COMSOL CONFERENCE 2019 CAMBRIDGE

FEM Simulations in Loudspeakers design: focus on mechanical and Acoustical-Structural analysis

G. Spatafora, E. Capucci, C. Corsini, R. Toppi
Faital SPA, Via Bruno Buozzi, 12, 20097 San Donato Milanese MI - ITALY

Introduction

Finite Element simulations are becoming an increasingly powerful tool in the transducer design process. The development of user-friendly software and predefined interfaces able to mimic physics setup to predict the loudspeaker complex behaviour helps designers in product development and improvement.
During product development, designers have to take into account several aspects at the same time, such as the kind of electro-acoustic device (i.e. cone loudspeakers, compression driver, etc.), the frequency range and the industrial application/area of applicability.
This work is focused on the mechanical analysis of both non-moving (test case 1) and moving parts (test case 2) and the acoustic-solid interaction problem (test case 3). FAITAL team develops simulations for several areas (i.e. magnetic circuit analysis [1] and thermal analysis [2]), but we will not face these topics in the current study.

Mechanical behaviour of non-moving parts

Test case 1: basket flexural load

Both in the automotive and professional audio industry, customers provide technical specifications related to the application and typical usage needed to develop our loudspeakers. Eigenfrequency and quasi-static analysis, resistance to vibration and shock, fatigue and Power Spectral Density analyses are usually required.
If needed, non-linear effects in material behaviour or pre-stress conditions, due to the counterpart mounting configurations, have to be considered. These tests are usually simulated using a parametric geometry to highlight its influence on the mechanical response.
The validation phase of the product development process consists of several tests. We develop virtual models of these tests to avoid undesired failures. In this work, we focused our attention on the application of a flexural/torsional load to a loudspeaker basket. The main aim was to ensure basket stability (i.e. to maintain the centring between cone/spider and coil) avoiding undesired failure.
Figure 1 resumes the technical requirements received from the customer. Figure 2 shows the developed model with the boundary conditions applied to solve the Solid Mechanics equations.

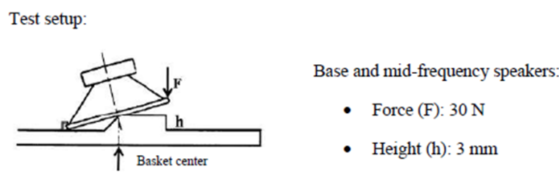

Figure 1 *Test setup required by the customer*

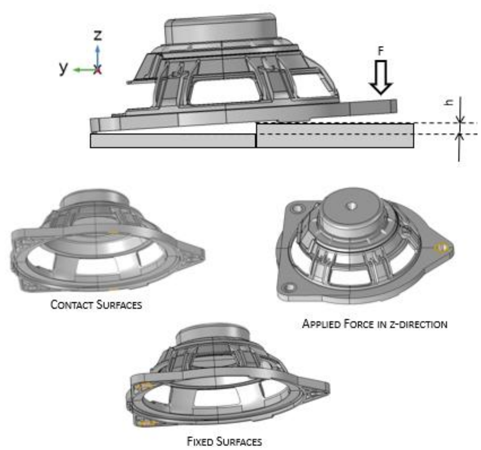

Figure 2 *Comsol Multiphysics model: boundary conditions*

Safety attribute was added to check the material failure. The Safety attribute to the *Linear Elastic Material* (our case) and *Nonlinear Elastic Material* nodes enables one to study the use of materials in the structure, particularly concerning safety factors. Many different failure criteria permit to evaluate the safety factors, including user-defined expressions. Thanks to the Safety node, the user can access post-processing variables for the safety factor, margin of safety, damage and failure indices. The calculated damage index was a simple check variable to identify a problematic spot and to check how slight geometrical modifications may change the mechanical response (Figure 3). For this simple model, a validation setup test was developed to check our chassis.

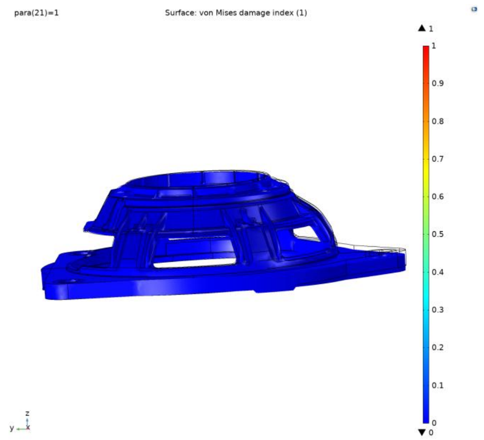

Figure 3 *Von Mises Damage index in deformed configuration, the solid black line refers to undeformed geometry.*

Mechanical behaviour of moving parts

Test case 2: mechanical evaluation of a compression driver membrane

A compression driver is a small specialised diaphragm loudspeaker. It is connected to an acoustic horn, a duct which radiates the sound into the air. It works in a "compression" mode: the area of the loudspeaker diaphragm is significantly larger than the throat aperture of the horn so that it provides high sound pressures [3].
Compression driver membranes can be made using metallic or polymeric materials, and they can have a dome or annular shape. Technical requirements to define the membrane geometry are related to the first resonance in free air and to have few modes in the desired range. Overall dimensions related to the magnetic circuit and driver dimensions are defined in a previous design step. The main goal of Test Case 2 was the development of a dome membrane; several geometric parameters (Figure 4) are involved in mechanical behaviour evaluation.

