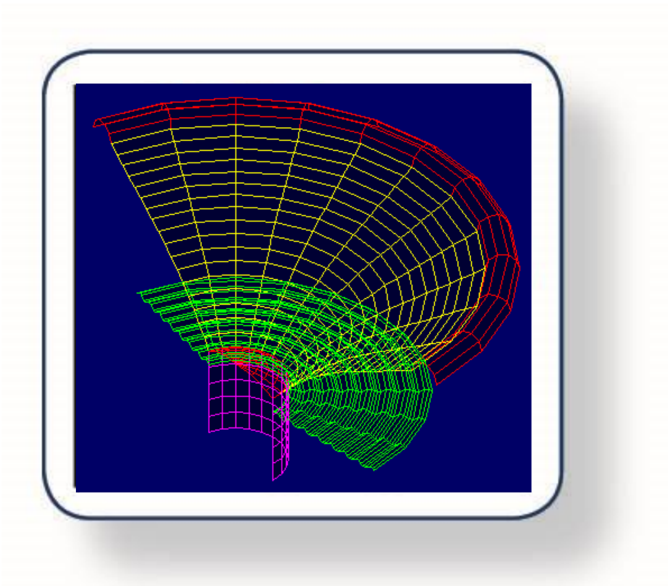
Manual



FINECone™

Acoustic Finite Element Dome/Cone Simulation Program

Contents

1. Overview

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

https://youtu.be/nlaGb67RPwc

1.1 DXF

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.

The geometry is defined by importing a DXF-file for example drawn with the free CAD program DraftSight and MUST be made ONLY with simple lines and arcs. The DXF-format is industry standard.

Figure 1 shows the geometry of a typical woofer on the left and how the DXF file would look like to the right.

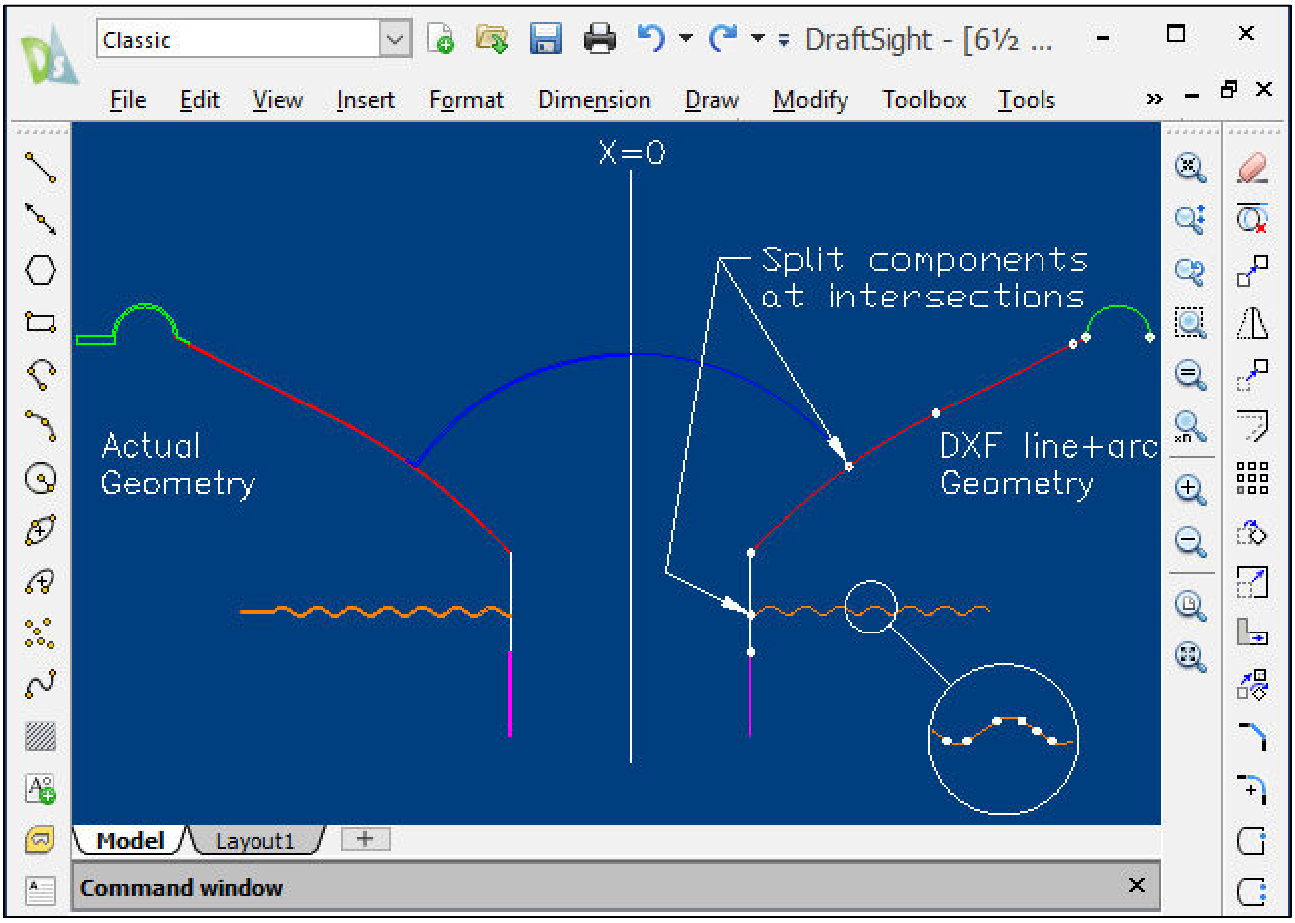


Figure 1 - DraftSight drawing - Left: Actual geometry - Right: DXF line + arc geometry

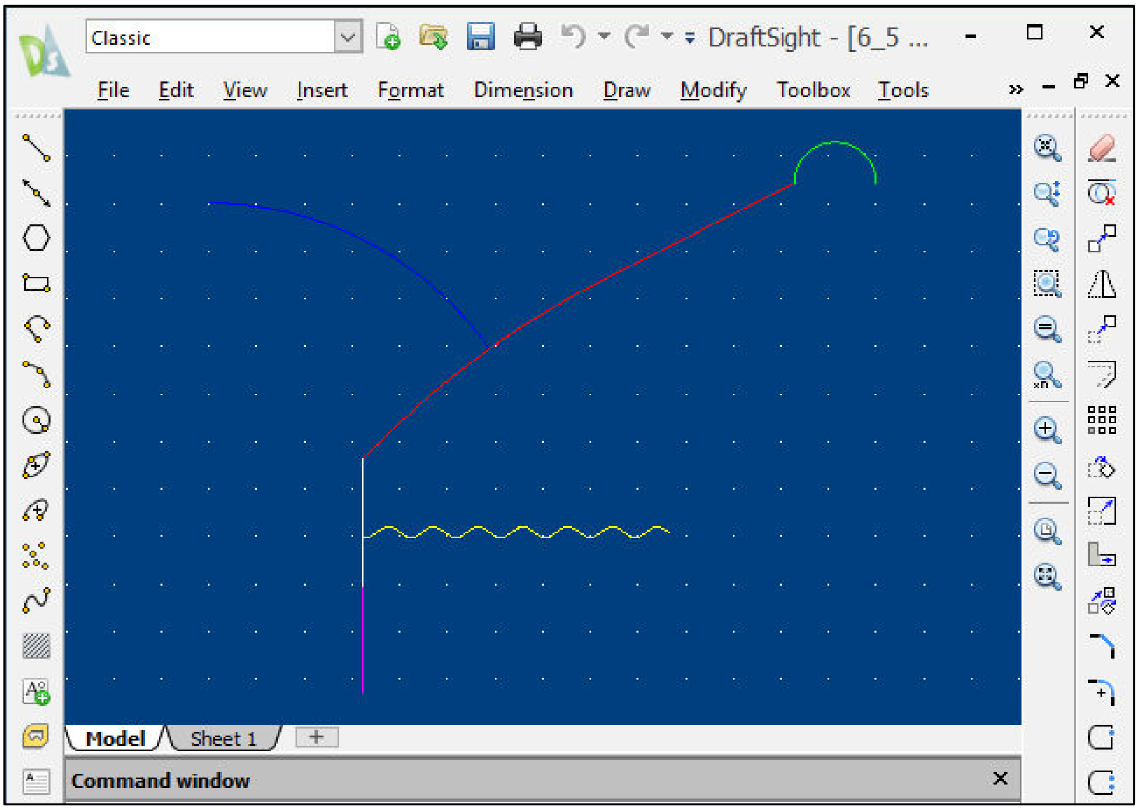


Figure 2 - FINECone reference DXF in DraftSight

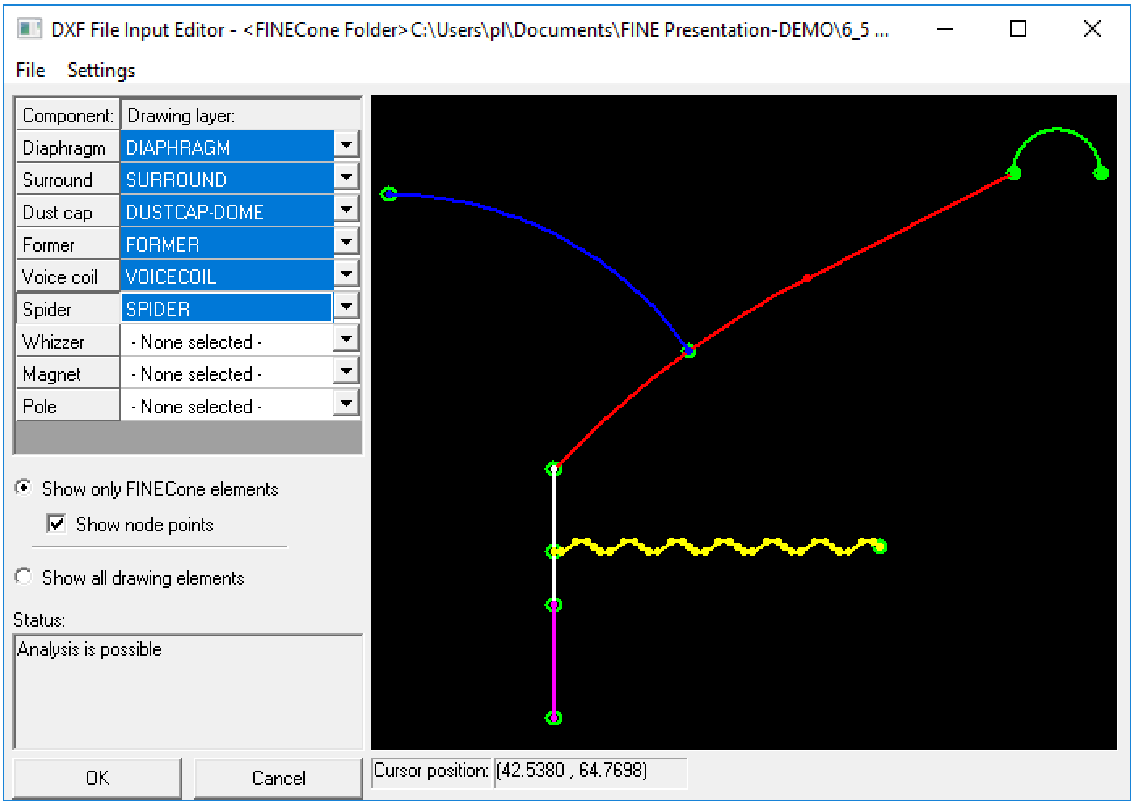


Figure 3 - FINECone Geometry

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 2. 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 3, 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.

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.

1.1.1 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 it’s 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 28.
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.

1.2 Project information window

1.2.1 Geometrical properties

Figure 4 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.

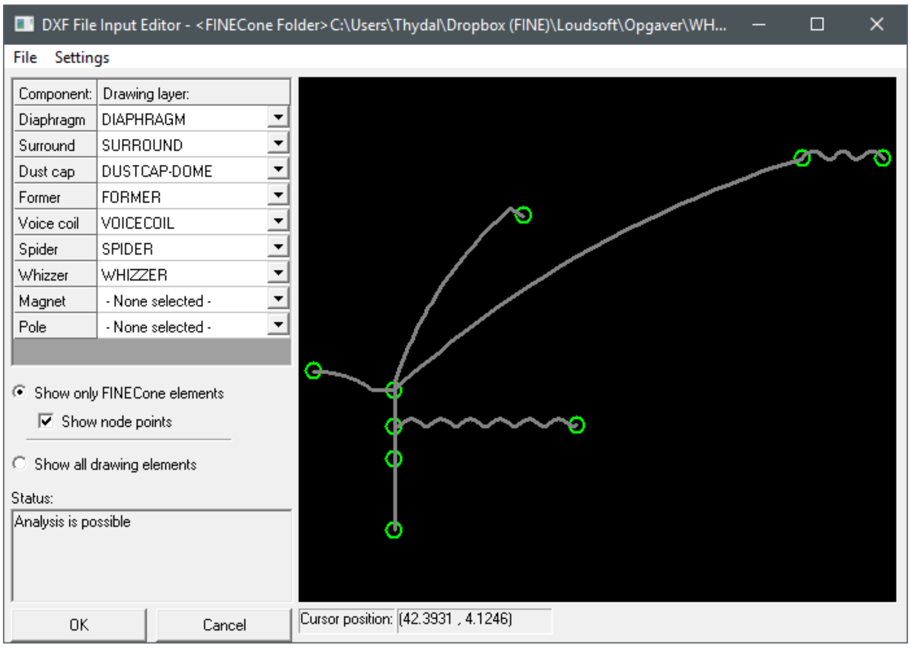


Figure 4 - The geometrical properties of a 6.5-inch woofer with a whizzer cone

1.2.2 Material properties

The Material Properties window 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.

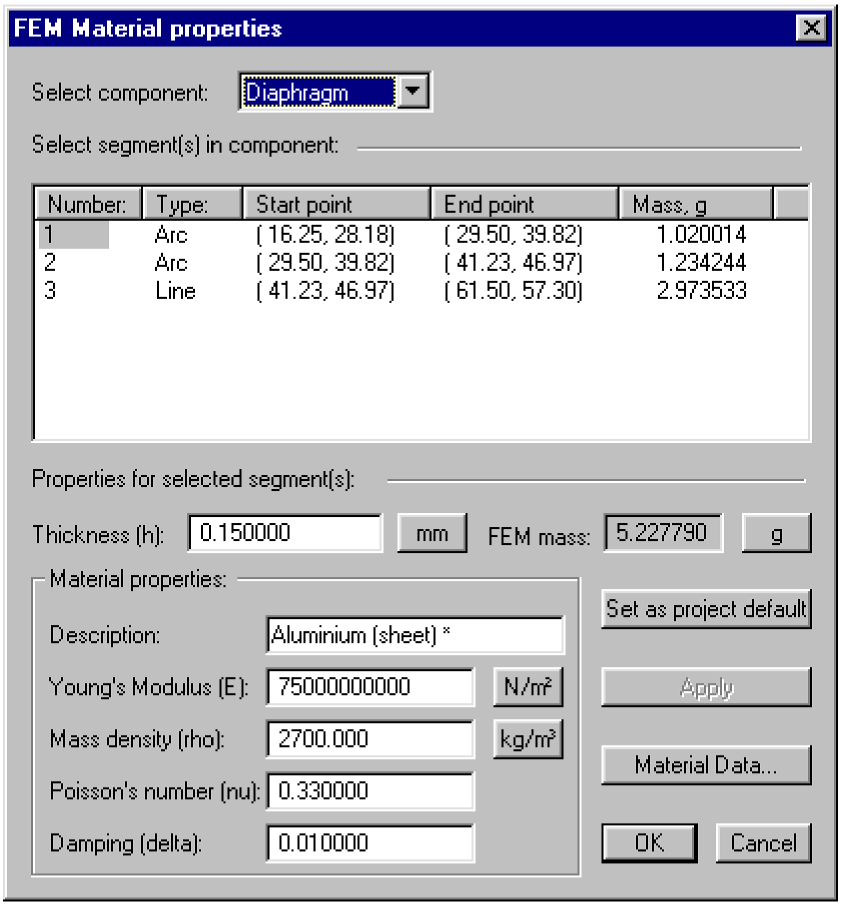


Figure 5 - Material properties can be defined for each individual segment in each component

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.

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 5. 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.

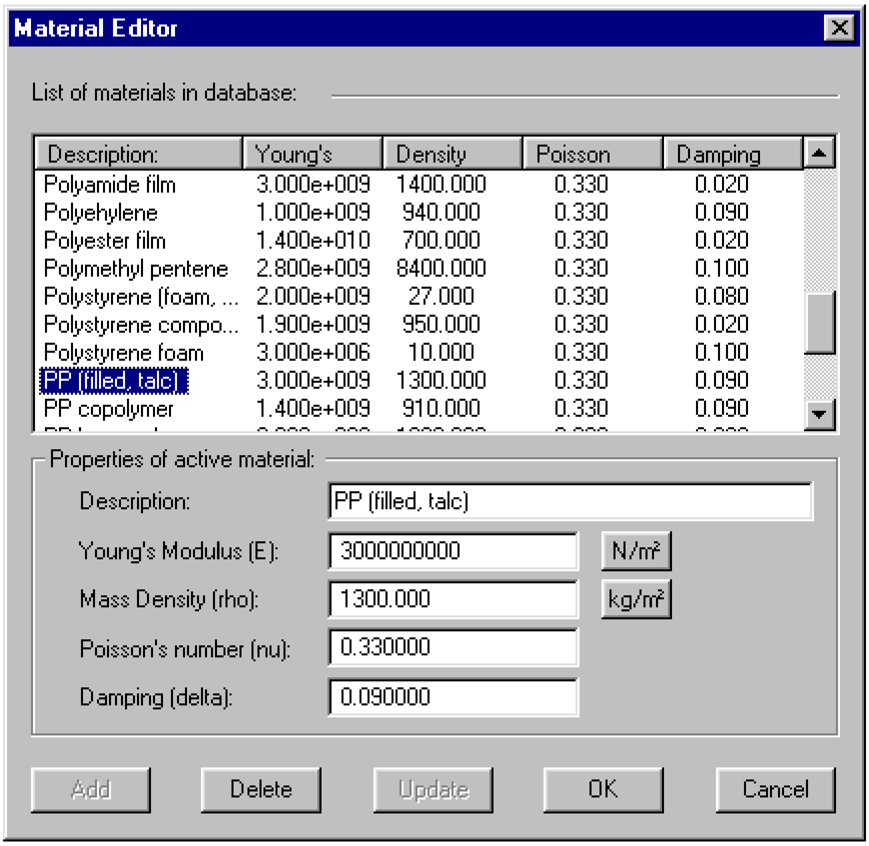


Figure 6 - Material database with many predefined, common materials

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.

1.2.3 Lumped parameters

In the Lumped Parameters mode, the components are modelled as simple and ideal masses and compliances (inverse stiffness). This mode is normally used to quickly simulate a response without break-up. The ‘General’-tab can be used to check the Thiele-Small parameters.

Note that these values may not be correct when a real FEM simulation is performed.

