A short comparison between LOUDSOFT and COMSOL for transducer engineers

Nico Germanos

nico@fink-audio.com; nico@physical-lab.com

Abstract. This white paper provides a brief comparison between COMSOL and LOUDSOFT to help new engineers understand the differences between the programs. Both programs are suitable for loudspeaker development and, in addition to the workflow, highlights and limitations should also be considered. Nowadays, simulations in the loudspeaker sector are indispensable and the choice of software used should be carefully considered.

I. INTRODUCTION

Transducer design is a very special field of engineering. Over the last 20 years, a lot has changed in the approach to loudspeaker development. While it used to be unavoidable to create dozens of prototypes, modern simulation methods are now used to achieve remarkable results and to drastically reduce the number of prototypes. This ultimately saves a lot of time and money and provides insights into the behavior of the speakers that were previously unthinkable. In my many years of experience as a consultant, I had a lot of contact with various software, but two providers particularly stood out. Firstly COMSOL, formerly known as FemLab[1] and secondly LOUDSOFT [2] Software by Peter Larsen. In addition to COMSOL and LOUDSOFT, there are of course a lot of other providers of special software, which I don't want to go into in detail here because I specialize in these two software packages

II. SOFTWARE PACKAGES

COMSOL is an all-in-one solution that has a modular structure. In addition to the basic module, the acoustics, AC/DC and structural mechanics modules are also recommended or required for loudspeaker development. In addition, the optimization and CAD import modules or the CAD design module are also highly recommended. You can build your model in COMSOL or simply import it from your desired CAD software. I personally use Solidworks for highly complex geometry's and the COMSOL Design Module for simple structures although you can also build complex models in COMSOL directly. Each of the COMSOL modules mentioned above is based on its own physics and many modules can be physically coupled, making multiphysics simulations possible. In addition to the modules mentioned, COMSOL offers a large number of modules for all types of physics problems.

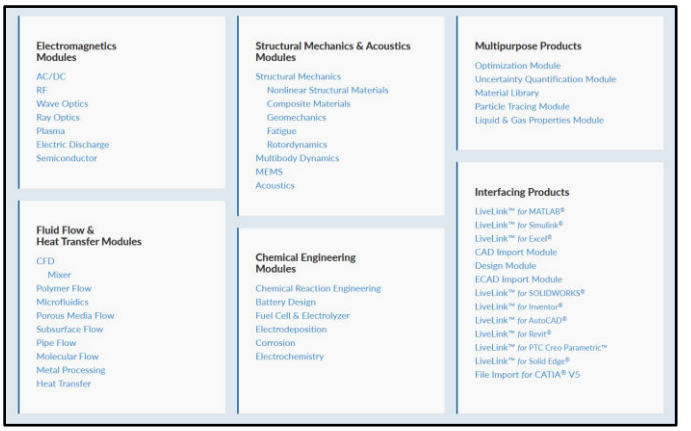

Figure 1: Overview of the available COMSOL modules [1]

LOUDSOFT, on the other hand, offers different software programs that specialize in different areas of transducer design. You can import and export results so that the workflow can be kept very straight forward. In the case you want to develop transducers the basic programs are:

- FINEBox PRO
- FINEMotor PRO
- FINECone
- FINESuspension

You can buy these software packages individually, so you can choose what fits best for your work. In addition to the software packages mentioned, there are a number of other software and hardware products available to cover the entire development and validation process. This is called the FINECircle.

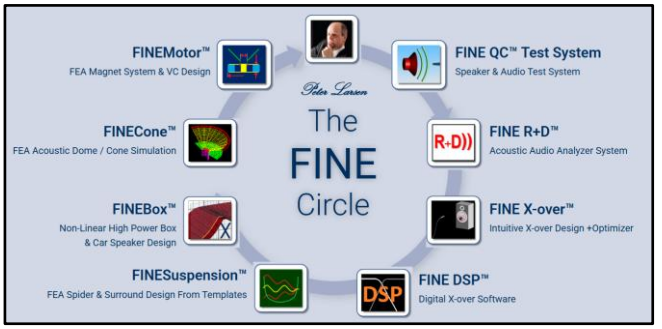

Figure 2: Overview of the available LOUDSOFT products [2]

FINEBox is an enclosure design utility with a lot of helpful features like inverse FINEBox that helps you to find specific target TSP parameters. FINEMotor, on the other hand, is a powerful software program for motor design. FINECone is a FEA software package where you can import your 2D rotationally symmetrical drawing and analyse the design using finite elements. There is also a modeler that helps you if you want to start with traditional speaker designs as you just need to set up the desired dimensions of your parts. The vibroacoustic model solves among other things for the acoustic SPL in different angles and distances. FINESuspension is a specific FEA software package for calculating the nonlinear behavior of spiders and surrounds.