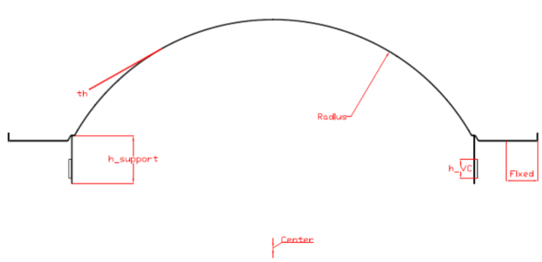

Figure 4 *Geometric parameters: dome*

The developed model solved Solid Mechanics equations in the frequency domain. We developed a parametric model, using geometrical settings highlighted in Figure 4. Then, eigenfrequency analysis is solved for each geometry, and a Model Method evaluates the mechanical results. A Method allows one to extend the functionality of the software by writing code inside the COMSOL Multiphysics graphical user interface (GUI). The developed method permits to create a rank based on all the simulated membrane according to the technical requirements. The final table can be checked by the users that can select the geometry for 3D analysis and rocking modes evaluation.

The eigenfrequency analysis of compression driver membranes together with the development of the magnetic circuit is only the first part of a more complex design phase. Membrane mechanical response is used to develop the compression chamber and the phase plug. The design of the phase plug is the most complicated step. Works by Dodd, Oclee-Brown and Bezzola ([4,],[5], [6], [7] and [8]) covered the main steps needed during the design process.

Acoustic Structure interaction

Test case 3: 6'' woofer product development

Acoustic structure interaction models are developed to evaluate the acoustic performance of virtual prototypes. Comsol Multiphysics provides the *Acoustic-Solid Interface* to reproduce the reciprocal influence between the movement of the diaphragm and the acoustic pressure of the air surrounding the speaker.
Typical output variables are impedance, sound pressure level evaluated in axis at 1m and the near field.
Model complexity is strictly related to three factors:

1) *Physical size vs wavelength*

Our domain of simulation can be a single driver, a driver in a cabinet/box or the acoustic evaluation of an entire room or cockpit. The mathematical approach can vary, according to the model size, from pressure acoustic equations to Geometrical Acoustics approaches (i.e. theatres acoustic analysis).

2) *Study type*

The typical analyses developed to check the loudspeaker behaviour are in the frequency domain. Nevertheless, the evaluation of some critical parameters (i.e. Total Harmonic Distortion) requires time-domain analysis. Moreover, the rigours mechanical evaluation of moving parts needs to consider several nonlinearities (i.e. geometrical nonlinearities, material nonlinearities).

3) *Physics*

Lumped-element models have long been used to evaluate the fundamental vibration and radiation characteristics of moving-coil loudspeakers. The traditional low-frequency (i.e. Thiele-Small) models combine and simplify several driver elements. They can predict only a single mechanical resonance wherein the diaphragm (e.g., cone and dust cap) and the inner portion of the surround move as a piston.
The development of lumped models valid in large signal is a crucial topic in research [9]. Several models take into account thermal effects, and the displacement varying

parameters have been introduced in the equivalent circuit to describe time-variant and nonlinear mechanisms. The resulting models lead to nonlinear differential equations which have been solved by using several methods [9]. *Electric Circuit Interface* (AC/DC module), in COMSOL Multiphysics, can solve the lumped-element models (an example here: [https://www.comsol.it/model/ lumped-loudspeaker-driver-using-a-lumped-mechanical-system-55901](https://www.comsol.it/model/lumped-loudspeaker-driver-using-a-lumped-mechanical-system-55901)).

The Acoustic Module provides several interfaces; in loudspeakers analysis, *Pressure Acoustic Interfaces* (frequency and time domains) and the *Thermoviscous Interface* are used for an accurate analysis of acoustic propagation in geometries with small dimensions.

The user may choose levels of details in the model development:

- Lumped model: The most straightforward way to check loudspeakers behaviour is to implement the acoustic analogous for the magnetic circuit, mechanical contribution and acoustical response by using the *Electric Circuit Interface* (AC/DC Module).
- Acoustic radiation: magnetic circuit and mechanical contribution are lumped (*Electric Circuit Interface*), whereas pressure acoustics (*Pressure Acoustic Interface*) is solved using FEM. In this way, the acoustic variables consider only the shape of moving parts.
- Vibro-acoustic model: the magnetic circuit is modelled using a lumped circuit (*Electric Circuit Interface*); mechanical and pressure acoustics are solved using FEM.
- Electro Vibro-acoustic model: FEM is used to model all contributions.