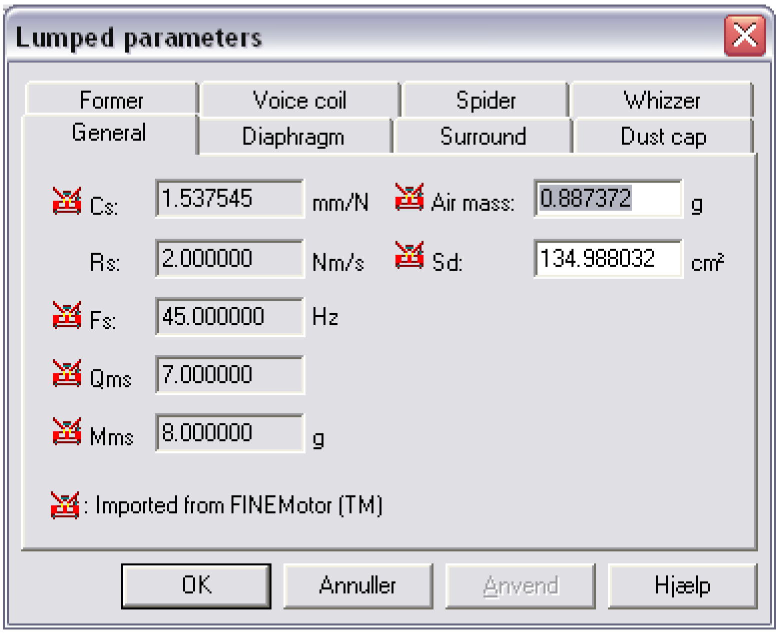


Figure 7 - Some of the Thiele-Small parameters. The red loudspeaker icon indicates that the values are imported from FINEMotor

In Figure 8 is shown the tab for surround as an example. Note the mass may be calculated using the Finite Element Method (FEM). The mass factor is only used in Lumped Parameters mode and specifies how much of the surround is contributing to the actual moving mass. The compliance can be input from FINEMotor or calculated using FEM. The resistance is the mechanical damping of the component and may be calculated using FEM.

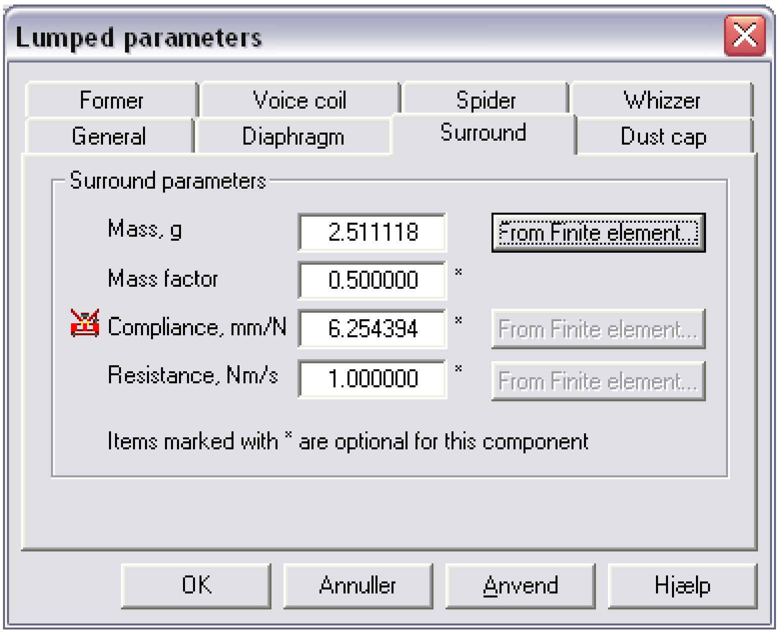


Figure 8 - Surround tab shown with mass calculated from the finite element method

1.2.4 Electrical properties

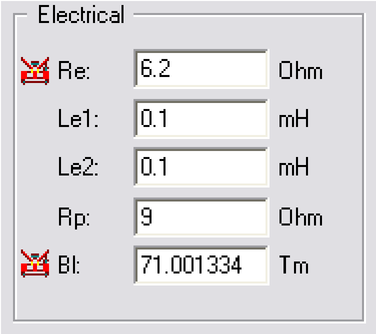


Figure 9 - The electrical parameters with Re and Bl imported from FINEMotor

In the Electrical Properties window, the user can input information about the voice coil of the loudspeaker which is used for simulating the impedance of the loudspeaker.

- 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. The symbol next to Re indicates import from FINEMotor.
- 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.

The user can also choose to import a measured impedance curve by

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.

1.2.5 Frequency Range

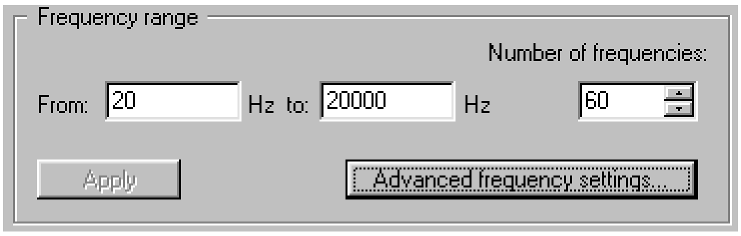


Figure 10 - The frequency range and how many sections the frequency range is divided into

The default frequency range is from 20 to 20000 Hz. The calculated frequency points are determined by the number of frequencies. These are by default logarithmically spaced over the selected frequency band. Use a low number (10) of frequencies to reduce calculation time until the mechanical parameters are satisfactory. Then you may 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. 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.

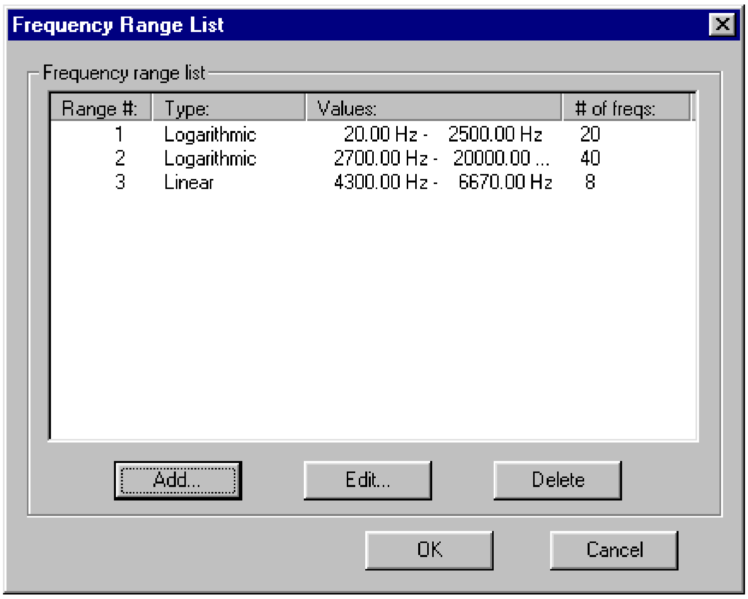


Figure 11 - List of pre-set frequency ranges

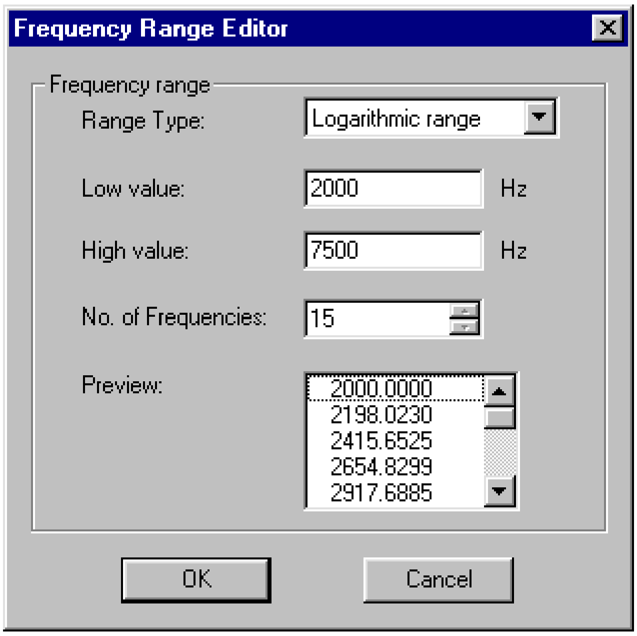


Figure 12 - Window allowing you to pre-set new frequency ranges

1.3 Impedance window

The impedance window in Figure 13 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'.

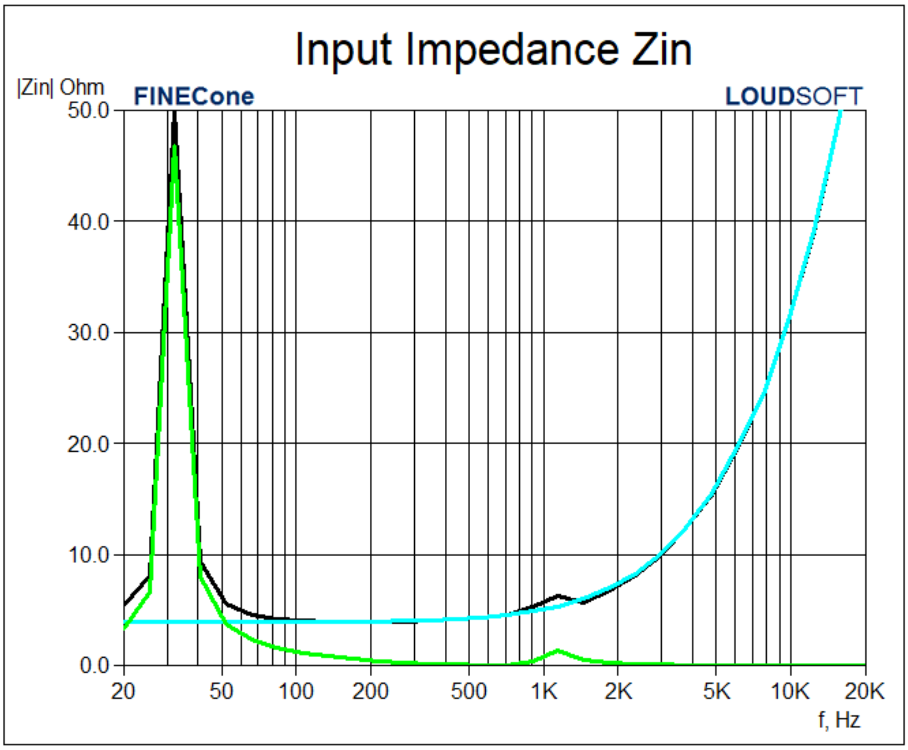


Figure 13 - Impedance window showing the impedance of a 10-inch subwoofer

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

1.4 Sound Pressure Level window

The Sound Pressure Level window shown in Figure 14 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.

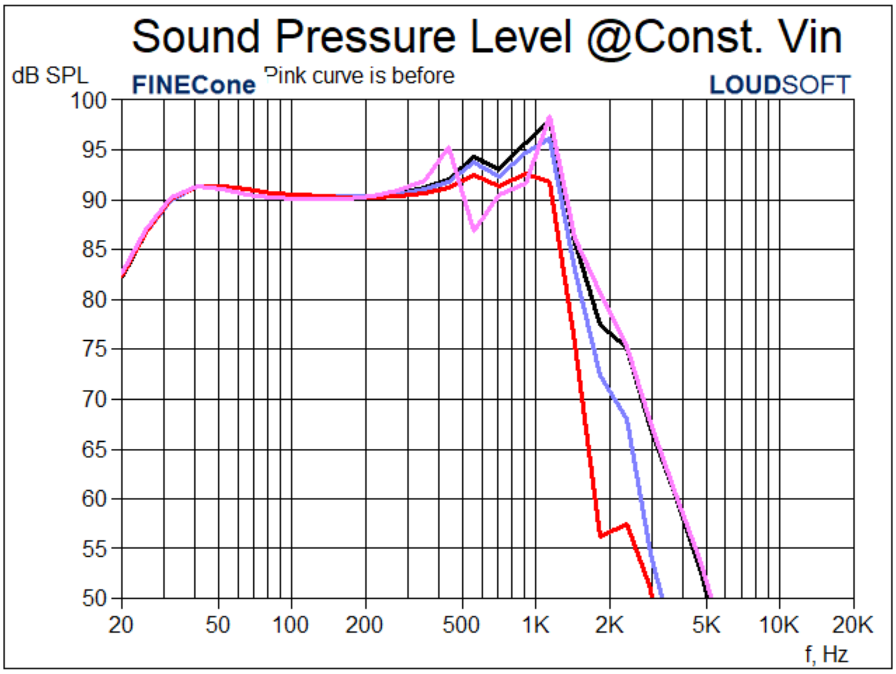


Figure 14 - The Sound Pressure Level window shows the on-axis as well as two off-axis simulated curves of a 10-inch subwoofer. A measured frequency response has been imported.

1.5 Animation window

The animation window is a useful tool to visualise the behaviour of the individual parts of the loudspeaker at different frequencies. This can help pinpoint what component is breaking up and causing fluctuations in the frequency response.

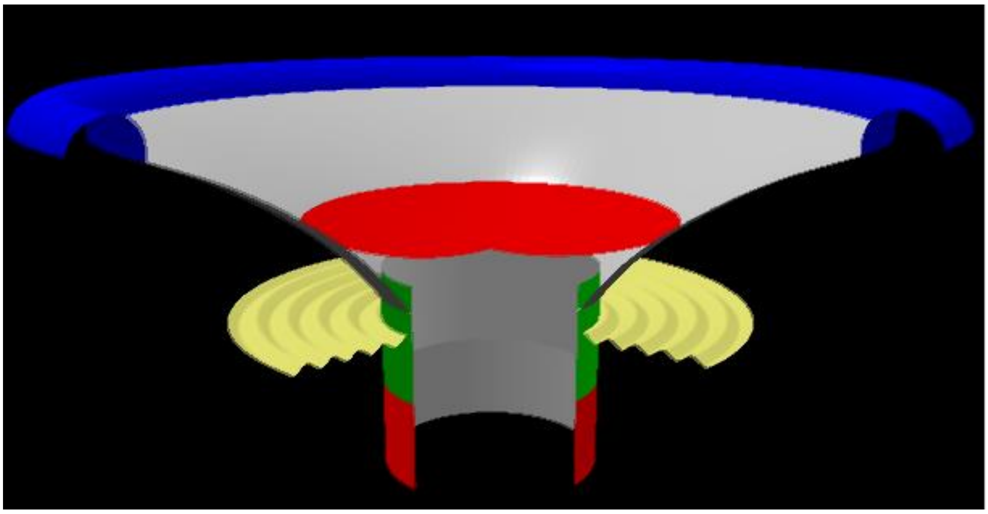


Figure 15 - The 3D animation window makes it easier to visualise the behaviour of the cone at different frequencies.

The first drop down menu shown in Figure 16 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.

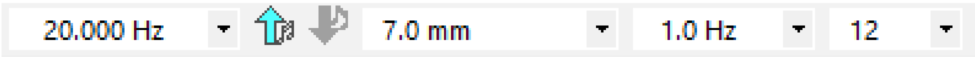


Figure 16 - Drop down menus for the animation window

The first button from the left in Figure 17 starts and stops the animation. The buttons are only click-able when the animation window is selected. The second button from the left 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 third button from the left toggles the mesh of the model on and off. The fourth button from the left allows you to zoom and move the model around in the animation window. The last button from the left controls image smoothing.

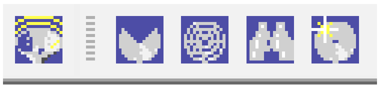


Figure 17 - Buttons for how to display the animation

1.6 Toolbar

1.6.1 Import from FINEMotor

It is possible to import the Thiele-Small parameters from FINEMotor by clicking on . This is highly useful when simulating a new driver that is still not able to be physically measured. The magnet and pole properties can then be edited using these two buttons .

1.6.2 Window selection

The different windows available in FINECone can be toggled on and off using the buttons shown in Figure 18. 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 is the animation window.

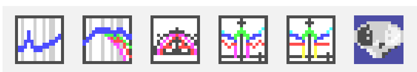


Figure 18 - The six buttons for the different windows in FINECone

1.6.3 Include/exclude mechanical parts

It is possible to include or exclude different parts of the model using the buttons shown in Figure 19. This can be useful if you for instance have measured the loudspeaker with the surround on and with the surround cut off in order to measure the compliance of the spider. Or if you have two identical drivers with and without a whizzer cone and wish to see the difference.

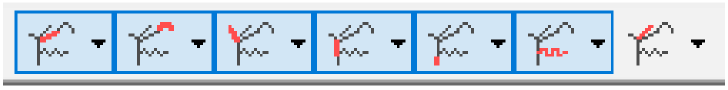


Figure 19 - Include or exclude different parts of the model

It is also possible to keep all the components included in the model, and then instead only exclude them when calculating the frequency response. This can be 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 20.

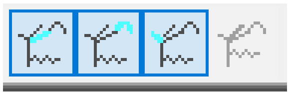


Figure 20 - Include or exclude the contribution of different parts of the model in the frequency response

1.6.4 2D plot

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

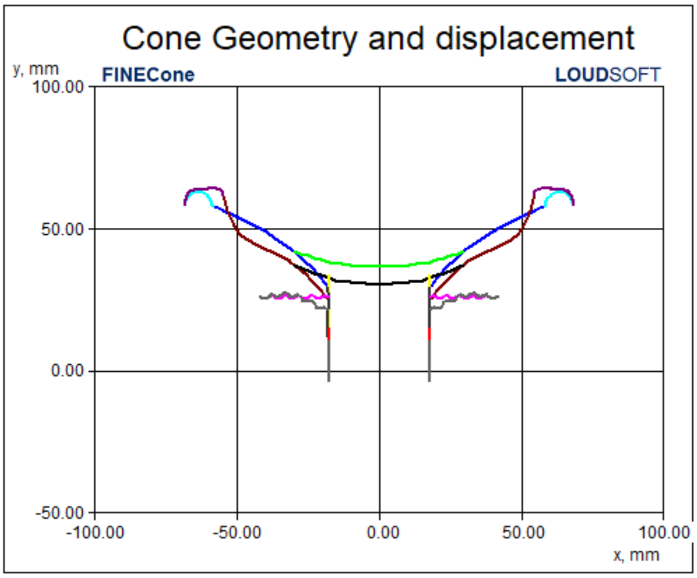


Figure 21 - 2D plot showing the model when at rest and when driven at 1863 Hz

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.

