

Numerical Simulation and Validation of Cabin Aiming and Cool-Down of a Passenger Car

Somnath Sen and Mayur Selokar
Subros Limited

ABSTRACT

Maintaining thermal comfort is one of the key areas in vehicle HVAC design wherein airflow distribution inside the cabin is one of the important elements in deciding comfort sensation. However, the energy consumption of air conditioning system needs to stay within the efficient boundaries to efficiently cool down the passenger cabin otherwise the vehicle energy consumption may get worsened to a great extent. One approach to optimize this process is by using numerical methods while developing climate systems.

The present paper focuses on the numerical study of cabin aiming and cabin cool-down of a passenger car by using computational fluid dynamics (CFD). The main goal is to investigate the cabin aiming with a view to figure out the minimum average velocity over the passengers at all vent positions. Cabin aiming ensures substantial amount of airflow reaches to the passengers as well as every corners of the cabin across the wide climatic range. After the cabin aiming, airflow field inside the cabin was predicted and then the transient behavior of temperature field during cool down was predicted considering appropriate cabin heat loads including that of solar irradiation and cabin soaking. The study shows that flow field on passengers as well as inside the cabin strongly influences the overall cool down rate.

Steady state and transient analysis were carried out in STAR-CCM+ software to figure out the flow field and temperature field respectively. The numerical prediction of flow field, cabin cool down rate were further validated with that of experimental results and it proves to be very useful tool to use at the early stage of design cycle to optimize the HVAC system in terms of performance, development time and cost.

CITATION: Sen, S. and Selokar, M., "Numerical Simulation and Validation of Cabin Aiming and Cool-Down of a Passenger Car," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(1):2016, doi:10.4271/2016-01-0251.

INTRODUCTION

Thermal comfort of the occupants in a vehicle cabin is of paramount importance due to occupant's health and safety. In addition to that, with more stringent requirements for reduced fuel consumption and exhaust emission, larger 'cabin volume'/surface area' ratio, changes to less efficient environmentally safe refrigerant and reduced condenser air-flow are posing significant challenges to occupant thermal comfort. However assessment of the thermal comfort in a vehicle is very complex due to fast transient behavior of hot soak and cool-down, and so for elevated asymmetric thermal environments associated with highly non-uniform air velocity and temperature distribution. This makes it necessary to investigate the air-flow field and the temperature distributions inside the passenger compartment in the design stage in order to ensure enhanced thermal comfort conditions for the passengers. Analysis tools for the temperature and velocity distributions in passenger compartments can provide design directions during the early stage of vehicle development process which will lead to shorter development time and substantial cost reductions.

Zhang *et al.* [1] and Zhang *et al.* [2] performed a thorough experimental and numerical study on the thermal comfort of sedan car models. The relationship between the cooling load and inlet temperature, outside temperature, air flow velocity, as well as the effect of glazing material were investigated. Han *et al.* [3] combined the solar load, refrigeration cycle analysis, CFD, and human physiology model. A simple PMV analysis was done, but it was not regarded as the main criterion for thermal comfort. In a study by Alexandrov [4], factors such as the temperature and velocity at inlets, the size and location of inlets and outlets, outdoor temperature, and air velocity were investigated. A comparison of different scenarios in terms of both the thermal comfort and energy consumption was not performed in that study. Bayraktar [5], [6] investigated the thermal management in a military vehicle compartment.

Han [7] used a three-dimensional model of a passenger car compartment and numerically simulated the thermal environment inside the compartment. Kilic and Sevilgen [8] used different types of boundary conditions on the human body surfaces to determine the suitable boundary condition for evaluating thermal comfort. Numerical results were in good agreement with the experimental data.

used in their study. Ishihara and Hara [9] measured the air-flow velocity distributions in the car compartment by visualization and then compared them with their numerical simulation results. Aroussi and Aghil [10] used a passenger compartment with one-fifth scale model of a vehicle cabin and they studied the characteristics of the fluid flow inside the cabin. Wan and van der Kooi [11] simulated the air-flow and temperature field in a car passenger compartment, especially around the air supply inlets and they found the optimal case for the air supply. Hagino and Hara [12] investigated the factors affecting the passenger's thermal comfort, such as air temperature, air-flow, sunlight radiation, and they proposed a method to forecast passenger's satisfaction. Currle [13] used CFD method to simulate the temperature and the air-flow field in a car's passenger compartment, and discussed the passenger's comfort by considering the passenger's thermal model, the natural convection, the convective heat transfer and the radiation. H. Zhang et.al. [14] used the computational fluid dynamics (CFD) method to simulate three-dimensional temperature distributions and flow field in a car compartment with and without passenger's. They validated their numerical model by making comparison with the experimental data. Jonas Jonsson [15] developed a numerical model that integrates solar load when computing the temperature distribution in a passenger cabin.

