Interactive Design of an Electrostatic Headphone Speaker Using COMSOL Server™

B. A. Marmo[1], M. P. Snaith[1]

[1]Xi Engineering Consultants, Edinburgh, United Kingdom

Abstract

Interactive Design of an Electrostatic Headphone Speaker Using COMSOL Server™
An electrostatic headphone includes many interrelated design elements that affect the frequency response of the headphone and the users listening experience. Electrostatic speakers work by driving a changed elastic membrane between two parallel conductive wire meshes to which an AC signal is applied. Design elements such as the material properties of the membrane, speaker size, AC and DC signal levels, the initial tension in the membrane and open area of the conductive wire meshes all affect the sound characteristics of the speaker. A fully-coupled MEMS-acoustic interaction model that included a graphic user interface was developed that allowed our client to optimize their speaker using virtual tools and thereby reduce expensive and time consuming prototyping.
The dynamics of the electrostatic speaker is a true multiphysics phenomenon involving the non-linear relationship of structural mechanics, electrostatics, electrical conduction and acoustics. The dynamics are further complicated by the strongly non-linear electrostatic driving force that decreases with the square of the distance between charged surfaces. The vibrating membrane of the electrostatic speaker, the non-conductive spacer plates that separate the membrane from the wire mesh and the body of the headphones were modelled as MEMS materials. These components were fully coupled to the surrounding acoustic domain that was modelled as air. The wire mesh was modelled in the acoustic domain as a perforated plate allowing the propagation of the acoustic waves. The model was excited in the frequency domain by applying a constant electrical potential to the membrane and harmonic electrical potentials to the wire meshes that were 180° out-of-phase (balanced drive). The model was solved at 1/12 octave intervals between 30 Hz and 40 kHz. A far-field analyzer was used to extrapolate the pressure field to 100 mm and the model results were compared to experimental data, where a microphone was placed at the same 100 mm distance away from the speaker in an anechoic chamber. Once the model results were validated using experimental data, a graphic user interface was designed using the COMSOL Multiphysics® Application Builder that allowed our clients, who had no modelling experience, to virtually test proposed design modification. The model is hosted on an Amazon™ server and our client can access the model using COMSOL Server™ software where they can model the effects of changing the size and thickness of components, their elastic and electric properties, and the thickness and aperture size of the wire mesh. Once the new model design has solved, the resultant sound field can be viewed (Figure 1) and the frequency of each modification can be compared (Figure 2).

Figures used in the abstract

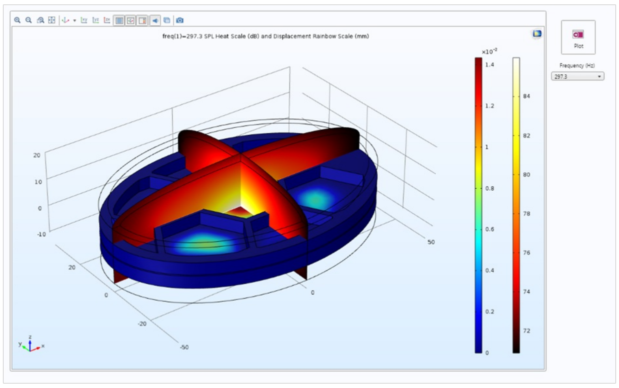

Figure 1: The sound field modelled by our client using COMSOL Server™. The underlying model is a fully couple MEMS-acoustic model.

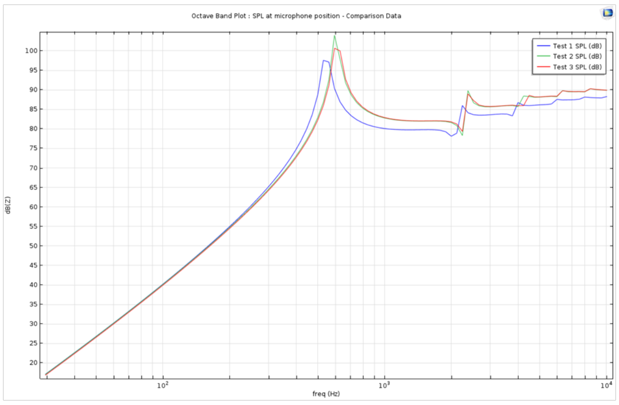

Figure 2: Comparison of modelled SPL from three design modifications by our client using COMSOL Server™.