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

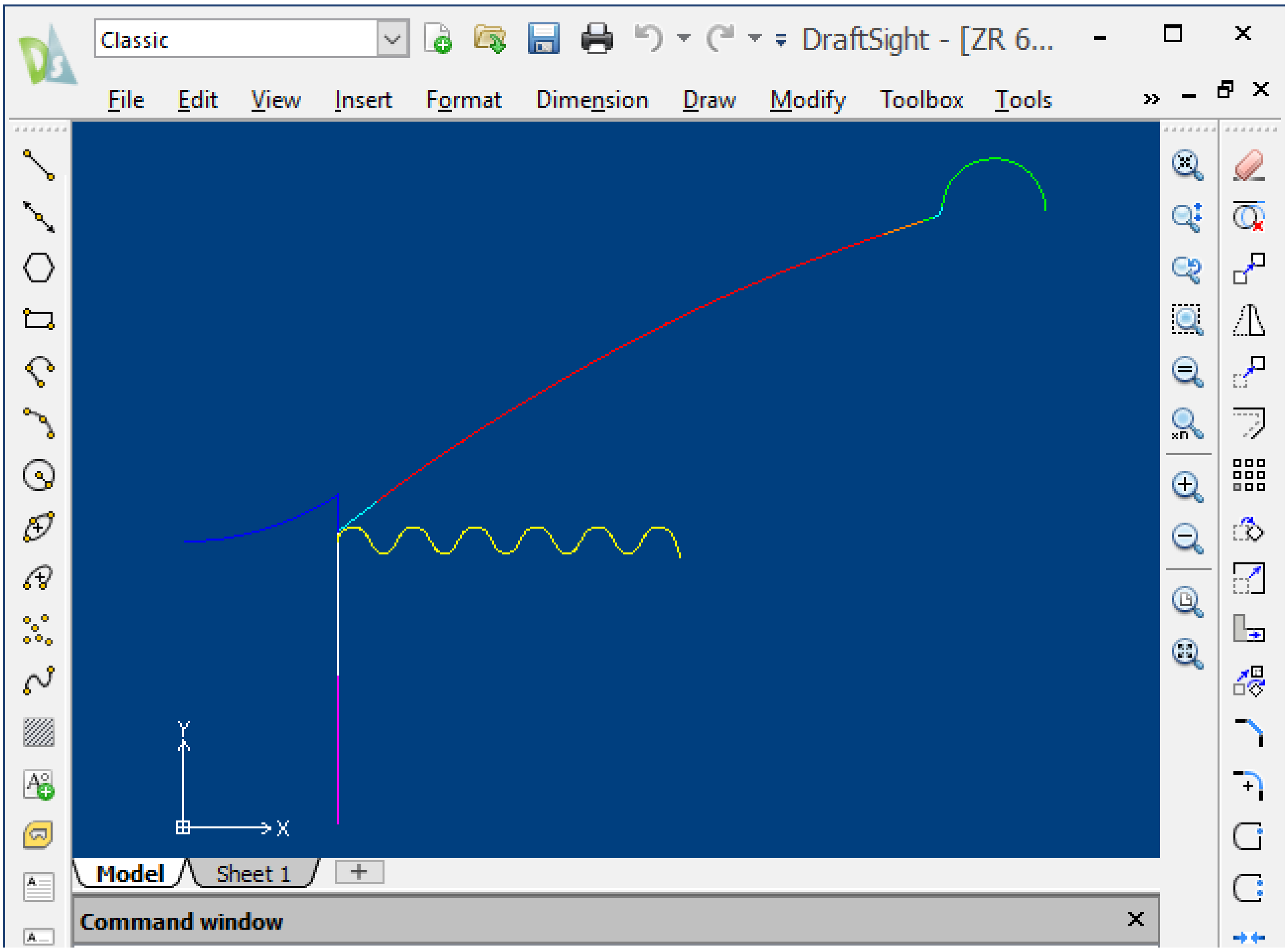


Figure 22 - DXF created using DraftSight

We recommend the free 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 below (Sec. **Error! eference source not found.**) how to make the DXF file.

Please note that the DXF file is simplified, especially regarding the VC former (Fig.1), 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, which is drawn as one line (orange) in the Diaphragm layer. The actual cone is glued to the first roll of the spider. Since we cannot model double lines, we have simply split the cone into two arcs, as indicated by the cyan colour.

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

Refer to Section 1.1.1 for hints on how to create a DXF file.

Step 3 - Start the simulation using the FINECone Wizard.

Press  button to start the FINECone Wizard.

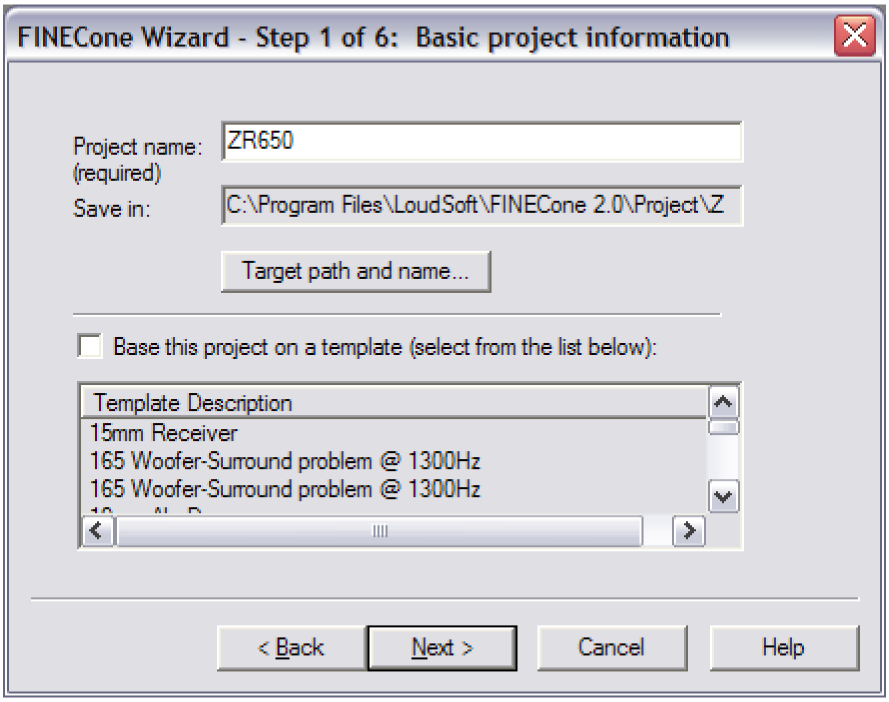


Figure 23. FINECone Wizard

If we had saved an earlier simulation as a template file (.FTE) we could select that as a start.

However, this analysis is new and we therefore continue with next.

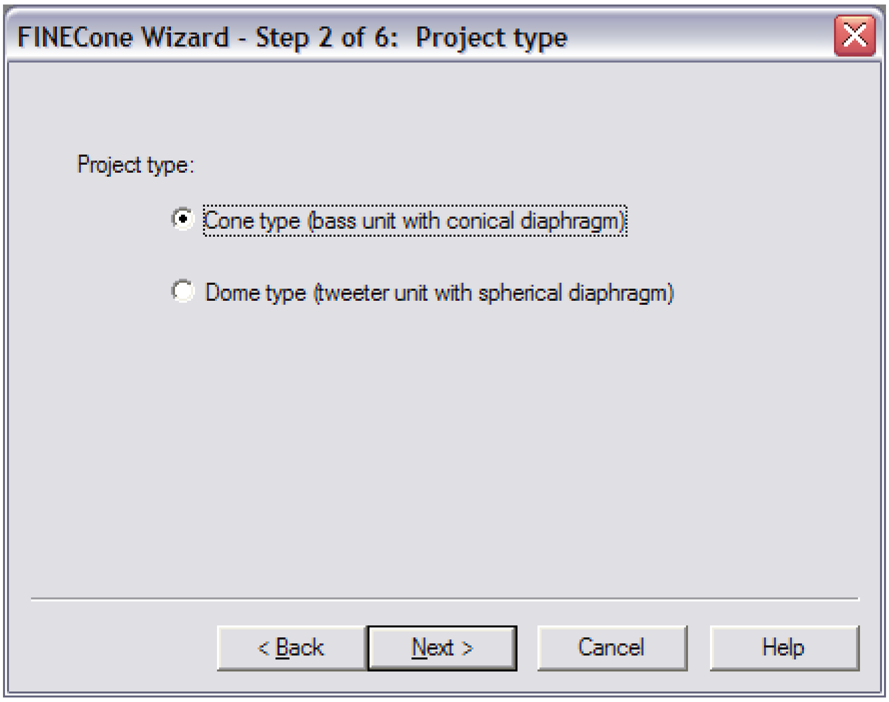


Figure 24. Choose Cone type (Project Type)

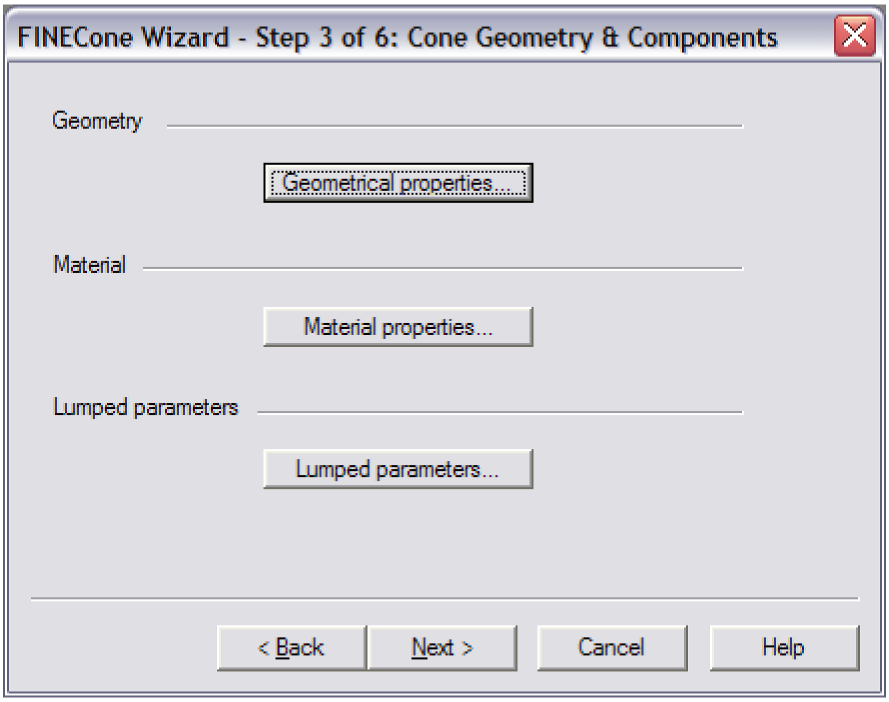


Figure 25. General FINECone Steps

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 26) 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 27.

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.

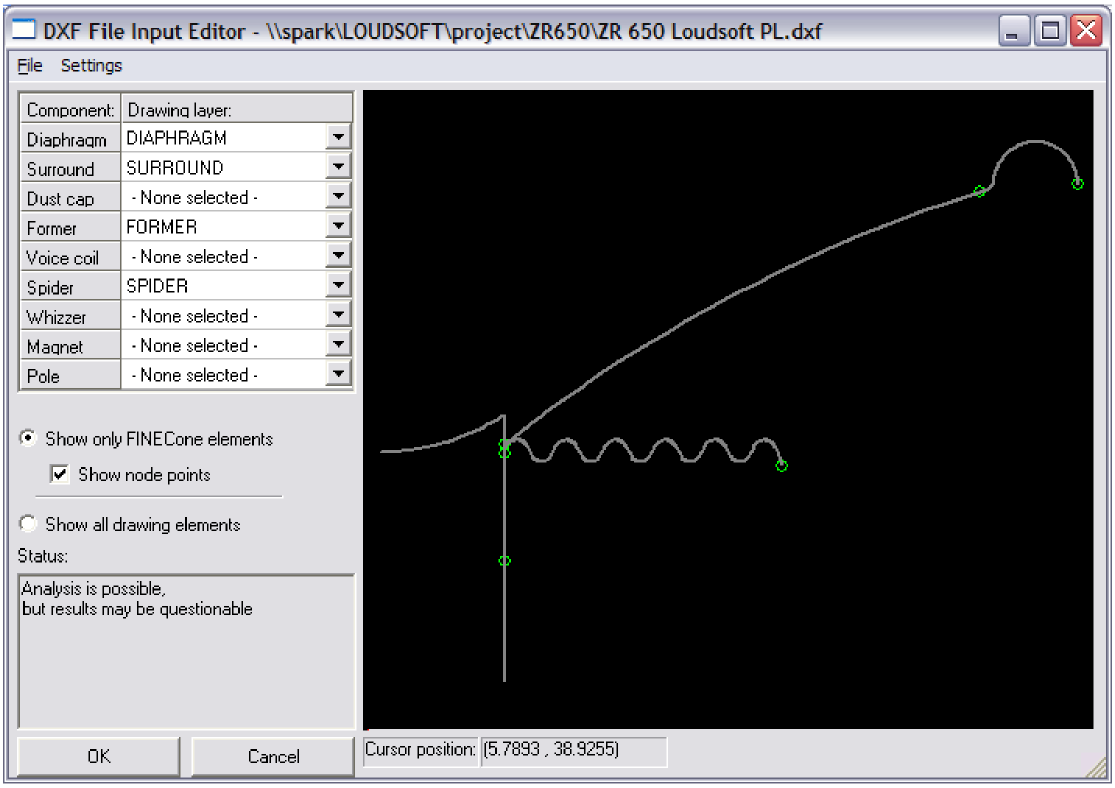


Figure 26. DXF Import -Input Editor

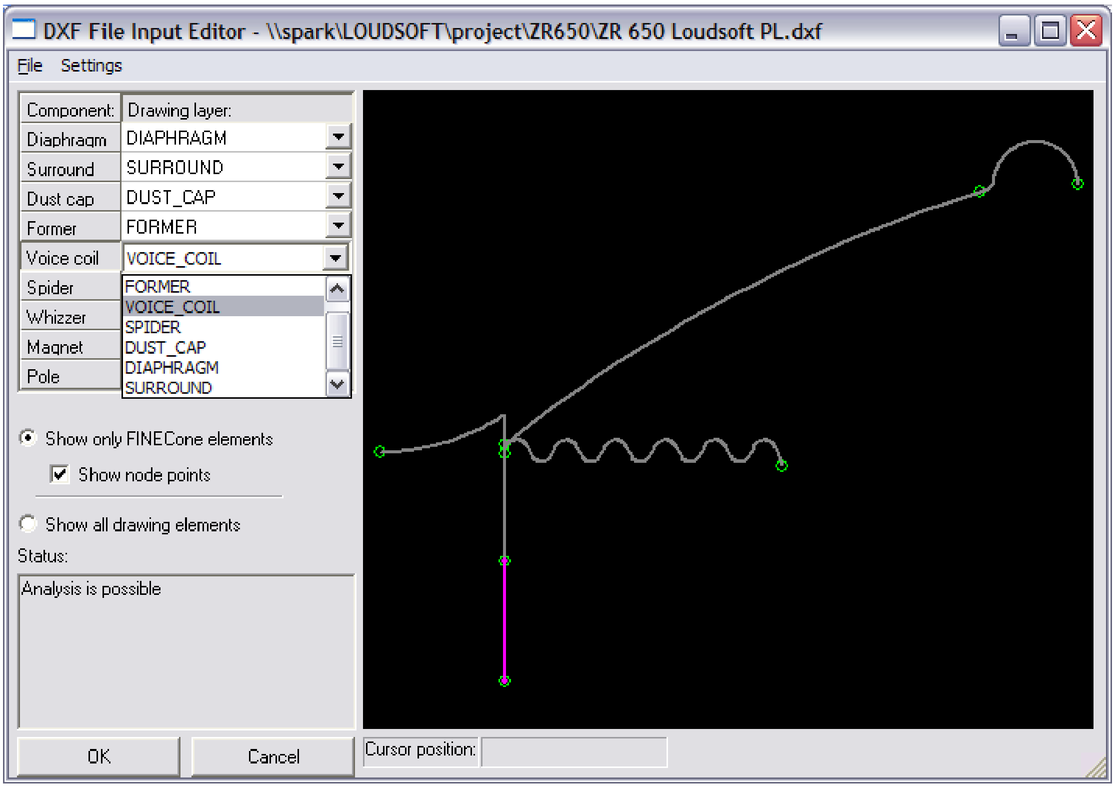


Figure 27. DXF layer - drop down menu

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.

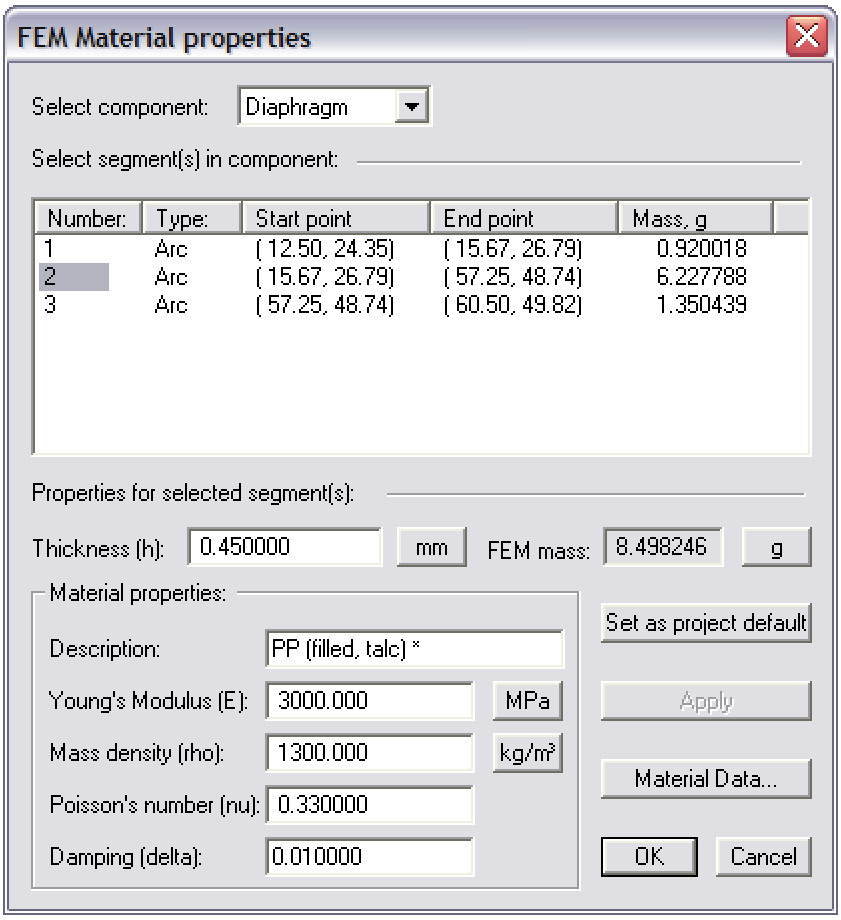


Figure 28 - The thickness of the various components are set in the FEM Material properties window

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.

