

A New Computational Tool for Automotive Cabin Air Temperature Simulation

Jiazhen Ling, Vikrant Aute, Yunho Hwang and Reinhard Radermacher
 University of Maryland

ABSTRACT

The thermal comfort inside automotive cabin has been extensively studied for decades. Traditional CFD models provide accurate simulation results of the air temperature distributions inside cabins but at a relatively high computation cost. In order to reduce the computational cost while still providing reasonable accuracy in simulating the air temperature profile inside a mid-sized sedan cabin, this paper introduces a new simulation tool that utilizes a proper orthogonal decomposition (POD) method. The POD method, an interpolation technique, requires only one set of multiple CFD simulations to produce a set of “snapshots”. Later, any simulations that require CFD runs to solve algorithm equation sets can be simplified by using interpolation between the snapshots provided that the geometry of the cabin keeps the same. As a result, the computation time can be reduced to only a few minutes.

CITATION: Ling, J., Aute, V., Hwang, Y. and Radermacher, R., "A New Computational Tool for Automotive Cabin Air Temperature Simulation," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(2):2013, doi:10.4271/2013-01-0868.

INTRODUCTION

The thermal comfort conditions in automotive cabins have been extensively studied by researchers and engineers for decades. Compared to the thermal load of the building, 50% of the automotive thermal load is due to the solar radiation, which can be up to 70 percent of the global sensible heat exchange between occupants and the environment [1]. The volume of automotive cabins is much smaller than those of buildings, which tends to make the air flow field in the cabin extremely non-uniform. For example, air velocity can be the highest at the outlet of air vents but becomes almost zero at passengers' backs due to the direct contact with the seat. Figure 1 shows a flow chart to describe the traditional automotive thermal comfort evaluation process. When simulating the air temperature profile inside the cabin, the most widely used method is the computational fluid dynamics (CFD). Due to the complicated heat transfer and fluid dynamic mechanisms, a fine meshing scheme is generally applied to facilitate the convergence of the calculation. The fine meshes, however, usually lead to a high computation cost for CFD simulations. Therefore, it is worth to investigate other approaches that can reduce the computation cost of CFD while still maintaining a reasonable accuracy. For this purpose, one reduced-order method known as the proper orthogonal decomposition (POD) is a good candidate. Figure 2 shows the benefit of applying reduced order methods by

comparing it with the lumped model method and the CFD simulations.

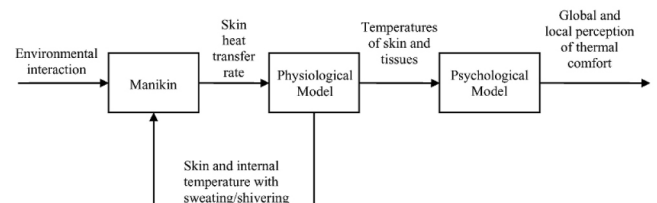


Figure 1. Flow Chart for evaluating the perception of human thermal comfort [2]

From Figure 2, it is clear that the reduced order method can decrease the computation cost as compared to the conventional CFD method while still providing a more detailed level of description than the lumped model can. The reasons for choosing the POD as the reduced-order method are best explained by Berkooz et al. [4]. The advantages of applying the POD method can be summarized as follows: it decomposes a system (which refers to the temperature field or velocity field inside cabins in this paper) into a set of optimum modes and such modes also guarantee a complete representation of the original system.