Computational fluid dynamics procedures have been applied in various studies on the important components of a HVAC (Heating-Ventilation-Air-Conditioning) system, [16, 17, 18]. Previously Taeyoung Han [19] performed numerical simulations of a two-dimensional, and a three-dimensional airflow in a passenger compartment. In a study by Alexandrov et al.[20], the authors used CFD to evaluate the effect of four HVAC design parameters on passenger thermal comfort in a simplified passenger compartment. They found that the location of the vents, and the air flow rate, were the most important parameters which influenced the thermal comfort of the passengers. Moreover, the position of the outlet in the rear of the car was found to play a significant role in rear passenger thermal comfort.

The present paper aims to provide a new and comprehensive model for estimating the cabin aiming and airflow distribution inside the vehicle. Although there are numerous studies on cabin airflow as mentioned above, the main emphasis of this cabin aiming is to account for the dynamicity of air vents and louvers which decides the spread/range of airflow distribution inside the cabin across all the climatic conditions. This ensures better thermal comfort at any vent angle by achieving a reasonable amount of airflow distribution over the passengers. Four different setups were used to ensure the spread of airflow emitted from duct. A passenger car model equipped with front and rear AC unit was considered for the study. The study also evaluated overall airflow field inside the cabin with two different air vent conditions considering appropriate heats loads including that of solar intensity and cabin soaking. The influence of airflow field on cabin cool-down was studied numerically by adjusting the vent angle to its appropriate positions. The cabin aiming and transient pattern was validated with experimental results. Steady state and unsteady state simulation in CCM+[21] were carried out in order to perform the airflow and temperature field.

CABIN AIMING

Every vehicle equipped with AC system consists of an IP duct which includes vents - two at the centre and two at sides, each vent equipped with number of louvers. From an airflow distribution point of view the centre vents emits more percentage of airflow compared to that of side vents in most of the cases. Each vent which has number of louvers can be adjusted horizontally or vertically. Typically side vents are aimed to cater with airflow requirement of driver and co-driver from left to right or vice versa whereas the centre vents are aimed to provide jet effect for the rear occupants. It is also utilized to cover front row passengers partly. This decides the spread of exit air and forms a conical field mostly covering from head to waist of occupants - most sensitive to comfort range.

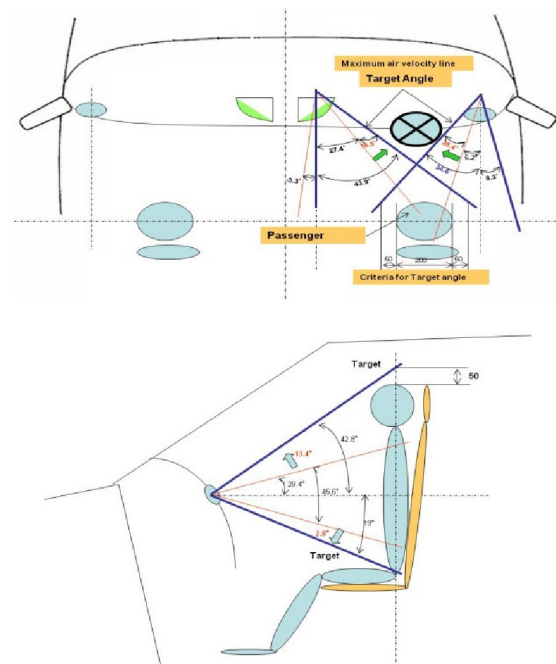


Figure 1. Vent spread angle

Figure-1 shows the spread angle of a vent. The side vent has got 1/5th portion from the centerline which delivers air to rear and rest of the portion covers the driver or co-driver as seen from horizontal spread. Similarly the centre vent has got more than 2/3rd portion aimed towards rear of vehicle while the rest is dedicated for the driver or codriver. The vertical spread angle of vent covers head to waist while most of the flow is adjusted to deliver above the chest. From human thermal comfort point of view such spread angle which covers most of the body above the waist strongly influences the human sensation. Any vent angle which is out of the above range or unable to spread that much angle may cause discomfort to the human body. The same spread angle also should be maintained for other centre and side vent.

While it is out of scope to decide the optimum spread angle, but with pre-defined spread angle the exit air should reach to the passengers and also to every corners of the cabin is the main focus of the present

study. This will help evaluate the reasonable amount of flow distribution at all vent spread angles as it is one of the most vital parameters for human comfort.