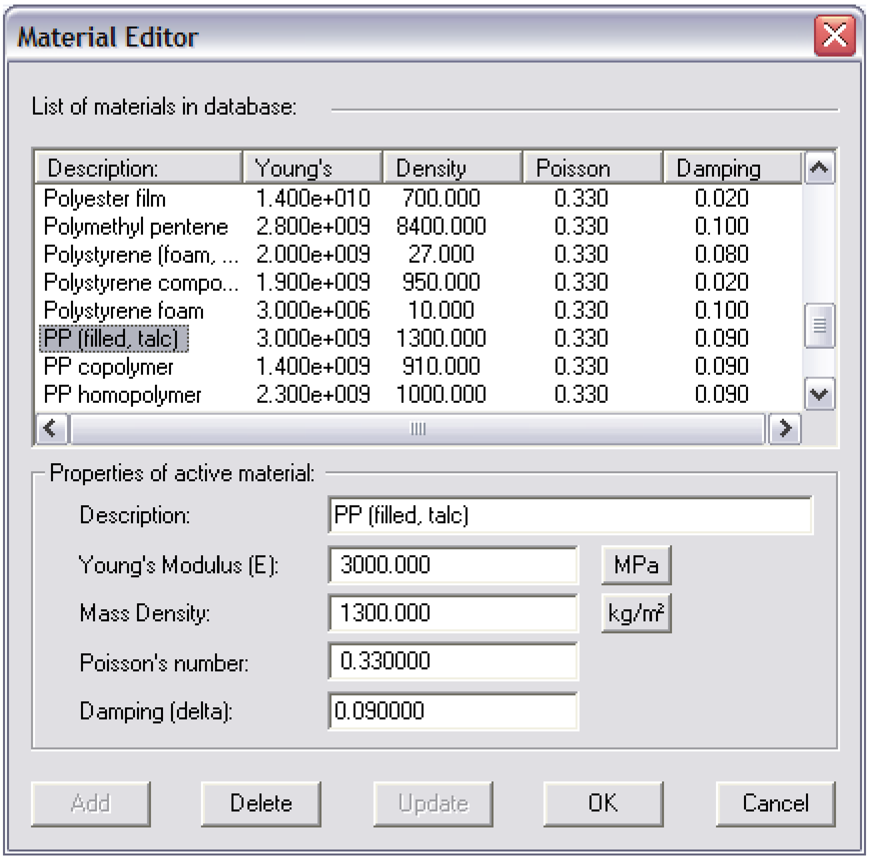


Figure 29 - The cone material is selected from one of the standard materials in the FINECone materials database

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

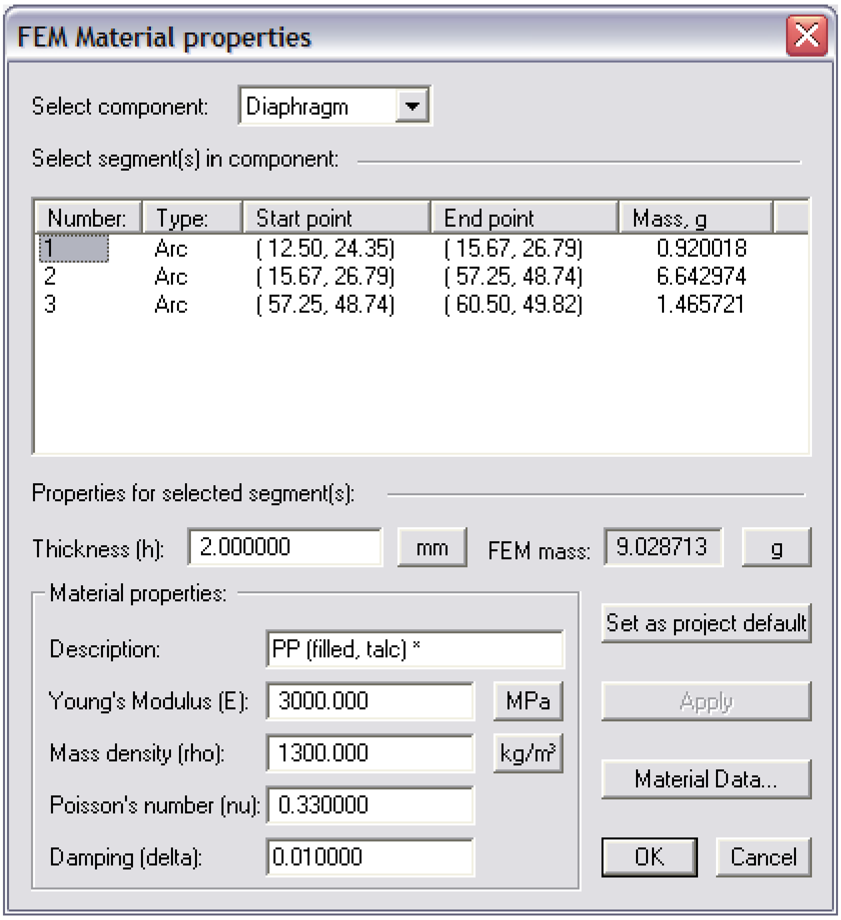


Figure 30 - Part of the cone is made thicker to account for the glue on the voice coil

The surround is 0.41mm rubber from the database.

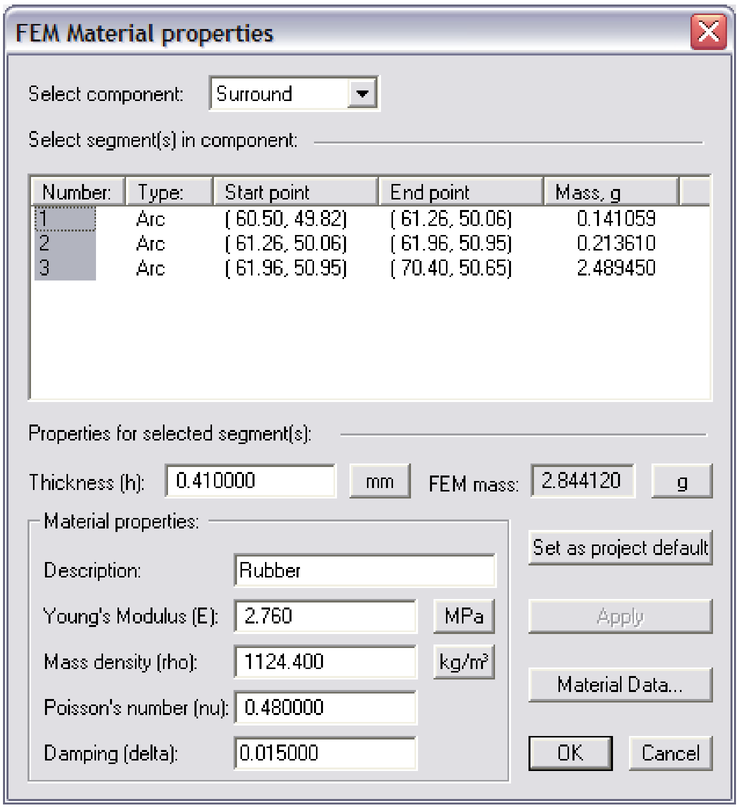


Figure 31 - The surround material is using the Rubber material which has the properties of the most common type of rubber used for surrounds

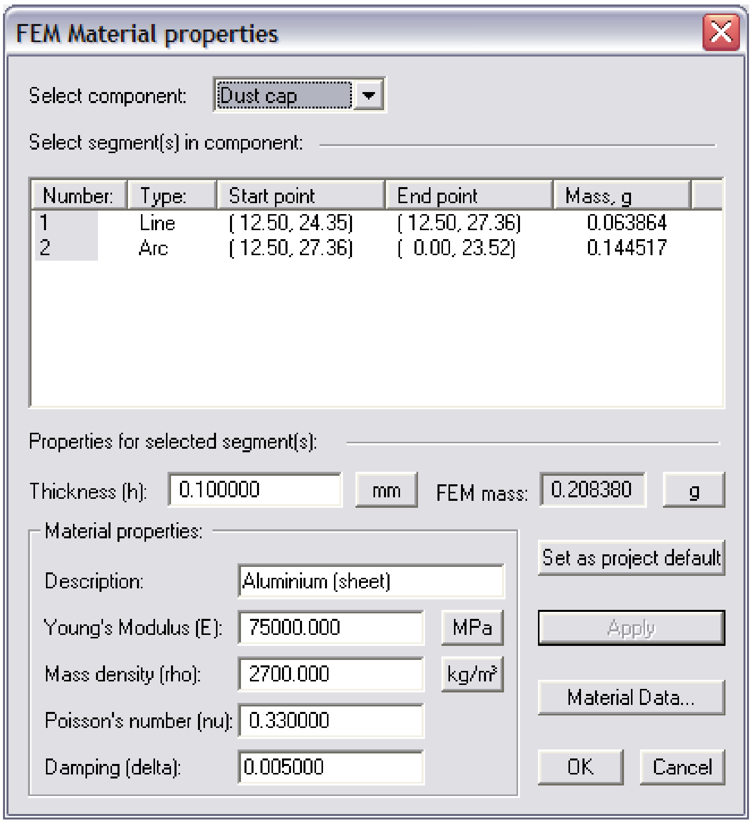


Figure 32 - For the dust cap aluminium is chosen

Note: Don't forget to press 

, whenever you have made a change.

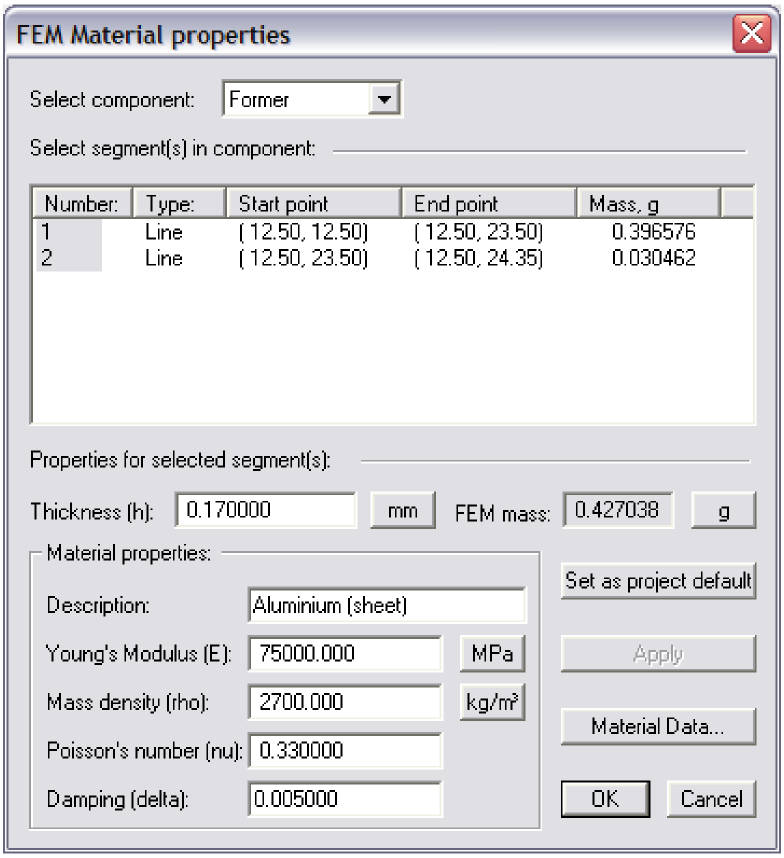


Figure 33 - Also for the voice coil former aluminium is used

The important parameter of the voice coil is the mass of the coil winding + former covered by it, see Figure 34. We then adjust the thickness to get the same mass as measured. The VC stiffness is not used.

Figure 34 - The voice coil used in the woofer we are simulating

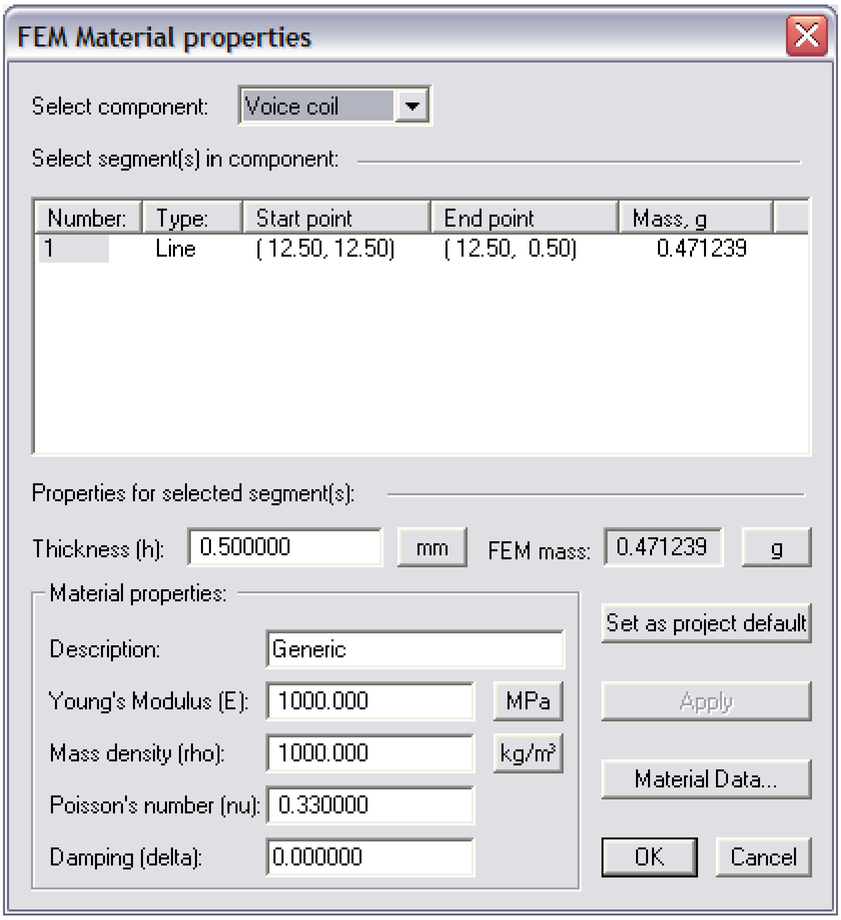


Figure 35 – For the voice coil we choose the 'Generic' material as we are going to tweak the parameters ourselves.

Since we do not know the accurate material of the spider we can use the 'generic' material found from the example file.

Remember to select all segments with CTRL+A.

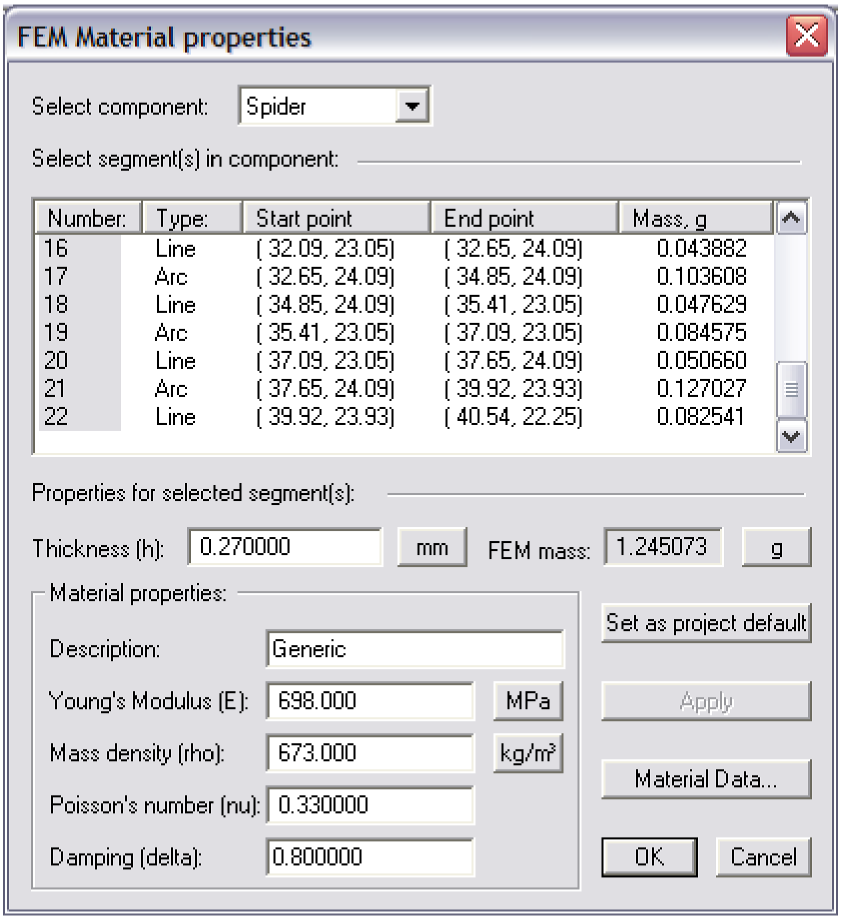


Figure 36 - For the spider the 'Generic' material is used as we do not know the precise material of the spider

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.

For the initial simulations we use the ‘Fast solution of differential equations’ setting to get faster calculation. The setting can be found in Tools/Program Options/Calculation.

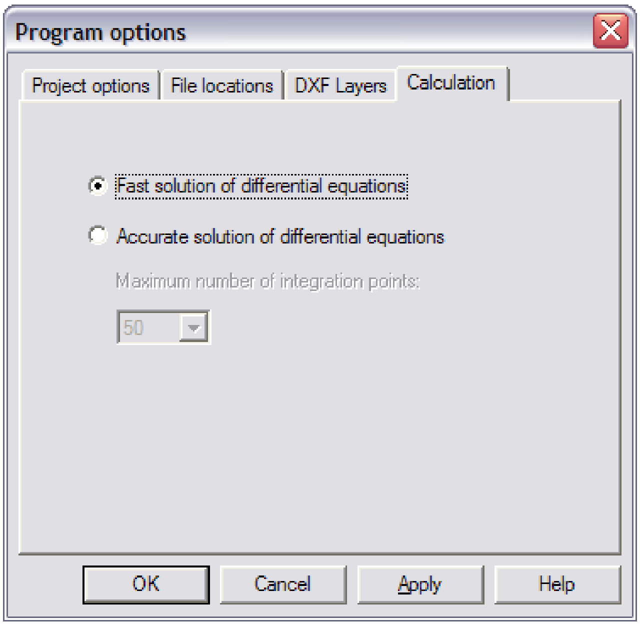


Figure 37 - Choose between quicker but less accurate and rough calculations or slower but more accurate simulations

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 [Lumped parameters...] button to get the lumped Mms values.

Input the Effective Cone Area Sd, and get the air mass calculated automatically. The air mass will automatically be included in the FEM calculations. 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.

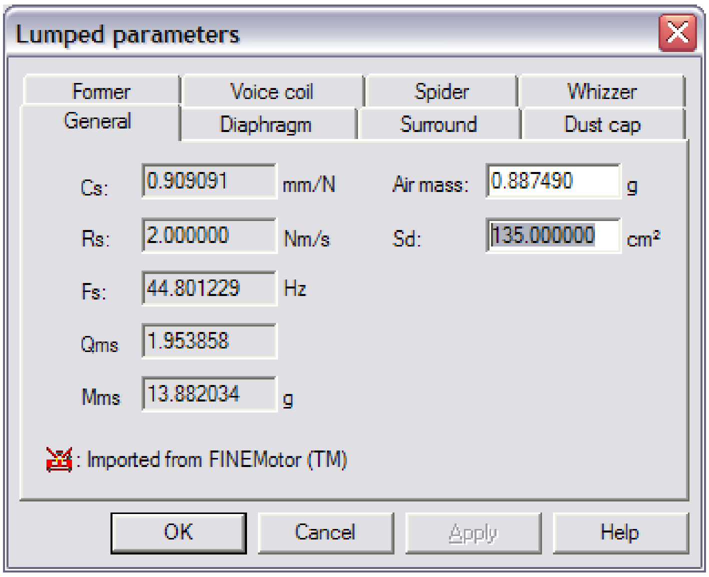


Figure 38 - The cone area is entered in the Lumped parameters window

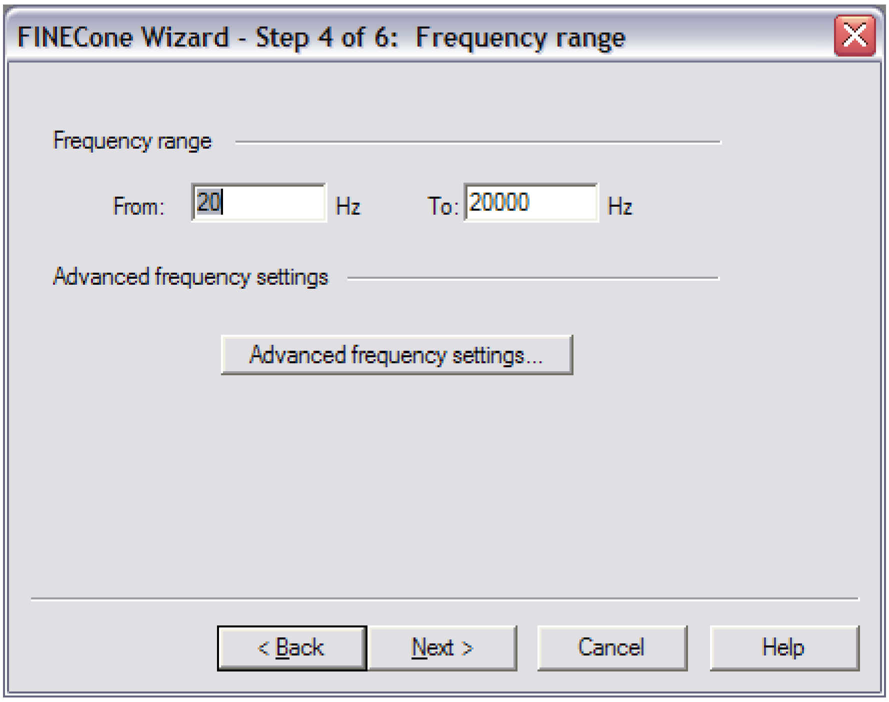


Figure 39 - The frequency range can be specified depending on the simulation

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.