III. WORKFLOW

There are fundamental differences in the way the programs work and how they are used. In the next chapters I will give a little insight into how both programs work.

A. LOUDSOFT

Before the actual work, the question often arises as to where to start. Regarding LOUDSOFT, it's your choice. If you have a specific enclosure design and you want to develop a fitting transducer, then FINEBox is the way to go. Of course, there are also dozens of different enclosure simulation packages on the market and a lot of them are free, but keep in mind that you should know how these programs work and what the limitations are. The good thing about LOUDSOFT is that you can easily export and import results from the other software packages that makes your life as an engineer much easier. Finding your specific parameters in FINEBox can be done in a few clicks. First you choose your type of enclosure. There are a lot of different types of systems available like closed, ported, bandpass and even more complex boxes.

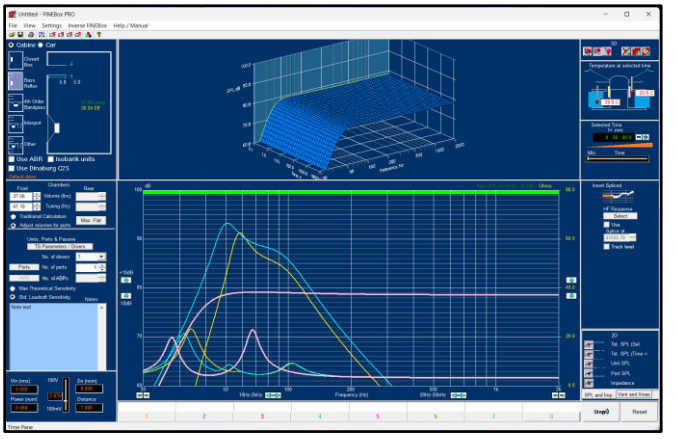

Figure 3: Start screen of FINEBox. On the left side you can choose from different kinds of enclosure designs.

Once you have chosen your enclosure design you can go ahead with the Inverse FINEBox button. After establishing the desired volume, tuning frequency and type of transducer you receive your final TSP for the transducer. This is a really helpful tool for quick estimation before going deep into the development.

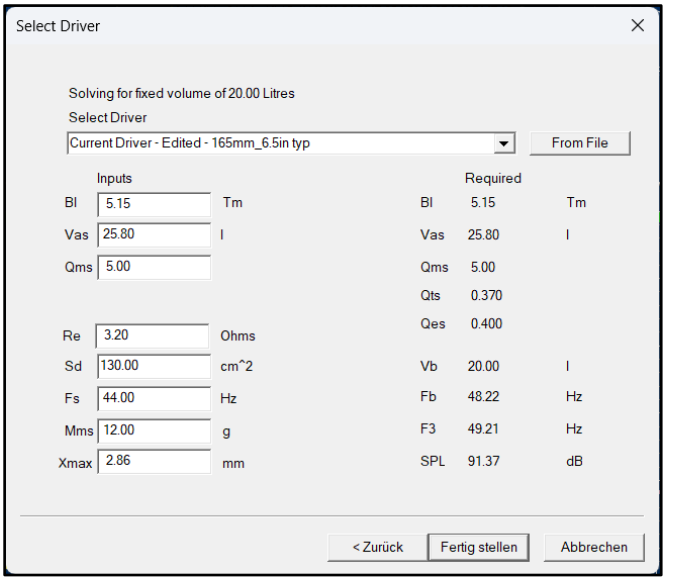

Figure 4: Inverse FINEBox

Now that you have target parameters you can continue with FINECone. FINECone is able to calculate the behavior of the moving parts using a simple, effective and very fast FEA algorithm. You can start with a predefined template where you can easily change design dimensions, or you can import a 2D CAD file. After this you define your materials and some parameters like Re, Le1, Le2, Rp and BL. Simply spoken FINECone calculates the vibroacoustic behavior in a defined distance and angle using a lumped element model that is coupled to a mechanical shell element model.

As already mentioned, this calculation is very fast and just takes some seconds (dependent of the resolution you choose). The good thing here is that you do not need to care about boundary conditions or the mesh of the model. Everything is done automatically to keep it as simple as possible. Of course, the model comes with limitations, but actual comparisons of a real transducer with the simulation show real good results.

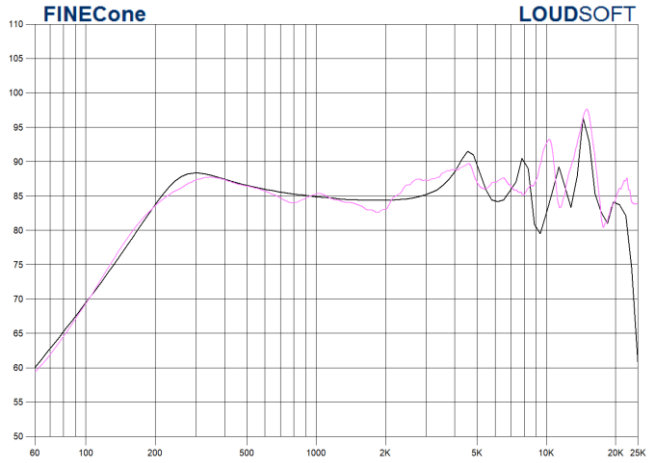

Figure 5: Comparison between a real fullrange driver and the FINECone simulation.

Please note that simulating moving parts with difficult materials like paper is quite challenging as the material properties can vary a lot. It is always helpful to adjust the simulation so that the parameters fit best after prototyping. A quite new feature of FINECone is the Geometry/ Shape Optimizer. Here you choose from different options to optimize the on-axis SPL:

- Optimize cone shape including dustcap
- Optimize cone shape excluding dustcap
- Optimize surround thickness including dustcap
- Optimize surround thickness excluding dustcap

There are three optimization strategies for the options mentioned above:

- Max deviation
- RMS Error
- Average Gradient

The optimizer is a nice and fast calculating feature that helps you find the best shape and thickness for your desired design in a predefined frequency range.

After you have found your initial design of the moving parts you can jump to FINEMotor. This program is made for quickly simulating motor designs for all kinds of transducers. Simply put, you model a motor design of your choice and then calculate the non-linear parameters such as the BL(x) curve and Le over x. FINEMotor is using FEMM [3], a free suite of programs for solving low frequency electromagnetic problems on two-dimensional planar and axisymmetric domains, to calculate the BL-curve over excursion of the problem.

FINEMotor then calculates the BLx factor and the Le over x curves. A nice feature is that you are able to change the coil length and the resulting BLx curve in real time. You also get the resulting TSP to see if you fit your design goals. This is unique and a really fast forward option for finding the best result.

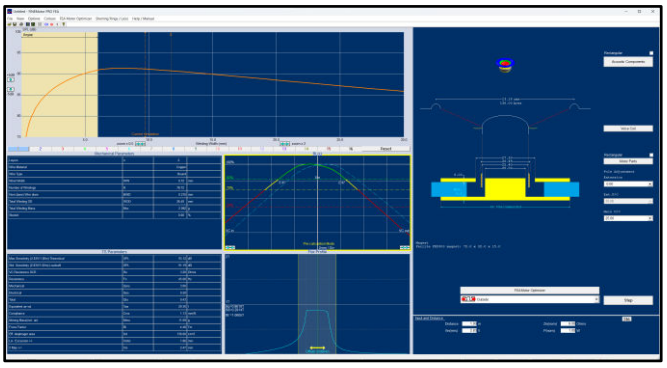

Figure 6: FINEMotor workspace

Of course, you can also add all kinds of different types of copper or aluminum rings into your design to decrease Le over x.

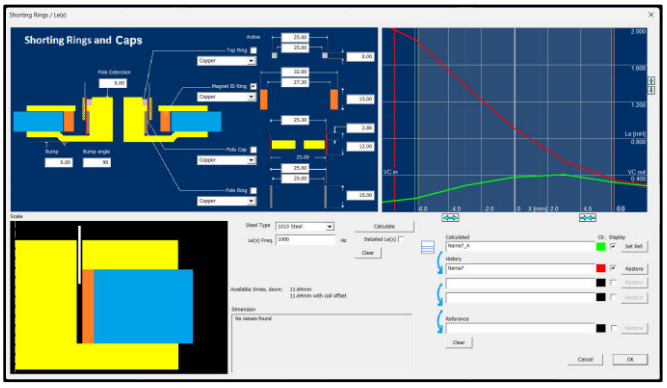

Figure 7: FINEMotor shorting rings menu

New is the FEA optimizer where you can choose from different options:

- Maximum B Field
- Flatness of the BL curve
- Symmetry of the BL curve

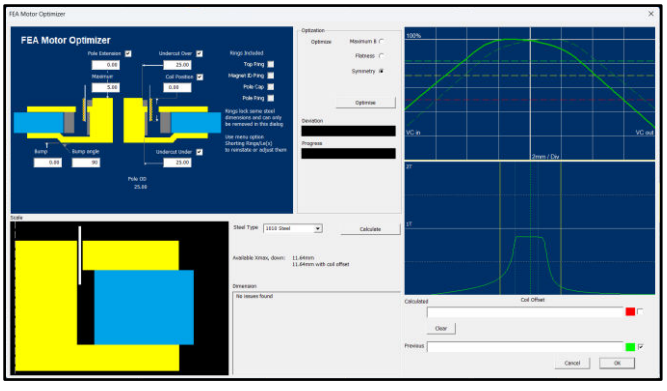

Figure 8: FINEMotor optimizer menu

If the parameter estimation or your targets between FINECone and FINEMotor do not fit, you can easily change them in both programs to get your final design. You can also import your FINEMotor results in FINECone to compare your simulations.

The last step you take is to develop a spider and surround design that matches your parameters in the models. This can be done in FINESuspension. This program features a nonlinear FEA analysis of spiders and surrounds. This feature makes the software extremely powerful and useful, especially for long throw woofers. By adjusting the spider geometry in a template, you can effectively increase the force of the spider over the excursion to prevent damage on the speaker. You can also import your results from the surround simulation to calculate the combined force over excursion. You are able to switch between the force, KMS or CMS results.

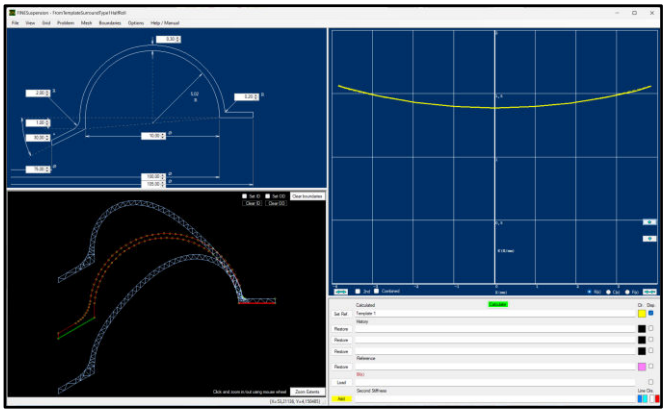

Figure 9: FINESuspension surround analysis

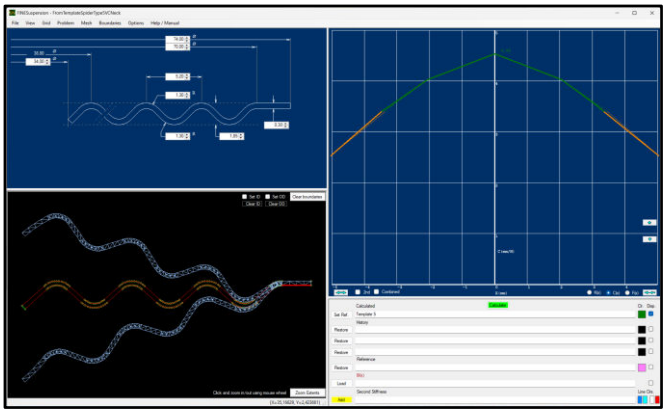

Figure 10: FINESuspension spider analysis

Of course, every software package comes with good documentation and a lot of examples to help you understand the workflow. Should problems still arise, Peter Larsen himself offers quick and ongoing support. This is not always a given with special software.

B. COMSOL

COMSOL is a modular software suite for engineers or scientists to help simulate and understand physical or multiphysical problems. In principle you buy the basic COMSOL software package and the additional physics module you need for your simulations [1]. Besides the physic modules a lot of additional modules are available as can be seen in figure (1).

The add-ons offer ranges from optimization, CAD design, Cad import or so-called live link modules for external programs such as Matlab, Excel, Simulink and various CAD programs. The live link products enable data exchange between the programs and make it possible, for example, to apply changes in 3D CAD programs directly to the COMSOL model. This is a really powerful feature.