The proposed test case is a vibroacoustic model of a single driver in the frequency domain. A prototype of 6'' driver for professional market showed an undesirable dip in the SPL. Hence, we started developing a vibroacoustic model to evaluate the mechanical response of the moving parts. In this specific case, we could work on geometrical variations on the cone and surround without introducing changes in main diameters and quotes related to the magnetic circuit. The developed model involved the attenuation of sound waves due to thermal and viscous losses near the walls. In loudspeakers, these losses are principally related to the air gap where the coil moves. COMSOL Multiphysics provides the *Thermoviscous Acoustic Interface* to take such losses into account. Specific multiphysics nodes were added to efficiently couple the *Thermoviscous Acoustic Interface* with the *Pressure Acoustics* and *Solid Mechanics* ones. The thermoviscous approach is computationally expensive; therefore, it is usually convenient to apply it only to the system components where this type of physics is relevant. These simulations can then be combined with simulations based on simpler physics that describe the rest of the system [10]. Figure 5 shows the complete tree developed for the model.

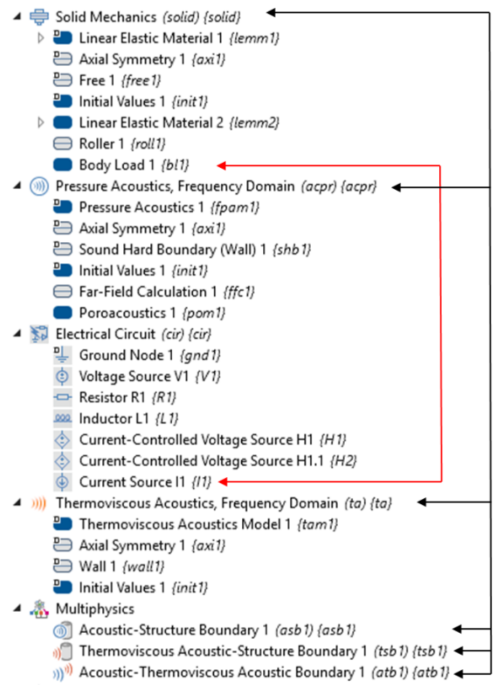

***Figure 5** COMSOL Multiphysics Tree*

The final developed geometry allowed to improve the SPL response, obtaining the new measure showed in Figure 6.

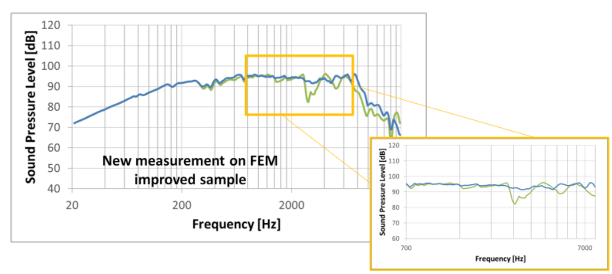

***Figure 6** Comparison between the two prototypes before (green) and after (blue) improvements introduced thanks to the FEM model.*

The final output was considered acceptable, and the new prototypes went through the next steps of the design process and validation.

Conclusions

The main aim of this work is to show how the development of FEM models may help during the design process of loudspeakers. In particular, COMSOL Multiphysics provides several tools to analyse the driver and its components in several areas of interest. Level of details can be chosen by the designers to accomplish their goals according to the needs in terms of results, effort and time.

References

[1] Baratelli M et al., "A finite element approach to predicting the effect of the demodulation ring in a loudspeaker," in CAE Conference, 2017.
[2] Baratelli M et al., "FEM Thermal Model of a Compression Driver: Comparison with Experimental Results," in AES, 2018.
[3] https://en.wikipedia.org/wiki/Compression_driver
[4] Oclee-Brown J “Loudspeaker compression-driver phase-plug design” University of Southampton, Faculty of Engineering and the Environment, Doctoral Thesis, 393pp, 2012.
[5] Oclee-Brown J et al. “Wideband Compression Driver Design, Part 1: A Theoretical Approach to Designing Compression Drivers with Non-Rigid Diaphragms” 139 AES 2015.
[6] Dodd M et al. “Wideband Compression Driver Design. Part 2, Application to a High Power Compression Driver with a Novel Diaphragm Geometry” 139 AES 2015
[7] Bezzola A “Audio Product Development at Samsung Audio Lab Better and Faster With Simulations” COMSOL Conference 2018.
[8] Bezzola A “Numerical Optimization Strategies for Acoustic Elements in Loudspeaker Design” 145 AES 2018.
[9] Klippel W “Measurement of Large-Signal Parameters of Electrodynamic Transducer” 107 AES 1999.
[10]https://www.comsol.it/blogs/how-to-model-thermoviscous-acoustics-in-comsol-multiphysics/