Manual

FINECone™

Acoustic Finite Element Dome/Cone Simulation Program

Contents

1. 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

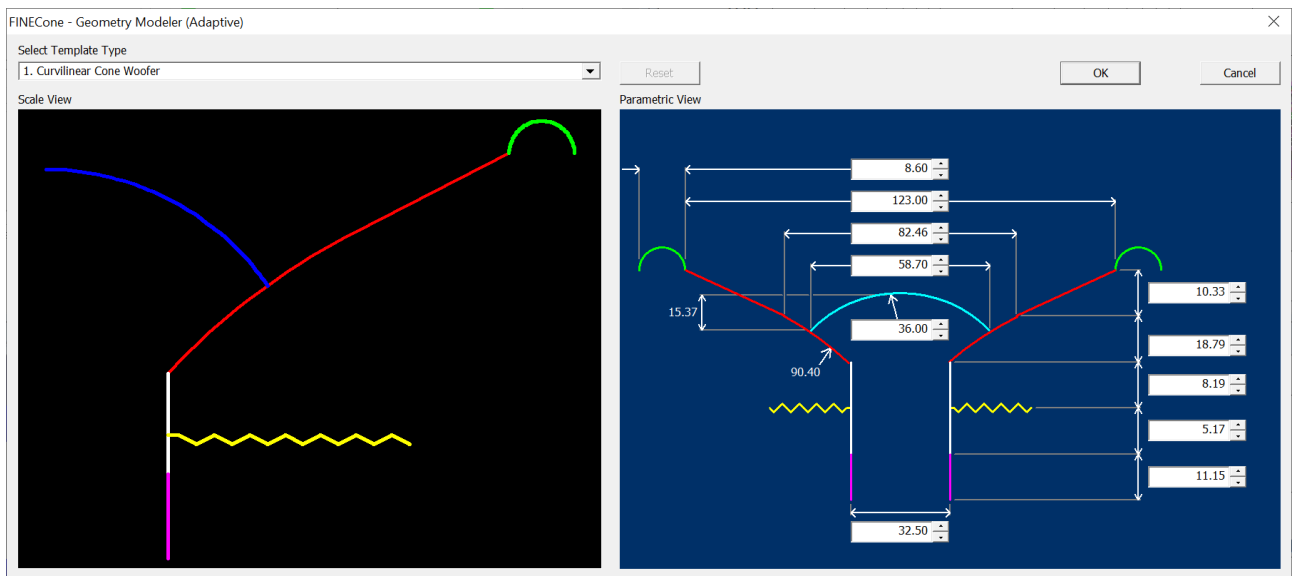## *1.1* Geometry Modeler 🏰 / *New*

Figure 1 - 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 Figure 2 you simply select the template type, which is closest to the speaker geometry you want to simulate.
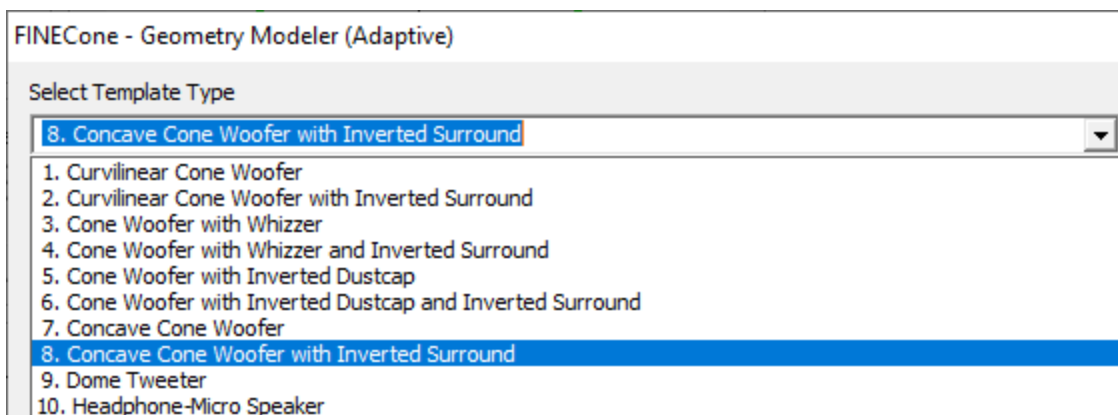

Figure 2 - Select Template from drop-down menu

Figure 1 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 window at right. 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 3. 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!
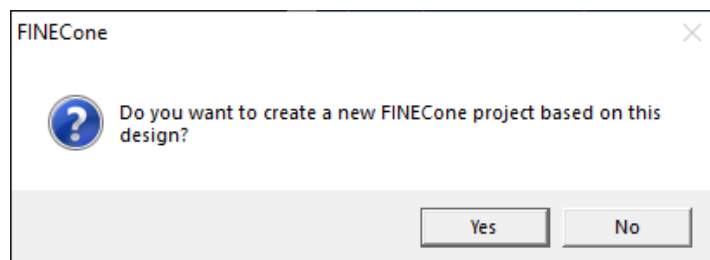
Figure 3 - Auto-creating your FINECone project

1.2   DXF geometry: How to change or make your own

If you said yes to automatic project creation, you 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 4 shows a section view of a woofer. The same woofer is shown at right when prepared for FINECone analysis.
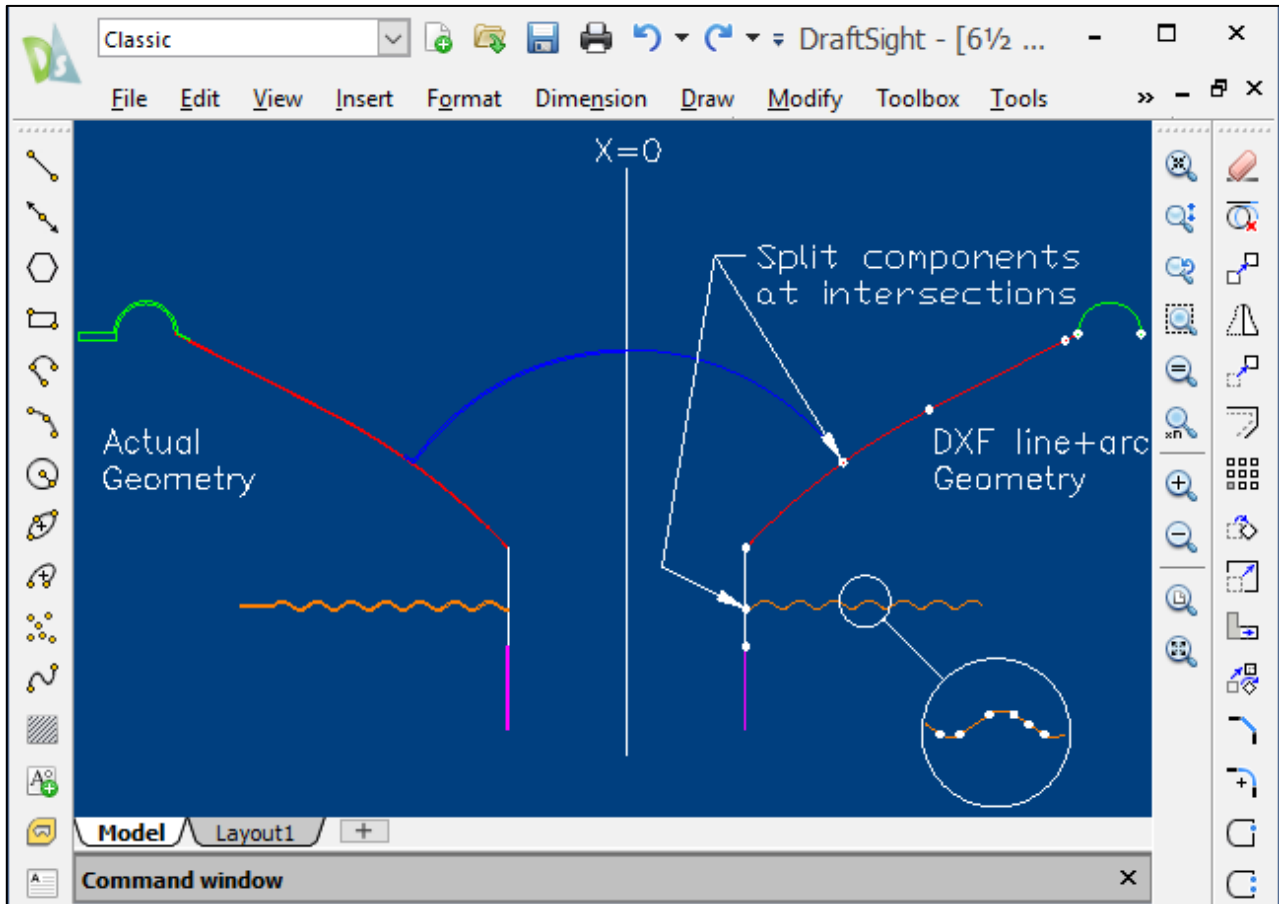
Figure 4 - DraftSight drawing - Left: Actual geometry - Right: DXF line + arc geometry

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 1.21) and Figure 4

    c. Be sure to keep the changed component in its correct layer

    d. You can for example split a line in a number of (small) segments, which allows you to model a tapering cone thickness

    e. Or you can change radii or intersections.

    f. 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 1.21) and Figure 4

    c. You can only define up to 7 components, see Figure 6

Figure 4 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.

An example DXF file is shown in Figure 5. Each component like cone and dust cap etc. are placed in their **own layer**.
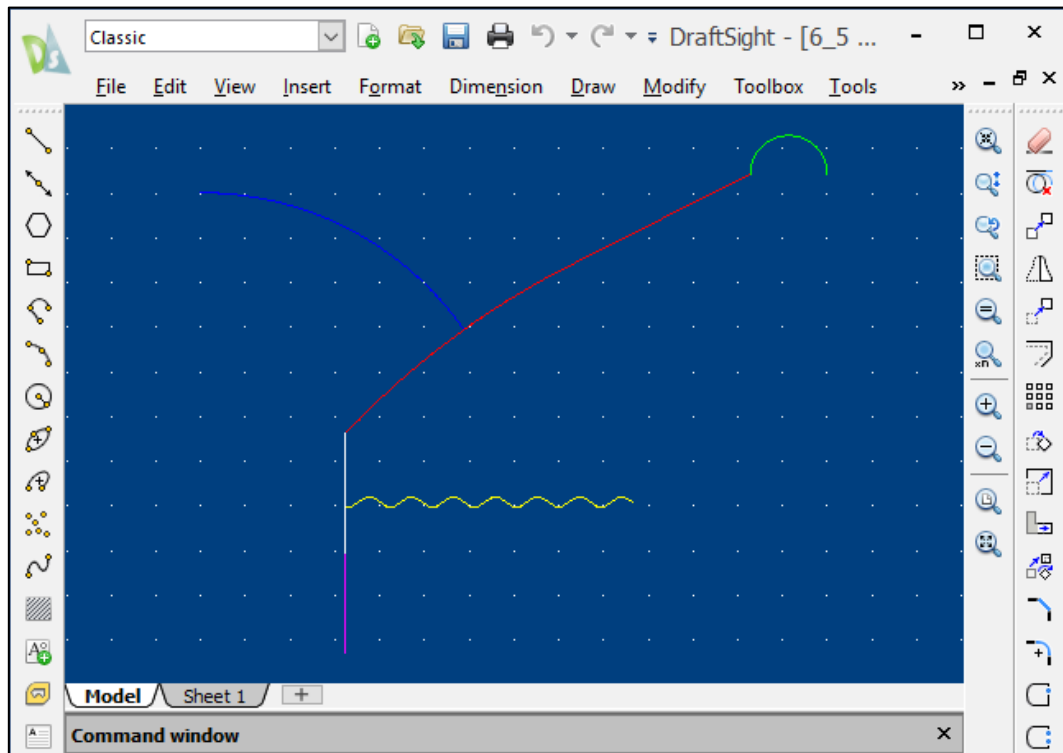

Figure 5 - FINECone reference DXF in DraftSight

1.2.1 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 or poly-lines 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 starting with 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 35.

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 safer 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 5. 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 7, 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 7 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 6)

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.
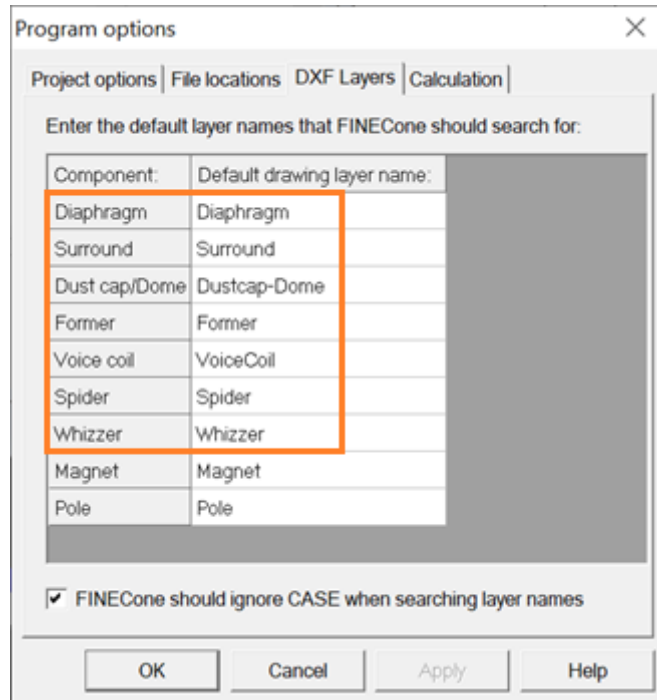
Figure 6 - DXF: Default Layer names (1-7 Main Acoustic components in frame)

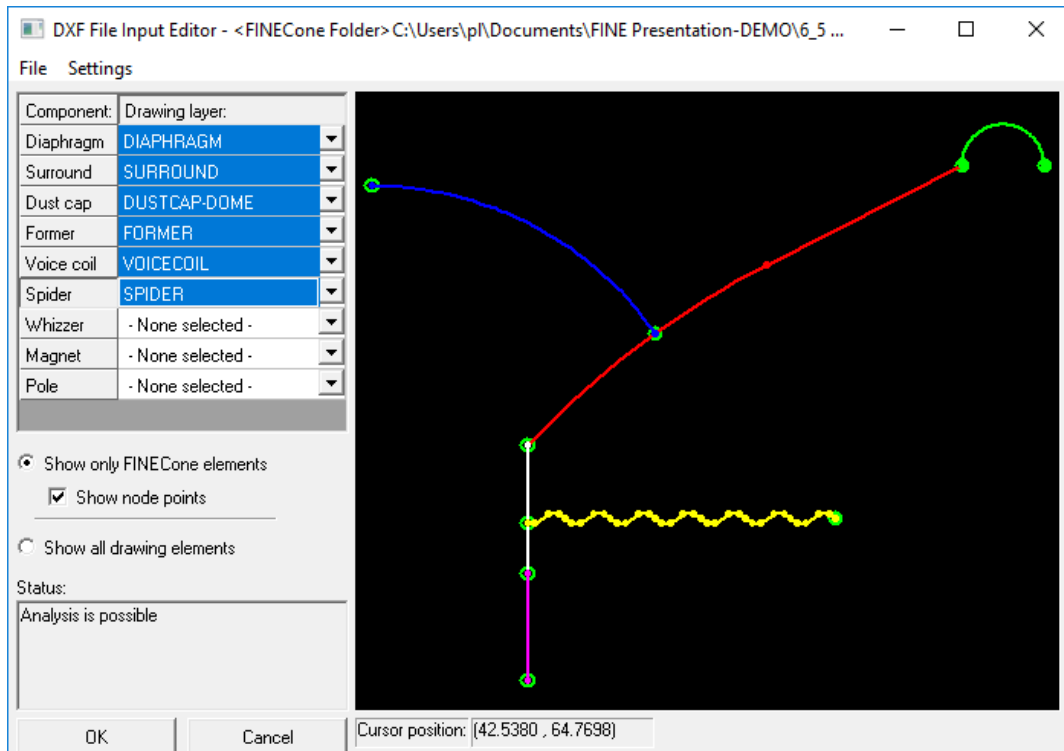The default layer names are defined in Tools/Program options/DXF layers, Figure 6

Figure 7 – Example FINECone Geometry

1.3   Project information window

1.3.1 Geometrical properties

Figure 8 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.
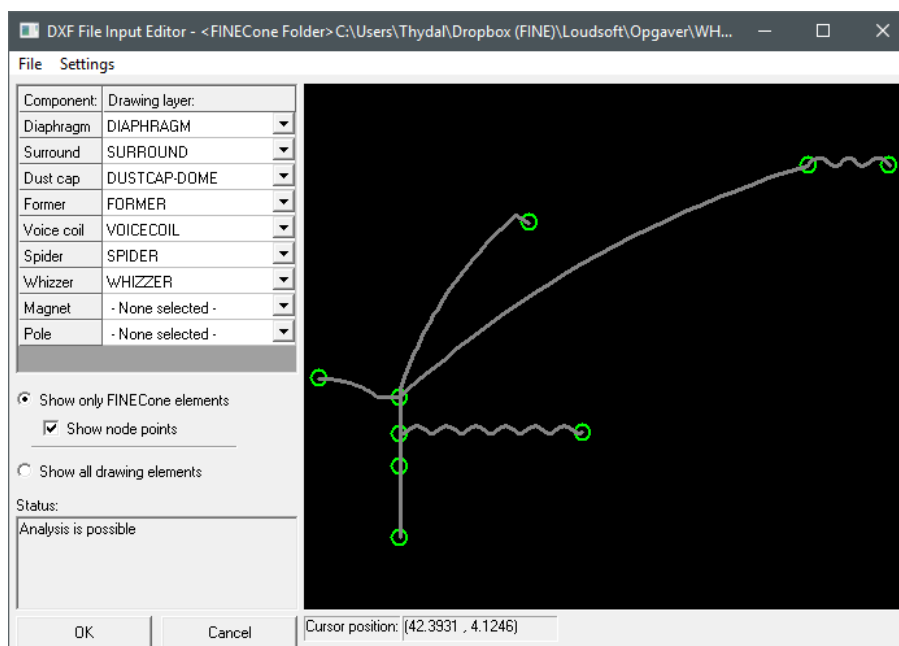

Figure 8 - The geometrical properties of a 6.5-inch woofer with a whizzer cone

1.3.2 Material properties

The Material Properties Window Figure 9 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.
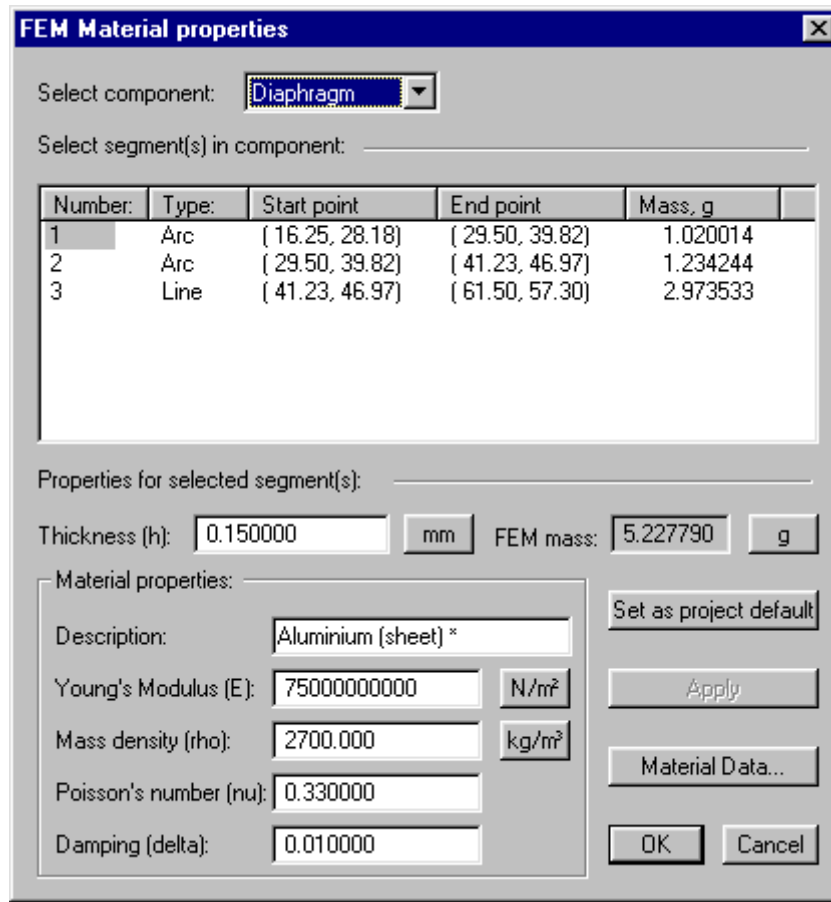
Figure 9 - Material properties can be defined for each individual segment in each component

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.