A detailed description of the functions of COMSOL would go far beyond the scope in this context, because the possibilities are almost limitless. The extremely large number of examples in the so-called application library makes it much easier for users to understand the software. The downside of the open structure of the software is the resulting complexity.

In the case of loudspeaker development, in addition to the basic software, the acoustic module, the AC/DC module and the structural mechanics module are also required. With these modules it is possible to examine speakers using different approaches. From a complete FEM simulation to the investigation using lumped elements, everything is possible. And all of this in 2D (also rotationally symmetrical) as well as in 3D.

When you start COMSOL, you can choose between a blank model or the Model Wizard, which will help you choose the right physics for your simulation task. You can also choose the multiphysics couplings between the modules if wanted. I highly recommend building your model step by step and not starting with all modules with its physics at once to reduce errors in your model. You then decide how you want to simulate the loudspeaker. You don't need the structural mechanics part of your model right from the start. You can also start with a simple acoustic approach and then add the mechanical part. There are several approaches:

- Fully coupled FEM model with or without structural mechanics (2D or 3D)
- Lumped elements model (magnetic System with coil) with or without structural mechanical coupling (2D and 3D)
- Lumped elements model (magnetic System only) with or without structural mechanical coupling (2D and 3D)
- And many more…

All approaches can be examined either stationary, transient or time explicit.

The procedure is best viewed using examples. The model was simulated using an axisymmetric 2D approach [4]. In this model, a coupling was established between the AC/DC module (for the motor design and coil), the acoustic module (for the air section) and the structural mechanics module for the moving and vibrating parts.

The COMSOL user interface is divided into the following areas:

- Main Menu
- Model Builder
- Settings Window
- Graphics Window

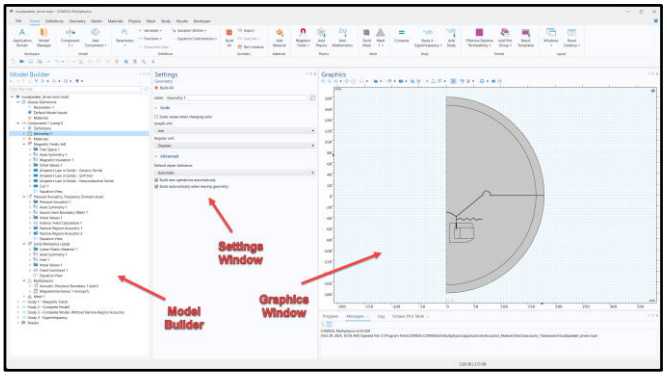

Figure 11: COMSOL Workspace

The model builder is a list of so-called nodes that you work on. The default nodes the components are:

- Definitions
- Geometry
- Materials
- Physics Interfaces
- Multiphysics
- Mesh

Here you usually work from top to bottom to define the model. The geometry node can also be used to import your CAD drawing directly into your component. COMSOL already provides a large number of predefined materials in the basic version that you can use for your material definition. After the model has been drawn and the materials have been assigned, you can work on physics. Here you have to define the model physically correctly. There are many boundary conditions and additional conditions available to define the model as precisely as possible. You have to work this out for each physical module. The modules can then be coupled via a multiphysics node.

In this example the following conditions were applied to the magnet fields node:

- Symmetry (as the model shows rotational symmetry)
- Amperes law in solids for the magnet
- Amperes law in solids for the iron
- Amperes law in solids for nonconductive solids for the former and the spider
- Coil node where you can define your coil

In the pressure acoustics node, you need to define the surrounding space. In this case its air. You also need to define some boundary conditions like walls. As you do not want the waves to be reflected from the surrounding air you need to define a so called PML (perfectly matched layer) in your definitions that is used to highly dampen incoming waves so that you virtually simulate a free space. Also, an exterior field calculation node was added to simulate the SPL in a defined distance and place. This is achieved by applying the source boundaries for the exterior field transformation set up by the Helmholtz–Kirchhoff integral. A very powerful feature for a transducer designer if you want to calculate your SPL at 1m, your polar pattern in every desired plane or even a balloon plot.

