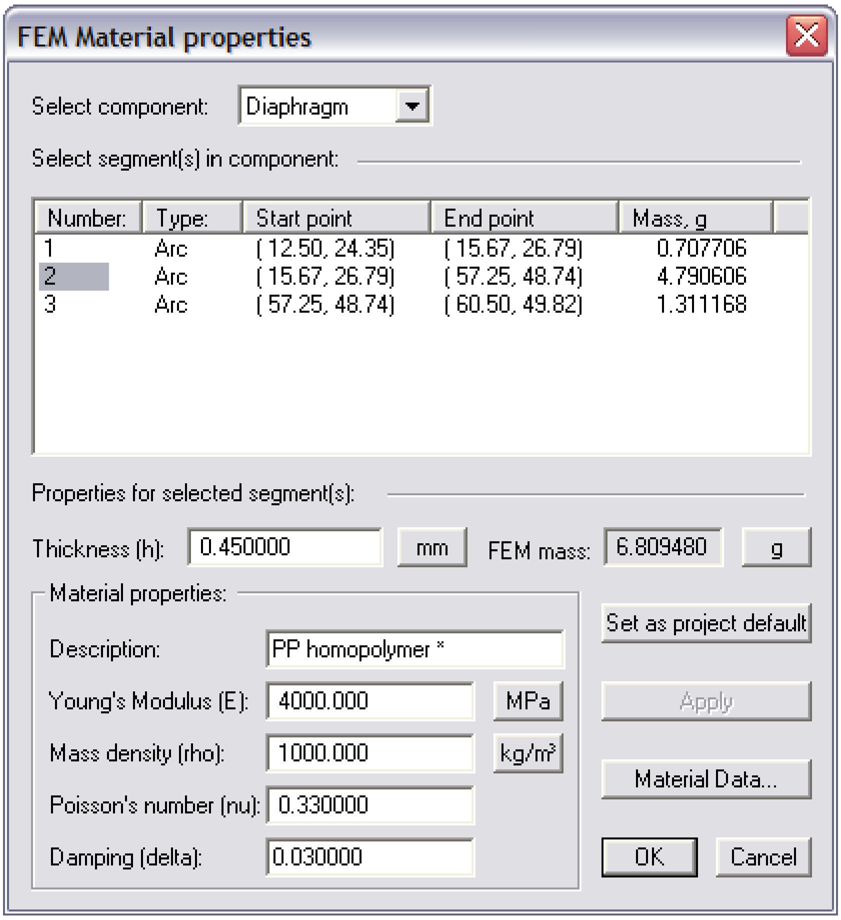


Figure 76 - The damping in the cone is changed to match the measured response

Then repeat this at the other disagreement until we find an acceptable agreement.

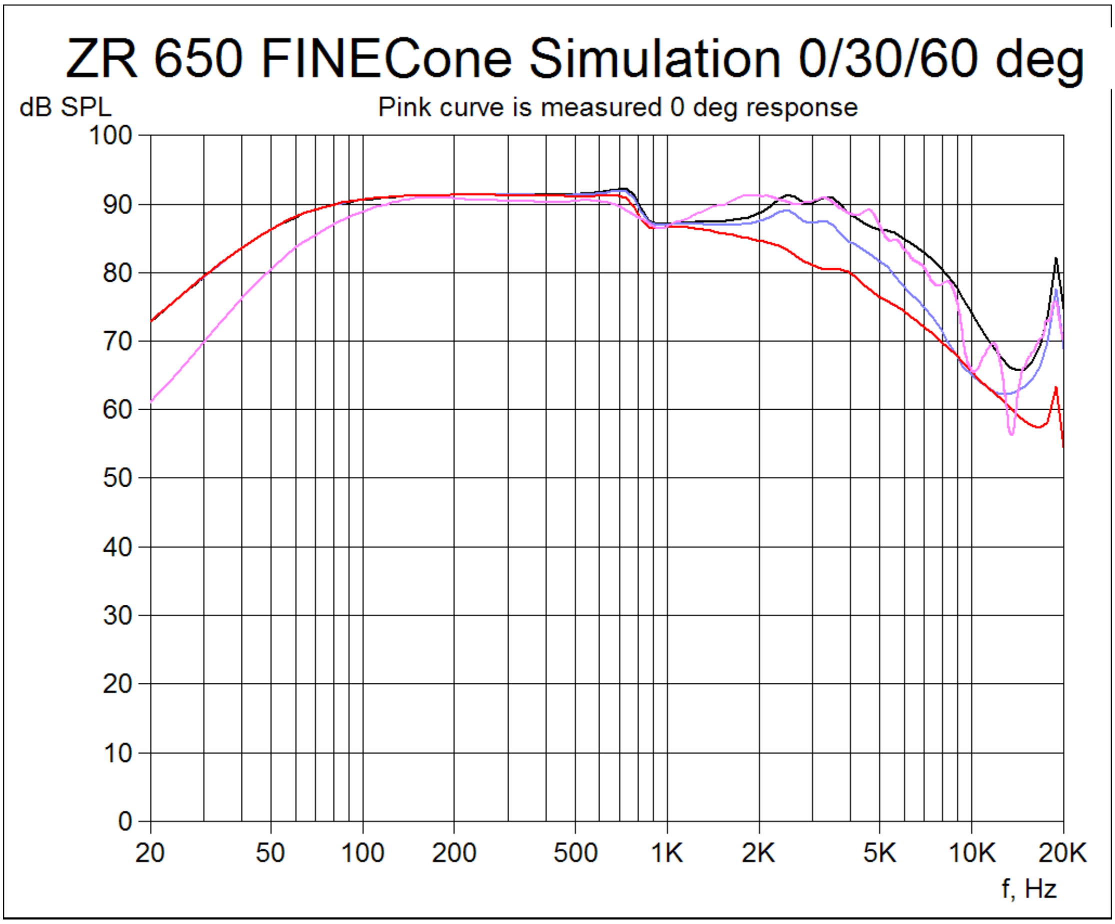


Figure 77 - The simulated response for on-axis, 30 degrees and 60 degrees are shown versus the measured response

The result is a good simulation of the actual measured ZR 650 response (pink curve). The 30 and 60 degree off-axis responses are also calculated and shown.

The simulation accuracy is focused between 100-10kHz. It is possible to increase the simulation accuracy considerably by splitting the cone into 5-7 segments and split the VC former in smaller segments. Examples made in this way can be found in the FINECone Project directory.

Note: Many simulations will show a lower SPL in the range 700-3000Hz. This is normal and a result of the chosen Rayleigh calculation method

2.4 Simulating a real 6.5" aluminium cone woofer

We will verify a FINECone model compared to a real driver to check the accuracy. The FINECone model can then be used to simulate new materials, cone shapes and many other things.

The actual driver is a 6.5-inch woofer in a plastic frame with a 90mm ceramic magnet and 33mm voice coil. It has a curved aluminium cone with a rubber surround and a large plastic dust cap.