1.3.3 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 up 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.
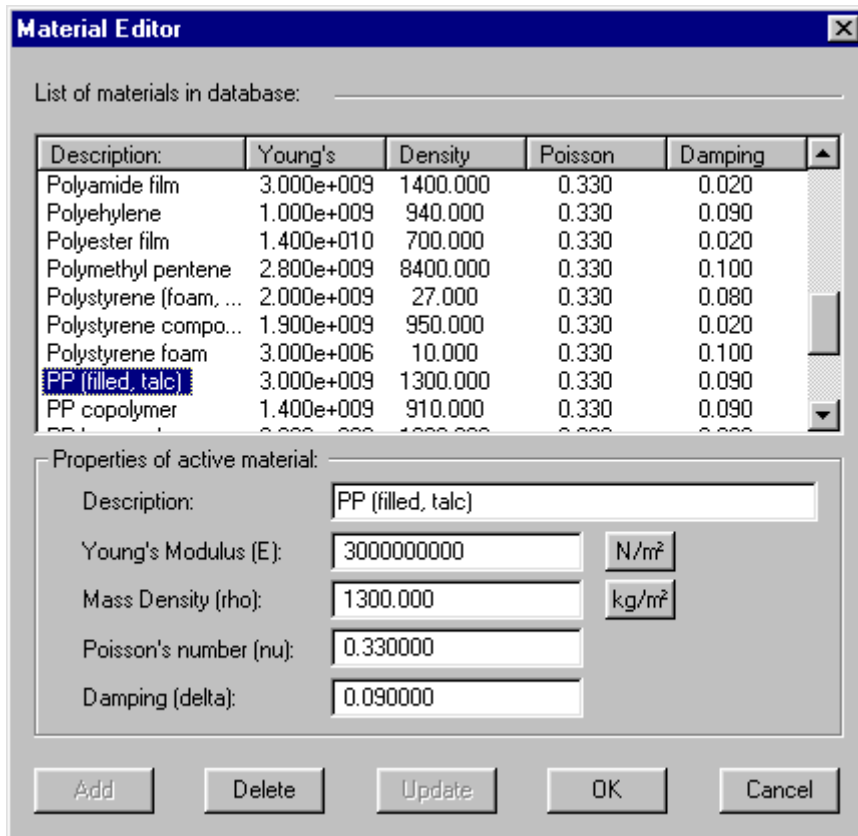

Figure 10 - Material database with many predefined, common materials

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.

1.3.4 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.

1.3.5 TS Parameters (Lumped elements)

Figure 11 - Thiele-Small (TS) parameters. The right column is imported from FINEMotor or FINE R+D. Press < to use Re, BL, le1, le2 and Rp in FEM calculations

The FINECone FEM (Finite Element Model) is very advanced and detailed Figure 11. In contrast the TS Parameters are using only simplified lumped elements. However, the TS parameter button 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.

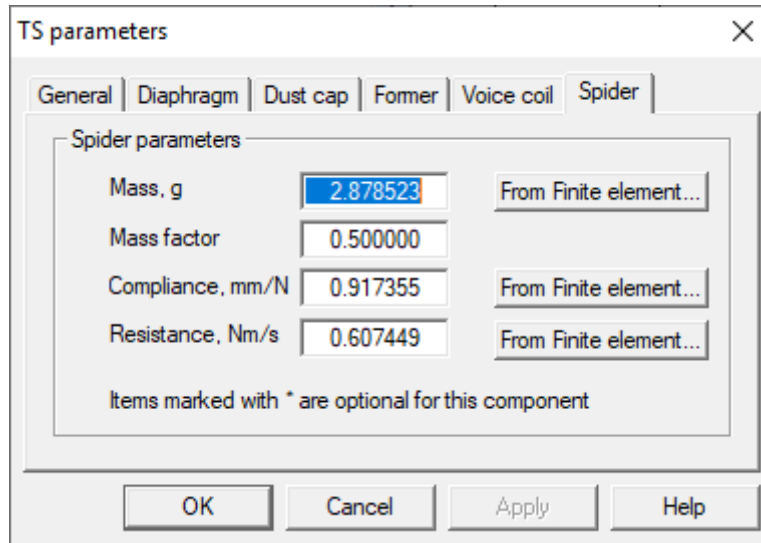*Note: The FEM model often uses Zmin instead of Re, as that matches the impedance curve better*

Figure 12 - 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 12. HOWEVER, you must first exclude the surround, see Figure 13. (This is to avoid having both spider and surround stiffness together). A spider stiffness ~700MPa is close to 1mm/N.

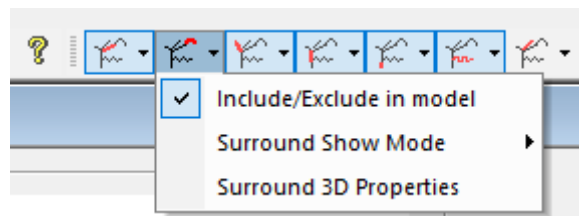Similarly, you may exclude the spider for calculating the surround stiffness.

Figure 13 - Exclude Surround

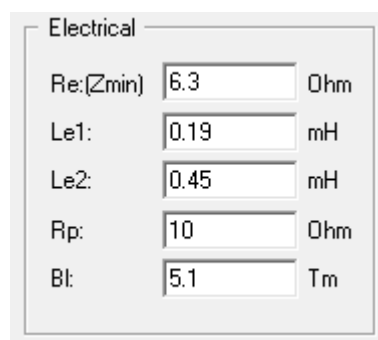## 1.3.6 Electrical properties

Figure 14 - The electrical parameters with Re and Bl imported (from FINEMotor)

In the Electrical Properties window Figure 14, the user can input basic loudspeaker parameters Re and BL. These can also be inserted from TS parameters, see section 1.3.4, 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.

In addition, the user can also choose to import a measured impedance curve by right-clicking the Impedance curve, and selecting: [Import measured Impedance], Figure 15.
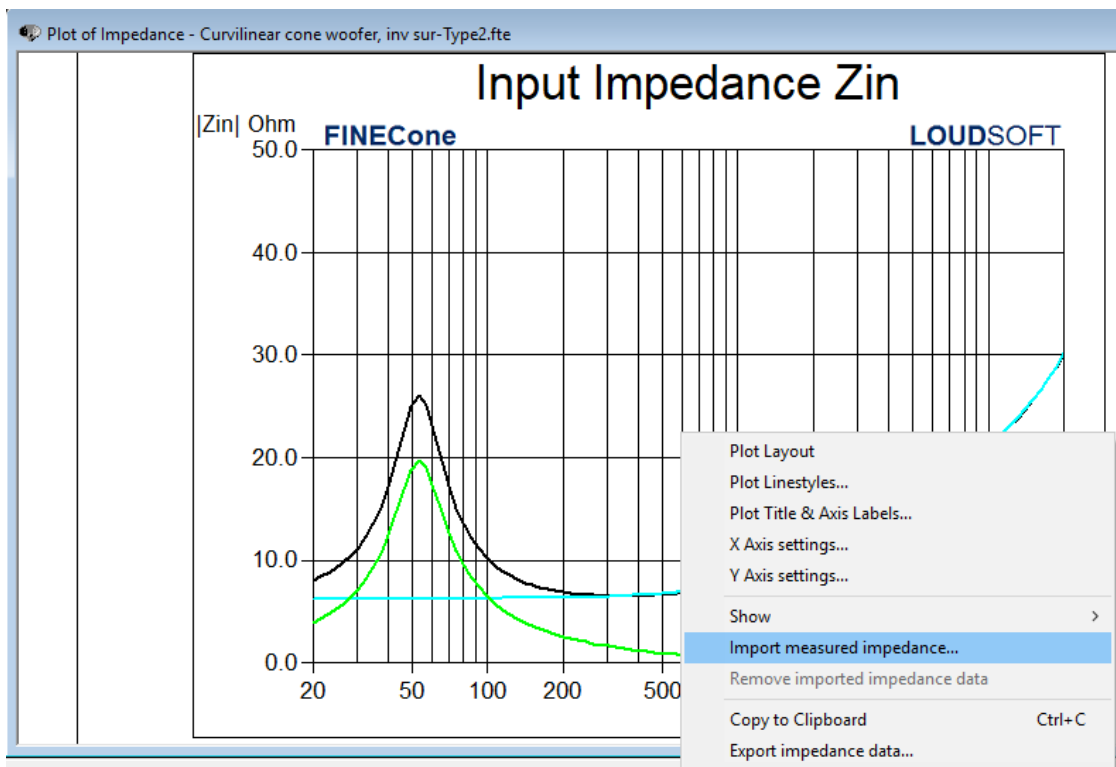

Figure 15 - Import measured Impedance curve

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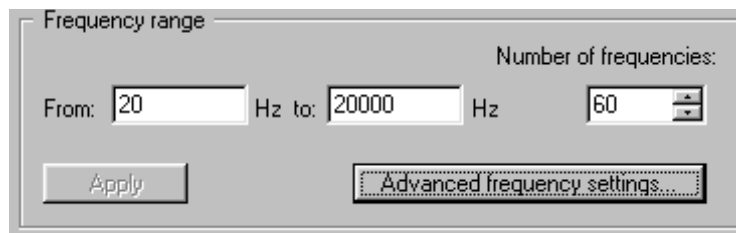
1.3.7 Frequency Range

Figure 16 - 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 16. 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 18.
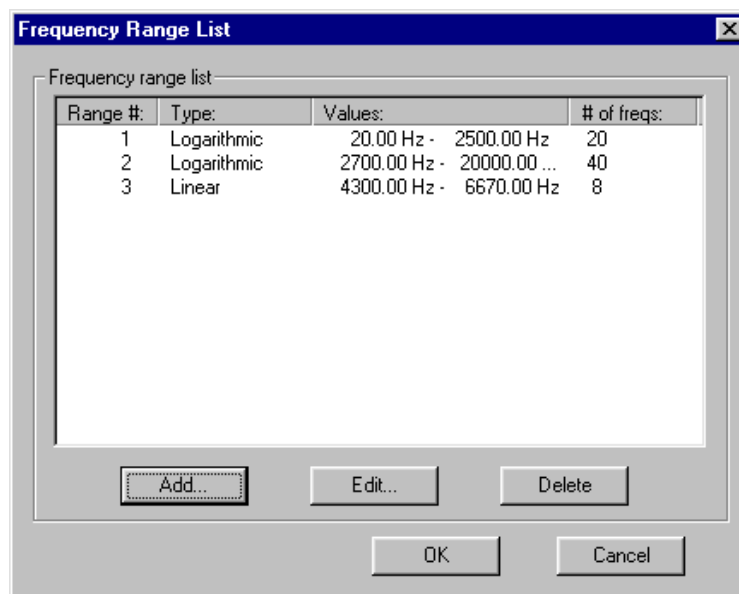

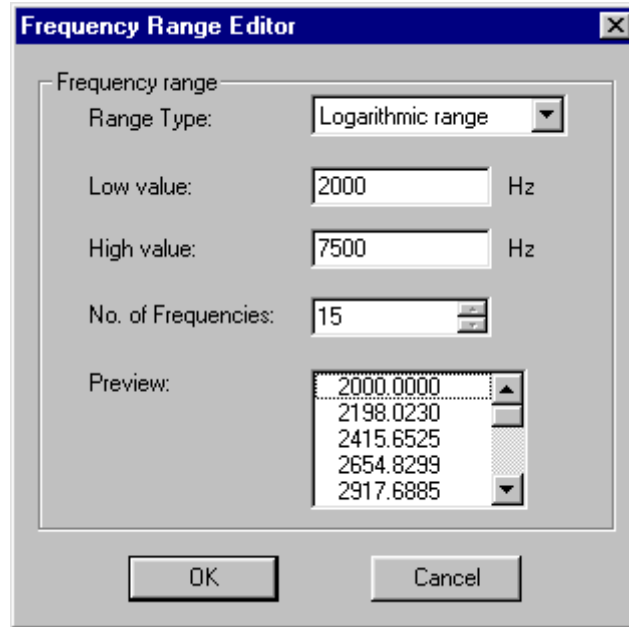**Figure 17 - List of pre-set frequency ranges**

Figure 18 - Window allowing you to pre-set new frequency ranges

1.4   Impedance window

The impedance window in Figure 19 shows three simulated curves. The green curve represents the mechanical impedance of the driver, which is mostly dominated by the suspension. The teal 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 1.3.5.
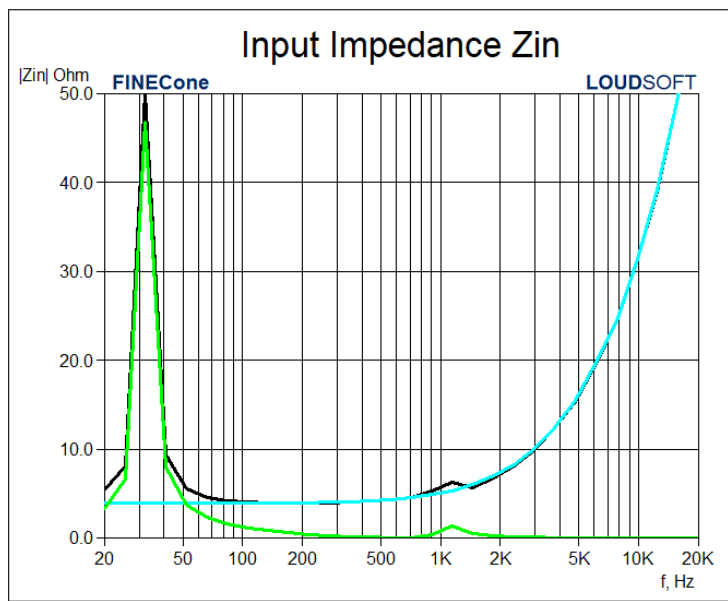
Figure 19 - Impedance window showing the impedance of a 10-inch subwoofer

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

1.5   Sound Pressure Level (SPL) window

The Sound Pressure Level window shown in Figure 20 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 right click on the window and select from the drop-down menu.
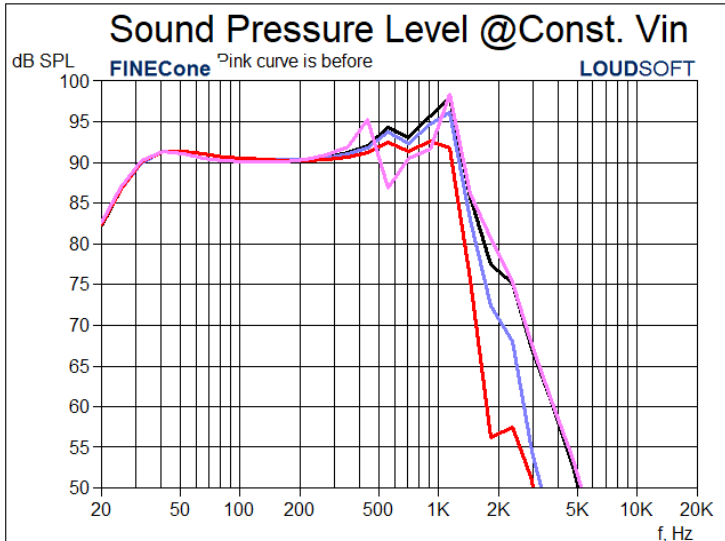

Figure 20 - The Sound Pressure Level window shows on-axis as well as two off-axis simulated curves of a subwoofer. A measured frequency response (Pink) has been imported by right-click.

1.6   Animation window / 3D Geometry + Displacement

Figure 21 – Select 3D Geometry + Displacement

The animation window (3D Geometry + Displacement) Figure 21 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.
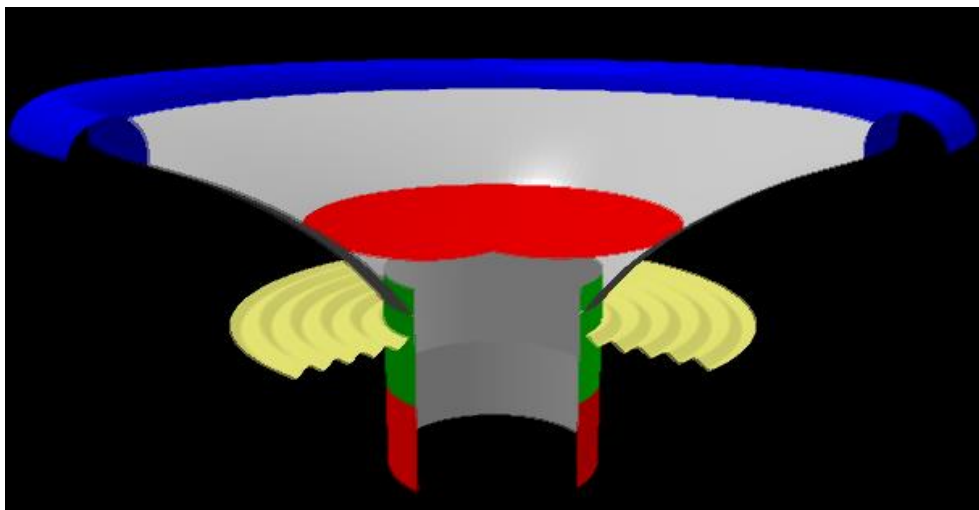

Figure 22 - The 3D animation window visualises the behaviour of the cone at different frequencies. Press to animate.

