Computational Analysis of Thermal Distribution within Passenger Car Cabin

Zeya Ahmad Quadri¹ & Jomon Jose²
¹Department of Aerospace Engineering, University of Petroleum and Energy Studies, Dehradun
²Department of Mechanical Engineering, The ICFAI University, Dehradun
E-mail : zeya.quadri@stu.upes.ac.in, jo12jose@gmail.com

Abstract – This paper on the thermal distribution within a passenger car cabin discusses and solves the problem of Thermal Accumulation within a car when it is parked under the sun for some period of time. It discusses the effects of inlet and outlet vents on the temperature distribution in the cabin. Studies show that the engine working temperature of the car has an impact on the cabin environment. In addition it has been identified that proper ventilation within the car reduces this problem.

In this paper, we use 3-dimensional computer simulations using CFD technique to examine these issues of Heat accumulation within the car. This paper identifies the causes of this common problem that most passengers face in their daily lives and aims to suggest ways of reducing the uncomfortable period of time within the car.

Keywords—CFD, solar radiation.

I. INTRODUCTION

Modern technology has brought a lot of changes to the cars we look around in the traffic. The cars have evolved not just in their exteriors but also in their interiors. The luxury segment cars are primarily about the inner comfort. They intend to improve the experience of the passenger within the car.

The problem faced by the cars today is the initial high temperature within the car cabin as we enter into them. This phenomenon of Thermal Accumulation is clearly evident when the car has been parked in the sun for a few hours. The car user is forced to wait for a period of around 2-5 minutes to allow the cabin environment to be like the ambient.

In a parked car with its Engine on, we can also notice a temperature rise within the car because of the heat leakages from the Engine Section.

The comfort provided by a car to the passengers travelling in it has become a major field of improvement and demand in the Automobile Industry. A perfect HVAC (Heating Ventilation -Air Conditioning) is being explored by the research scientists.

To understand this, knowledge about the Thermal Distribution within the car is essential. In this project a GAMBIT model of a two-seater car has been used to analyse in Ansys Fluent the various factors that affect the cabin temperatures.

This study shall present before the car engineers various aspects to be looked into while designing a car with not just an aesthetic value but which also delivers comfort to the passengers.

A. Working Principle

Thermal analysis of a passenger compartment involves not only geometric complexities but also strong interactions between airflow and the three modes of heat transfer, namely Heat Conduction, Convection and thermal Radiation [3].

Conduction is a mode of transfer of heat which occurs when a temperature gradient exists in a body or two or more bodies in the same phase. The heat transfer rate per unit area is proportional to the normal temperature gradient [5]:

$$\frac{\partial T}{\partial x} \sim \frac{q}{A}$$

When the proportionality constant is inserted:

$$q = -kA\frac{\partial T}{\partial x}$$

(1)
Where

\[ q = \text{Heat transfer rate} \]
\[ \frac{\partial T}{\partial x} = \text{Temperature gradient in the direction of flow} \]
\[ k = \text{Thermal conductivity of the material} \]
\[ A = \text{The area under consideration} \]

The minus sign is inserted so that the second principle of thermodynamics will be satisfied. The equation is called the Fourier’s law of Heat Conduction. Convection is the mode of heat transfer between any two different phases such as a solid and a liquid or a solid with a gas or vice versa.

The overall effect of convection is expressed by Newton’s law of Cooling.

\[ q = hA\Delta T \quad (2) \]

Where

\[ q = \text{Heat transfer rate} \]
\[ h = \text{Convection Heat transfer Coefficient} \]
\[ \Delta T = \text{Temperature gradient between the two bodies} \]
\[ A = \text{Area under consideration} \]

Energy transfer can also take place without a medium this mode of heat transfer is called Radiation. The mechanism in this case is electromagnetic radiation. Thermodynamic considerations show that an ideal thermal radiator or a black body will emit energy at a rate proportional to the fourth power of the absolute temperature of the body.

\[ Q_{\text{emitted}} = \sigma T^4 \quad (3) \]

Where

\[ \sigma = \text{Stefan-Boltzmann Constant} \quad (5.669 \times 10^{-8} \text{ W/m}^2\text{.K}^4) \]

B. Parameters:

Solar Radiations

The Solar Radiations entering the car through the windshield account for the temperature rise within the car on a sunny day. The total heat permission through the glass is expressed as the sum of the solar radiation transmitted through the glass and the inward heat flow by convection from the glass inner surface [3].

Prolonged periods in the sun can deteriorate the car interiors including trims, plastic moldings, upholstery and seat covers. If there are fancy leather interiors, they need to be taken care of because of the constant exposure to heat that can induce weathering in leather materials that may eventually result in cracking and fading of surfaces. Also, a car with leather seat-covers parked in the sun is good enough to cause serious burns if a person sits inside without allowing it to cool.

There are technologies to reduce the temperature when parked. The “Auto Cool” is one such device which consists of a solar-powered car fan that aims to keep the interior of the vehicle cool even after many hours baking under the sun. In this technology all a person needs to do is place the device on top of the window and the fan will get to work as long as there is sufficient sunlight.

It maintains the security level of car intact since there wouldn’t be any gap located at the top. Tests prove that the Auto Cool supposedly keeps a car cooler by up to 30 degrees when used.

Tinted glasses can also be used to keep the car-cabin cool and hence avoid the need for constant use of air conditioner while driving. As we all know, less use of air conditioner means more fuel efficiency. Also, tinted mirrors maintain decent comfort levels inside the car-cabin by preventing harmful heat from entering inside. Tinted glasses can prevent the passenger’s skin from harmful effects of sun’s heat and radiations. Tinted windows also prevent the driver from any glare emanating from roadside objects making the drive safer.

Engine Temperature

The engine of the car operates at very high temperatures because of the ceaseless explosion of the fuel between piston and the cylinder. This generates temperatures as high as 160°-195°F within the engine. The modern day car design is such that the heat within the engine is dispersed to the atmosphere from the openings in the bonnet of the car. In spite of such an efficient car design some amount of heat leaks in to the cabin through the openings in the dashboard and other parts thus affecting the thermal distribution within the car.

In a car parked with its engine on, we can notice the effect the Engine temperature has on the cabin climate, while for a moving car most of the heat generated is lost to the ambient because of its design thus negating the effect. This research intends to note the changes this factor brings about in the environment within the car.
Number and position of Inlets and outlets of air within the car

Studies in this field have detected that the increase in the number of air inlet/outlet vents leads to decrease in the hot zones within the car [3]. Fresh air enters the cabin via the windshield cowl area. If we open the hood and look at the cowl we shall see openings that allow the air to enter. The number and also the position of these inlets and outlets provide better ventilation within the car.

Better ventilation within the cabin ensures that the environment remains almost the same as the ambient.

In this experiment the inlets have been considered to be located symmetrically on the dashboard 15 cm away from the sidewalls and the outlets below the rear window 30 cm away from the sidewalls.

C. Applications of the study

- High levels of vehicle comfort are being increasingly demanded by users. This creates a new challenge for climate control engineers to develop the (HVAC) Heating Ventilation and Aero-Conditioning for a new vehicle, this study simplifies the process of the engineers [5].

- When a vehicle is parked under the sun, accumulated heat affects many interiors, thus bringing about their degradation this study can be used effectively to prevent this [1].

- This study will provide the scientists in the automobile sector the exact causes of the temperature rise within the car. It shall also suggest methods for improving the environment within.

- This study can be used further in any closed environments where temperature distributions play a vital role; for example Space shuttles etc.

II. LITERATURE REVIEW

Al-Kayiem [1] found out that when a car is parked under the sun there is degradation of its interiors and an uncomfortable period for the passengers. He experimentally with the help of thermocouples pointed out the areas of thermal accumulation within the car. The highest temperatures were noted in the spots near the windshields, the dashboard is seen to be a major heat source for the cabin environment and the primary sink for the external radiations. He suggested the installation of a sunshade which proved to reduce the temperature by about 27%.

Alexandrov [2] in his study indicated that both the external and internal parameters play an important role in determining the HVAC system for the car. He depicted the air flow patterns conjured up within the car due to these parameters. In his study he suggested that the development of zones of low air circulation can be reduced significantly by improving the inlet/outlet configuration.