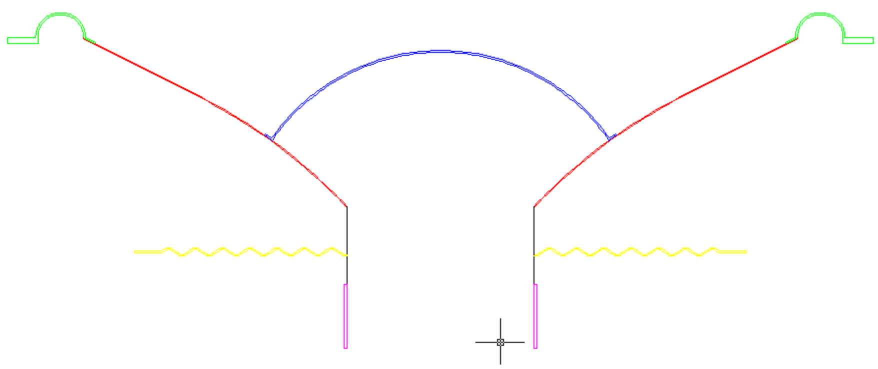
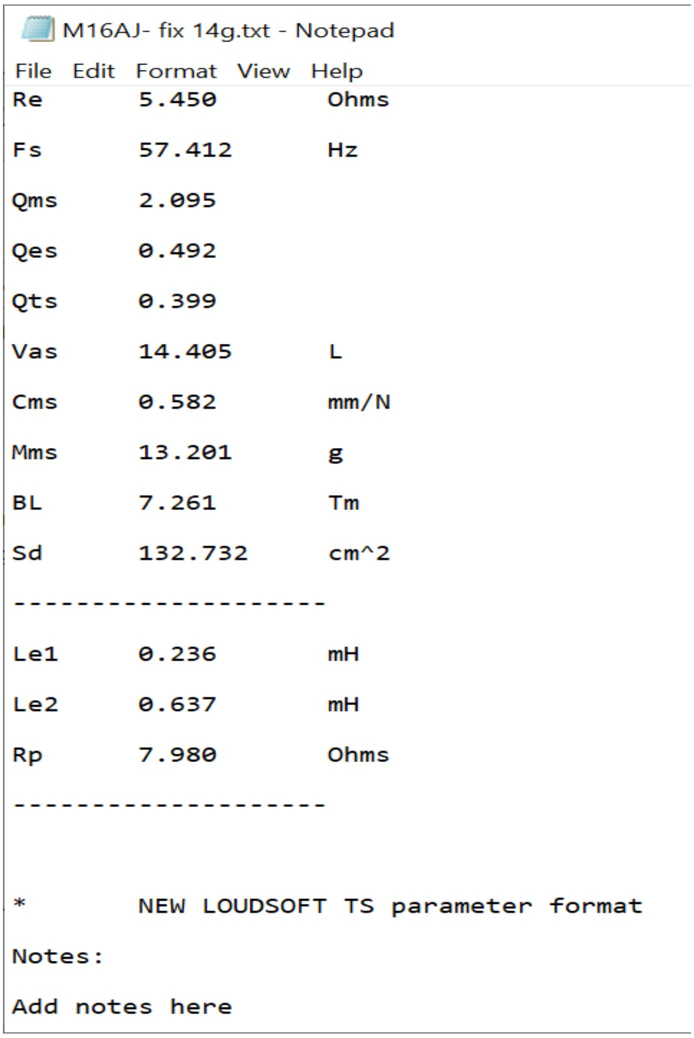


Figure 78 - 2D representation of the driver



M16AJ- fix 14g.txt - Notepad

File Edit Format View Help

```
Re        5.450        Ohms
Fs        57.412       Hz
Qms       2.095
Qes       0.492
Qts       0.399
Vas       14.405       L
Cms       0.582        mm/N
Mms       13.201       g
BL        7.261        Tm
Sd        132.732      cm^2
--------------------
Le1       0.236        mH
Le2       0.637        mH
Rp        7.980        Ohms
--------------------

*         NEW LOUDSOFT TS parameter format
Notes:
Add notes here
```

Figure 79 - TS parameter file in txt format (FINE R+D and FINEBox)

These are the steps in FINECone FEM:

- Define Geometry by importing DXF file
- Define Material Properties of speaker components using material database
- Define Electrical Parameters and import FINEMotor data if available

Since all meshing, number of elements, degrees of freedom and constraints etc. are done automatically by the software, we will just make a sketch of the geometry in DraftSight and import the DXF file into FINECone.

The model must be axi-symmetric and only the right half is used. This implies that the coordinate of the leftmost point is on the symmetry axis where X=0. Usually this is the midpoint of the dust cap. The DXF-drawing is shown in Figure 80.

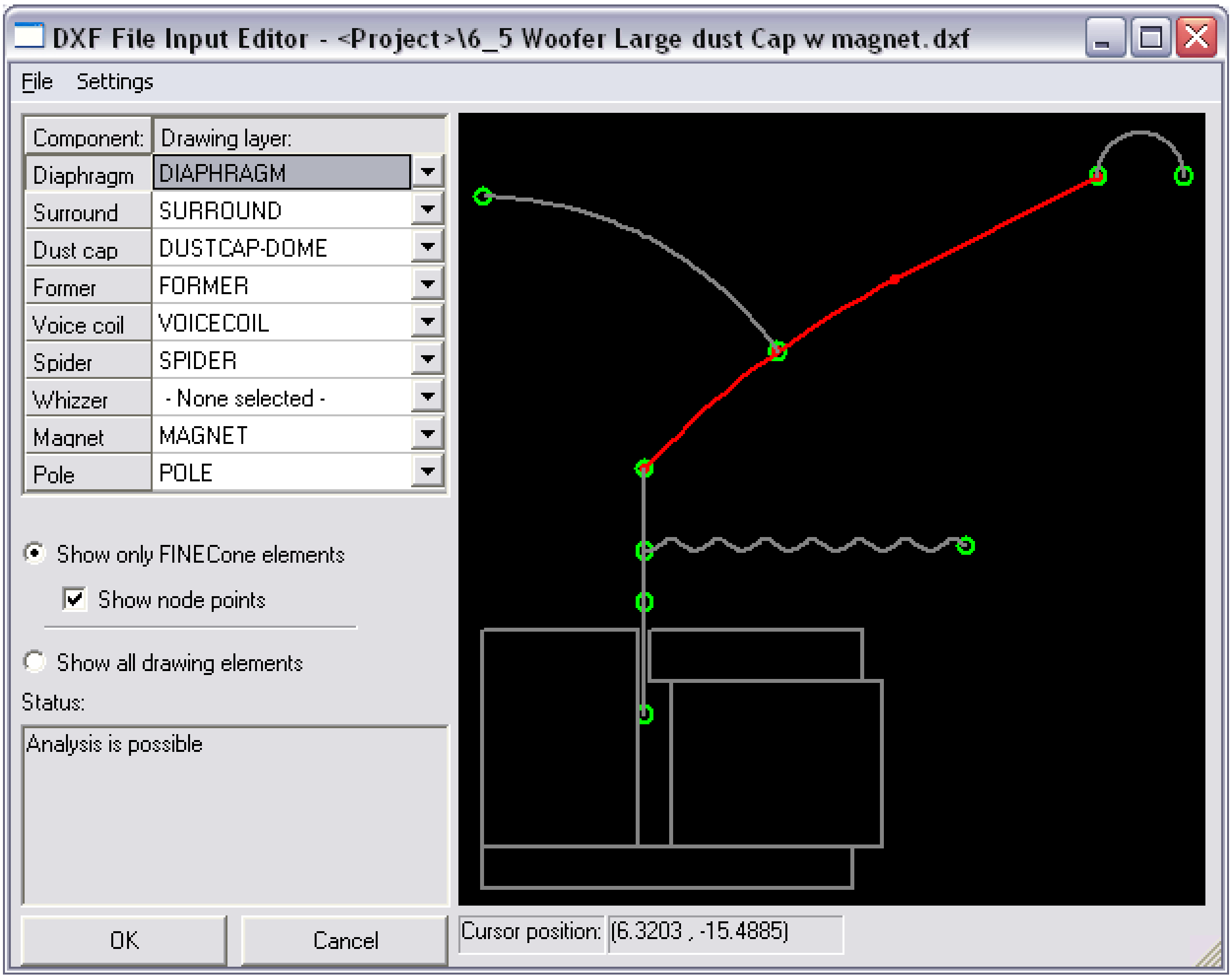


Figure 80 - DXF Import and Automatic Error Checking

We have used the default names for the layers and the entire drawing will be imported directly.

Note. You can change the default layer names in Tools/Program Options/DXF Layers

The Status window reports: Analysis is possible. This means that the DXF error checking has analysed the DXF file and found no errors. Click OK to proceed.

FINECone will now start the calculation using default parameters. These must be changed to give meaningful results in this case.