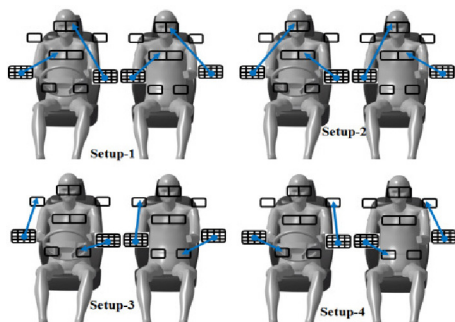


Figure 2. Cabin aiming setup

The above requirement is fulfilled by four different setups of vent exit air as shown in [figure-2](#) by changing the vent louvers directivity towards 4 major areas namely eyes, chest, shoulder and waist. Setup-1 & 2 covers the airflow aiming towards chest and eyes whereas the setup-3 & 4 guide the airflow towards waist and shoulders. Head being the larger surface area releases much more heat compare to the other parts of the body. Next comes the chest and other body parts which release subsequently lesser heat. Hence the setup 1 and 2 are very important in order to take care of the human discomfort due to heat released by head and chest areas. At the same time human waist, shoulder and hands etc. are subjected to get airflow through the other setup to take care of the uncomfortable part of the human thermal sensation. Therefore the aiming provision ensures that a human gets airflow in totality in any conditions by changing the vents direction. With all these setups occupants are subjected to get the airflow from face, chest and waist level. Air velocity on back, pelvis, thigh has very negligible effect on human comfort and that's why the whole concentration is given on head, arm, chest e.g. above the waist level. Based on the benchmark results from similar vehicle it is observed that air velocity within the range of 1.8-2.3 m/s over occupants is best satisfying the comfort level. Beyond the above range it may cause discomfort in a way that human gets less cold air so high cold air which anyway a concern for thermal sensation. Once the cabin reaches to the desired comfort range, blower speed can be reduced in order to maintain energy consumption within the efficient boundaries.

THERMAL COMFORT

Comfortable climate is usually defined as "the condition of mind that expresses satisfaction with the thermal environment" (ASHRAE 55, 1992) [22]. Thermal comfort is human being's thermal sensation of the surrounding environment, which expresses the level of satisfaction of thermal environment. Thermal comfort in vehicles is highly regarded as one of the most important factors when a vehicular thermal environment is designed. A comfortable thermal sensation could not only bring the occupants a great feeling and good physical states but also more concentration and motivation in work or life

which make a tremendous contribution to the work efficiency and life quality. For a vehicular environment, a comfortable temperature and constant fresh air is required. And in order to fulfill this requirement, a well-performed HVAC system is indispensable. Nowadays a basic standard heating, ventilation and air-conditioning system has been already suited in over 85% of new vehicles, which could meet the demand of thermal comfort of most occupants.

CABIN HEAT LOADS

A clear understanding of the heating and cooling loads, encountered by the passenger cabin, is a key prerequisite for an efficient design of any mobile AC system. The function of AC system is to compensate for the continuous changes of cabin loads in order to maintain the passenger comfort within a thermal 'comfort zone'. To do this, it is necessary to control the AC cycle to maintain the air quality within the desirable limits. The net heat gain by the cabin can be classified under various categories. The total load as well as each of these loads can either be positive (heating up the cabin) or negative (cooling down the cabin) and may depend on various driving parameters. The thermal loads are imposed to the vehicle cabin through various mechanisms. Generally, the thermal loads in the cabin are calculated based on a heat balance of the incoming thermal energy from different sources as shown in [figure-3](#). The maximum contribution of heat load comes from glass surfaces. The summation of all the load types will be the instantaneous cabin overall heat load gain. The mathematical formulation of the model can thus be summarized as.

$$\dot{Q}_{Tot} = \dot{Q}_{Met} + \dot{Q}_{Dir} + \dot{Q}_{Dif} + \dot{Q}_{Ref} + \dot{Q}_{Amb} + \dot{Q}_{Exh} + \dot{Q}_{Eng} + \dot{Q}_{Ven} + \dot{Q}_{AC}$$

Where $\dot{Q}_{Met}$ is the metabolic heat rate from humans, $\dot{Q}_{Dir}$ is the direct heat transfer into the cabin and $\dot{Q}_{Dif}$ is the diffused heat transfer, $\dot{Q}_{Ref}$ is the reflective heat transfer, $\dot{Q}_{Amb}$ is the convective heat transfer, $\dot{Q}_{Exh}$ is the exhaust heat transfer, $\dot{Q}_{Eng}$ is the heat transfer from the engine, $\dot{Q}_{Ven}$ is the heat loads due to ventilation and $\dot{Q}_{AC}$ is the heat load coming from the AC system.

In order to maintain the interior temperature constant, the heat removal by the air-conditioning $\dot{Q}_{AC}$ should be equal to the heat gain. According to the requirements for thermal comfort in summer conditions, interior air temperature should be in the range of 23 to 28°C (ASHRAE, 2003). Each thermal load is calculated assuming a quasi-steady-state condition.

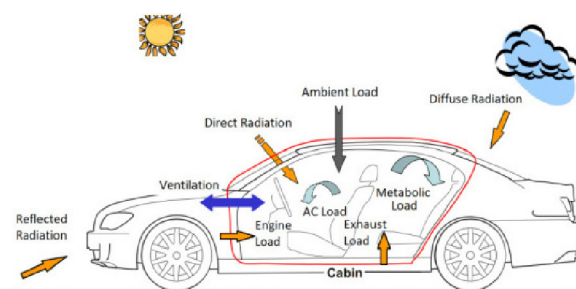


Figure 3. Cabin heat load