Summary:

- Symmetry (as the model shows rotational symmetry)
- Exterior field calculation for SPL@1m, polar patterns or balloon plots
- Narrow region acoustics 1&2 used for the gap between the magnet and the coil to calculate the thermal or viscous losses

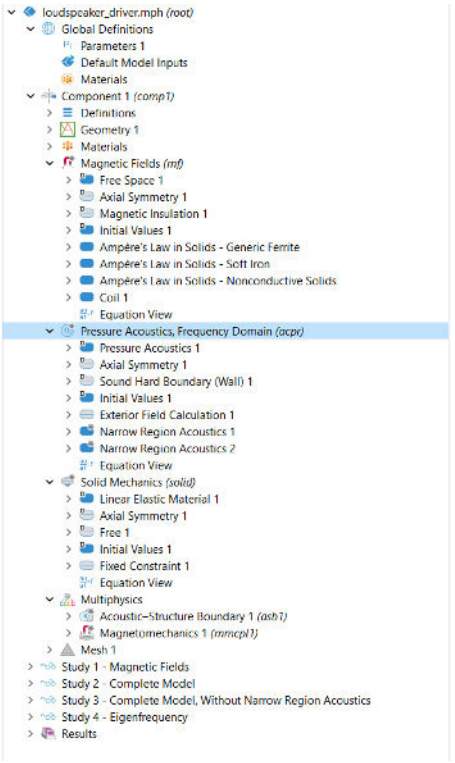

Figure 12: COMSOL Model Builder

The moving parts can be modeled in the structural mechanics node. Here you can define your type of material (linear, elastic, hyperelastic or nonlinear [and many more]). In this case a linear elastic approach was chosen for the membrane, the dust cap, the former, the coil, the surround and the spider. This works fine for low to medium excursions. In some cases, it might be helpful to describe your surround and your spider as a hyperelastic material.

After finishing your physical model, you need to build your mesh. The accuracy of the results depends very much on the quality of the mesh. While a mesh that is too coarse will produce inadequate results, a mesh that is too small can lead to a disproportionately long computing time.

COMSOL comes with an automated meshing functionality that is based on applied physics. This might work good for small models, but you definitely should learn to build up your own mesh to reduce computational time without losing information. When it comes to high fidelity models you definitely need to do the mesh by yourself.

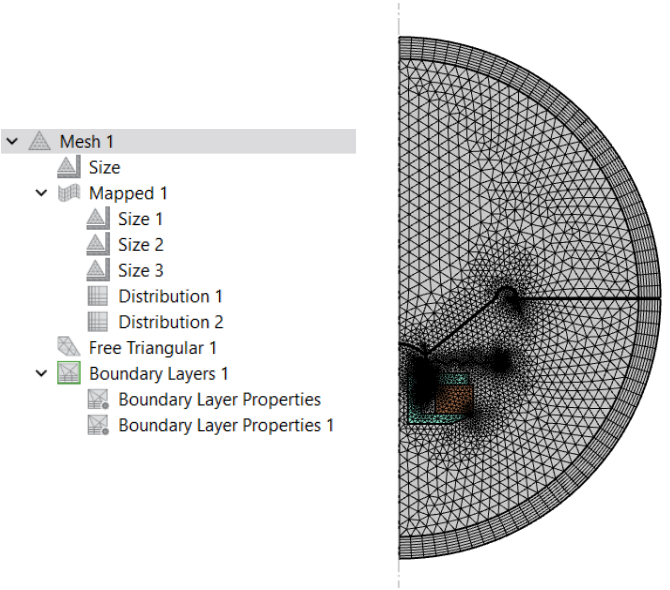

Figure 13: COMSOL Mesh example for the loudspeaker

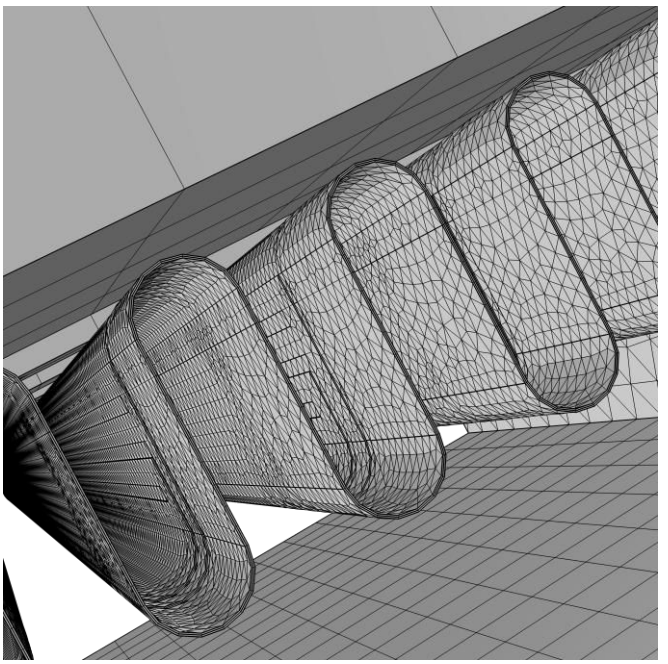

Figure 14: This examples shows a very detailed and complicated mesh of a circuit on a mylar foil in an air motion transformer

During calculation, you can have a look at the current log file and the progress. After your model is solved, you get some standard plots of the problem (you can choose if you want the standard plots to be computed).