Therefore, we select FEM Material Properties

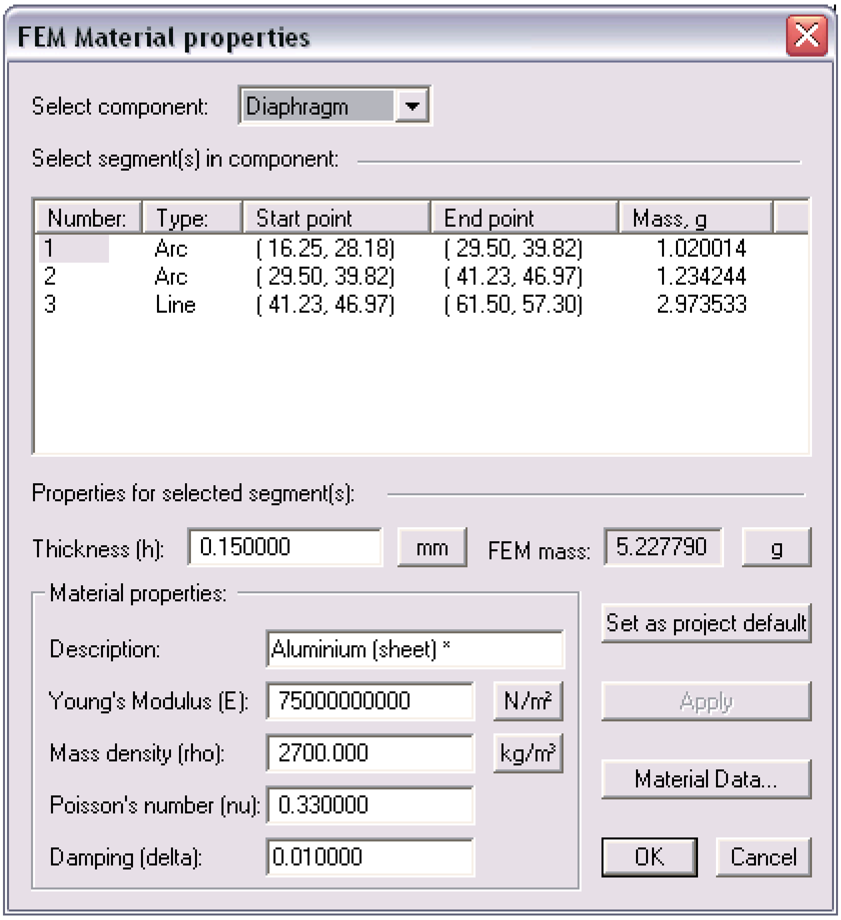


Figure 81 - The FEM Material properties window showing the diaphragm sections

The diaphragm is for this example divided into 3 segments. Basically, this diaphragm is designed with a large radius (arc) which is connected to a line (see Figure 80). The large arc, however, was divided into two arcs both connected to the dust cap.

The cone material is chosen as Aluminium [sheet]. The * indicates that the material from the database is changed by increasing the damping from 0.05 to 0.1 to model the actual speaker material correctly.

The surround material is obtained from the Material Database, shown in Figure 75, selected by the button on the lower right.

The “Rubber” material is selected which has the values for a typical surround rubber material.

Note: you can edit the materials or add new materials in the database at any time.

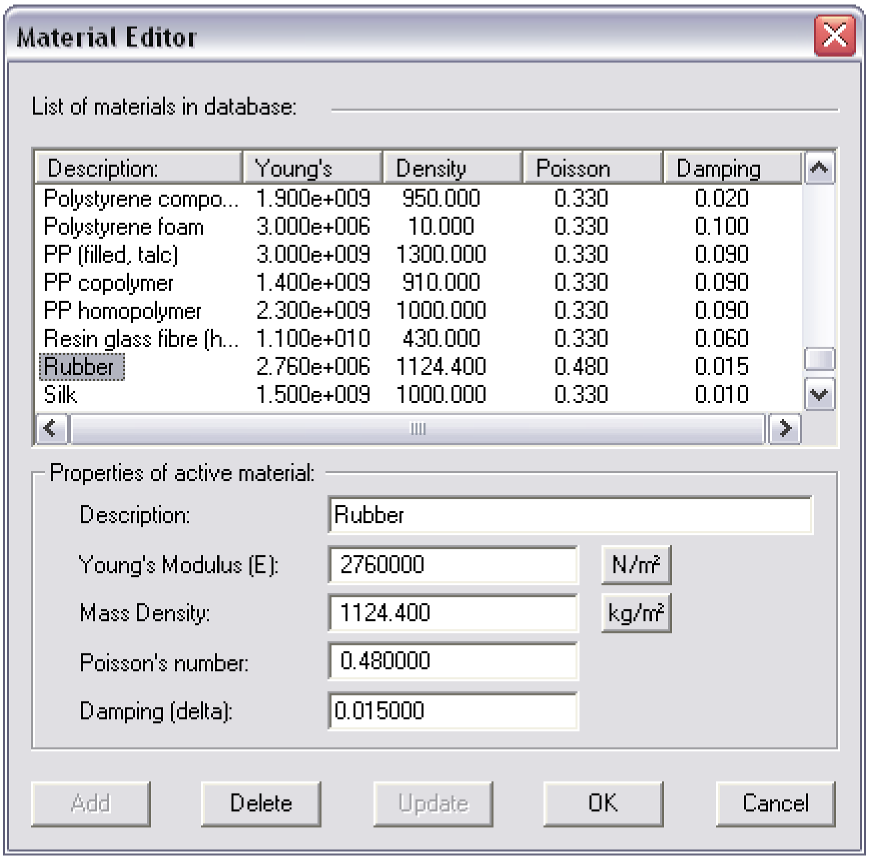


Figure 82 - Material editor window with a long list of pre-set materials

Now the electrical parameters should be entered. Here the values from FINE R+D were used first. To help the user to match an existing impedance curve, a measured impedance curve can be imported by selecting “Advanced Settings”

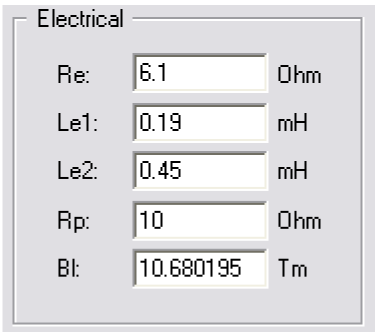


Figure 83 - Electrical Input

If you have a FINEMotor file (*.FM3) it can be imported directly into FINECone. See also TS parameters section 1.6.5. You may transfer Le1, le2 and Rp in the TS dialog, Figure 84.