Jalil and Alwan [3] put together a numerical study of a two dimensional, turbulent, recirculating flow within a passenger car cabin. Their study demonstrated the capability of CFD to accurately simulate the air flow within the cabin. They predicted that the increase of air inlets leads to the decrease of hot zones. It also leads to lower temperature gradient near the interior surfaces and a uniform temperature distribution.

Zhang, Dai and their team [4] in their study highlighted the problems and challenges faced by the Air Conditioner within the car using Fluent and applying a user defined function in accordance with the assumptions taken in the numerical analysis. In addition the air temperature distribution in a steady test situation is also numerically predicted by both transient and steady state model and the agreements between the data and the numerical analysis is satisfactory.

III. SIMULATION MODEL, METHODS AND BOUNDARY CONDITIONS

A. 3D Cabin Modeling

The entire space of the vehicle consists of the computational Domain and the inner surfaces of the compartment are the boundary conditions. The 3D model of the car was prepared using Gambit 2.3.16. The total volume of model is 4.26 cubic meters and a Face Area of 4.53018 x 10^-3 m². This study concentrates in the thermal distribution within a two seater car so as to reduce the complexities in the flow distribution of a four seater. The dimensions of the car are according to the ASHRAE standards. Inlet and outlet vents have been added to the structure at the specified locations.

B. Meshing Criteria

The meshing of the structure was completed in Gambit. Unstructured grids have been adopted, and most of the grids are tetrahedrons. The mesh has been given a spacing of 0.15 units for proper and uniform meshing. Tetrahedral meshing has been preferred over any other because it produces more accurate results than
quadilateral due to the non-uniform geometrical shape of the model under investigation. The meshing yields a total of 6892 tetrahedral cells, 14818 faces and 1707 nodes.

With the consideration of inlet and outlet vents meshing of the model has been done by tetrahedral meshes of size .08 units. A total of 52928 cells, 109445 faces, and 10684 nodes were formed.

C. Boundary Conditions for Solar Radiations

The inner surfaces of the car have been given boundary conditions. The Window glass is somewhat transparent and the major part of sun radiation can go through the glass. Solar radiations have been assumed to fall on the front wind shield, the side windows and the rear window. The momentum properties of the windows are considered to be a stationary wall with no slip condition. The thermal properties of the windows are set with an external temperature of 310K. The properties are that of air. The windows participate in solar ray tracing, the Absorptivity and Transmissivity are taken as the default values from the Fluent database. The other parts are considered as opaque walls which transmit no part of the solar radiations inside the cabin. It has been assumed that the other parts are completely adiabatic. They have zero heat flux and heat generation rate. No convection, radiation or any other mode of heat transfer takes place from across its walls. They do not take part in solar ray tracing. The fluid enclosed is considered as air with properties as provided in the Fluent database.

For solving the problem a pressure based solver with an absolute velocity formation and a steady time is used. The energy equation has been utilized to achieve results. The P1 Radiation model has been used to analyze the problem. In the P1 radiation model the directional dependence in Radiative Intensity Transport Equations (RTE) is integrated out, resulting in diffusion equation for incident radiation. This model boasts of benefits like low CPU demand, ease of working in applications where optical thickness is large.

The model has been analyzed when the car is positioned in the latitude and longitude of New Delhi at 2 Pm on the 10th day of the 4th month. Fair weather conditions have been assumed to get a conclusive result. The Sun direction vector has been calculated from the solar calculator with a sunshine factor of 1. The direct Solar Irradiation is considered as 1423W/m² and the Diffuse Solar Irradiation as 200W/m². The spectral fraction is 0.5. The residual monitors have been plotted with a storage capacity of 1000 iterations.

D. Boundary Conditions for inlet and outlet vents

The window panes are considered to be semi-transparent, thus allowing some amount of solar radiation to enter inside the car. The boundary condition in this analysis is same as the above section. In addition to that the Inlet vents have been set as velocity inlets and the outlets as pressure outlets. The flow within the cabin has been considered as laminar and steady and the experiment has been conducted in atmospheric pressure conditions of 101325 Pascals. A pressure based solver has been favored in this analysis. The radiation conditions are same as that of above. The P1 radiation model has once again been favored in this analysis. The material that makes up this 3D model is once again air with properties that are found in normal conditions.