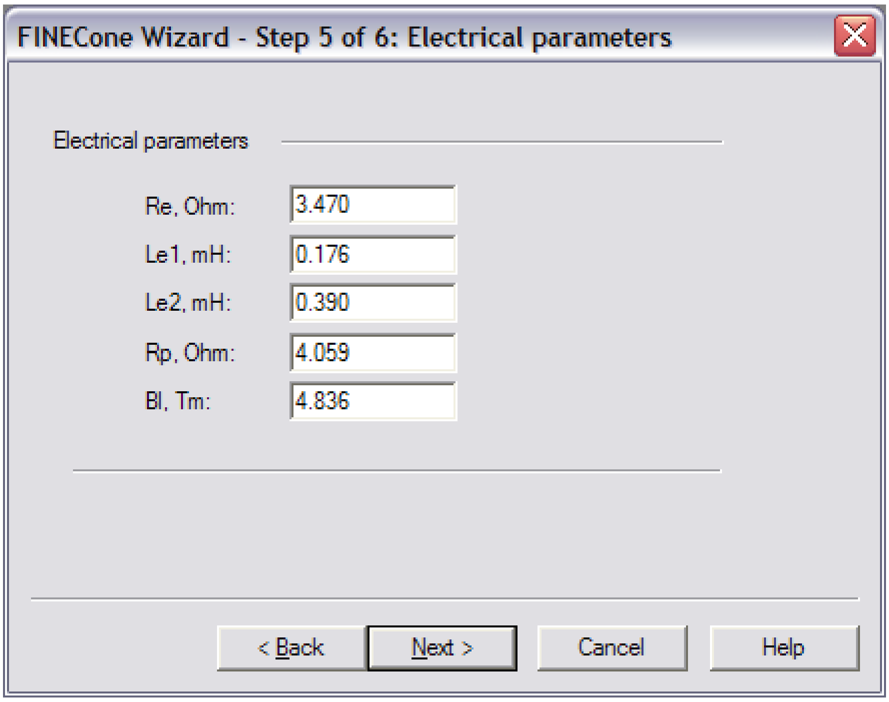


Figure 40 - The electric parameters found using FINE R+D are entered

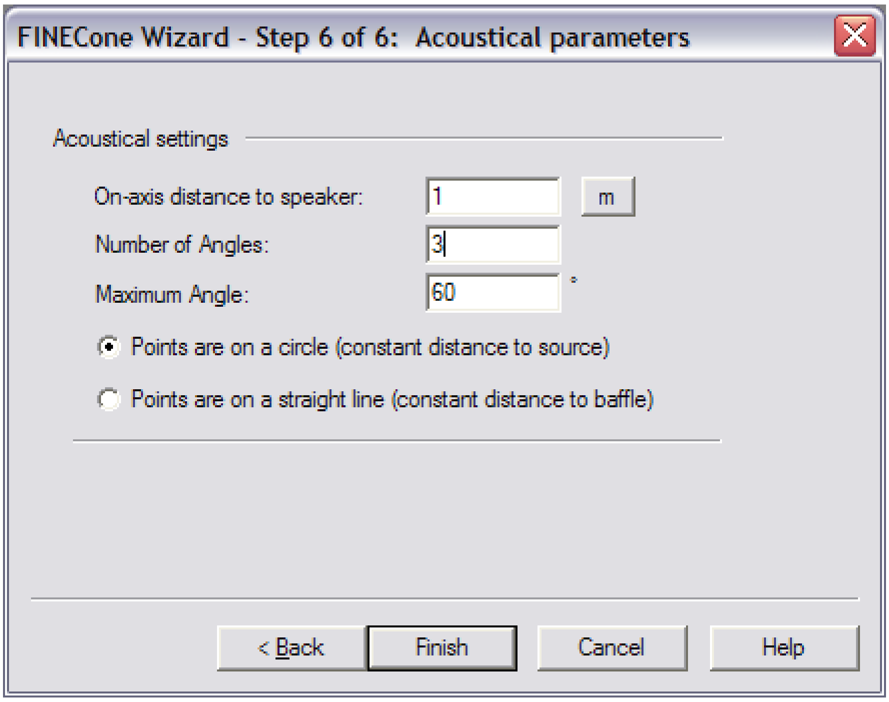


Figure 41 - Measuring points are specified as the last thing

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

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

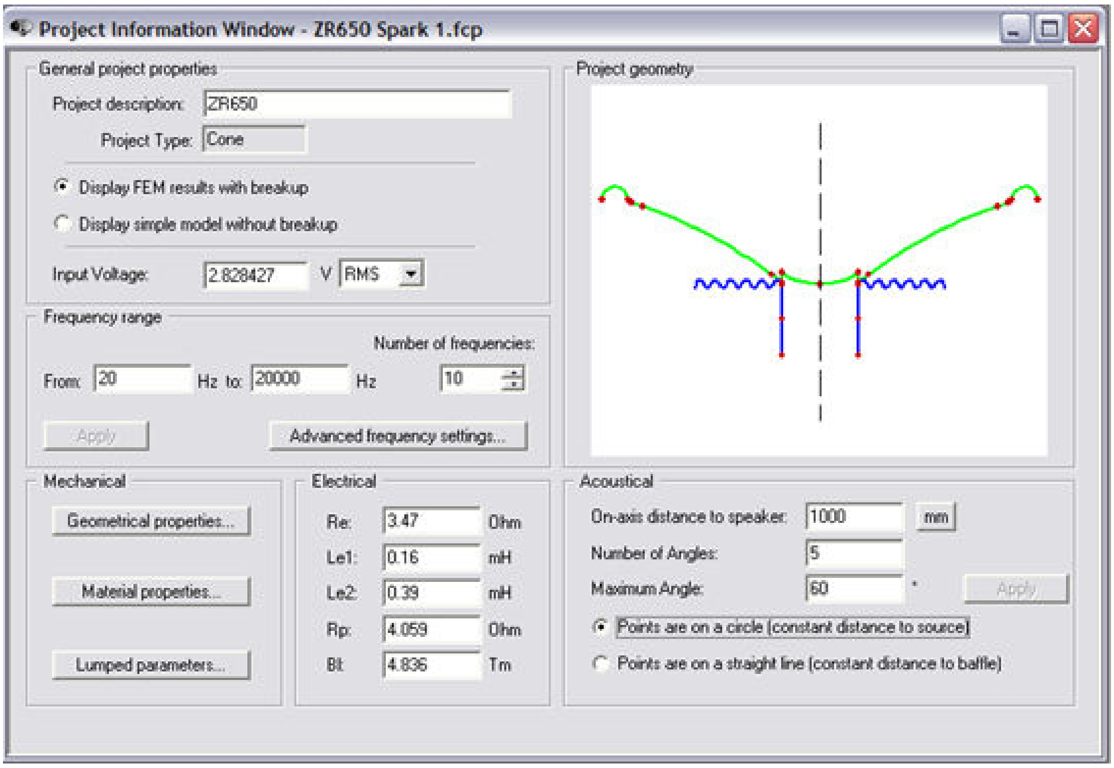


Figure 42 - This is the window you will see when you press finish if you have followed the instructions in the tutorial

Step 4 - Fit the impedance curve

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

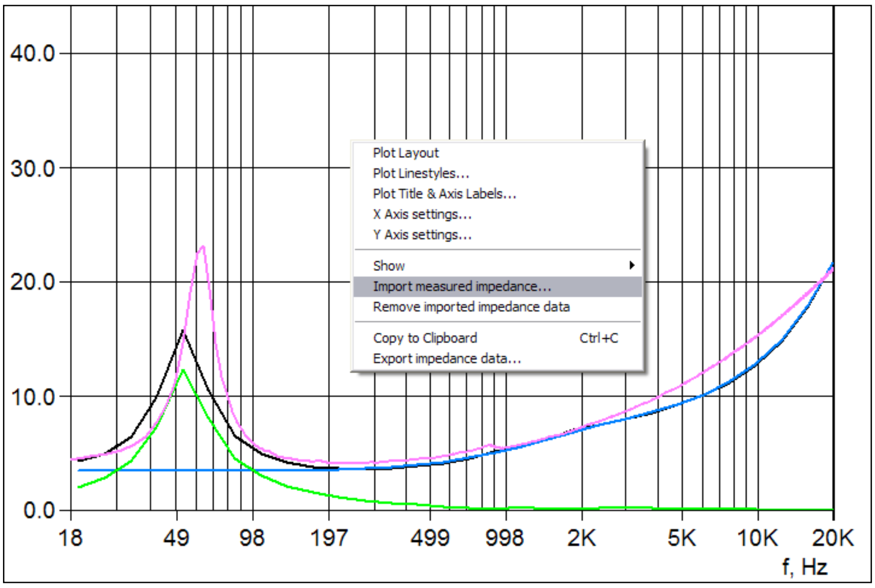


Figure 43 - The measured impedance of the driver is imported as a reference

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 software such as MLSSA.

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.

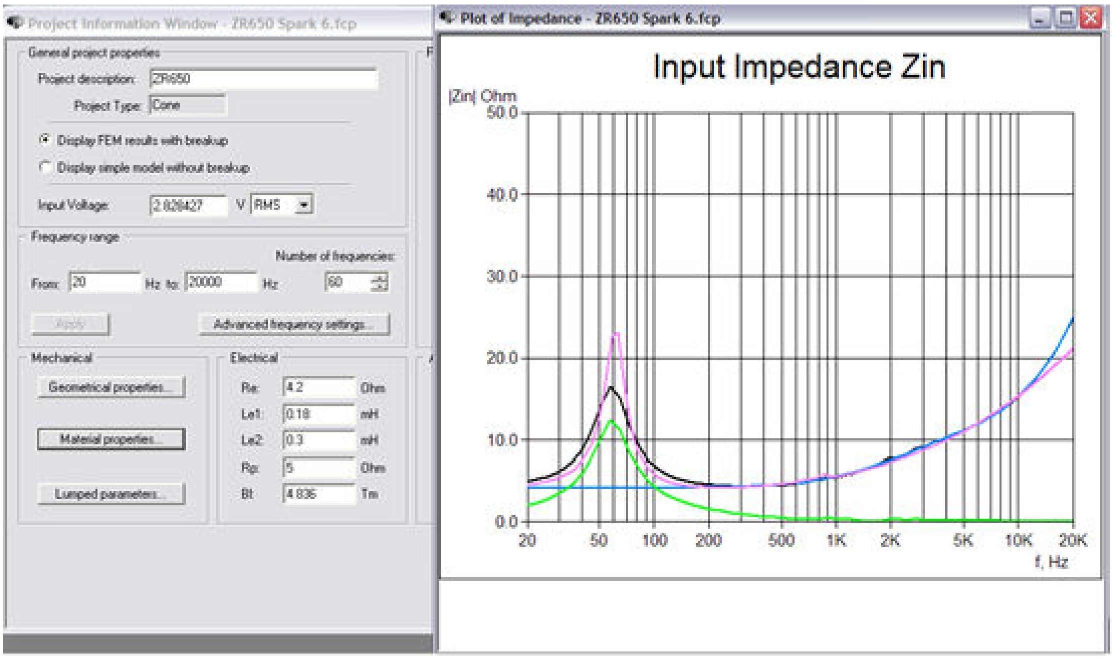


Figure 44 - After having imported the measured curve (the pink curve) the electrical parameters are tweaked to fit the simulated impedance curve

Step 5 - Fit the SPL curve

Finally, we work on the SPL curve

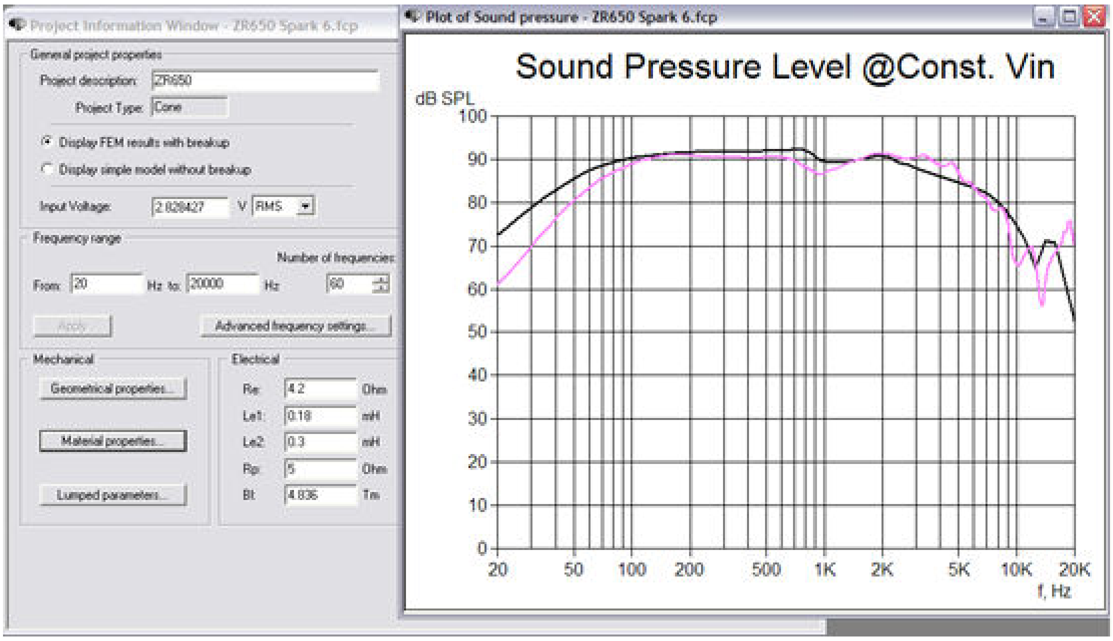
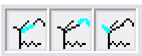


Figure 45 - The measured frequency response of the driver is imported into FINECone and is seen as the pink curve

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.

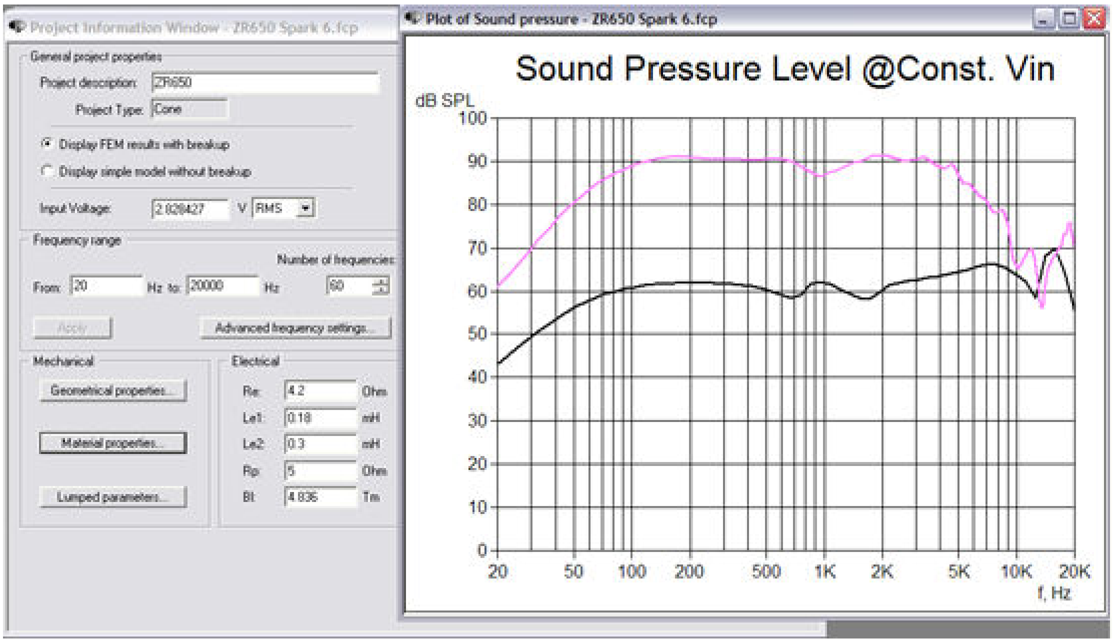


Figure 46 - Here the frequency response is shown for the dust cap alone

The dust cap only affects the very high frequencies.

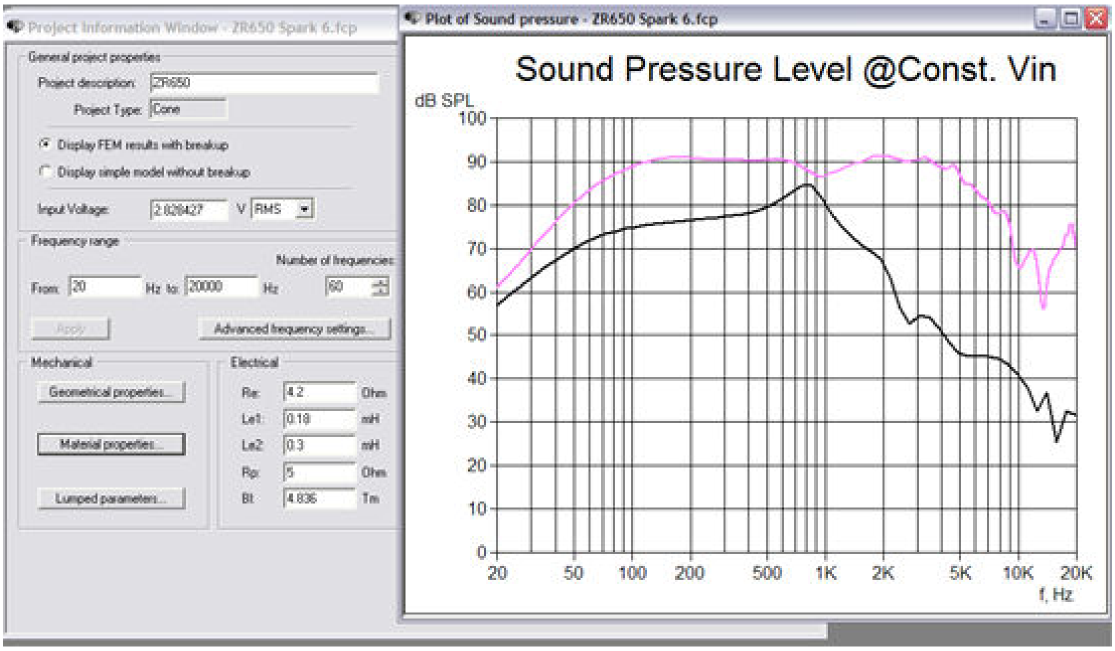


Figure 47 - The frequency response for the surround alone

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.