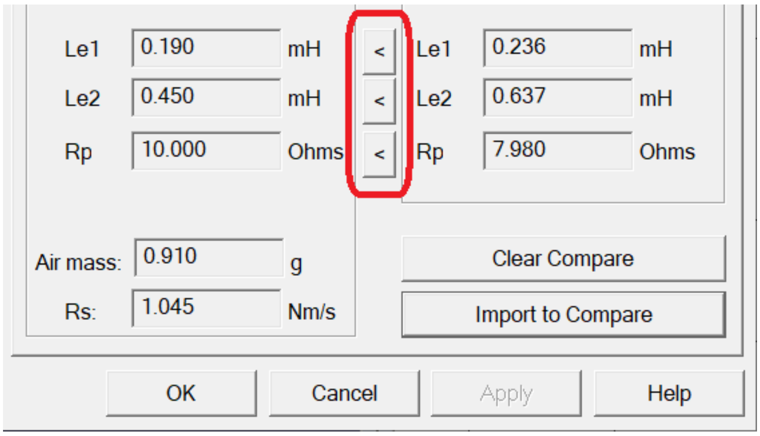


Figure 84 - TS Parameters: Including Le1, le2 and Rp from file

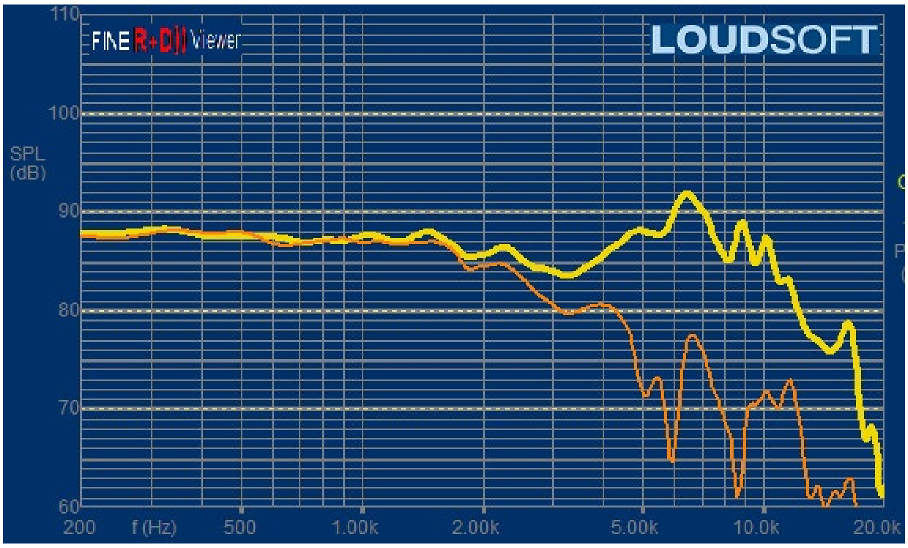


Figure 85 - The (0/30deg) response is measured with FINE R+D. The on-axis curve is imported in FINECone

The resulting frequency response is shown in Figure 86.

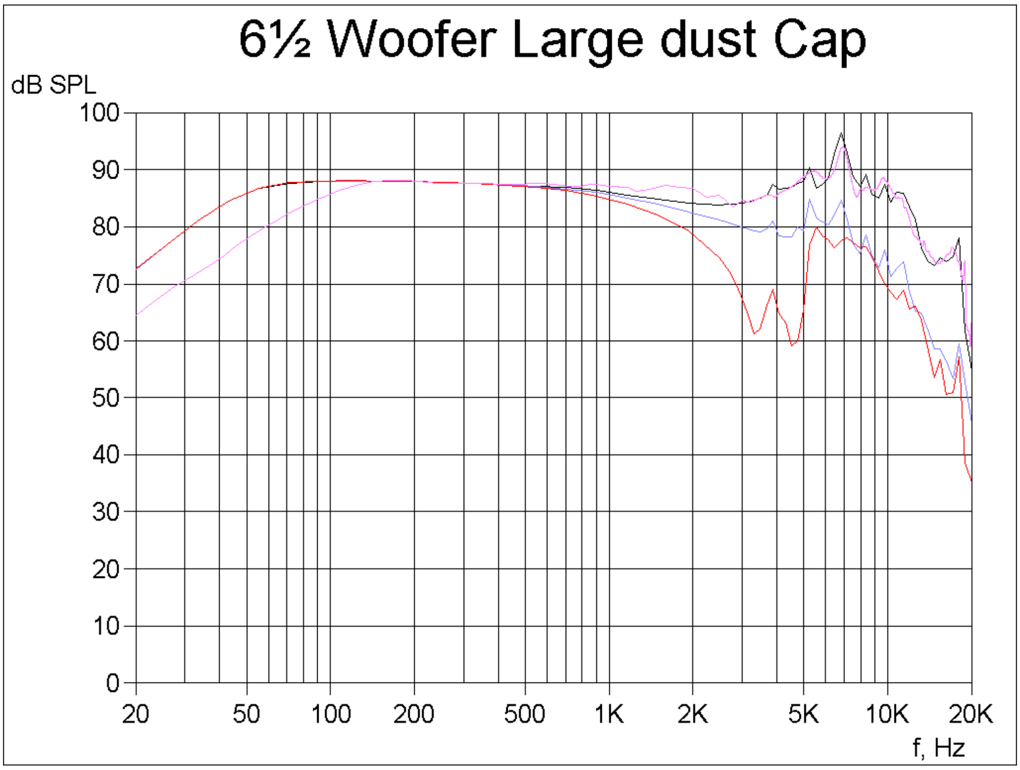


Figure 86 - The agreement between the calculated FINECone response (black) versus the measured response (magenta) is remarkable at high frequencies (break-up region)

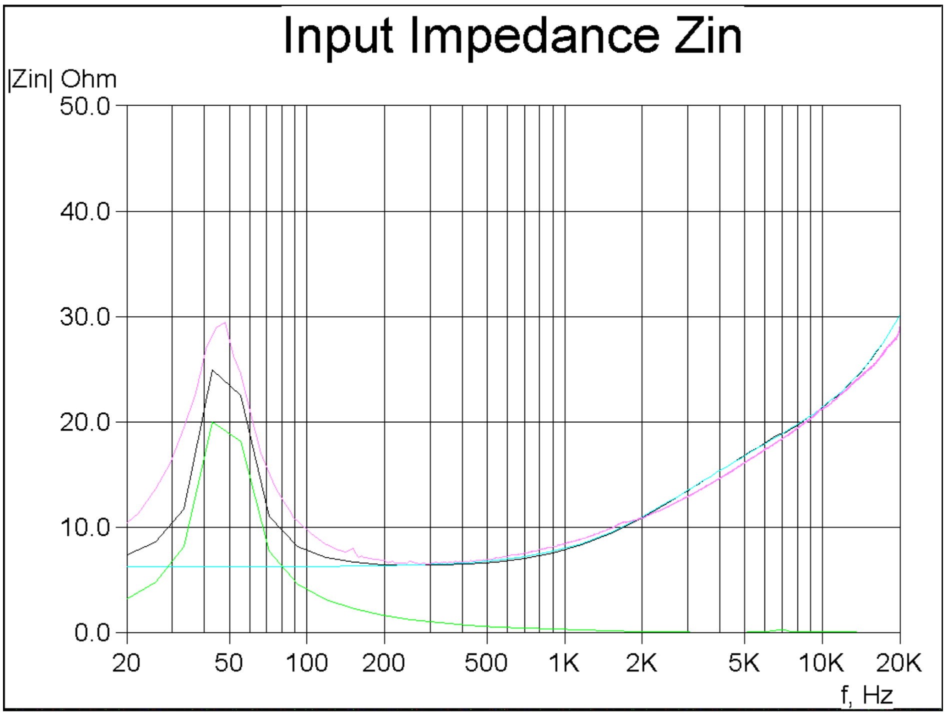


Figure 87 - The blue curve is calculated electrical impedance, the green curve is the mechanical impedance, and the black curve is the blue and green curves summed up to give the total impedance. The magenta curve is the imported impedance curve for comparison.

We now look at the 3D animation to get an idea of where the break-up is happening in the driver for the peak around 7 kHz. The frequency is set to 7162 Hz in the 3D animation drop down menu and the result can be seen in Figure 88.

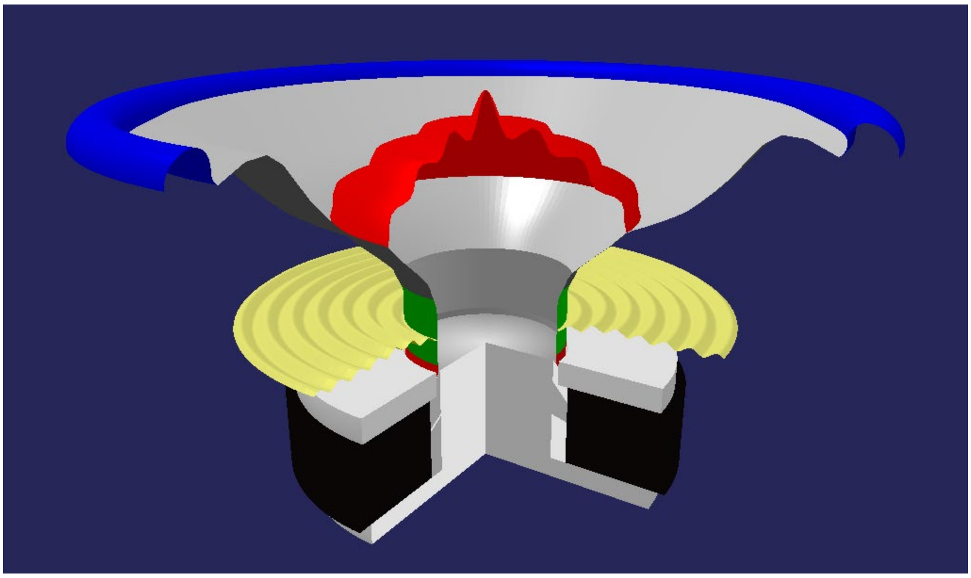


Figure 88 - This is the 6.5" woofer break-up animated at 7162Hz, which is the frequency of the large peak. Note the heavy break-up in the outer part of the cone.

The amplitude is set to 7mm as shown in Figure 89. Selecting the actual amplitude is not ideal for high frequencies as the excursion is extremely small.

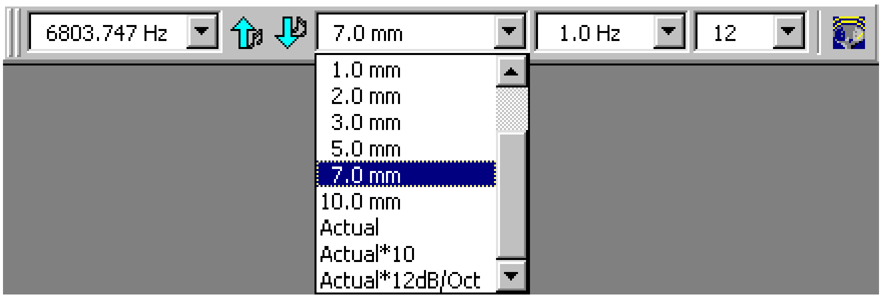


Figure 89 - The amplitude is set to 7.0 mm to better visualise the breakup

Excluding every part from the model except for the dust cap by using the buttons shown in Fig. 90, we can now see that it is providing the main contribution to the peak around 7 kHz as shown in Figure 91.

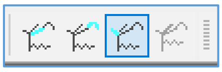


Figure 90 - Excluding all parts of the model except for the dust cap

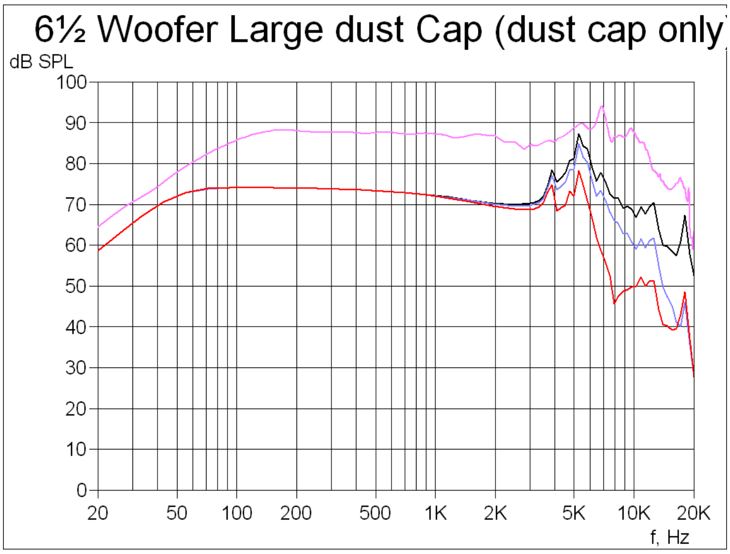


Figure 91 - Response of the large Dust Cap ONLY. The dust cap has a large peak around 5kHz.

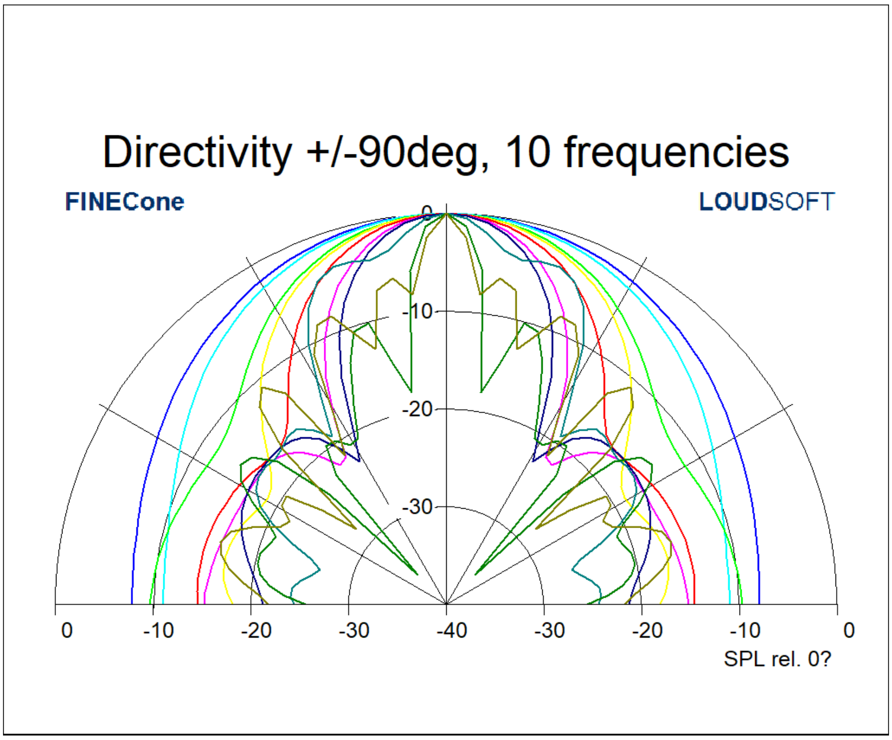


Figure 92 - Directivity at 10 frequencies

For displaying the directivity, it is recommended selecting 10 frequencies, and limited frequency range Figure 92. See the Directivity example: Curvilinear cone woofer -Dispersion Plot 2-20kHz.fcp

2.5 165mm woofer with edge problem

In the following example we have modelled a 165mm woofer which has a severe response problem around 1300Hz.

The measured response is imported and shown as the pink response. The low end measured response is different from the FINECone simulation because the driver was measured in a small baffle.