The button at right in Figure 24 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 in 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 23 - Drop down menus for the animation window

The first button from the left in Figure 24 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 25.
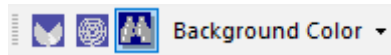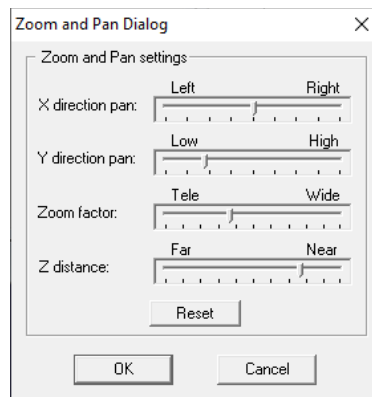

Figure 24 - Buttons for displaying the animation

Figure 25 - Zoom and Pan settings

1.7   Toolbar

1.7.1 Import from FINEMotor

It is possible to import the Thiele-Small parameters from FINEMotor by clicking on ![icon], see also 1.3.4. 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 ![icon].These are only for display.

1.7.2 Window selection

The different windows available in FINECone can be toggled on and off using the buttons shown in Figure 26. 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 26 - The six buttons for the different windows in FINECone

1.7.3 Include/exclude mechanical parts

It is possible to include or exclude different parts of the model using the buttons shown in Figure 27. 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 in order to find the compliance of the spider (see also figure 12). Or if you have two identical drivers with and without a whizzer cone and wish to see the difference.

Figure 27 - 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 28.

Figure 28 - Include or exclude the contribution of different parts of the model in the frequency response

1.7.4 2D plot

FINECone can also show 2D plots of the model with and without displacement Figure 29. This method may be preferred to the 3D animation as it can make it easier to locate the breakup.
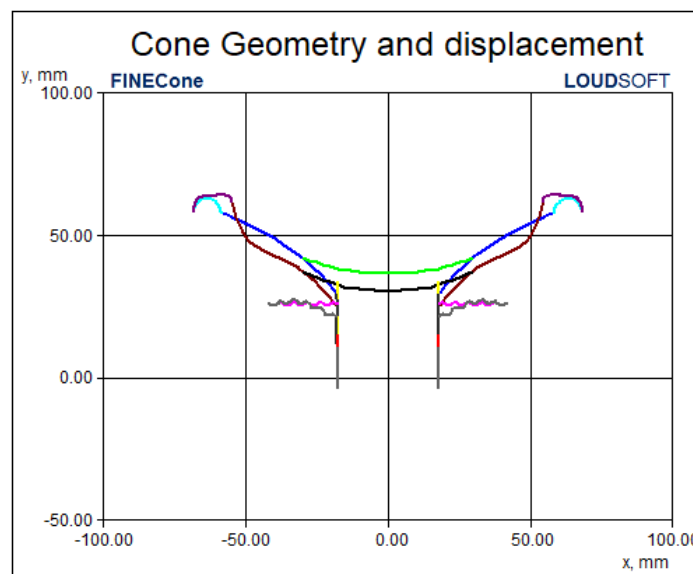

Figure 29 - 2D plot showing the model when at rest and when driven at 1863 Hz

2. Design Examples

2.1   Simulation of a 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 quasi-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 1.1** or creating your own DXF file, section1.2.

We recommend the CAD software DraftSight for making DXF files. 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 1.2.1 how to make the DXF file. Please note that the DXF file is simplified, especially regarding the VC former (Figure 4 and Figure 5), 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.
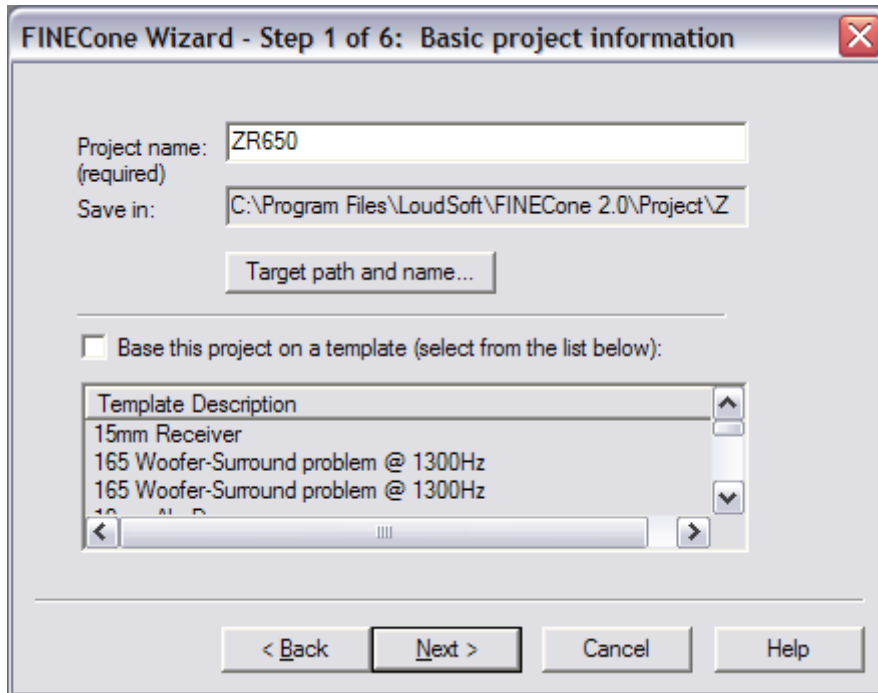
Press ✏ button to start the FINECone Wizard.

Figure 30. FINECone Wizard

If you cannot use the **Geometry Modeler section 1.1**, we strongly recommend selecting a template file (.FTE), or an earlier simulation as a start.

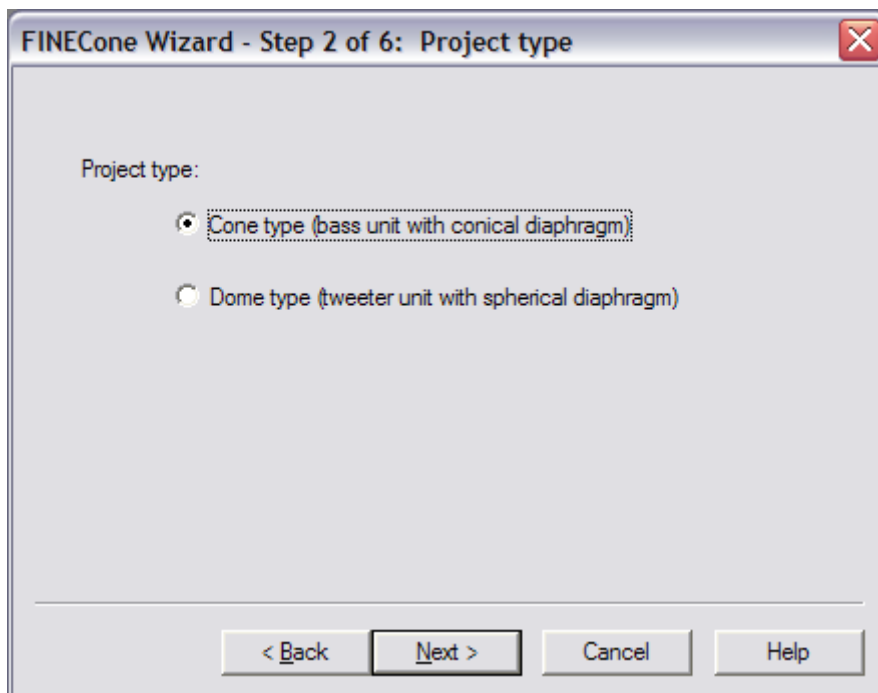However, if the analysis is new, we therefore continue with next.
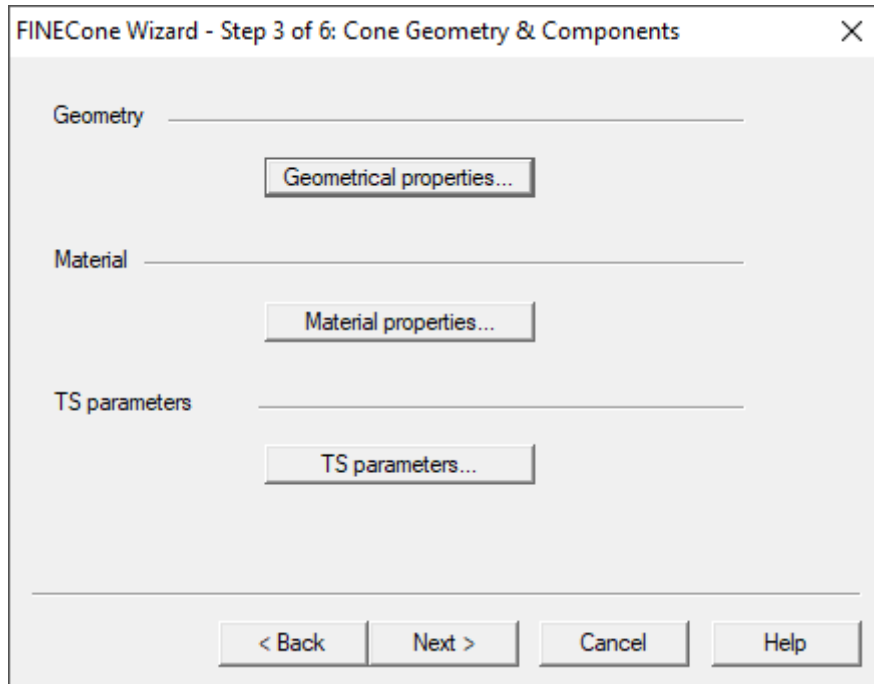

Figure 31. Choose Cone type (Project Type)

Figure 32. General FINECone Steps

The 3 buttons indicate the general procedure:

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

Press ⌜Geometrical properties...⌝ button to input the geometry of the driver.

After opening the DXF file (Figure 33) we find that two rows are not selected because the names of those layers are not the same as the default names. We have to choose dust cap and voice coil layers by finding the layer where the component is from the drop-down menu as shown in Figure 34.

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.
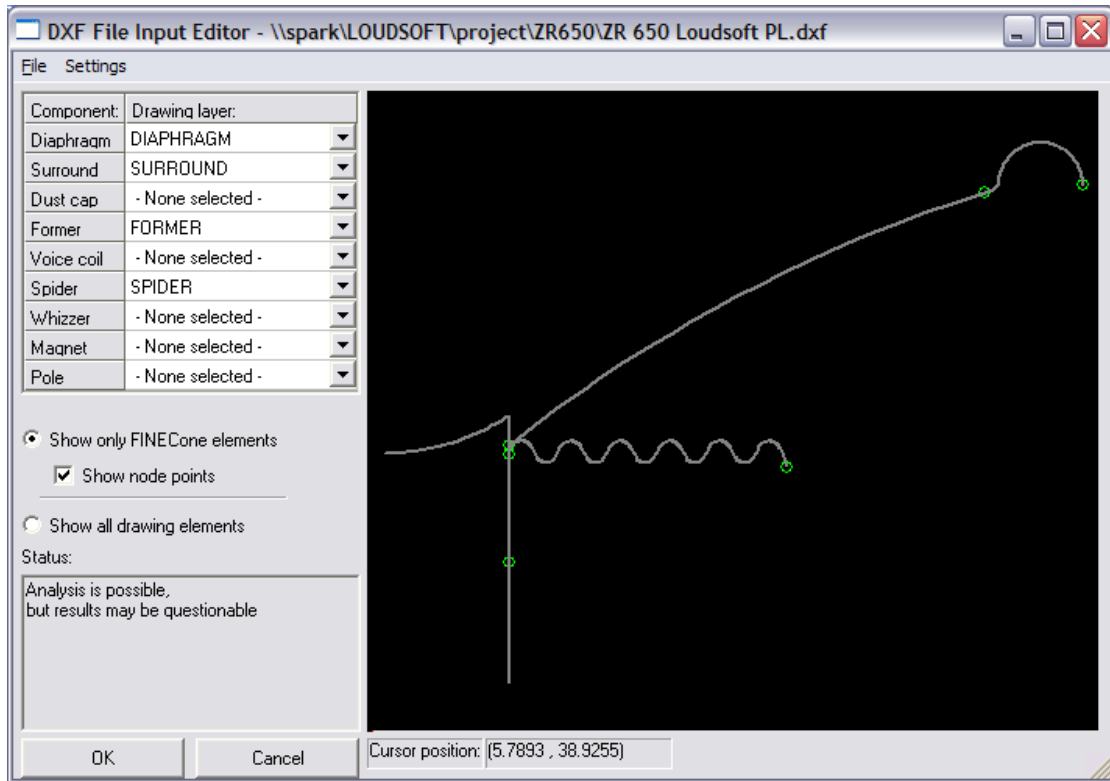
Figure 33. DXF Import -Input Editor
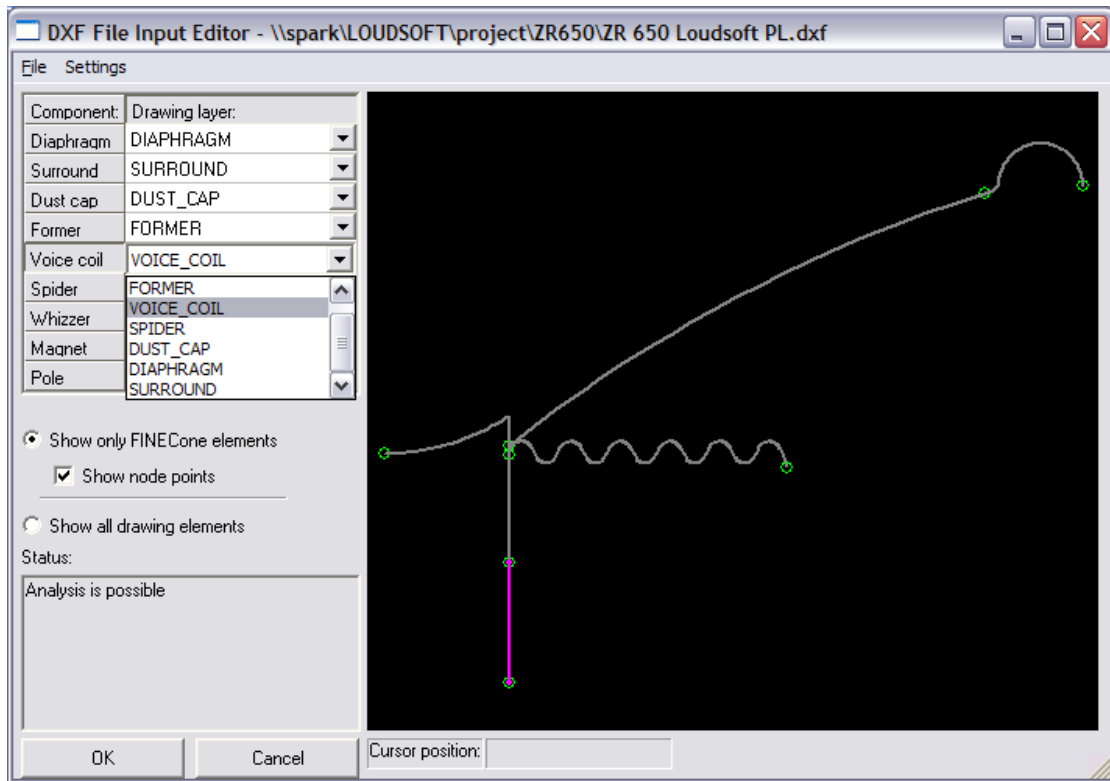
Figure 34. DXF layer - drop down menu

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.
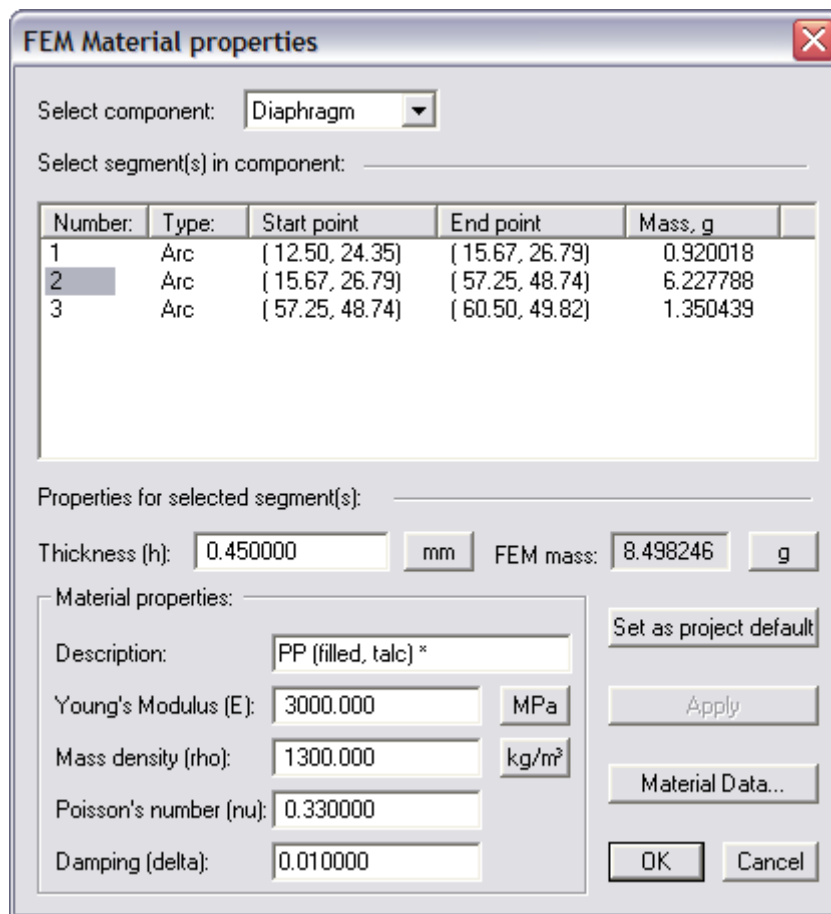
Figure 35 - The thickness of the various components is set in the FEM Material properties window