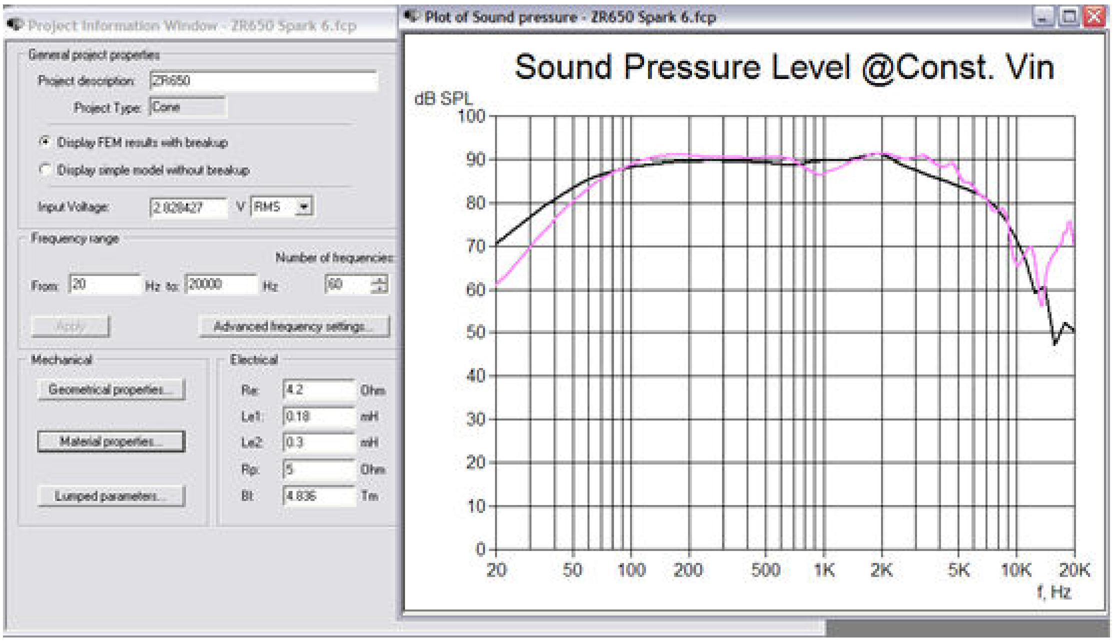


Figure 48 - The frequency response for the cone alone

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

Note: During the first simulations, the fast calculation is usually good enough. It saves lots of time.

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.

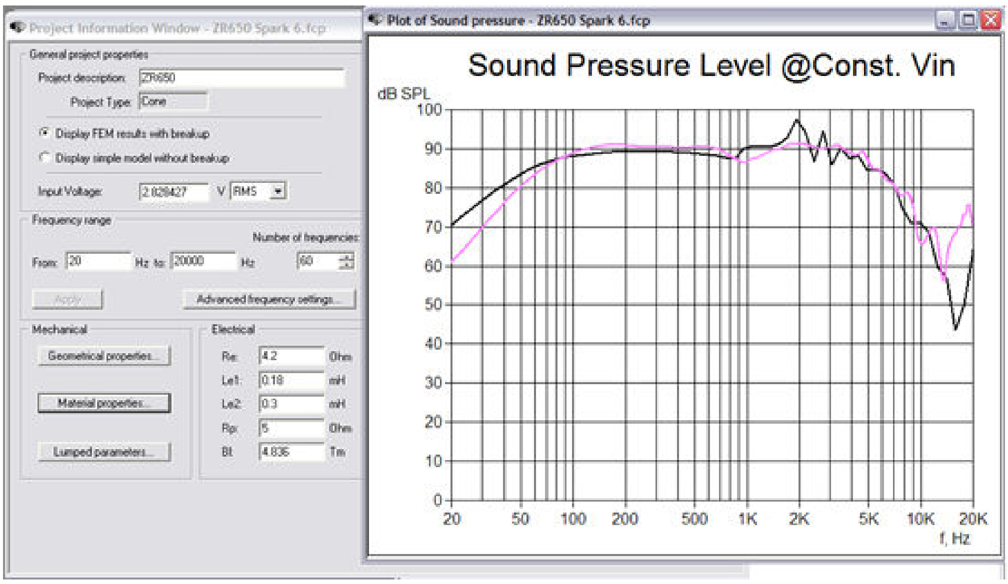


Figure 49 - The overall frequency response with the damping of the cone reduced

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

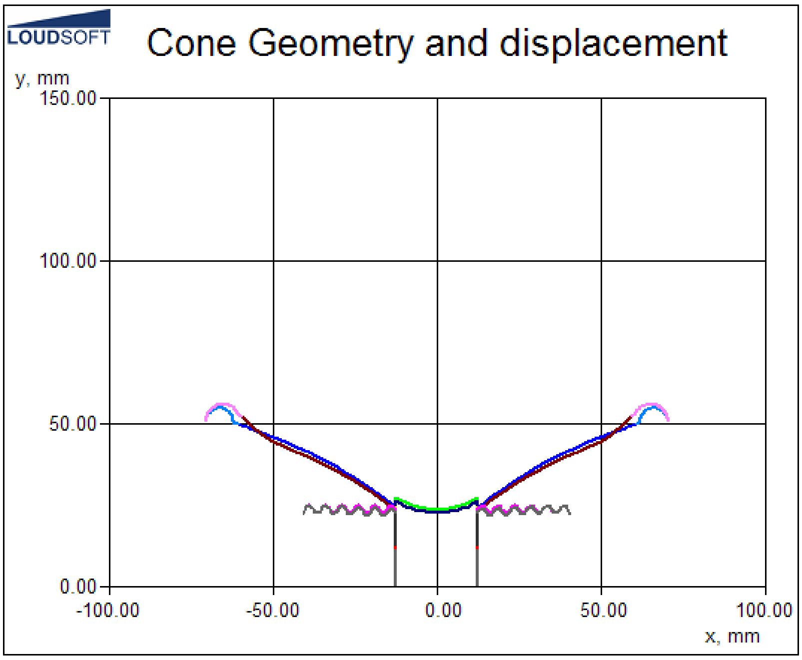


Figure 50 - A 2D plot of the driver at rest and when moving at the selected frequency.

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

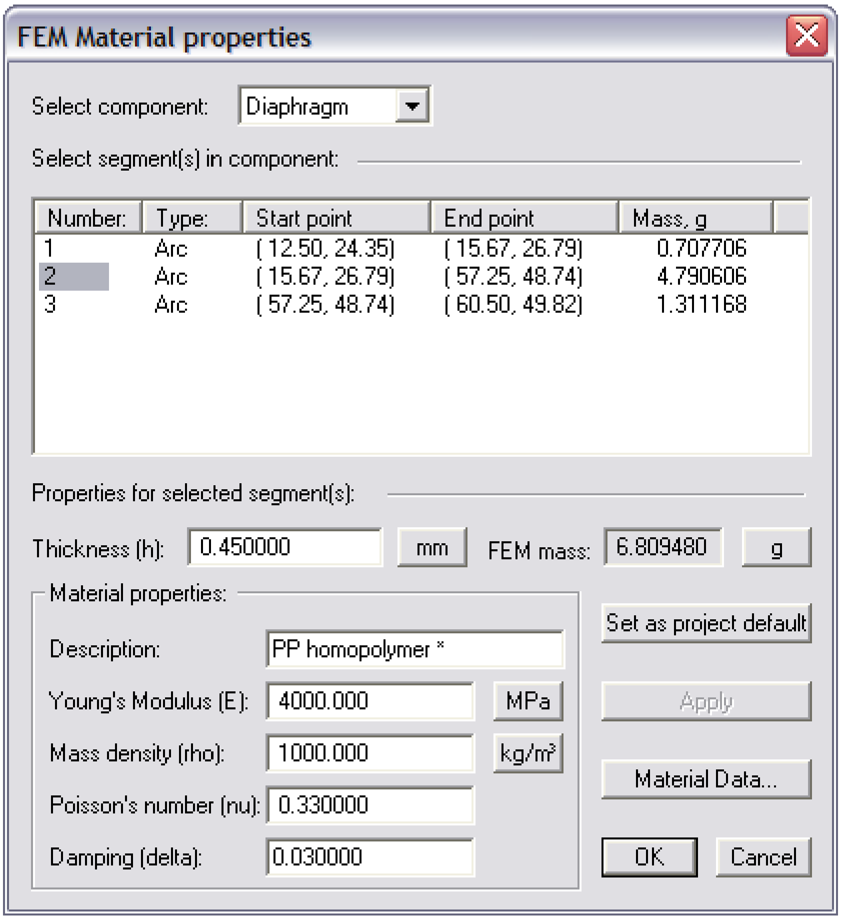


Figure 51 - The damping in the cone is changed to match the measured response

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

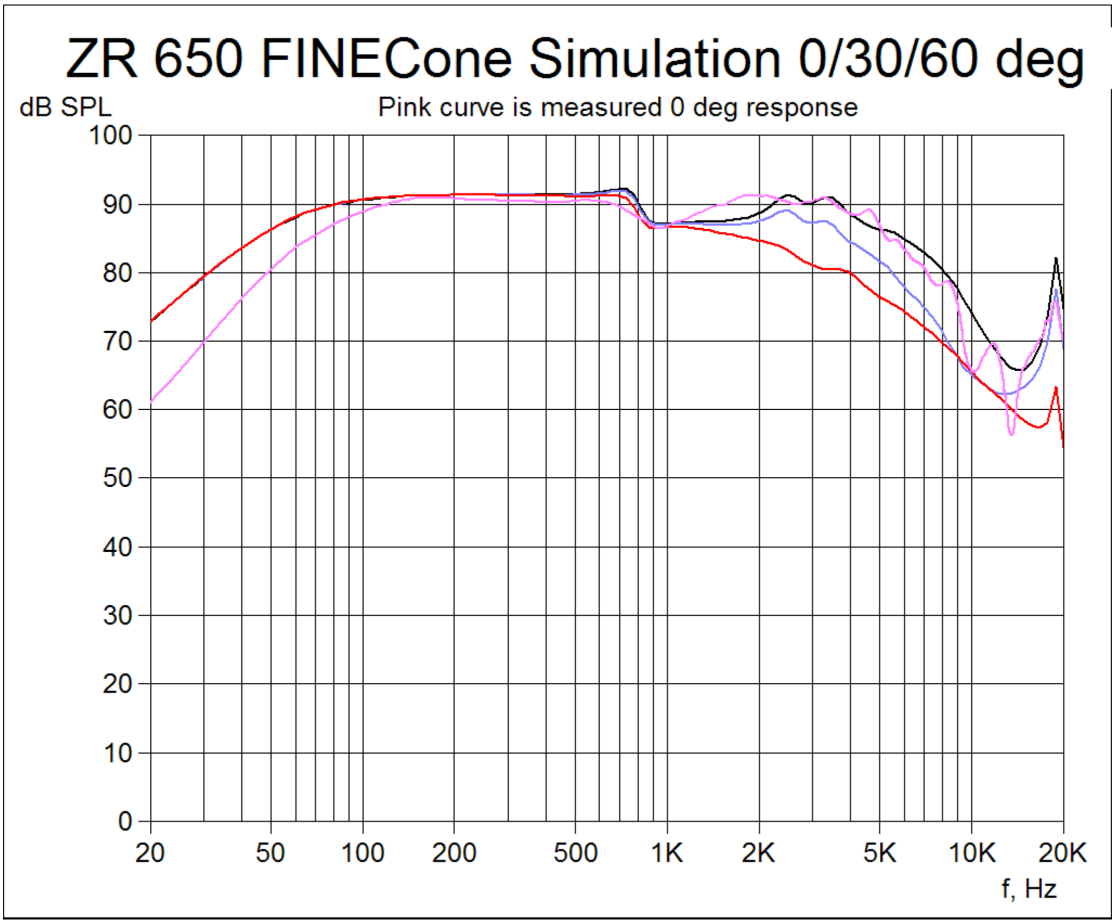


Figure 52 - The simulated response for on-axis, 30 degrees and 60 degrees are shown versus the measured response

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 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.

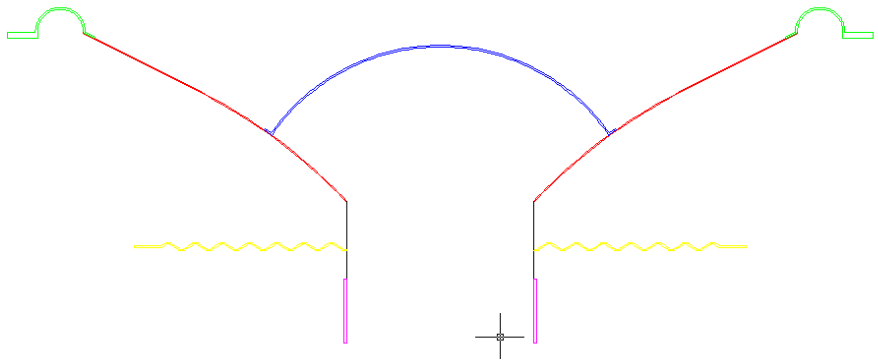


Figure 53 - 2D representation of the driver

Thiele-Small Parameters		
Parameter	**Value**	**Unit**
Fs	47	Hz
Re	5.50	Ω
Qms	4.93	-
Qes	0.49	-
Qts	0.44	-
Le1	0.2	mH
Le2	0.47	mH
Rp	4.91	Ω
Vas	23.93	Ltrs
Mms	12.64	g
Cms	907	m/N
Bl	5.65	Tm

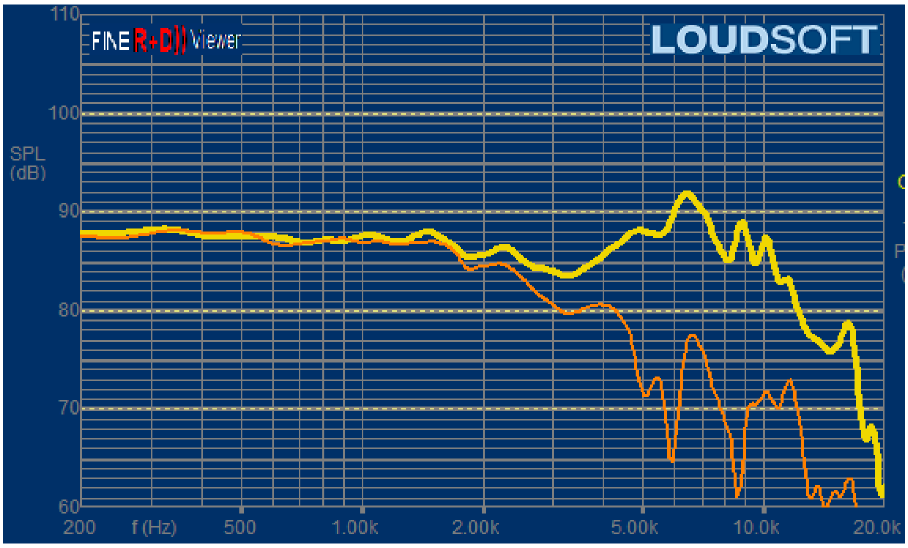


Figure 54 - The (0/30deg) response is measured using MLSSA. The on-axis curve is imported in FINECone

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 55.