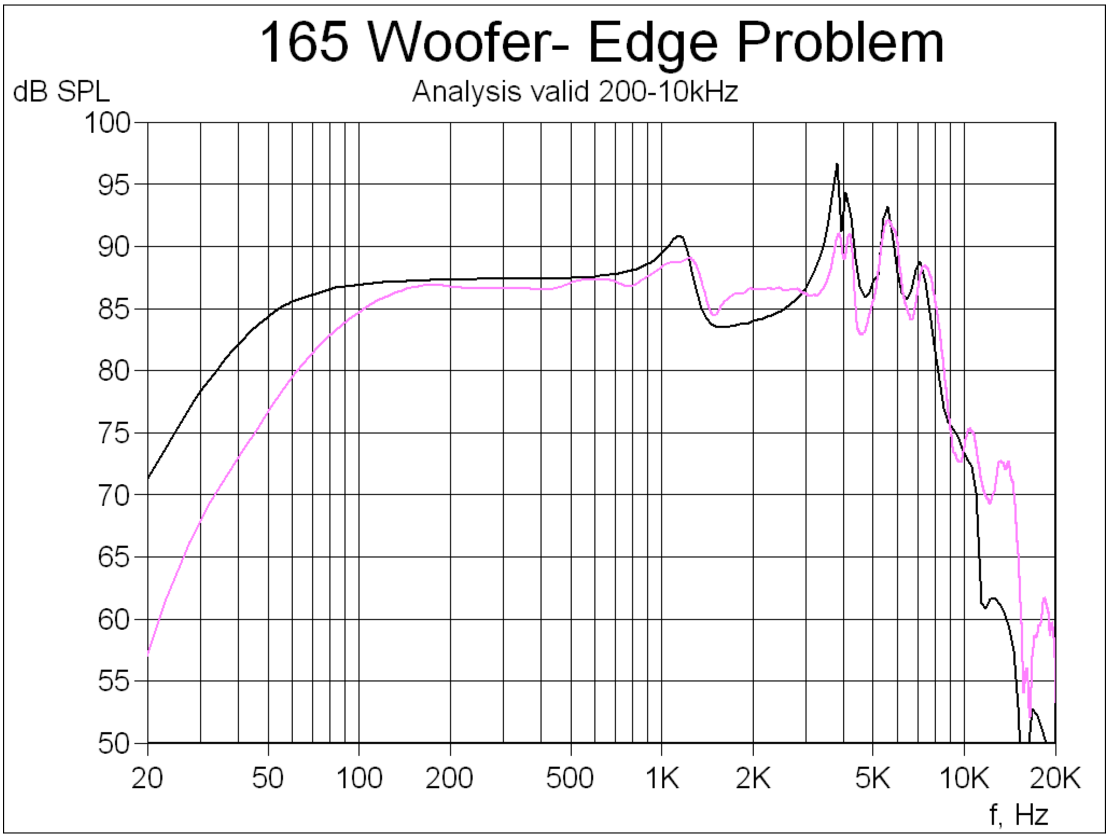


Figure 93 - Frequency response of the 165 mm woofer with an edge problem

The FINECone simulated response in Fig. 93 fits the imported measured response quite well. There is much break-up from 3-8 kHz, but we will concentrate on the peak and dip around 1300 Hz since it is quite annoying and very difficult to handle in the crossover.

The Project Geometry is shown in Fig. 94. The red dots indicate intersections between segments. Note that we have split the surround into 5 segments. All 5 segments have the same thickness 0.4mm, which can be seen in the FEM Material properties in Fig.95.

Note: Many simulations will show a lower SPL in the range 700-3000Hz. This is normal and a result of the chosen Rayleigh calculation method

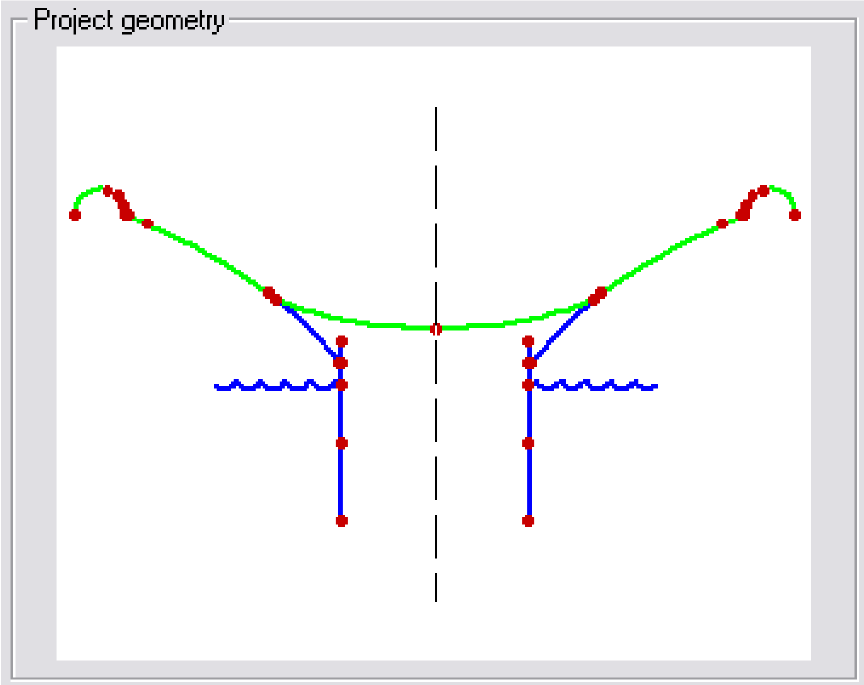


Figure 94 - Geometry for the 165 mm woofer with 5 segments in surround

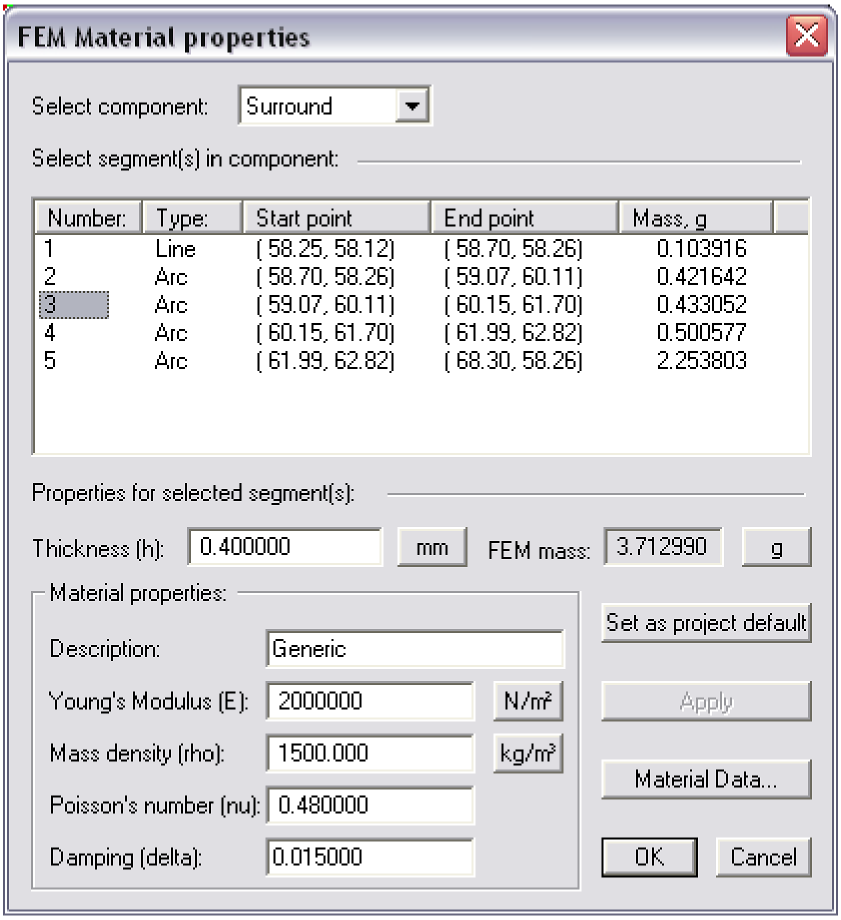


Figure 95 - Material properties for the surround

To find out what is happening around 1300 Hz we have this time used 2D animation, which is sometimes better at showing where the maximum movement of the components is. Fig. 96 shows the cone edge and surround is moving excessively (brown curve).

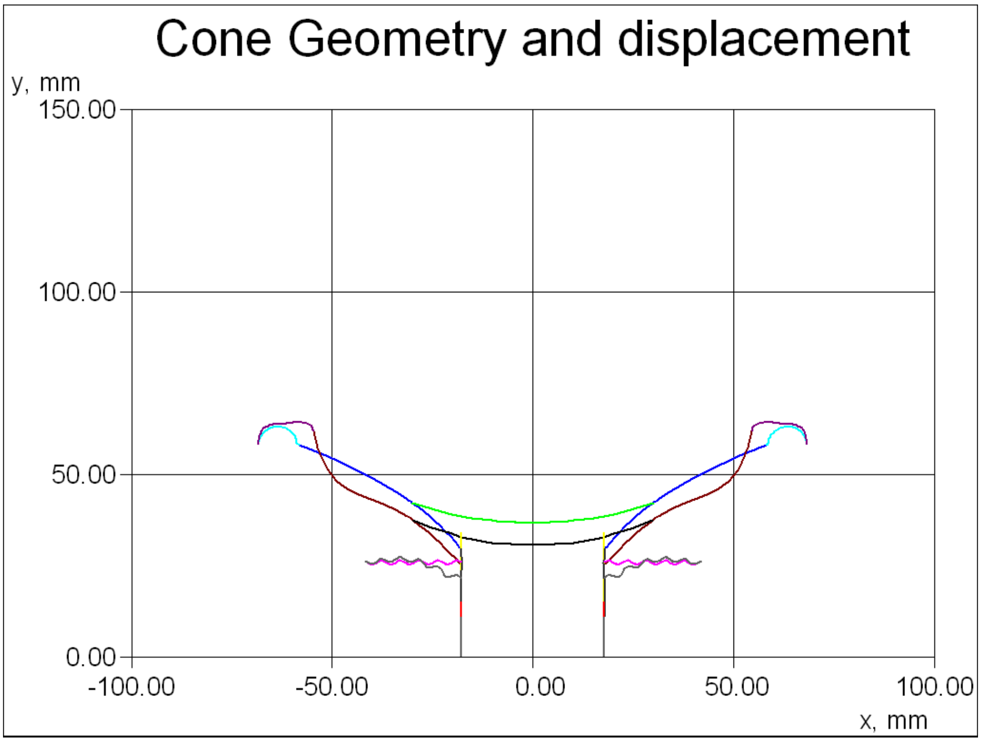


Figure 96 – 2D plot with displacement (brown), maximum at cone edge.