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.
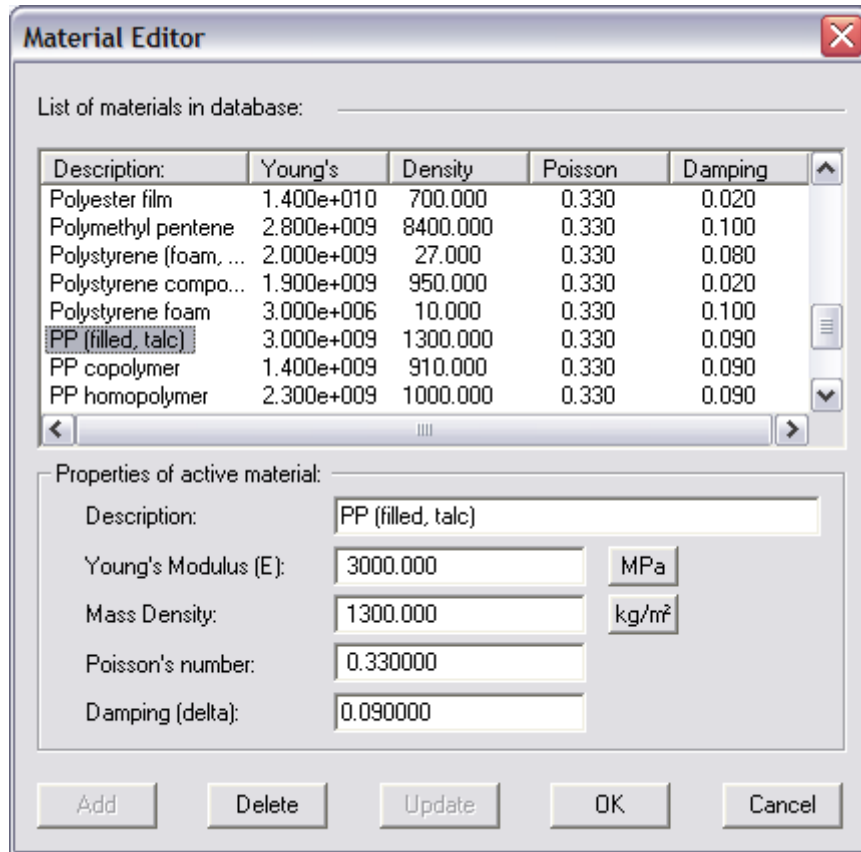
Figure 36 - The cone material is selected from one of the standard materials in the FINECone materials database

The influence of glue will be taken into account. 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 37, where the first cone segment (1) is specified with 2mm thickness. We may later change the stiffness by adjusting Young's Modulus
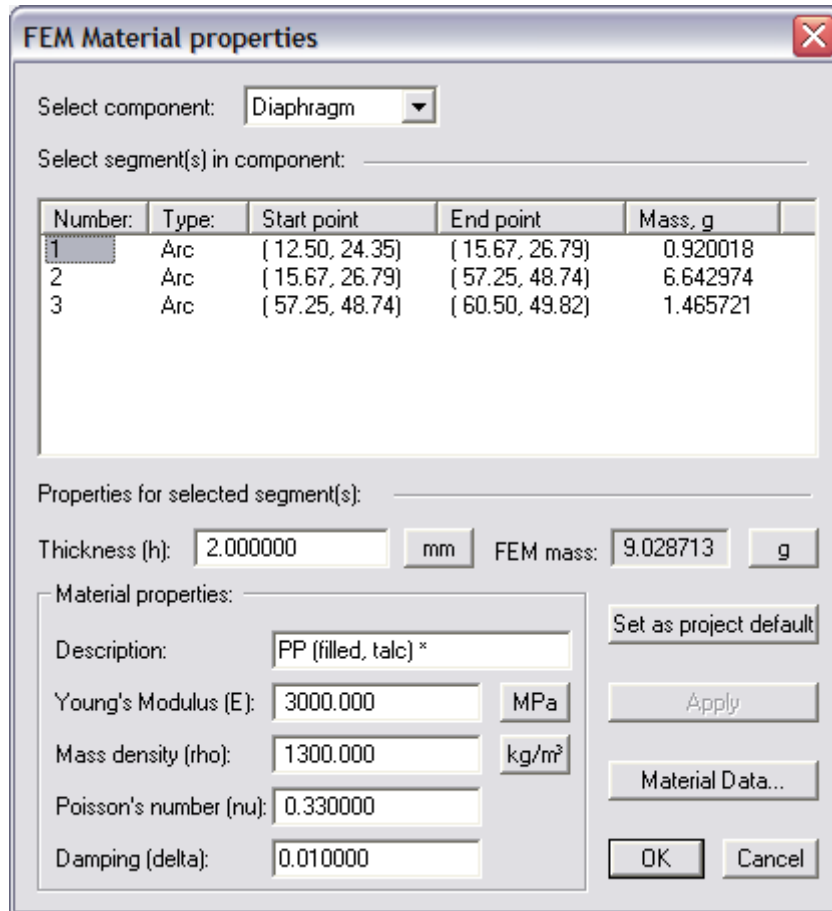
Figure 37 - Part of the cone is made thicker to account for the glue on the voice coil

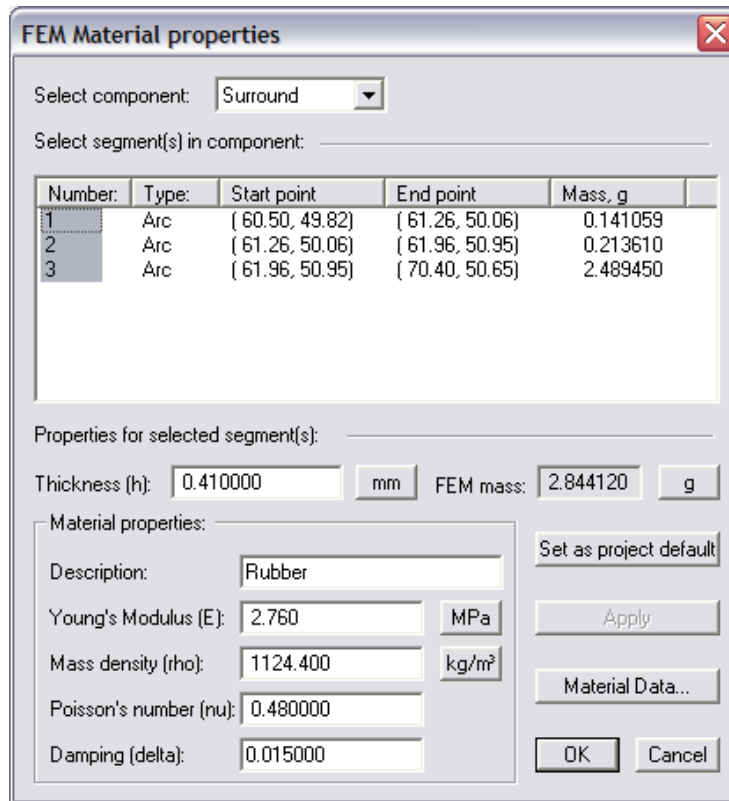The surround is 0.41mm rubber from the database.

Figure 38 - The surround material is using the Rubber material which has the properties of the most common type of rubber used for surrounds
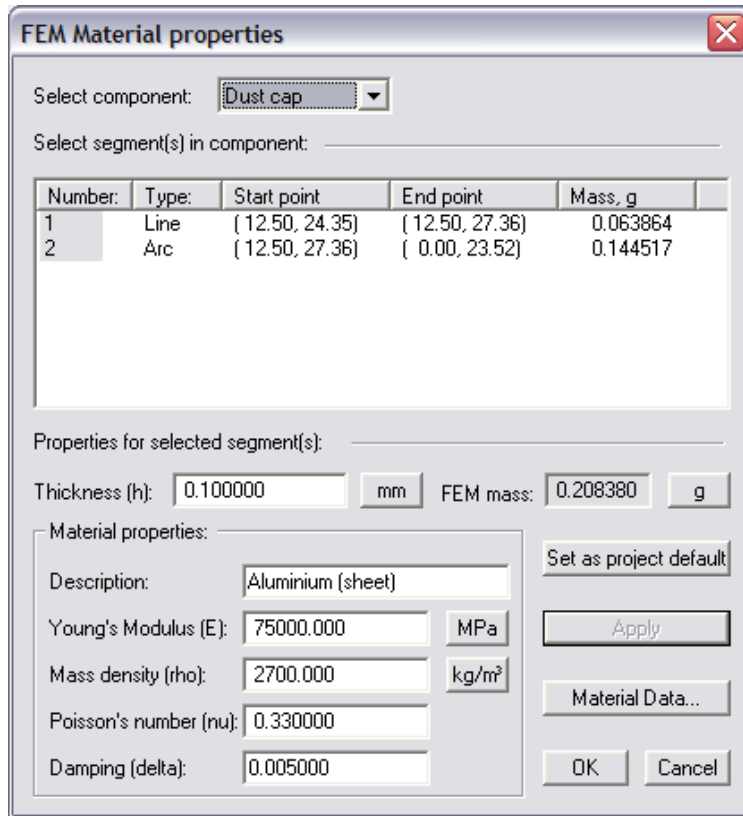
Figure 39 - For the dust cap aluminium is chosen

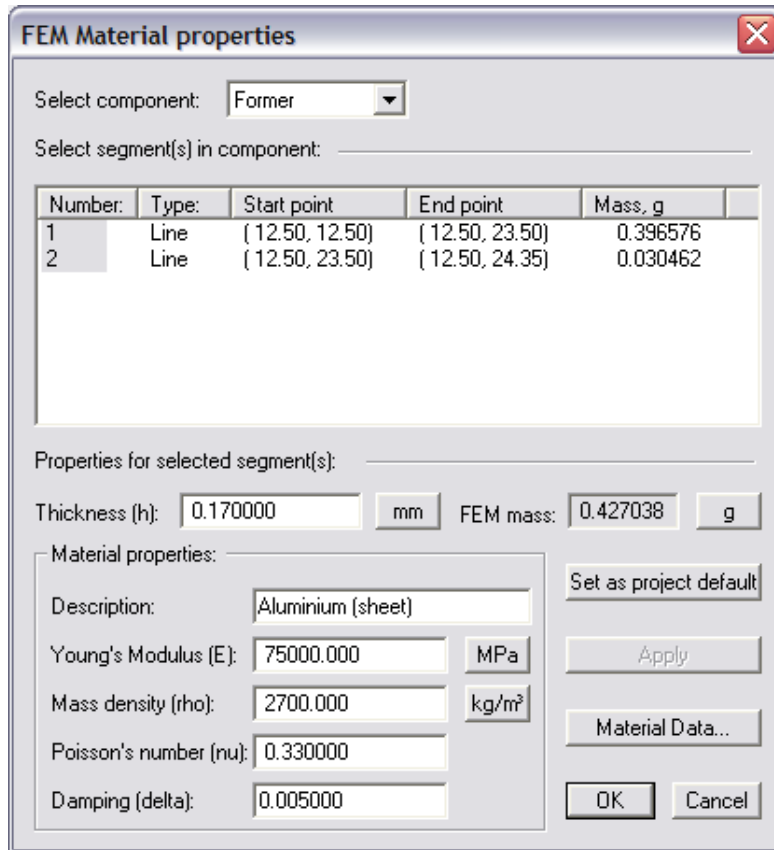*Note: Don't forget to press* [Apply]*, whenever you have made a change.*

Figure 40 - 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 41. We then adjust the thickness to get the same mass as measured. The VC stiffness is not used.

Figure 41 - The voice coil used in the woofer we are simulating
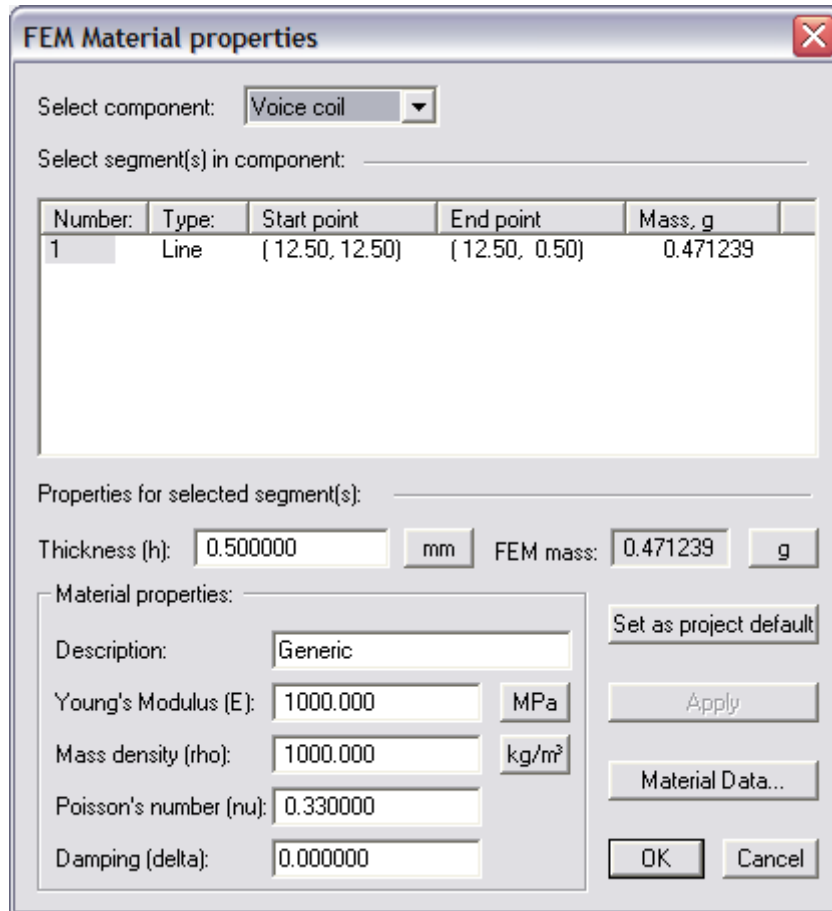
Figure 42 – For the voice coil we choose the 'Generic' material as we are going to tweak the parameters ourselves.

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.
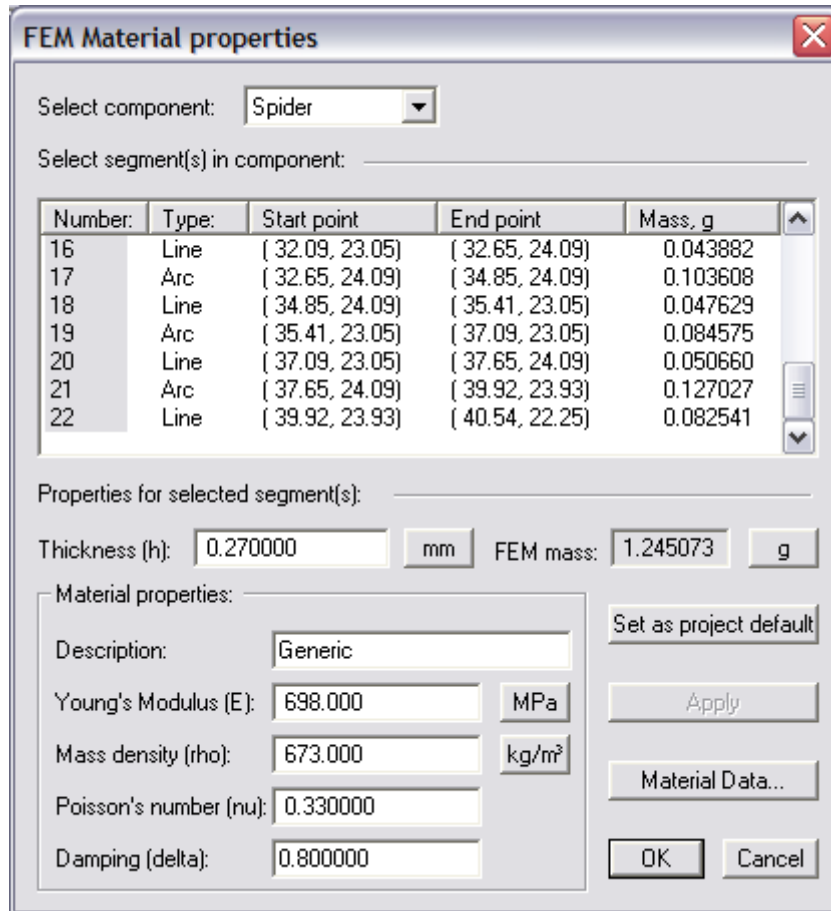
Remember to select all segments with CTRL+A.

Figure 43 - 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.

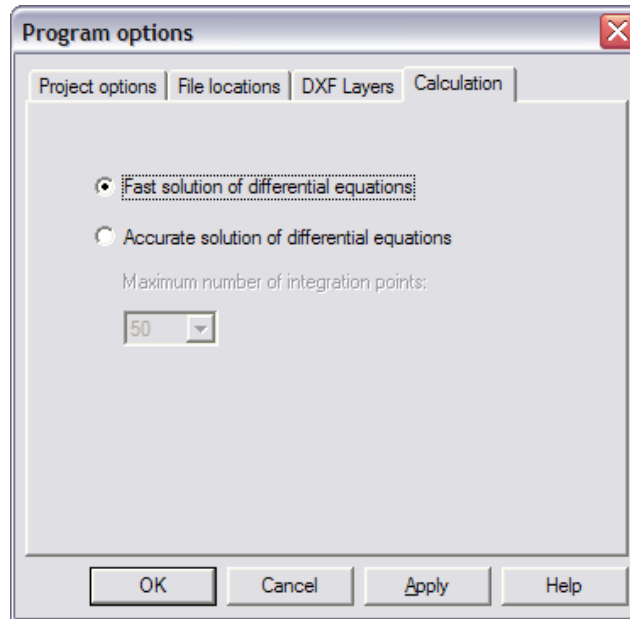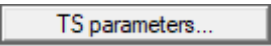(if using a slow PC, you may use the 'Fast solution of differential equations' setting to get faster calculation. The setting can be found in Tools/Program Options/Calculation).

Figure 44 - 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 1.3.4 and 1.3.5.