The results node gives a huge number of possibilities to show your desired results or variables. The possibilities here seem to be infinite.

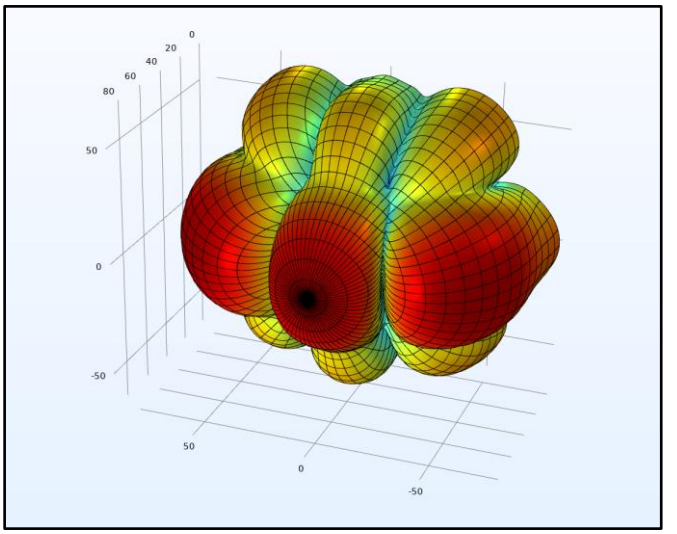

Figure 15: COMSOL balloonplot of a tweeter at 19kHz

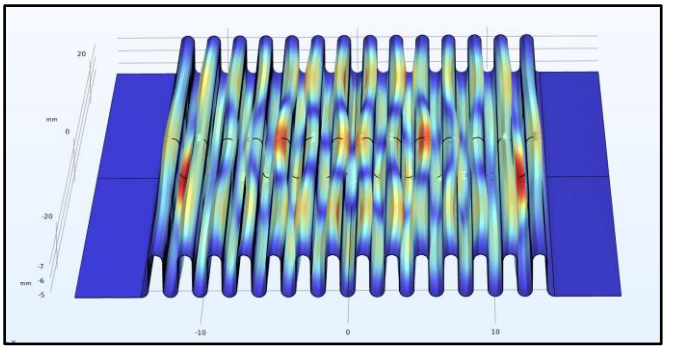

Figure 16: Displacement of an air motion transformer foil at 9kHz

Another approach for simulating loudspeakers is to use lumped elements. That means that you couple an electrical circuit node describing your system with your pressure acoustics node and your solid mechanics node. If needed, you can simply import your Spice model into COMSOL. This way you can choose if you want to lump the whole system or just parts of your loudspeaker. For example, you can lump the whole speaker or just the magnet system with or without coil. This might be a good technique if you have a specific magnet system you want to use or if you have a given magnet design with coil. You can choose what fits best.