There are many ways to correct this problem, for example by changing the cone profile to a larger cone angle or change the geometry or thickness of the surround. Here we will change the thickness of the inner part of the surround.

In Material Properties we select segments 1, 2 and 3 and change the thickness to 0.8mm. After Apply and OK the calculations are done automatically.

The new simulation, shown in Figure 90, exhibits a much smoother response around 1300 Hz. The pink curve is showing the response before the change was applied. That response was exported as an FSIM file. This file was then imported after the changed surround was calculated.

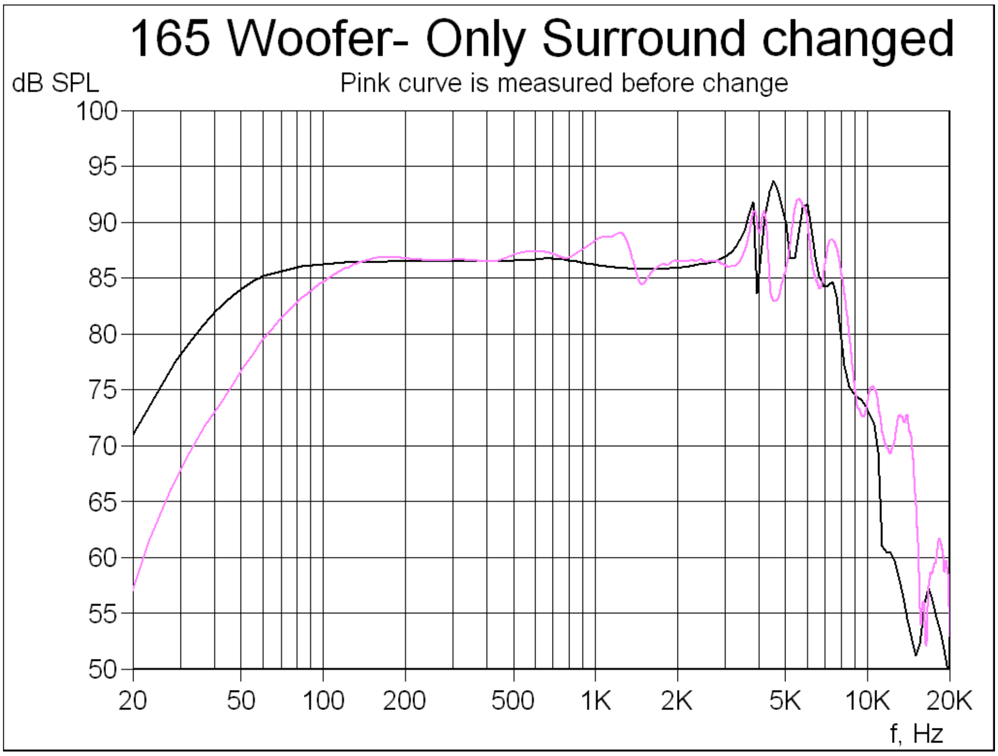


Figure 97 – 165 mm woofer with increased thickness of inner surround

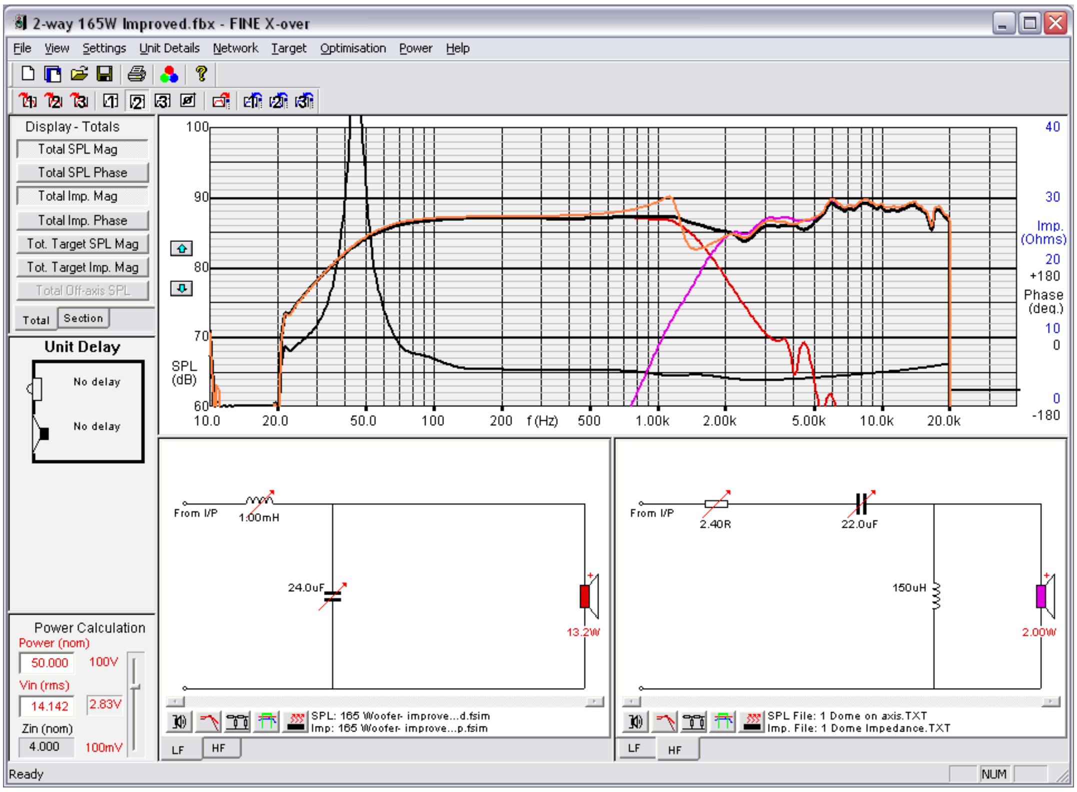


Figure 98 - FINE X-over using the frequency response and impedance exported from FINECone. Orange curve is with the bad woofer

Fig. 98 shows a screen plot from FINE X-over where we have used the exported responses from FINECone as input for the woofer section. The orange response is using the 165W before the simulated change. The final response (black) is much improved.