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.
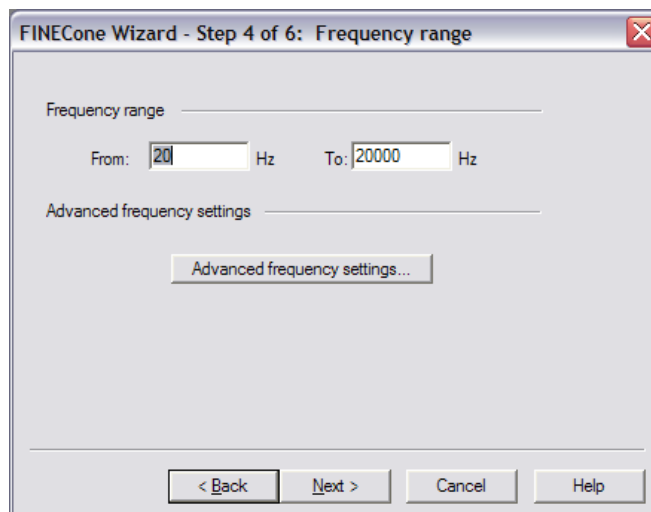
Figure 45 - The frequency range can be specified depending on the simulation

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.
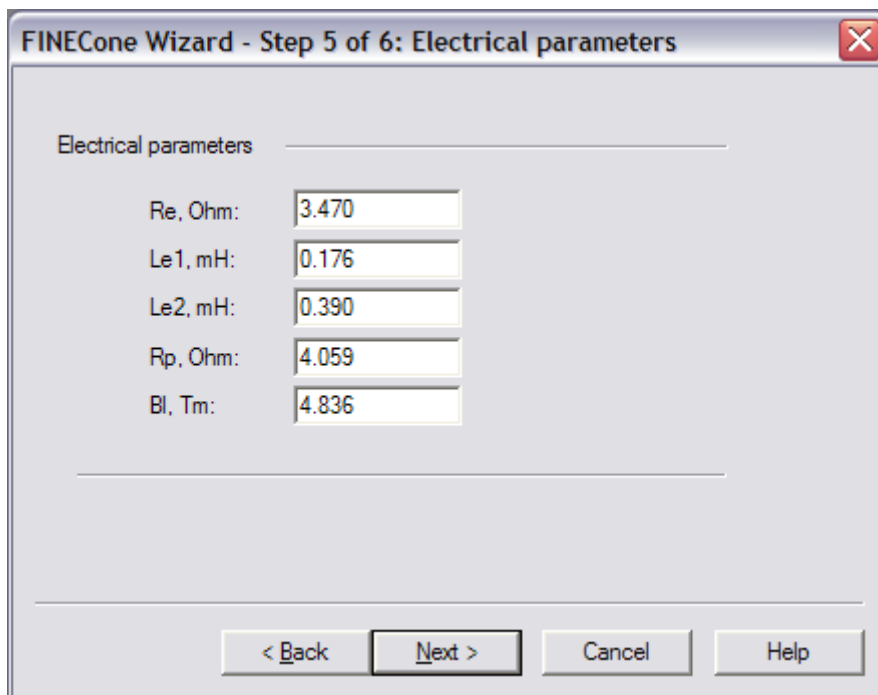

Figure 46 - The electric parameters found using FINE R+D are entered
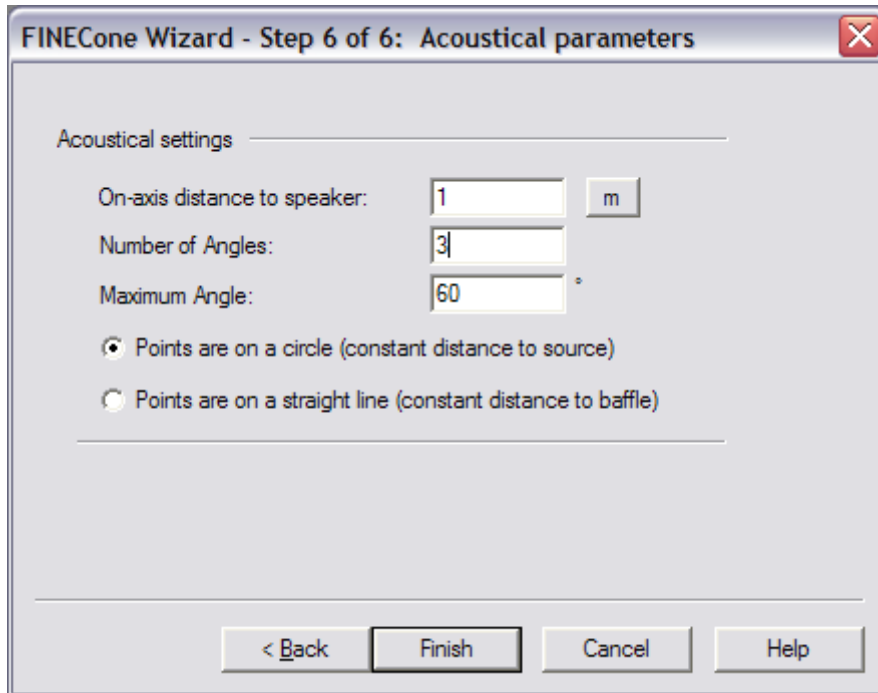
Figure 47 - Measuring points are specified as the last thing

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

The Finite Element (FEM) calculation is done automatically after Finish is pressed.
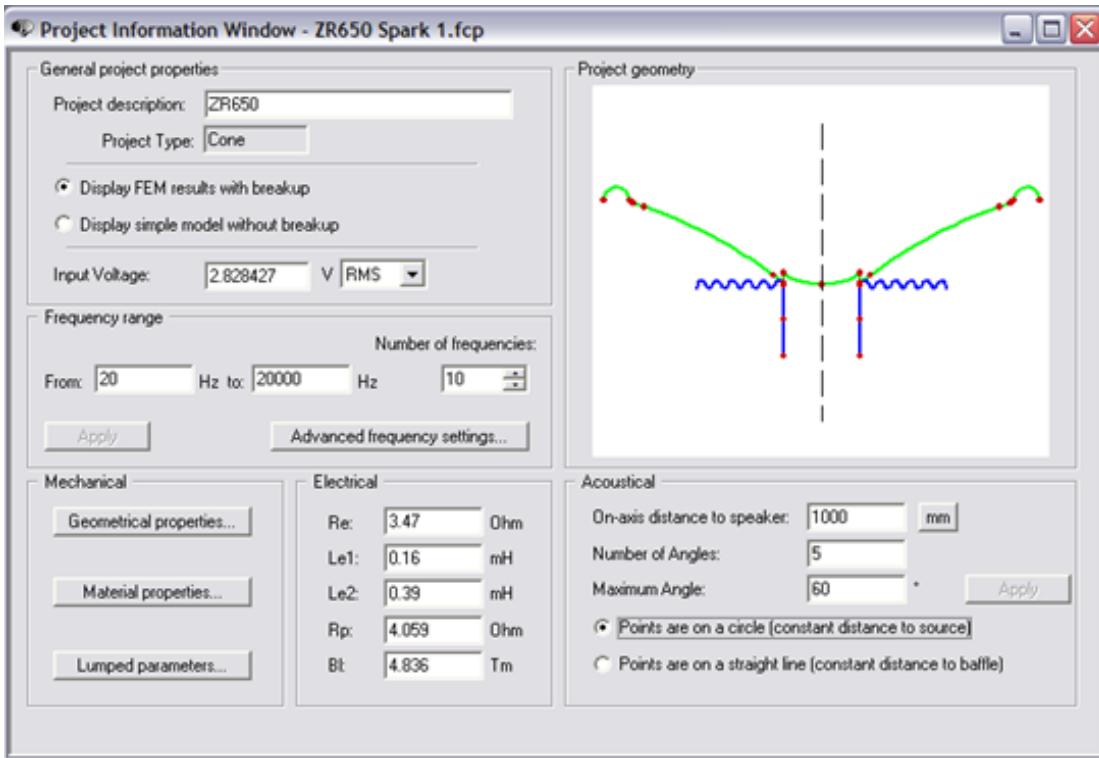
Figure 48 - This is the window you will see when you press finish if you have followed the instructions in the tutorial

Step 4 - Fit the impedance curve

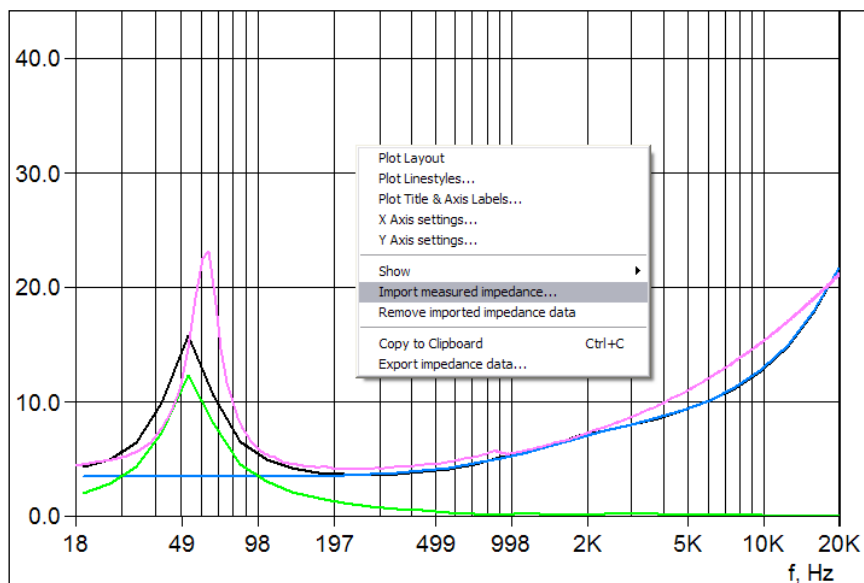Now is the time to take a look of the impedance curve we have calculated in FINECone.

Figure 49 - 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.
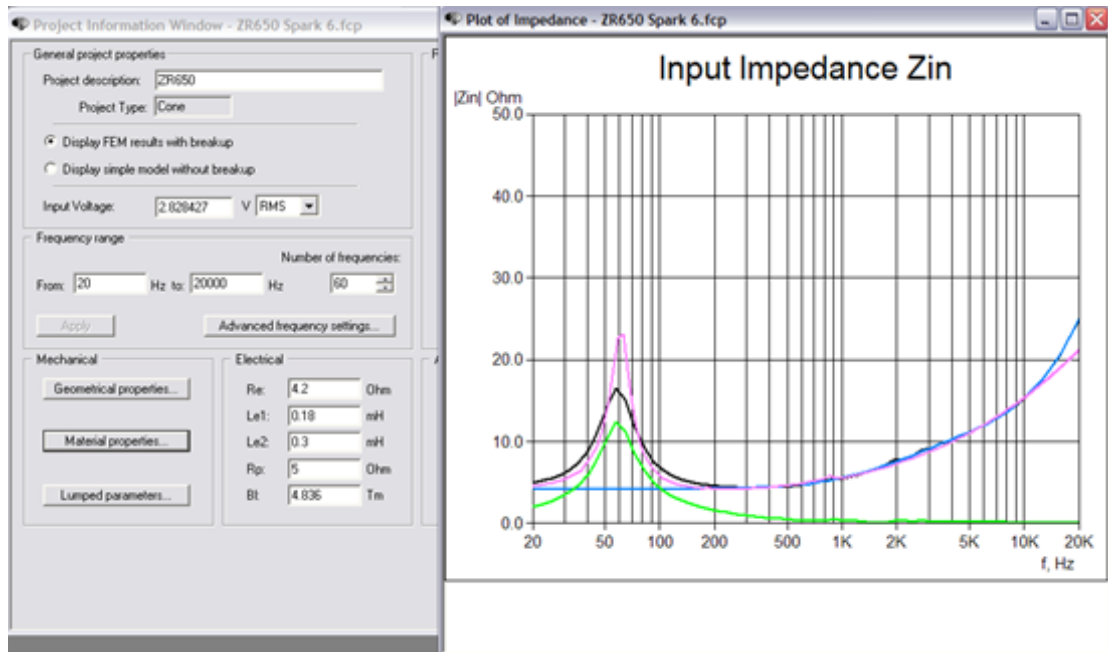

Figure 50 - After having imported the measured curve (the pink curve) the electrical parameters are tweaked to fit the simulated impedance curve

Step 5 - Fit the SPL curve
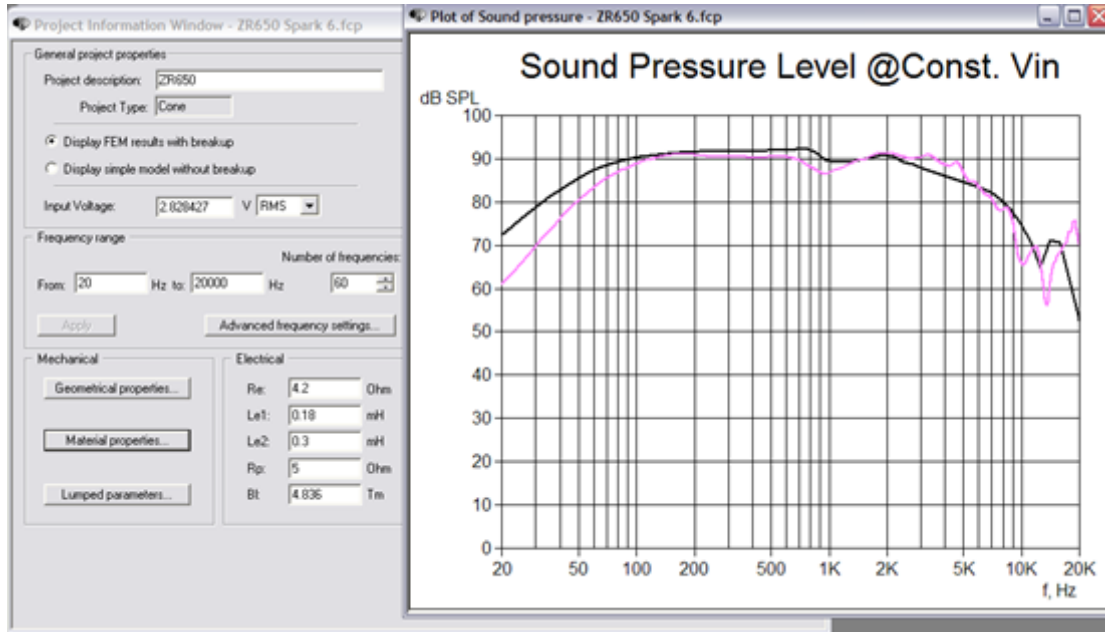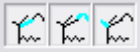
Finally, we work on the SPL curve

Figure 51 - 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.
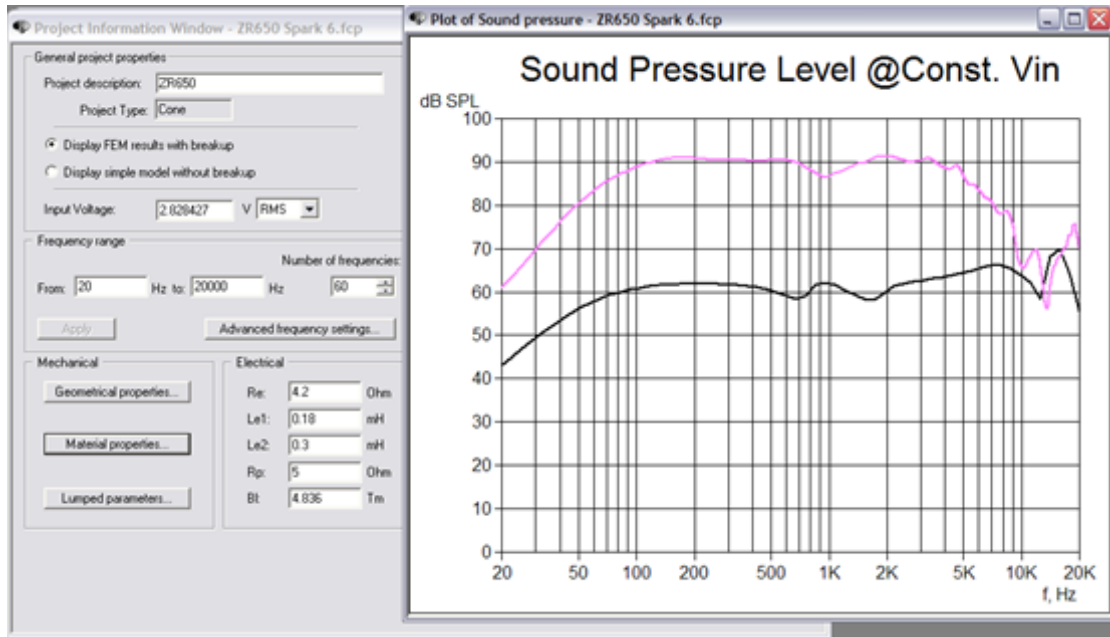
Figure 52 - Here the frequency response is shown for the dust cap alone

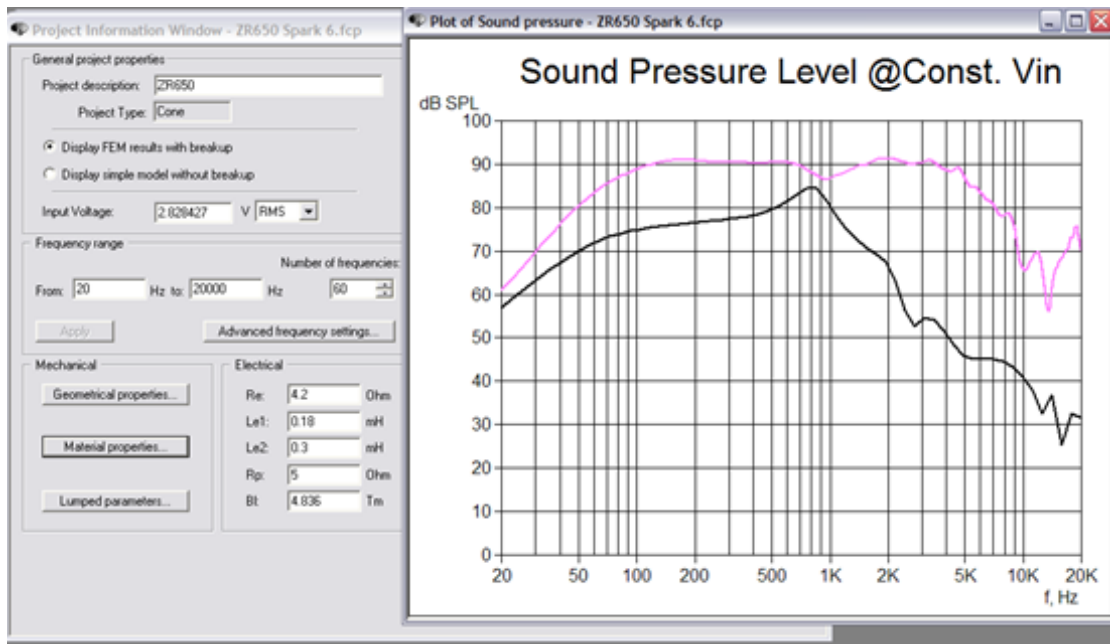The dust cap only affects the very high frequencies.