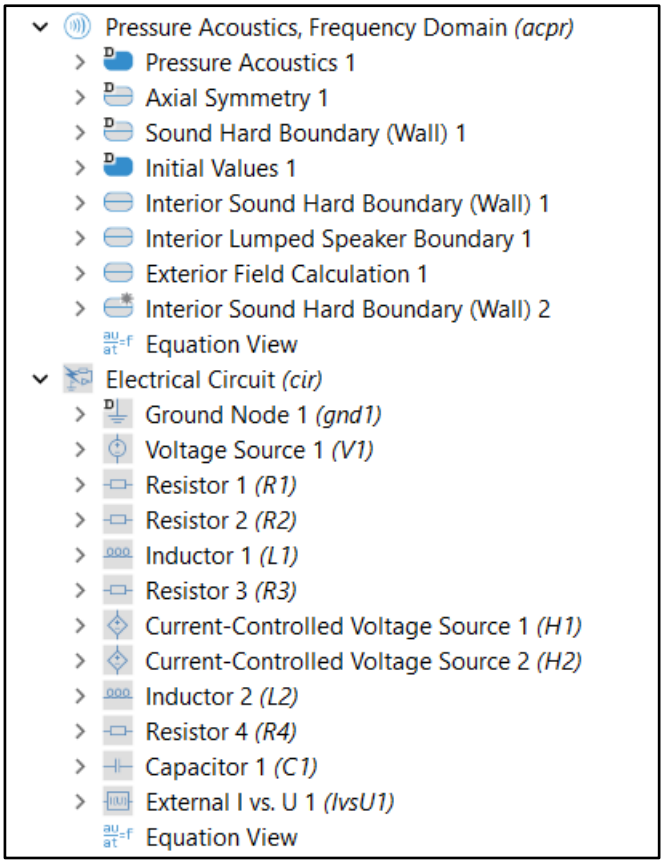

Figure 17: Example of a lumped model. It is a version of the so-called Leach model. Other models like a simple LR, the LR-2, the LR-3, or the so-called Wright model can be set up by modifying the electrical circuit [5].

IV. CONCLUSIONS

Comparing COMSOL with LOUDSOFT is not an easy task. On the one hand you have a huge modular software suite with COMSOL and on the other hand you have very specialized software packages for specific problems from LOUDSOFT. From the perspective of a speaker designer both have their advantages and disadvantages. You can solve a lot of problems with LOUDSOFT software although it comes with limitations. It's very easy to use and very fast. The support is fast and very friendly and helpful. If you use COMSOL it takes much more time, and you are able to simulate your loudspeaker with lots of details. You can build and describe your model in any way you want but you need a high level of understanding of what you do. Believe me when I say that you will spend a lot of time searching for failures in your model when you start working with COMSOL. The support from COMSOL is also very fast and helpful.

When I want to study a speaker design, I usually start with a very simple model in FINECone from LOUDSOFT to see what I can get with different designs. It is extremely fast, and the optimizer helps you find a good starting point for ongoing investigations. You can also see your TSP and other parameter changes in real time. If I find a satisfying starting point for the project that will meet the customers' demands I proceed with more details. Then I proceed with the motor design. This works extremely fast in FINEMotor. Especially with the optimizer and the option to add different kinds of copper rings to your design.

Afterwards I go on with COMSOL if needed. This way you save a lot of time for your COMSOL model when starting with a really good shape of the cone, dust cap and surround. Of course, you can also start directly in COMSOL, but as mentioned normally customers want fast results or suggestions so this is a good way to save some time. For difficult or special transducers, I start directly in COMSOL. Here you can start with a simple model and expand it piece by piece. This avoids failures from my point of view. You are also able to expand your speaker model with a waveguide or horn or any type of enclosure you want. If you want to optimize the shape or topology of your design, you can use the very powerful optimization module. However, using it is quite complex and time-consuming to take advantage of all the possibilities. Even if you miss some physics you can build for example your own boundary conditions in the physics builder.

So, if you are designing standard drivers and nothing special the LOUDSOFT products will make you very happy. If you want to simulate transducers with enclosures or waveguides or very special drivers, you should go for COMSOL.

From my point of view, I highly recommend to use both! And if you want to learn COMSOL properly, then I strongly recommend attending the Online COMSOL Master Course at the Multiphysics Modelling School of the University of Malaga [6].

Finally, I would like to point out that Comsol and Loudsoft are not the only tools offered for loudspeaker development. In addition to these, there are also tools from Ansys, Siemens NX, SpeakerLab, Akabak, SimScale and many more...

Comsol and Loudsoft are programs I use almost daily, therefore this paper is based exclusively on them.

REFERENCES

[1] The COMSOL® Software Product Suite. (n.d.). COMSOL. https://www.comsol.com/products (n.d.).

[2] LOUDSOFT. (n.d.-b). Loudspeaker design Software. https://LOUDSOFT.com/

[3] Home Page: Finite Element Method Magnetics. (n.d.). https://www.femm.info/wiki/HomePage

[4] Loudspeaker Driver — Frequency-Domain analysis. (n.d.). COMSOL. https://www.COMSOL.com/model/loudspeaker-driver-frequency-domain-analysis-1369

[5] Lumped Loudspeaker Driver. (n.d.). COMSOL. https://www.COMSOL.com/model/lumped-loudspeaker-driver-12295

[6] Multiphysics Modeling School. (2025, October 2). Multiphysics Modeling School: Consulting, Projects and Training. https://www.multiphysics.uma.es