2.6 38mm headphone transducer

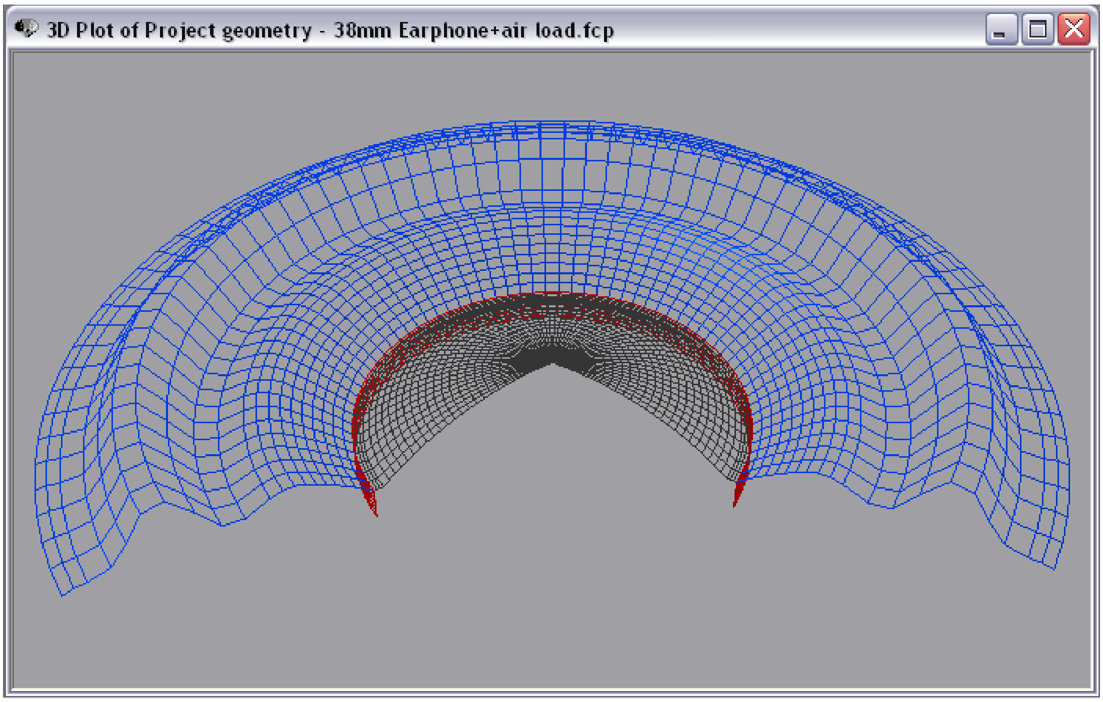


Figure 99 - 38mm Headphone transducer simulated in FINECone with break-up at 3165 Hz

The 38mm headphone was first modelled in FINECone with only the main acoustical parts: Diaphragm inside (dome) and diaphragm outside (surround) and voice coil. The diaphragm is 25u PEI which is used for both dome and surround since the diaphragm is made in one piece.

The resulting response is here shown as the pink curve in Fig. 100. There is serious break-up from approximately 3000 Hz and the first mode is shown as 3D animation in Fig. 99. This first break-up mode is showing up in the middle of the outer diaphragm (surround) where it is almost flat.

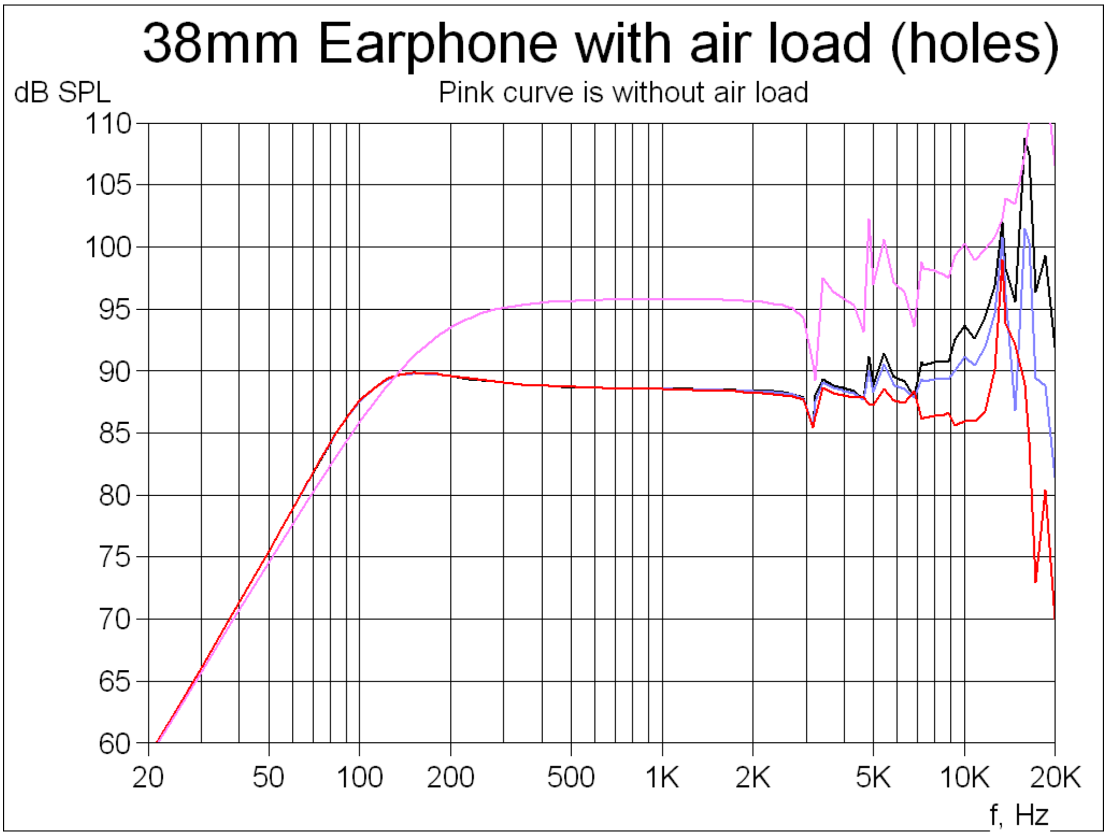


Figure 100 - A 38mm Headphone simulated frequency response with air load (rear holes)

The actual transducer has several holes behind the outer diaphragm/surround all covered with a cloth acting mainly as damping material. The net effect of this may be calculated as an effective air load mass using the well-known Helmholtz formula. We can incorporate this air load mass in the FINECone simulation by adding it as “Air load” in Lumped Parameters. The main curve in Fig.100 is showing the resulting response, which is some 7 dB lower in SPL due to the extra load mass.

We also note that the effective Fs is reduced from approximately 180 Hz down to 100 Hz with the air load mass.

Demo video:

https://youtu.be/nlaGb67RPwc

2.2 Appendix A

The DXF files can be read in Notepad or other text editors.

Fig. 101 shows the first lines of a DXF file created by FINECone. The line 999 is showing which FINECone Template type is used. Here it reads LOUDSOFT_FINECONE_TEMPLATE003, meaning that the FINECone Template type was 003.

Micro34.dxf - Notepad

File Edit Format View Help

```
 999
LOUDSOFT_FINECONE_TEMPLATE003
  0
SECTION
  2
HEADER
  9
$ACADVER
  1
AC1009
  0
ENDSEC
  0
SECTION
  2
TABLES
  0
TABLE
  2
LTYPE
 70
     1
  0
LTYPE
  2
CONTINUOUS
 70
     0
  3
Solid line
 72
    65
 73
     0
```

Figure 101 - FINECone Template type 003, shown in Notepad

2.3 Appendix B

Exporting frequency and impedance responses are by default in the standard LOUDSOFT format FSIM and are slightly smoothed. These import well into other FINE software, like FINE R+D, FINE X-over and FINE DSP etc.

The legacy FINECone format can be varied in many ways, that are different from the FSIM. In that case you can export the exact responses from FINECone, se Fig.102. However, these may not be read correctly in other software.

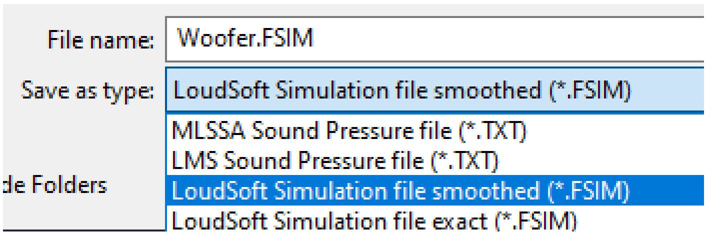


Figure 102 - Alternative export formats

LOUDSOFT

www.loudsoft.com