Figure 53 - 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.
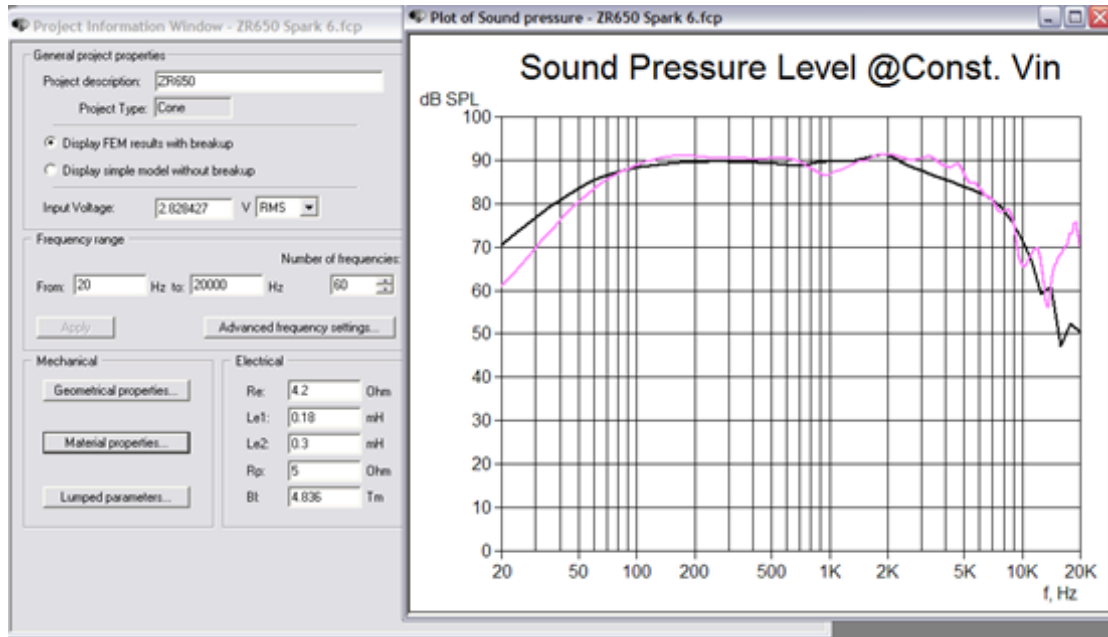
Figure 54 - The frequency response for the cone alone

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.
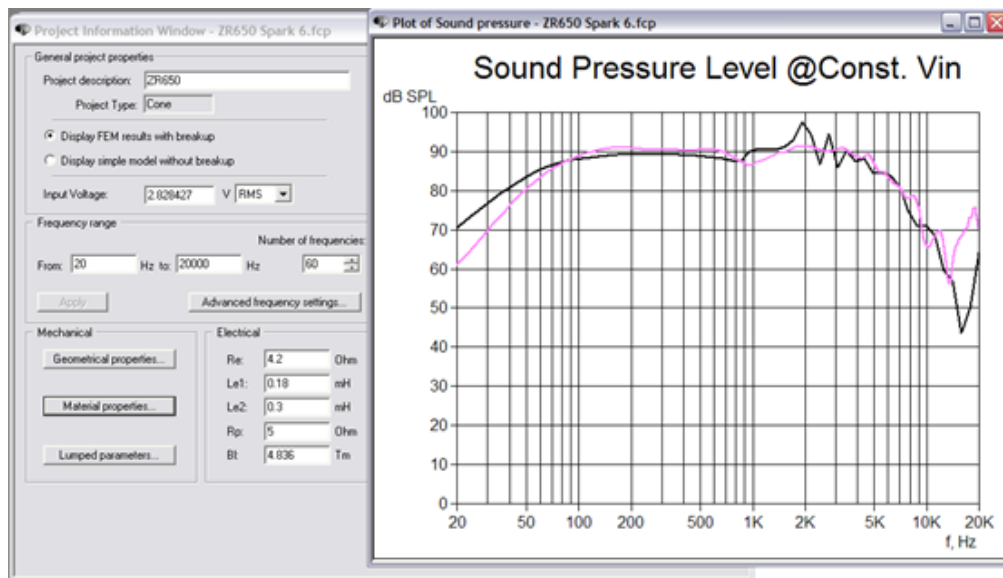

Figure 55 - The overall frequency response with the damping of the cone reduced

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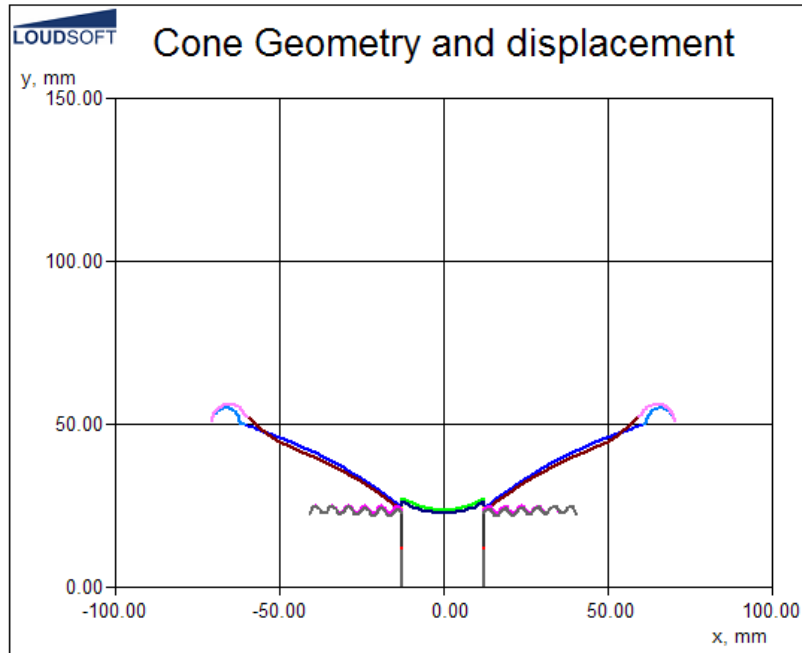

Figure 56 - A 2D plot of the driver at rest and when moving at the selected frequency.

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.

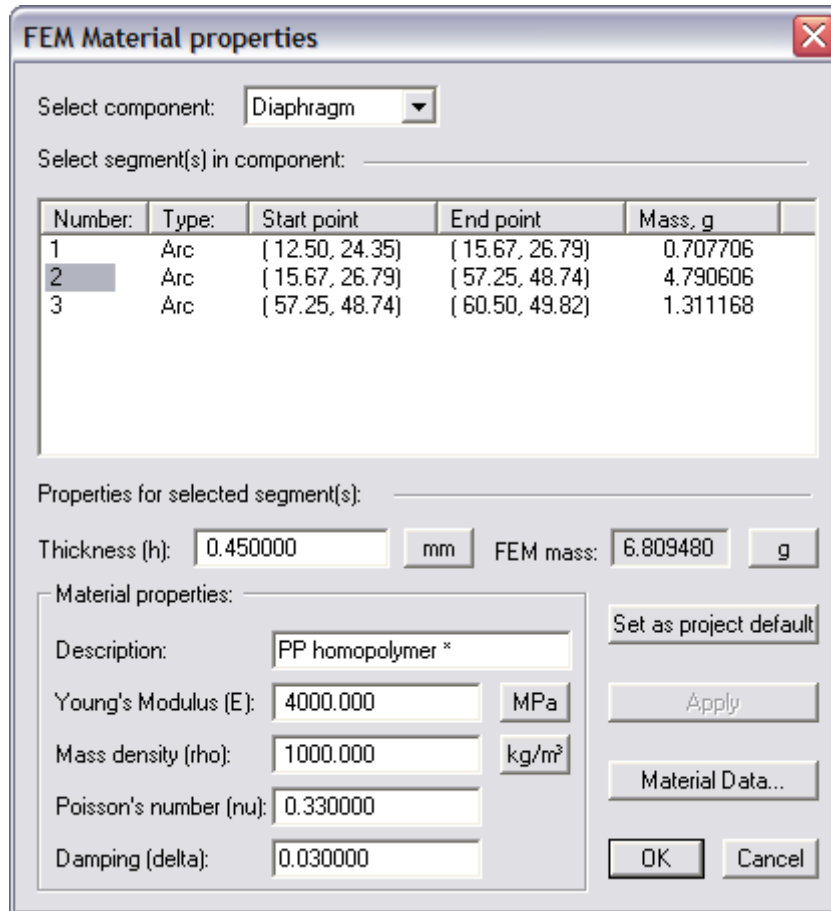Finally, we find the correct damping of the cone, which is lower than the Material data

Figure 57 - 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.
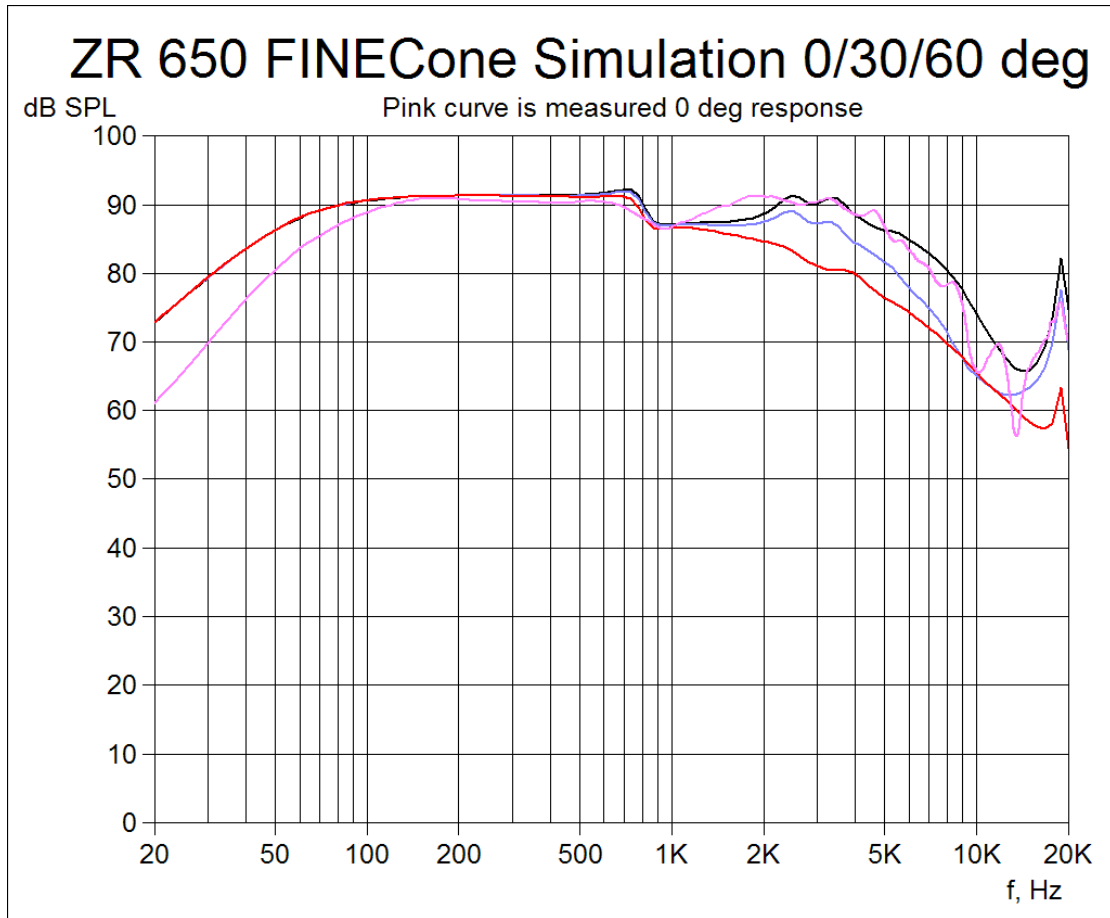
Figure 58 - The simulated response for on-axis, 30 degrees and 60 degrees are shown versus the measured response

The final 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 also 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

2.2   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.
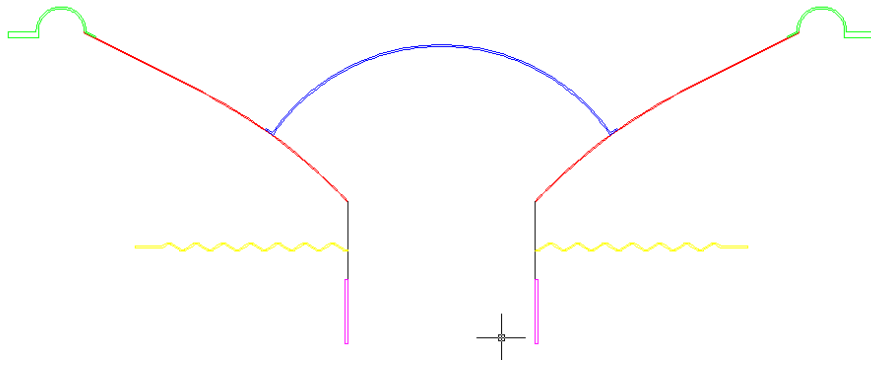
Figure 59 - 2D representation of the driver


```
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 60 - 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 61.
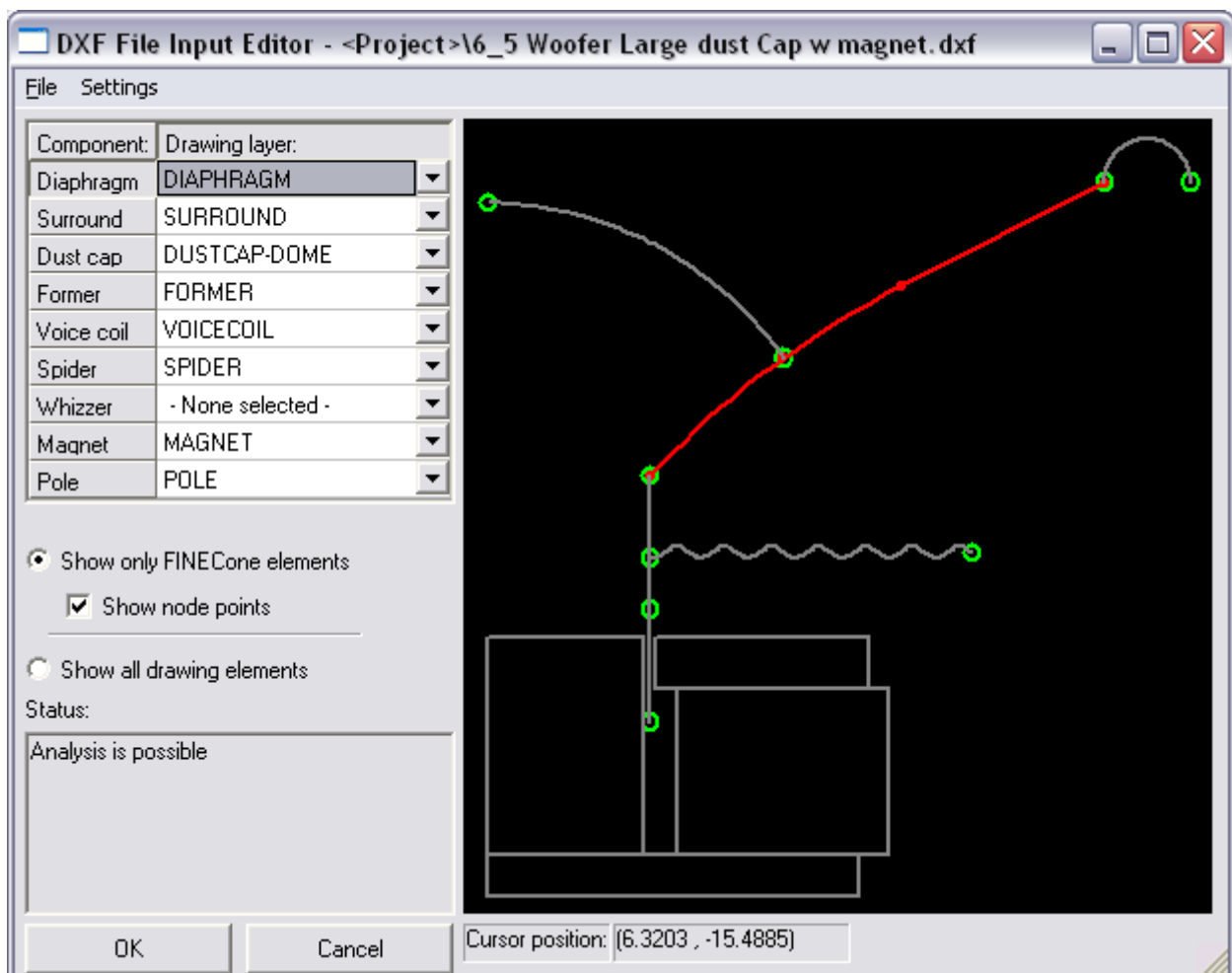

Figure 61 - DXF Import and Automatic Error Checking

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.

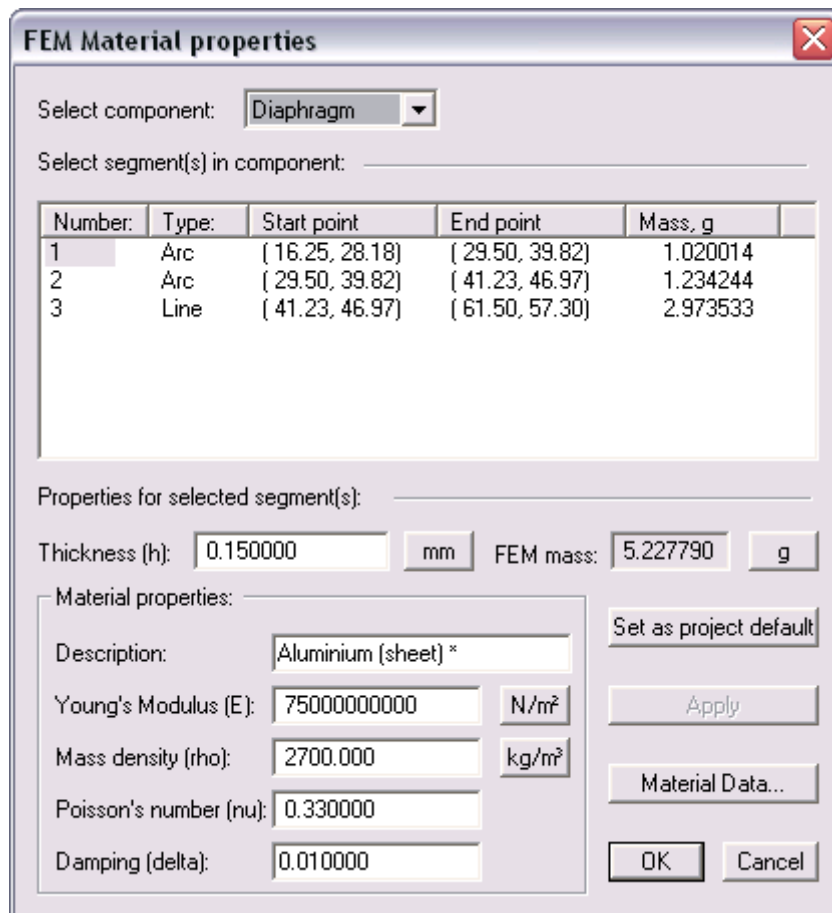Therefore, we select <u>FEM Material Properties</u>

Figure 62 - 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 61). 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 in order to model the actual speaker material correctly.

The surround material is obtained from the Material Database, shown in Figure 63, 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.
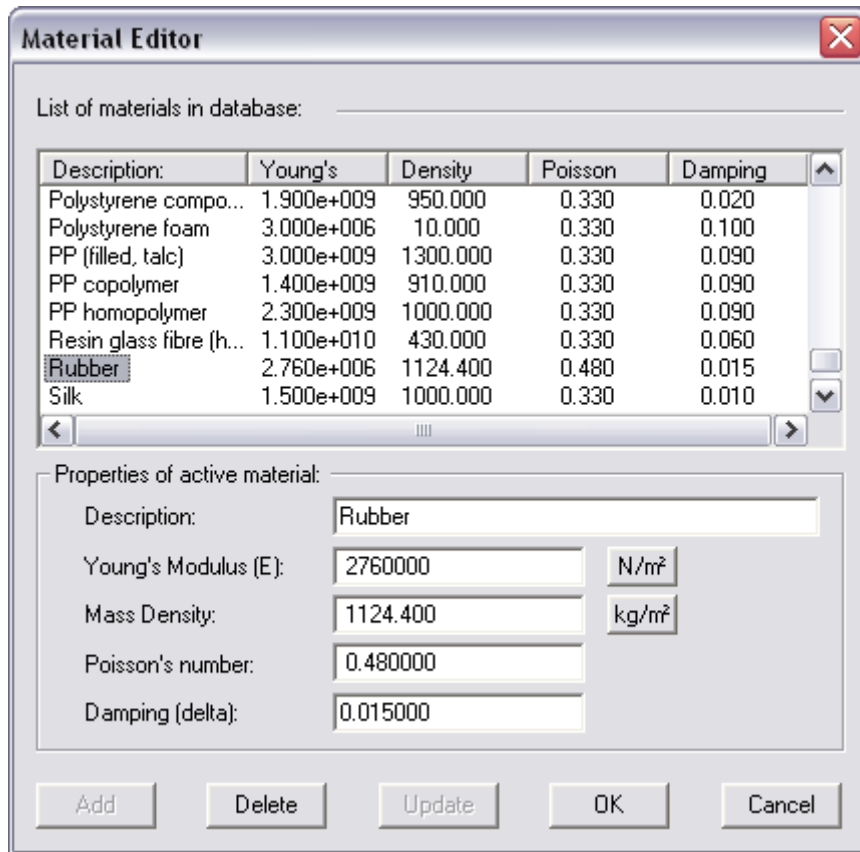

Figure 63 - Material editor window with a long list of pre-set materials

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"
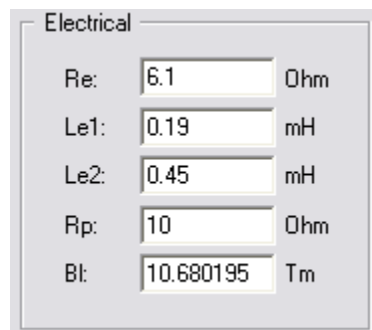

Figure 64 - Electrical Input

If you have a FINEMotor file (*.FM3) it can be imported directly into FINECone. See also TS parameters section 1.3.4. You may transfer Le1, le2 and Rp in the TS dialog, Figure 65.
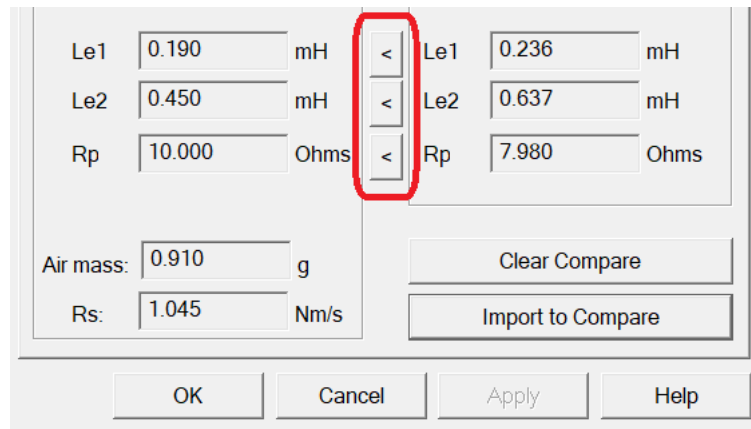

Figure 65 - TS Parameters: Including Le1, le2 and Rp from file
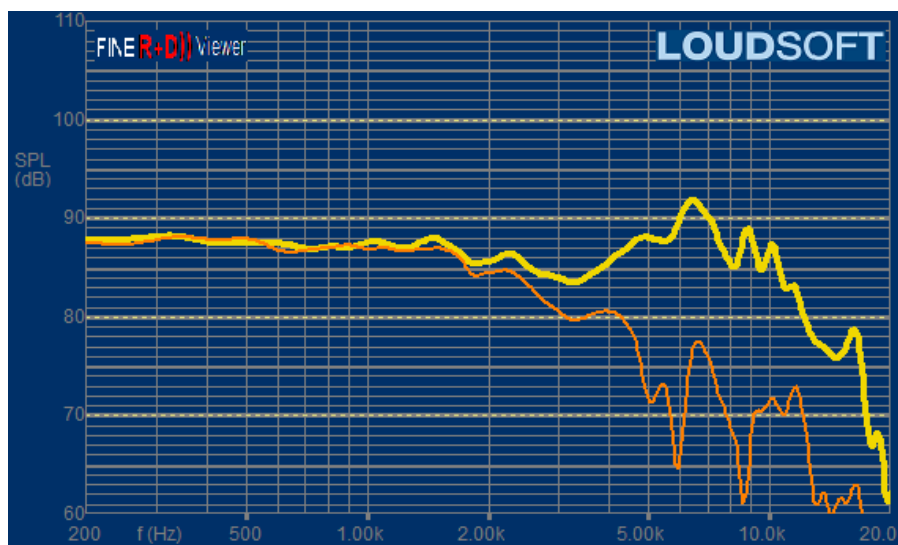

Figure 66 - The (0/30deg) response is measured with FINE R+D. The on-axis curve is imported in FINECone

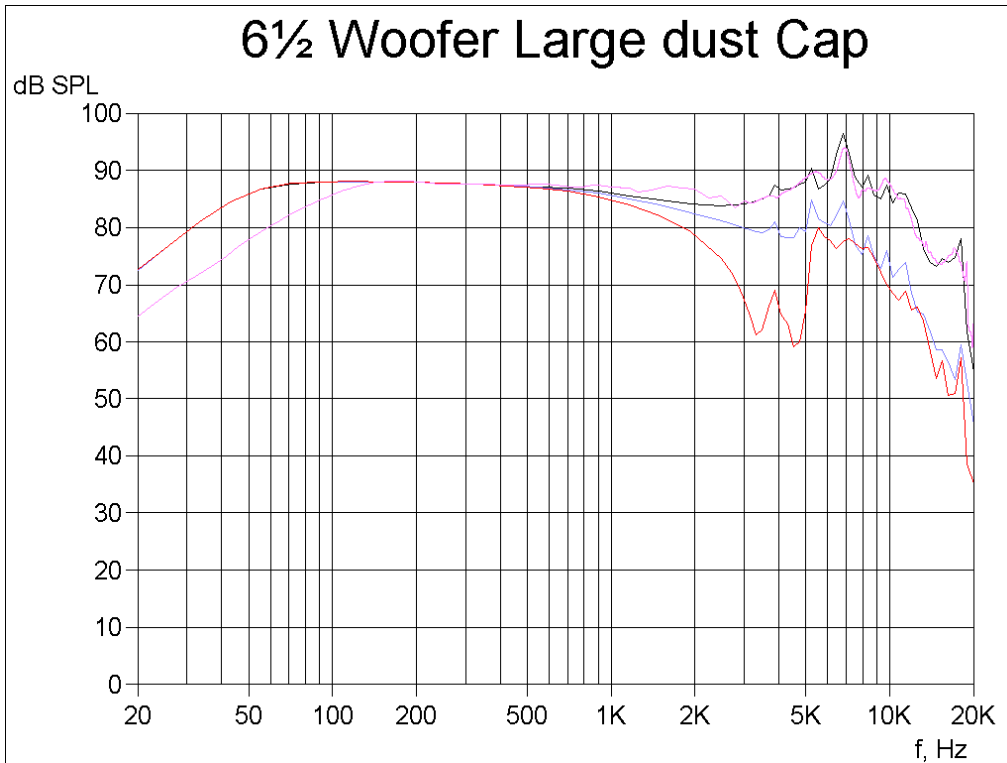The resulting frequency response is shown in Figure 67.

Figure 67 - The agreement between the calculated FINECone response (black) versus the measured response (magenta) is remarkable at high frequencies (break-up region)
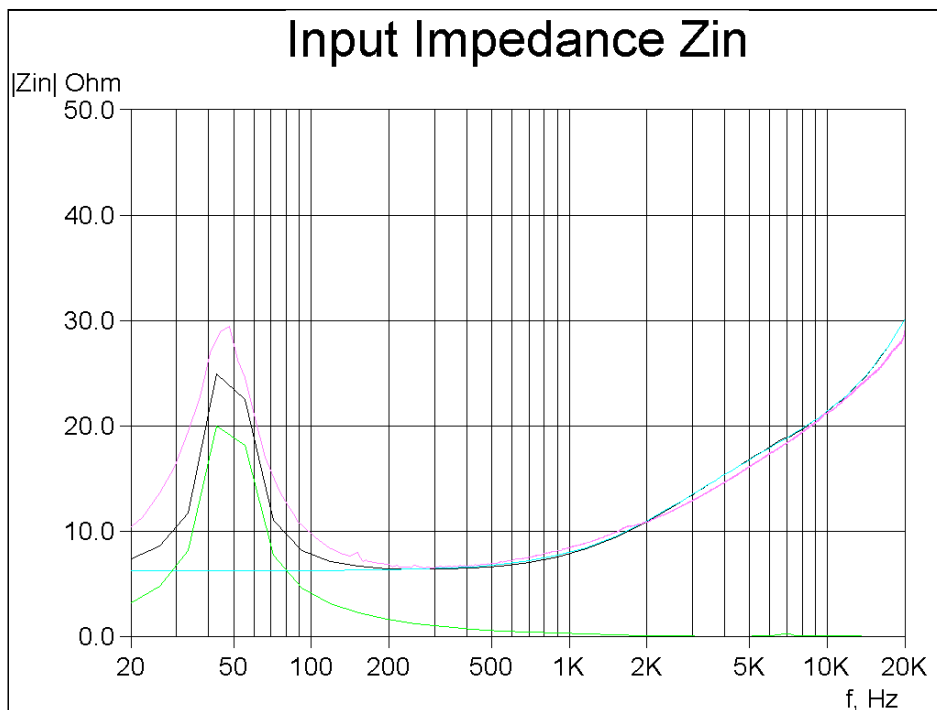

Figure 68 - The blue curve is calculated electrical impedance, the green curve is the mechanical impedance, and the black curve is the blue and green curves summed up to give the total impedance. The magenta curve is the imported impedance curve for comparison.

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 69.
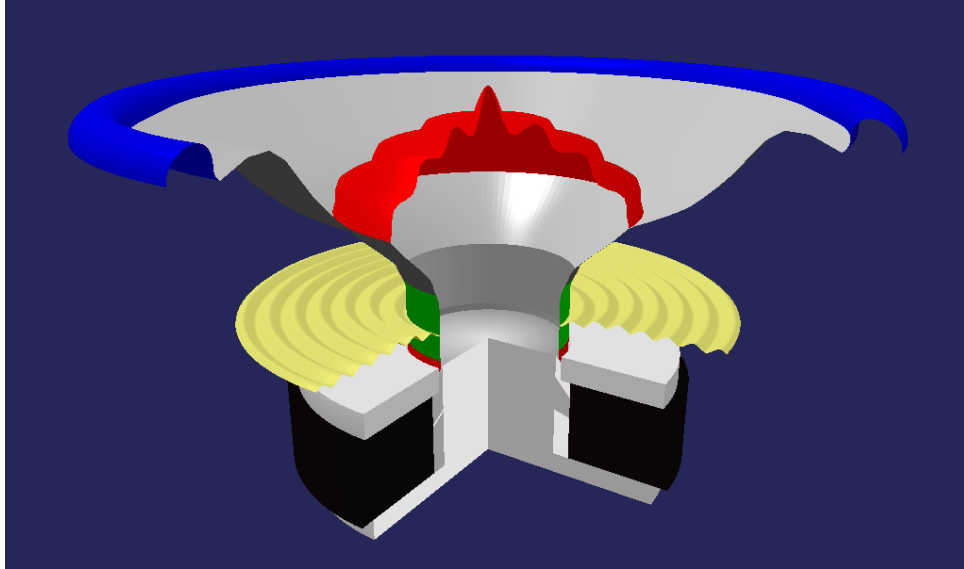

Figure 69 - This is the 6.5" woofer break-up animated at 7162Hz, which is the frequency of the large peak. Note the heavy break-up in the outer part of the cone.

The amplitude is set to 7mm as shown in Figure 70. Selecting the actual amplitude is not ideal for high frequencies as the excursion is extremely small.
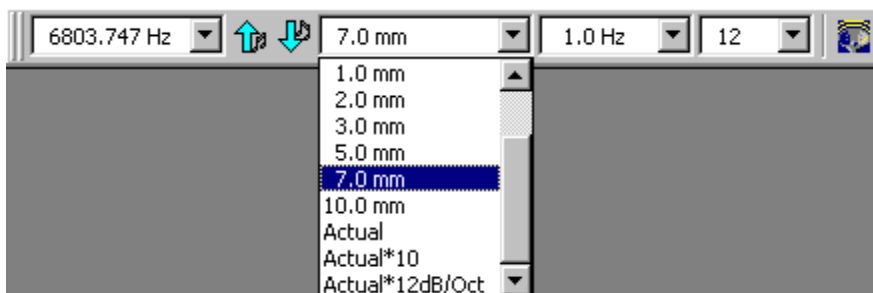

Figure 70 - The amplitude is set to 7.0 mm to better visualise the breakup

Excluding every part from the model except for the dust cap by using the buttons shown in Figure 71, we can now see that it is providing the main contribution to the peak around 7 kHz as shown in Figure 72.