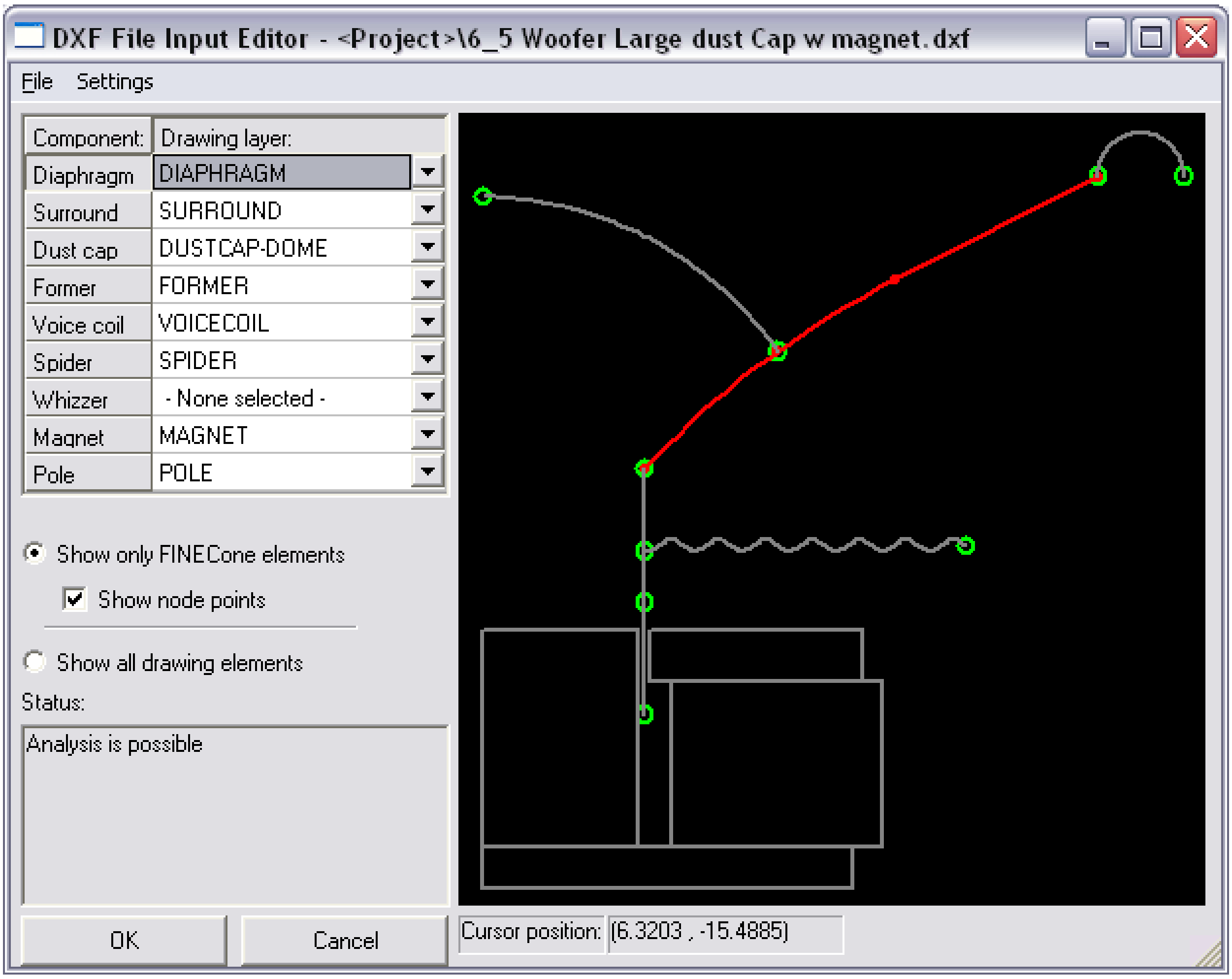


Figure 55 - DXF Import and Automatic Error Checking

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

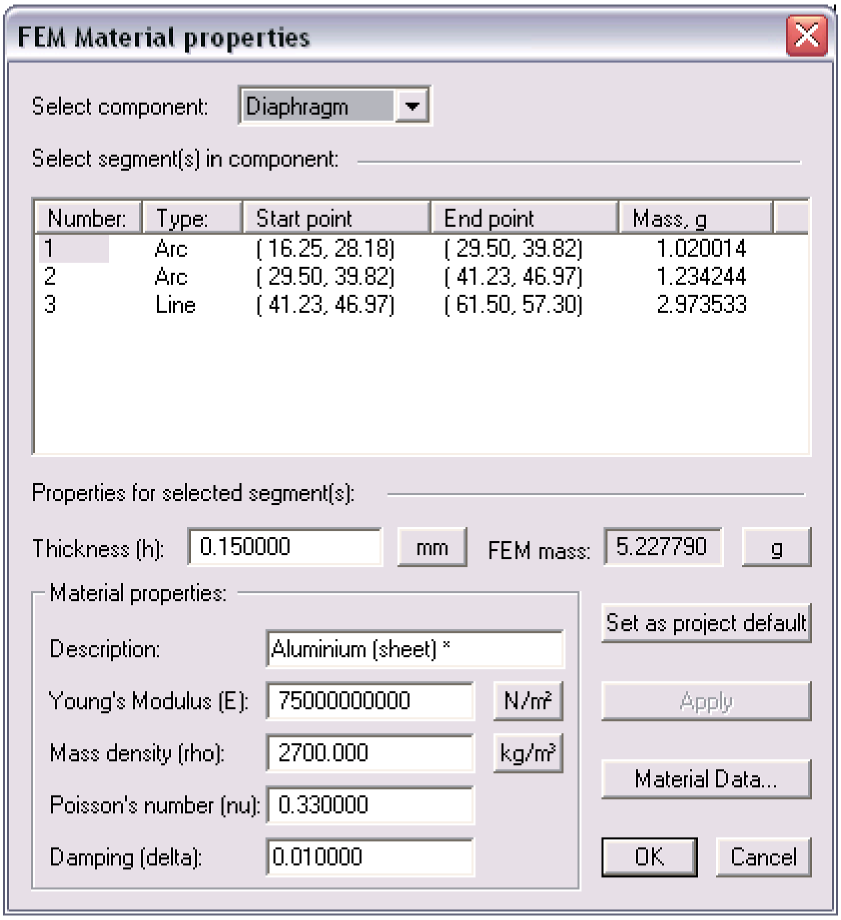


Figure 56 - The FEM Material properties window showing the diaphragm sections

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 55). 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 57, 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.

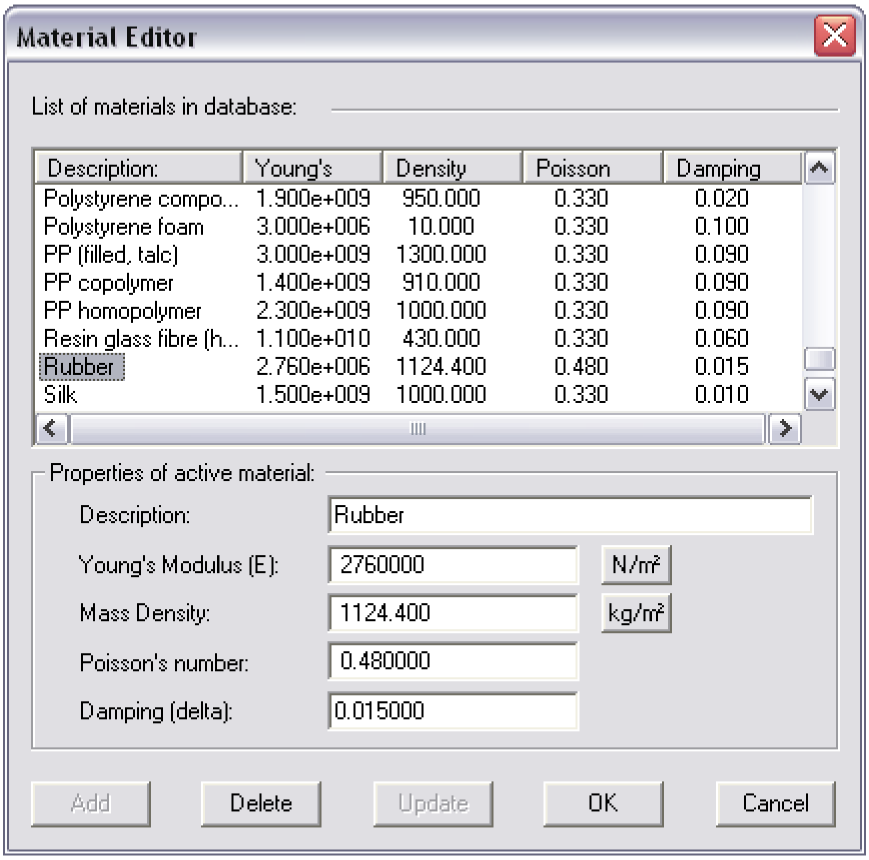


Figure 57 - Material editor window with a long list of pre-set materials

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”

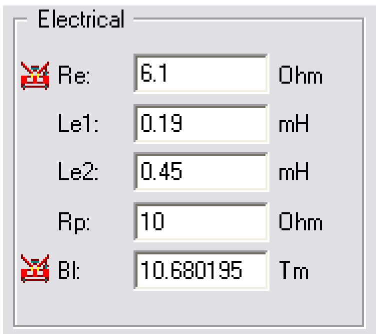


Figure 58 - Electrical Input with FINEMotor inputs

If you have a FINEMotor file (*.FM2) from the latest version it can be imported directly into FINECone. The parameters in Figure 58 marked with [icon] are imported From FINEMotor.

The resulting frequency response is shown in Figure 59.

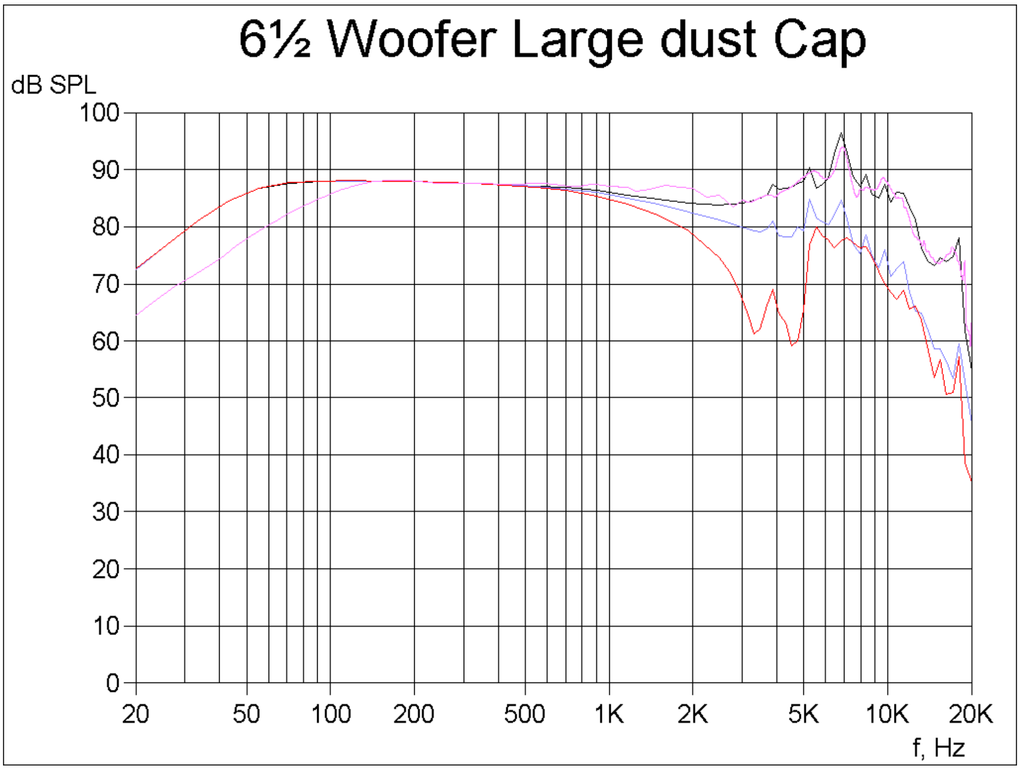


Figure 59 - The agreement between the calculated FINECone response (black) versus the measured response (magenta) is remarkable at high frequencies (break-up region)

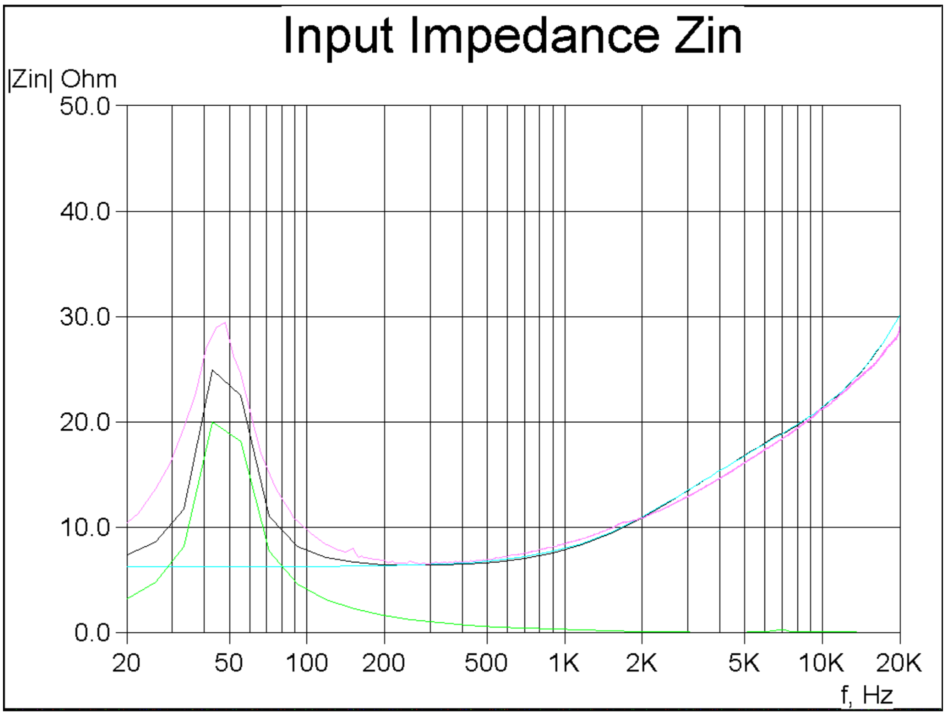


Figure 60 - The blue curve is calculated electrical impedance, the green curve is the mechanical impedance, and the black curve is the blue and green curves summed up to give the total impedance. The magenta curve is the imported impedance curve for comparison.

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 61.

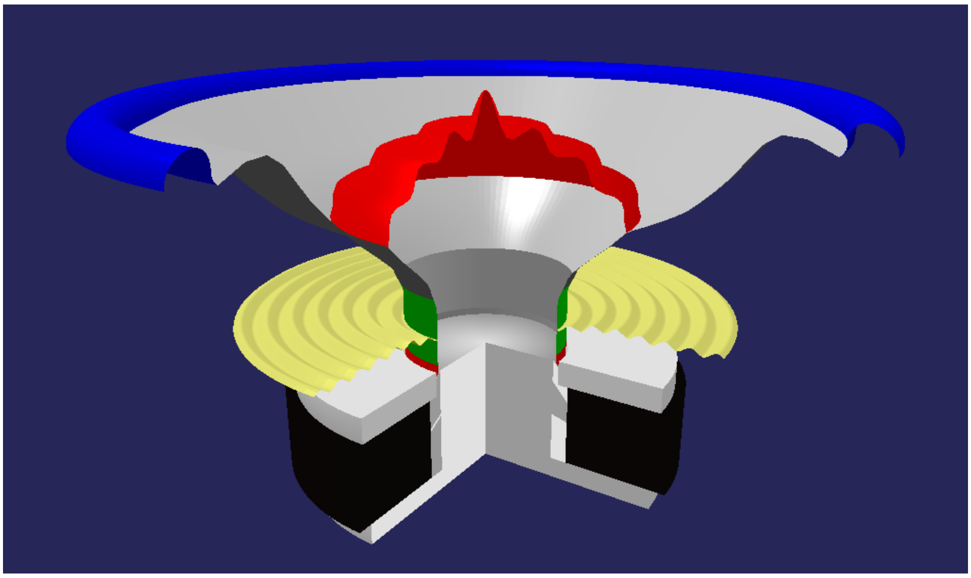


Figure 61 - This is the 6.5” woofer break-up animated at 7162Hz, which is the frequency of the large peak. Note the heavy break-up in the outer part of the cone.

The amplitude is set to 7mm as shown in Figure 62. Selecting the actual amplitude is not ideal for high frequencies as the excursion is very small.

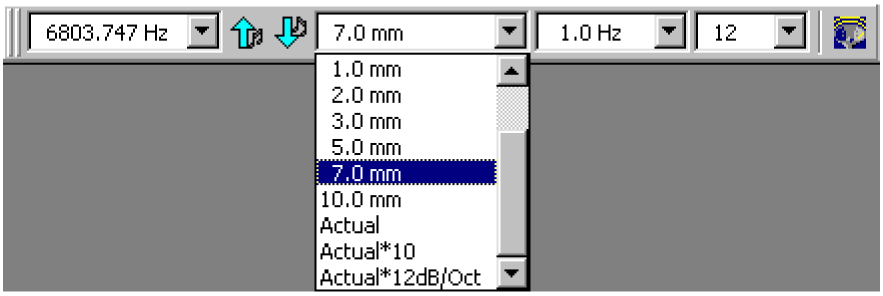


Figure 62 - The amplitude is set to 7.0 mm to better visualise the breakup

Excluding every part from the model except for the dust cap by using the buttons shown in Figure 63, we can now see that it is providing the main contribution to the peak around 7 kHz as shown in Figure 64.

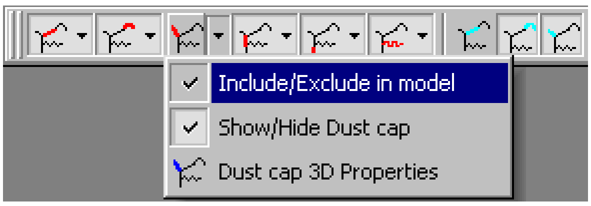


Figure 63 - Excluding all parts of the model except for the dust cap

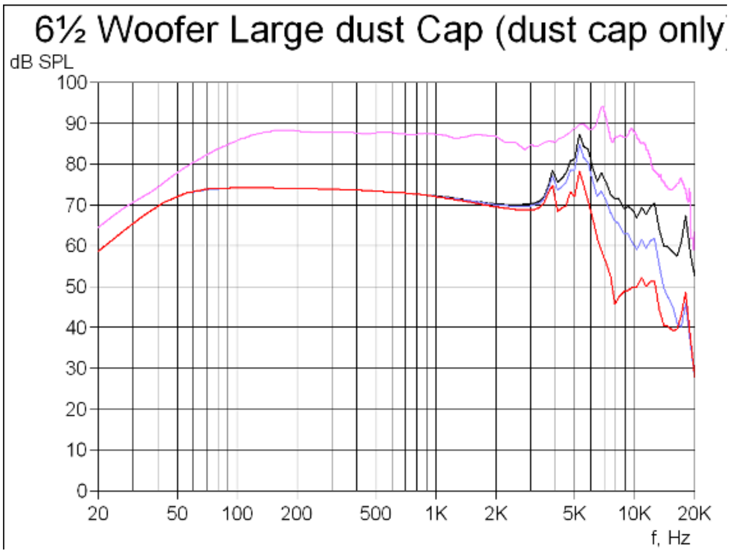


Figure 64 - Response of the large Dust Cap ONLY. The dust cap has a large peak around 5kHz.

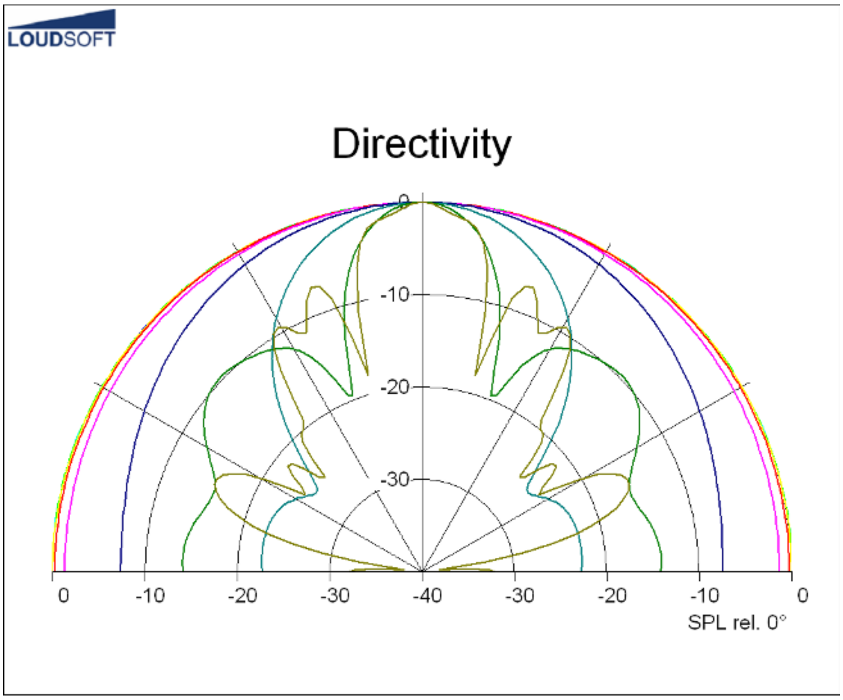


Figure 65 - Directivity at 10 frequencies

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.