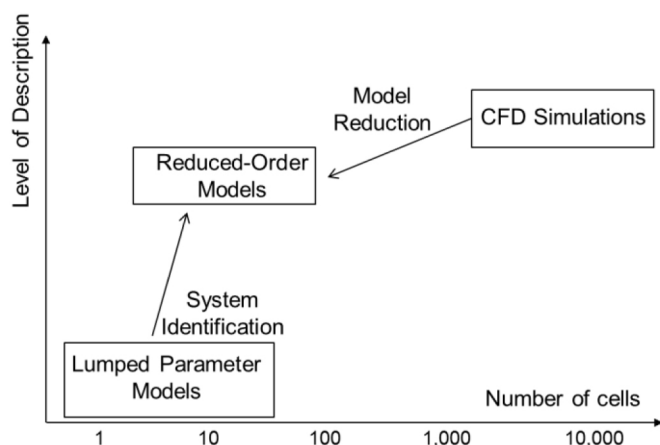


Figure 2. Comparison of reduced-order models with CFD simulations and lumped parameter models [3]

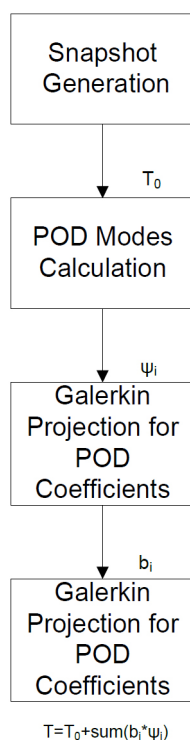


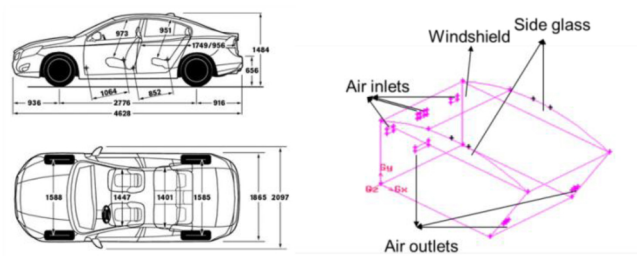
Figure 3. Flow chart of the POD method

Figure 3 shows the flow chart of the POD method. The POD simulation is an interpolation method because a set of snapshots generated through CFD simulations is required at the beginning. The results later generated from the POD simulations are based on these snapshots. The sophistication of the method reflects on the fact that it projects the POD modes onto a set of governing equations rather than using linear, exponential or other traditional curve fitting approaches, which typically lack a good physical description of the system. The POD method has been utilized to predict the thermal comfort in an office setting [5] and shows

encouraging results in terms of both accuracy and reduction in computation cost. In this paper, the POD method is being applied to the prediction of the air temperature distribution inside cabins. Its potential of replacing conventional CFD method is hereby evaluated.

PREPARATION OF SNAPSHOTS

As mentioned in the previous section, in order to conduct the POD calculation, a set of snapshots has to be prepared. The purpose of creating the snapshots is to provide a “database” for the later POD simulations. Since the creation of the snapshots requires CFD simulations, the computation cost is relatively high. However, it is a one-time effort so that the later interpolation can significantly reduce the calculation time. The snapshot preparation starts with the determination of the cabin size. Figure 4 shows the dimensions of a mid-size sedan adapted in the study.



(a) Cabin dimensions

(b) Gambit sketch

Figure 4. Dimensions and Gambit sketch of the selected cabin ((a) source: http://www.cartype.com/pages/4693/volvo_s60_sedan_2011)

The drawing was then simplified and used to create a model in ANSYS Gambit [6]. Theoretically, the POD method can be applied to the case that includes passengers in the cabin, because it is an interpolation method by nature. However, in the current research, the CFD model does not contain any passengers in the cabin. Four supply air outlets are located around the dashboard, and three air vents are located at the lower position of the dashboard and the back of the cabin. The dimensions of the air vents in three different car models as well as their supply air velocities were measured and tabulated in Table 1. The air velocity was measured by using a hand-held anemometer which has an accuracy of 3%. Each air outlet was divided into three segments from the top to the bottom and the average value of the three readings was used as the air outlet velocity. The final dimensions and the maximum air velocities at the vents were then chosen to be 12.7 cm by 7.62 cm and 10 m/s, respectively.

To set the boundary conditions in the CFD calculations, thermal loads on different cabin surfaces have to be decided. Zheng et al. [7] introduced a quick method to estimate the thermal loads of different surfaces in an automotive cabin. In this paper, the vehicle cooling load validation testing was