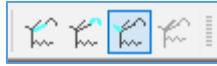
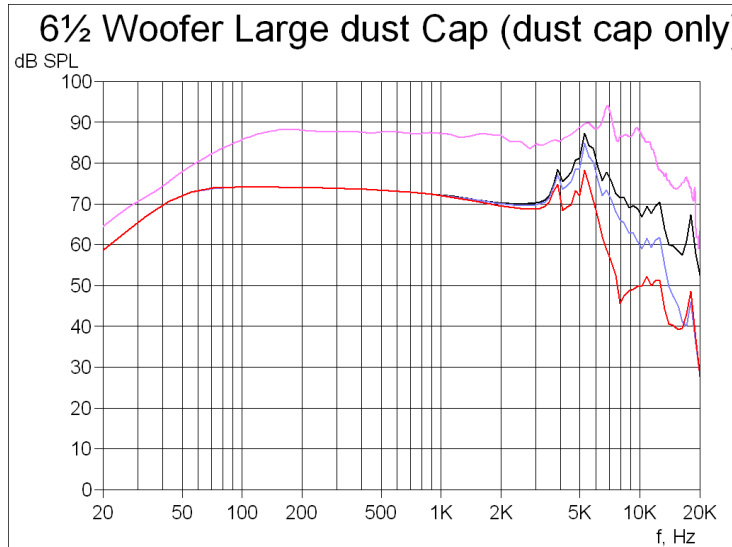**Figure 71 - Excluding all parts of the model except for the dust cap**

Figure 72 - Response of the large Dust Cap ONLY. The dust cap has a large peak around 5kHz.
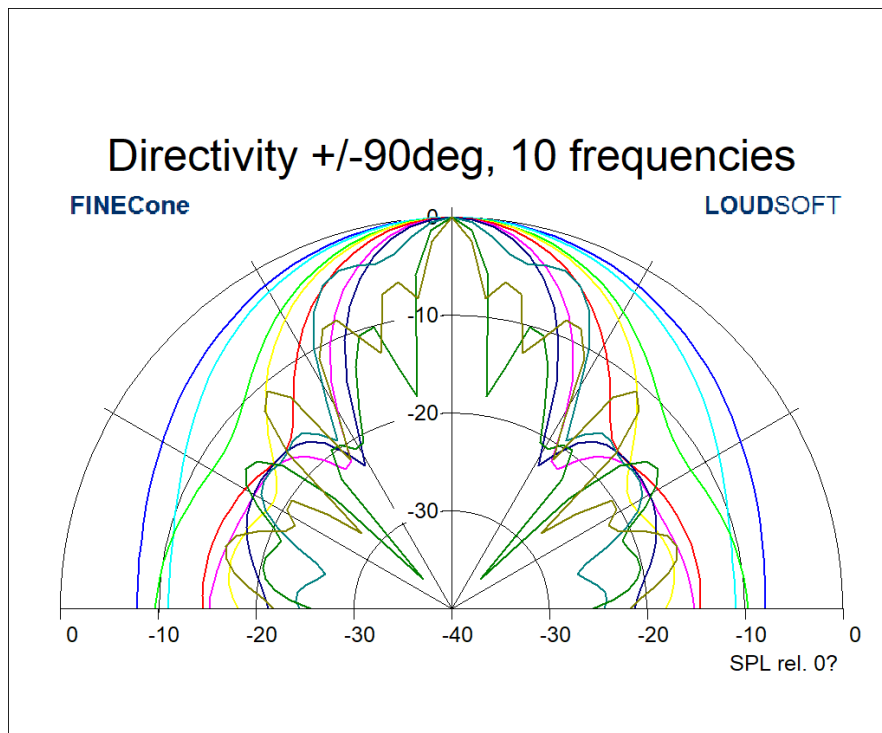

Figure 73 - Directivity at 10 frequencies

For displaying the directivity, it is recommended selecting 10 frequencies, and limited frequency range Figure 73. See the Directivity example: Curvilinear cone woofer -Dispersion Plot 2-20kHz.fcp

2.3   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.
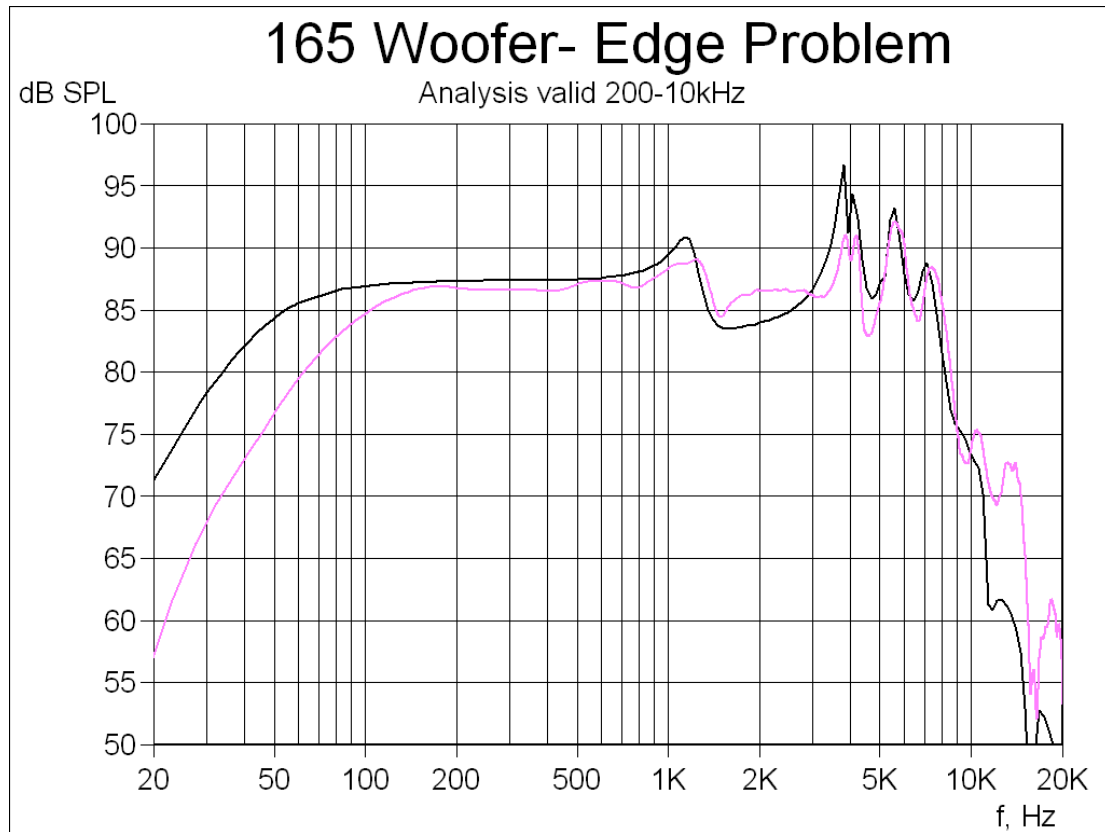

Figure 74 - Frequency response of the 165 mm woofer with an edge problem

The FINECone simulated response in Figure 74 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 Figure 75. 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 Figure 76.

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
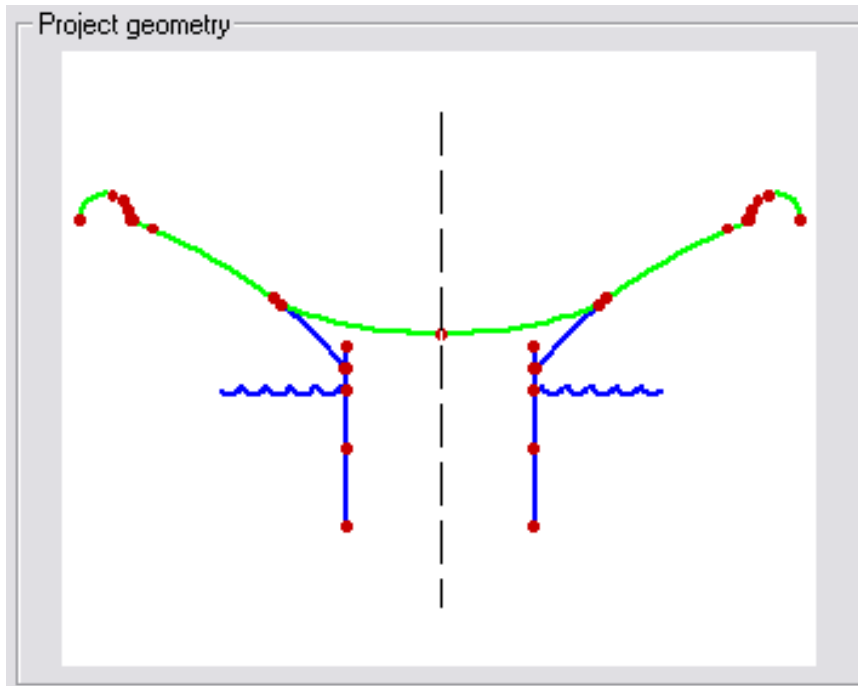
Figure 75 - Geometry for the 165 mm woofer with 5 segments in surround

Figure 76 - Material properties for the surround

In order 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. Figure 77 shows the cone edge and surround is moving excessively (brown curve).
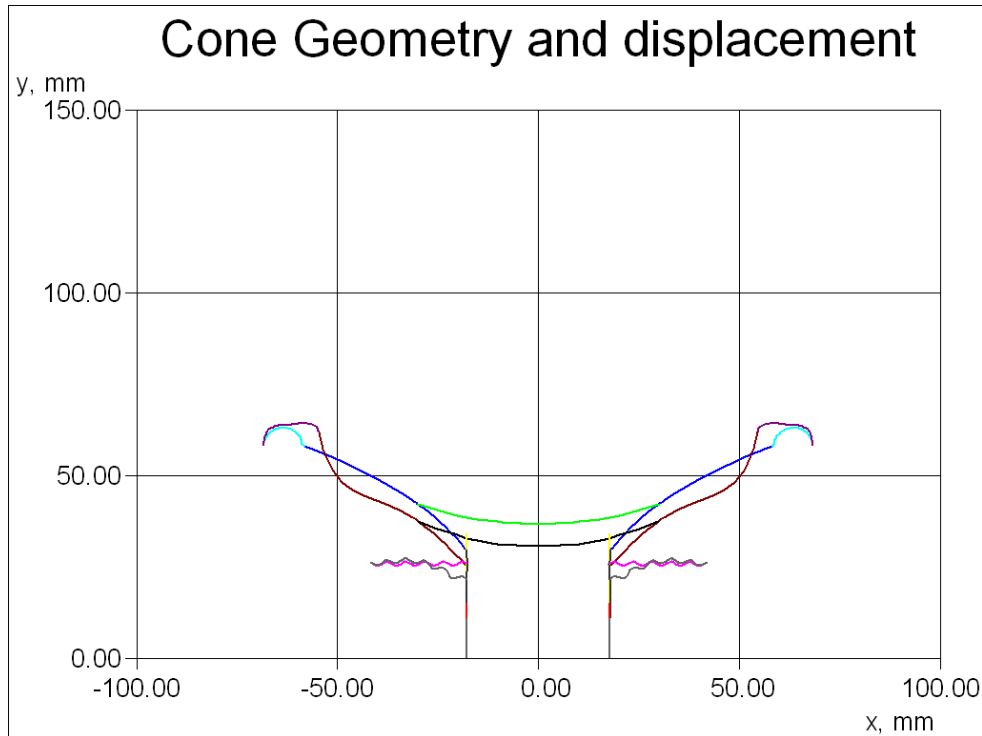

Figure 77 – 2D plot with displacement (brown), maximum at cone edge.

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 78, 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.
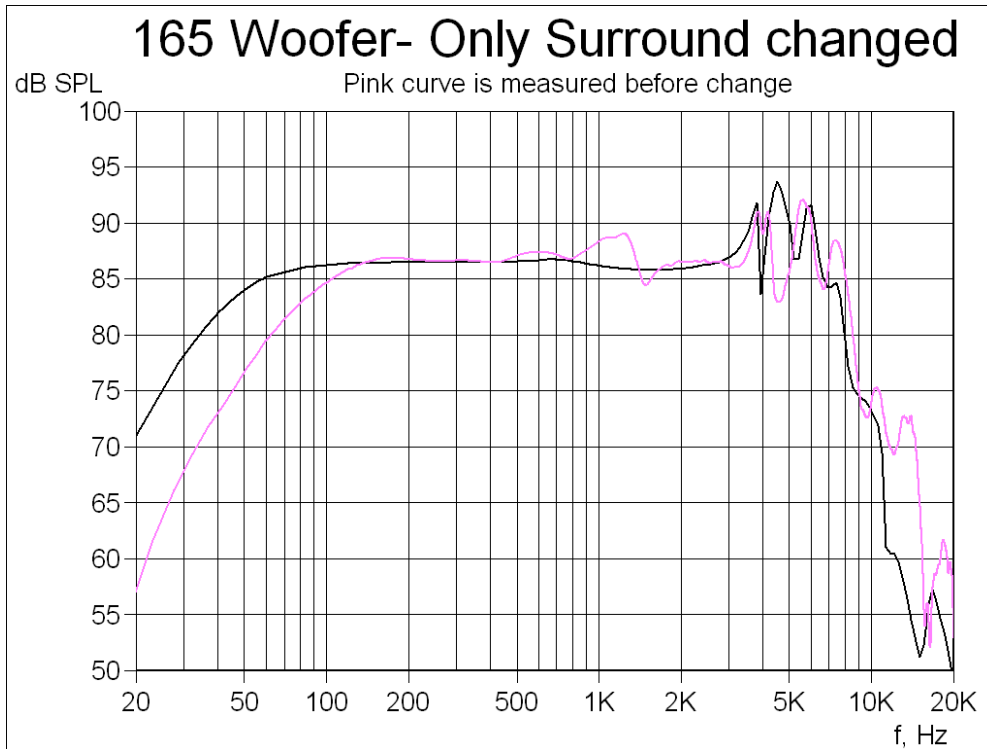
Figure 78 – 165 mm woofer with increased thickness of inner surround

Figure 79 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.
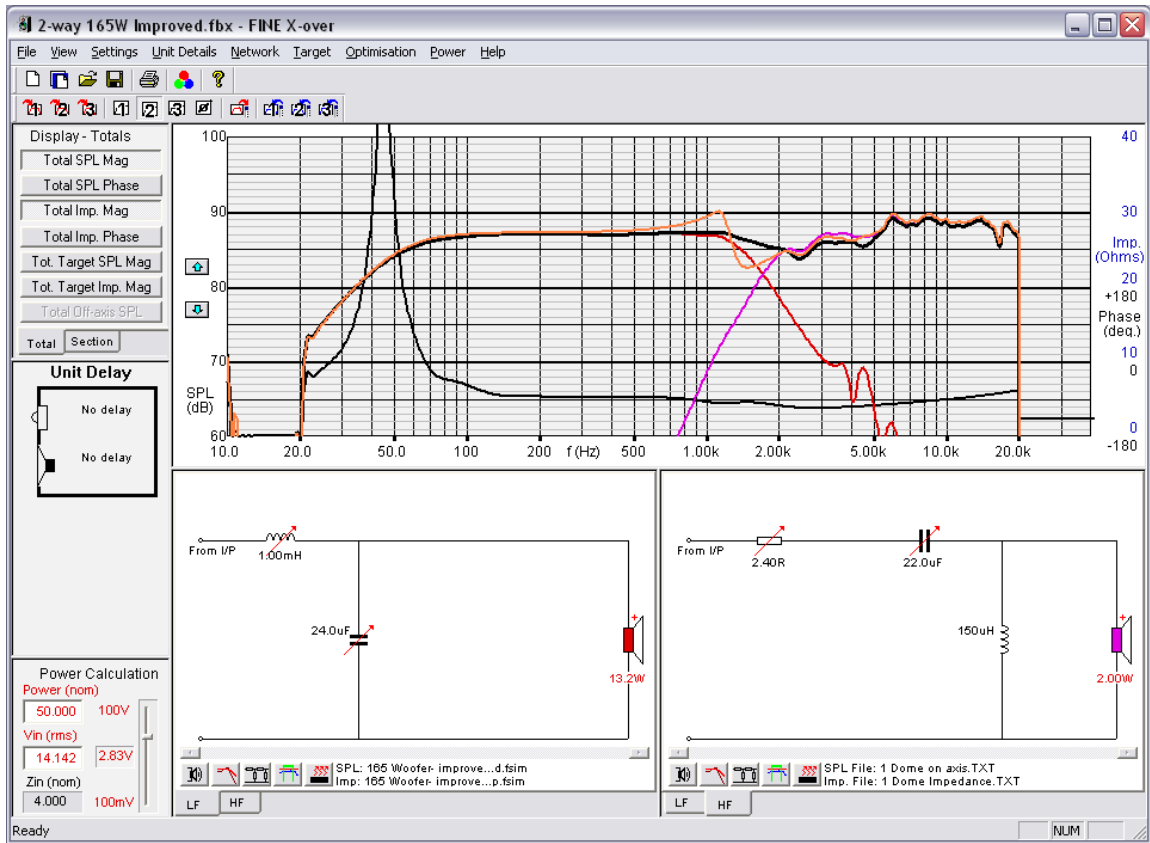
**Figure 79 - FINE X-over using the frequency response and impedance exported from FINECone.
Orange curve is with the bad woofer**
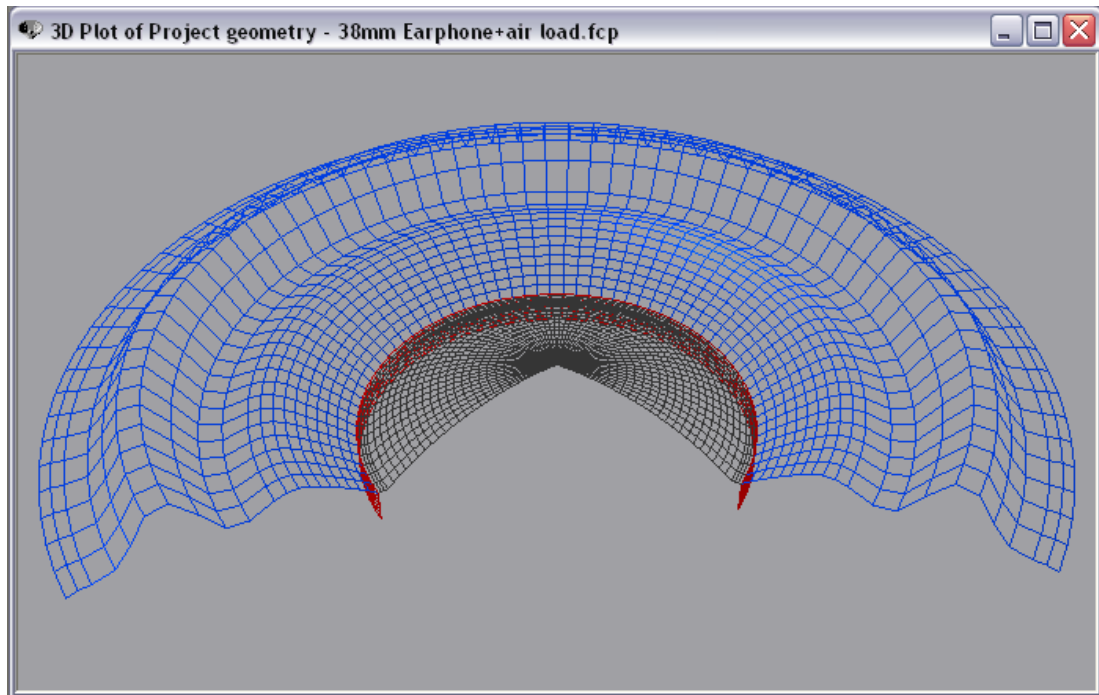
2.4 38mm headphone transducer

Figure 80 - 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 Figure 81. There is serious break-up from approximately 3000 Hz and the first mode is shown as 3D animation in Figure 80. This first break-up mode is showing up in the middle of the outer diaphragm (surround) where it is almost flat.
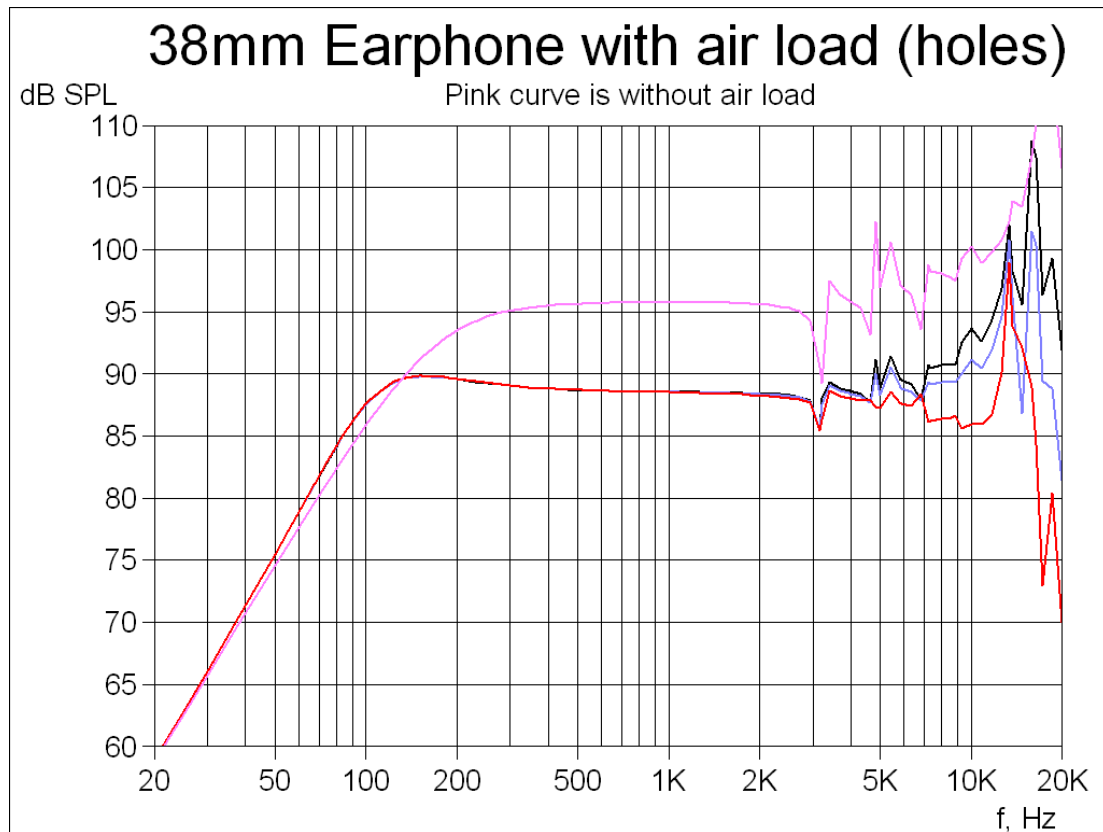
Figure 81 - A 38mm Headphone simulated frequency response with air load (rear holes)

The actual transducer has a number of 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 Figure 81 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

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

Figure 74 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 82 - FINECone Template type 003, shown in Notepad

LOUDSOFT

www.loudsoft.com