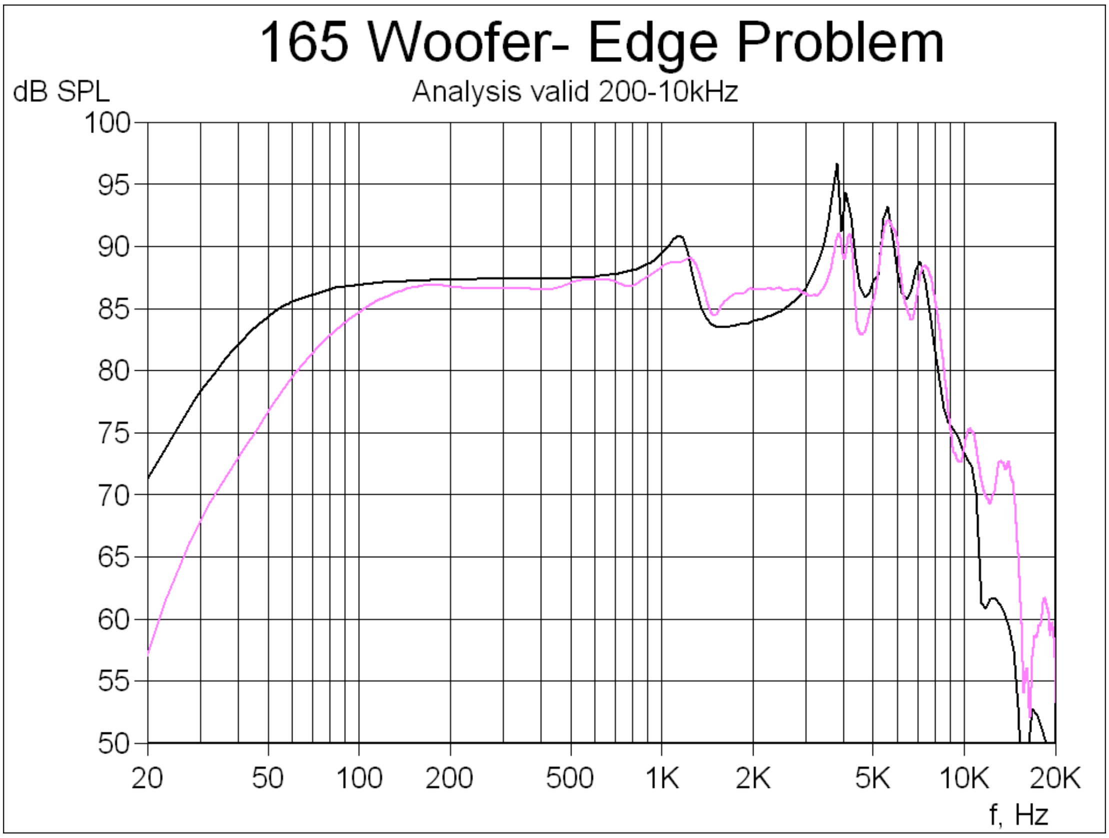


Figure 66 - Frequency response of the 165 mm woofer with an edge problem

The FINECone simulated response in Figure 66 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 67. 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 68.

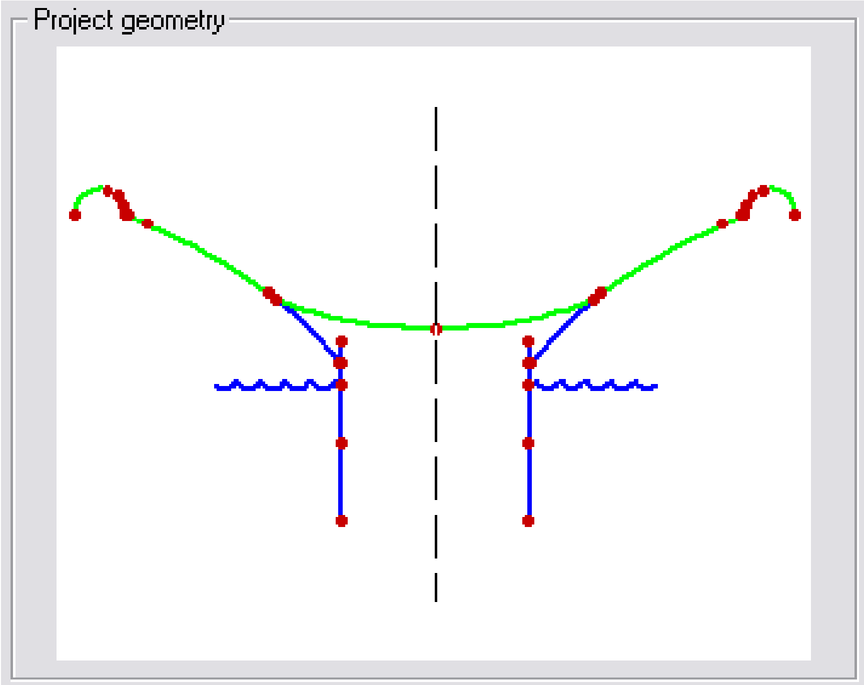


Figure 67 - Geometry for the 165 mm woofer with 5 segments in surround

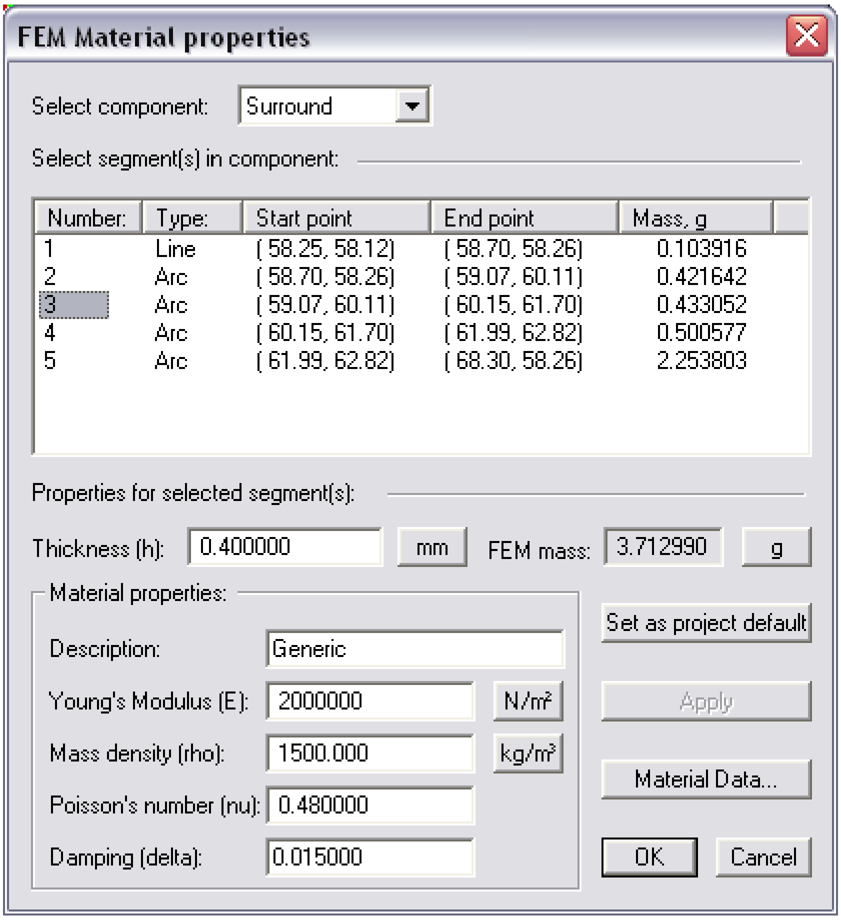


Figure 68 - Material properties for the surround

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 69 shows the cone edge and surround is moving excessively (brown curve).

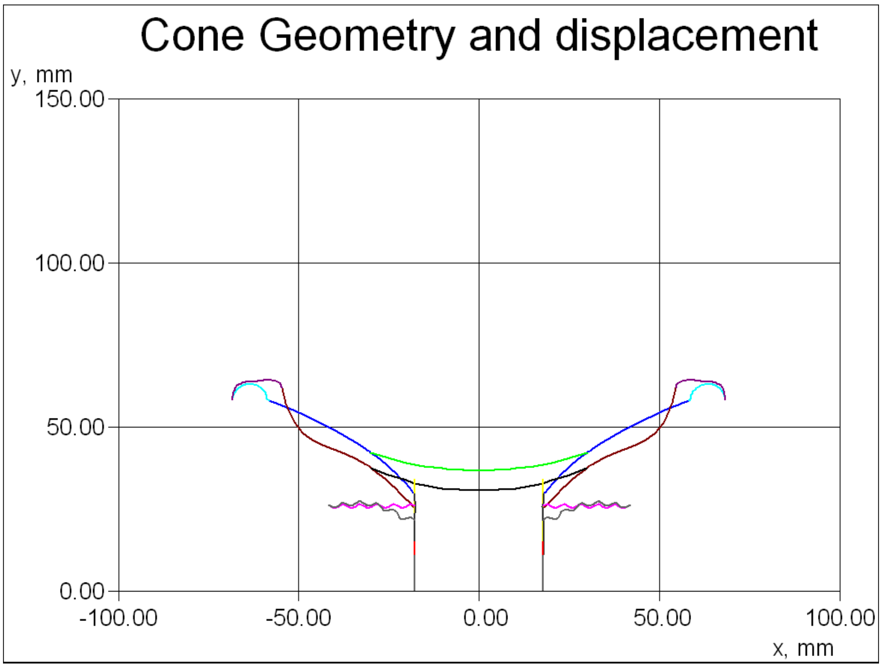


Figure 69 – 2D plot with displacement (brown), maximum at cone edge.

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 70, 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.

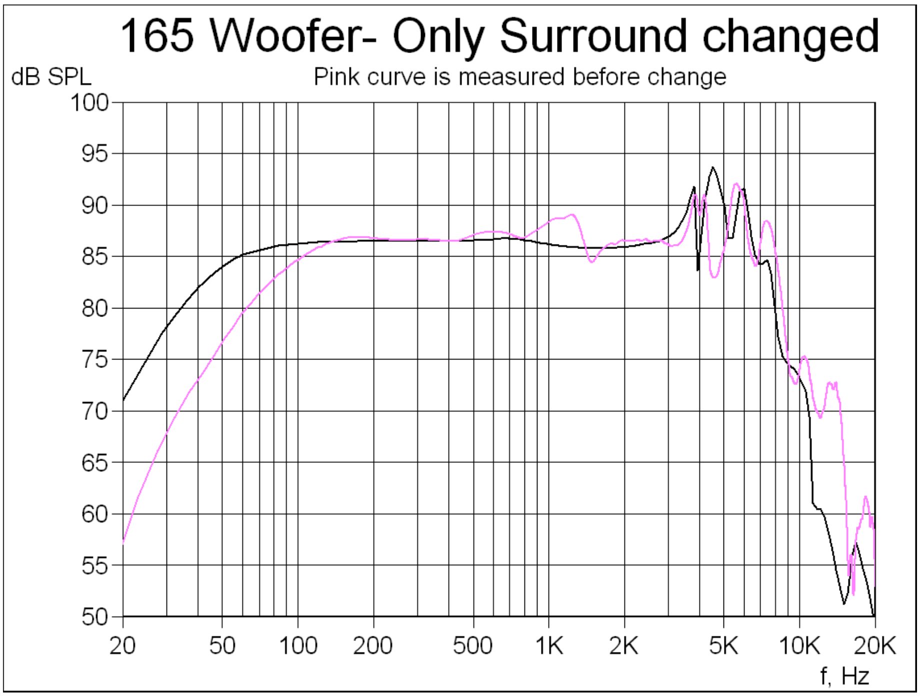


Figure 70 – 165 mm woofer with increased thickness of inner surround

Figure 71 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.

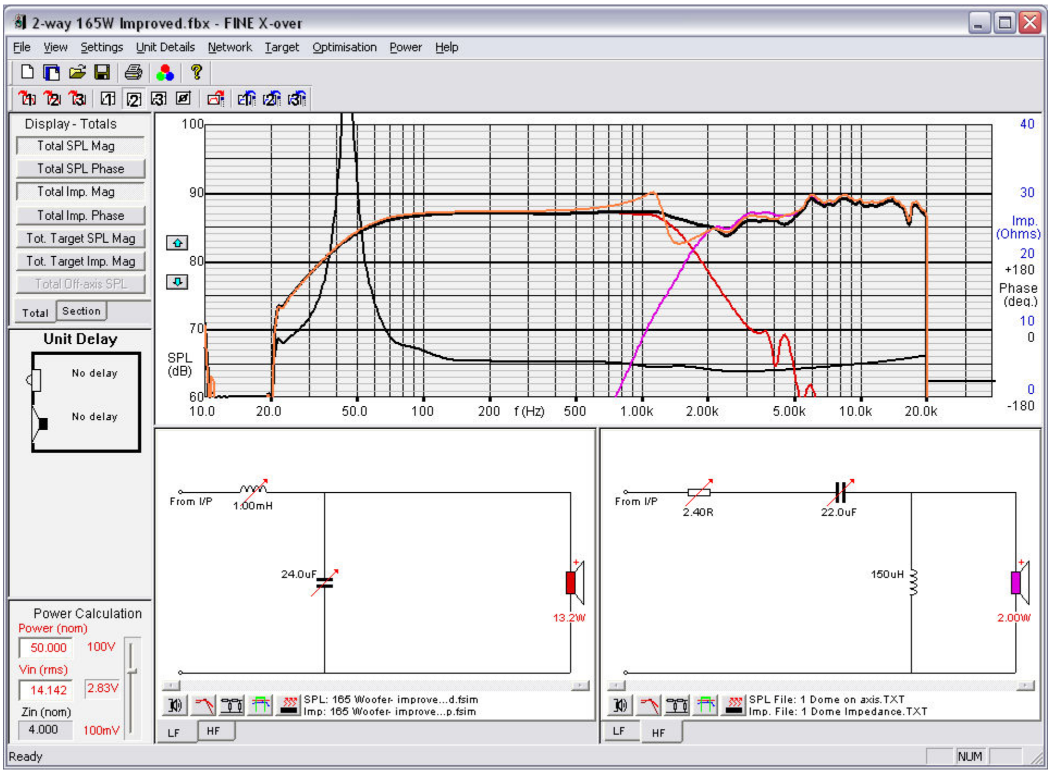


Figure 71 - FINE X-over using the frequency response and impedance exported from FINECone. Orange curve is with the bad woofer

2.2 38mm headphone transducer

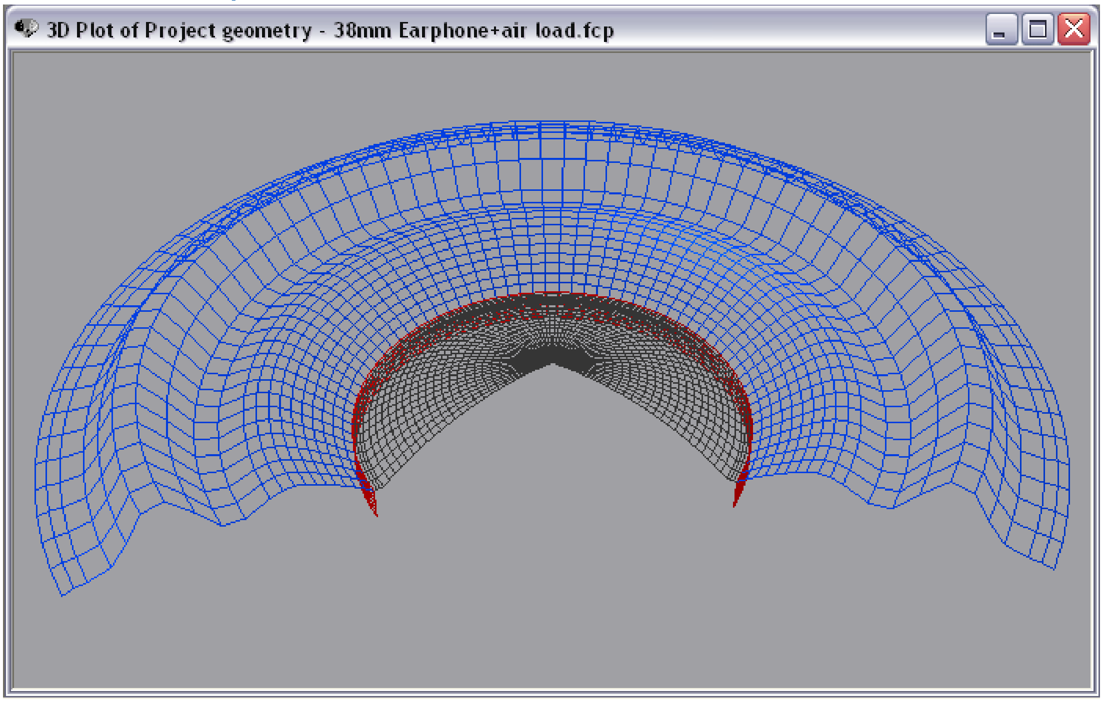


Figure 72 - 38mm Headphone transducer simulated in FINECone with break-up at 3165 Hz

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 73. There is serious break-up from approximately 3000 Hz and the first mode is shown as 3D animation in Figure 72. This first break-up mode is showing up in the middle of the outer diaphragm (surround) where it is almost flat.

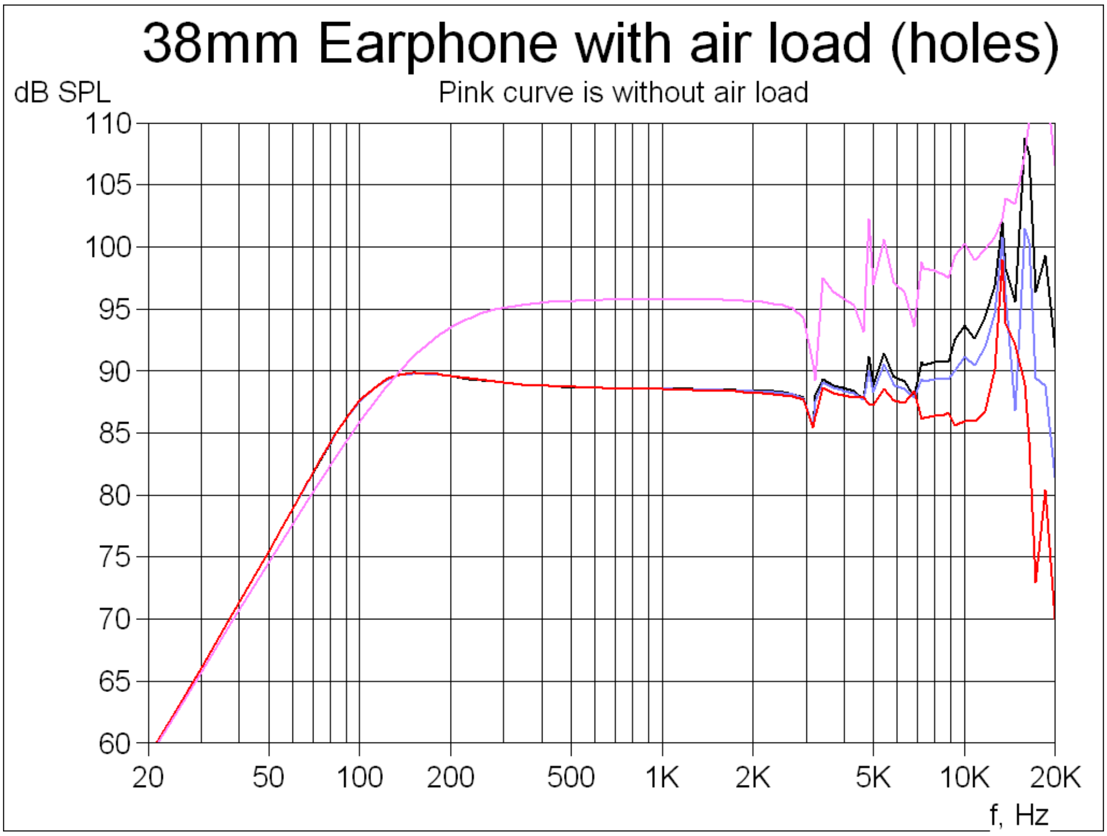


Figure 73 - A 38mm Headphone simulated frequency response with air load (rear holes)

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 Fig. 24 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

www.loudsoft.com