E. Boundary Conditions for Engine Temperature

The Engine operates at very high temperatures. The Ignition chambers reach up to 2000-3000K at the time of ignition. The car has inbuilt cooling systems which do not allow the heat generated to affect the cabin environment. The Thermostat and Radiator systems help to keep the temperature in control. The heat absorbed by the cooling water causes it to gain temperature. This water circulates within the cabin. The boundary temperatures for this problem have been taken as 353K. The ambient temperatures have been taken as 300K. The engine temperature seeps in through the area under the dashboard. Adiabatic conditions prevail within the car.

F. Inlet and Outlet Conditions

The air temperature and velocity at the inlets play an important role in determining cabin climate [2]. Steady state conditions are prevalent within the cabin. The ambient temperature is considered as 300K while the inner temperatures are relatively high due to the solar radiations falling on the window panes. The inlet velocity vents blow air at a velocity of 8m/s in the X direction, while .95m/s in the Y and Z direction at a temperature of 289K. The external back flow temperature at the pressure outlet i.e. the outlet vent is 310K. The X component of the outlet vent is .95 m/s.

G. Inlet and outlet Locations

In order to establish the role of the size, number and location of HVAC system inlets and outlets, a 3D simulation for various positions of inlet and outlet have been run. There are two inlets in the cabin, each of size (3X80 cm) located symmetrically on the dashboard 15cm away from the sidewalks. Two outlets of size
(7.5X3.5 cm) are provided below the rear window at a
distance of 30cm from the sidewalls [2].

IV. RESULTS AND DISCUSSION

A. Temperature variation due to solar radiation

The meshed model was exported to the simulation
software Fluent 6.3.26. This solver yields the following
results:

From the Fig 1 and Fig 2 it is evident that the
passenger seats, roof of the car which had perfect
adiabatic conditions also notice a 3K rise in temperature.

The Fig 3 gives us the Cross Sectional view of the
cabin environment. From the figure we can deduce that
the air near to the windshield, side windows, and back
windows has very high temperatures and it decreases as
we move inside. The high temperature on the windshield
face is bad for the material on the dashboard. The
dashboard tends to absorb this heat and it begins to act
as an internal heat source for the cabin [1]. The results
indicate that the temperature increase due to radiation is
more pronounced than that areas affected by convection.
B. Temperature variation due to number and position of inlets and outlets

The openings were provided for the inlet and outlet vents and the temperature of the stream of air from the inlet vent is 289K. The simulation was conducted when the sun rays fell on the window panes as in the previous section.

Figure 5: Contours of static Temperature (K) with inlets

The Figure 5 depicts the boundary conditions that have been imposed on the model. The blue spots show us the air inlet velocities at a temperature of 291K. These cold air inlets change the environment within the car.

Figure 6: Contours of Static Temperature (K) viewed from X Plane

Figure 7: Contours of Static Temperature (K) when cut along Z Plane

An area of air circulation is formed in front of the passenger seat. The Fig. 6 depicts the velocity profile of air within the car. These areas of air circulation are evident from the differences in velocities of the air flowing within the car.

Figure 8: Contours of velocity magnitude (m/s) along the x plane

V. CONCLUSION

A simulation of a 3D car cabin model has been performed with the help of simulation software such as Fluent6.3.26. The car has been subjected to various external and internal parameters. The car cabin environment was analyzed when it was kept in Delhi at 2 pm on the 10th of April. The Solar calculator was used to determine the intensity of radiations falling on the window panes. To depict the inlet and outlet vents in a car, slits with specific dimensions were cut in the model and cold air was allowed to enter into the cabin.

When the Solar radiations fall on the window panes the areas around them experienced the highest temperatures and a porous slit positioned above the windows could act as a vent for hot air. From the simulation it was evident that the location of the vents plays a vital role in determining the cabin environment. The vents if positioned towards the center of the dashboard tend to create a cool environment in the front part of the passenger seats. The velocity inlets help to negate the high thermal accumulation due to the incoming Solar Radiations.

So the HVAC (Heating Ventilation and Air Conditioning) system in the car plays a crucial part in determining the cabin environment. Future studies can take the help of the simulation software to make the ventilation system more efficient.
VI. REFERENCES


