**Introduction**: COMSOL was used to evaluate the performance of a speaker coupled to an automotive door. The Structural Mechanics solver was used to perform the simulation of the speaker membrane displacement for a rigid and a non-rigid door structure. The Pressure Acoustic module was used to simulate the sound pressure in the vehicle.

**Computational Methods**: A 3D model of a door structure was created and investigated under two conditions. Firstly, a structure was assumed to be fully rigid and the speaker was represented as a rigid piston[1]. Secondly, material properties were applied to the door components, as well as to the speaker geometry (non-rigid case).

Simulated cone displacements were then used to calculate sound pressure in the cabin. Cabin itself was described with frequency dependent absorption coefficients[2].

**Results**: The comparison between the *in situ* measurement and the simulation data shows that the non-rigid boundary condition allows to reach good simulation accuracy below 500 Hz. Differences at around 30 Hz come from panorama roof, not included in the simulation model.

**Conclusions**: It has been shown that interaction between the speaker and the vehicle door can be successfully modelled using COMSOL Multiphysics®. This type of simulation can help to optimize a sound system at the early stage of the design.

**References**: