International Journal of Engineering and Manufacturing Science. ISSN 2249-3115 Vol. 7, No. 1 (2017)

2-D CFD analysis of passenger compartment for thermal comfort and ventilation

Ajay Giri [1*], Brajesh Tripathi [2] and H.C. Thakur [3]
[*1, 3] School of Engineering, Gautam Buddha University, Greater Noida, U.P, India
[2] Department of Mechanical Engineering, Rajasthan Technical University Kota, India

Abstract- The temperature of passenger compartment gets increased very high during summer, if a car is parked in open atmosphere for long duration. It is very uncomfortable for the passengers and driver during start of the driving. This may be fatal to life also when a child is left inside during parking due to some reason because of heat accumulation and ventilation problem. In this study, a 2-D numerical analysis of passenger compartment of a small ALTO™ car is carried out considering an inlet vent in between roof and front windshield while an outlet vent in goods cabin. Governing equations for velocity and temperature are solved numerically with CFD software (Ansys Fluent V.14.0) with SIMPLEC algorithm to find velocity and temperature distributions. A two dimensional incompressible, laminar, steady flow analysis with Boussinesq's approximation is carried out with varying inlet velocities and constant ambient temperature. It is found that air circulation in rear of the compartment is more in comparison to front and it gets improved with the increase in the inlet air velocity. There is improvement in temperature distribution and finally reduction in temperature difference between compartment and ambient temperature with the increase in inlet velocity. This shows that provision of vents in the compartment can reduce the problem of heat accumulation and ventilation and provide better thermal comfort to the passengers.

Keywords: Passenger compartment, Ventilation, Heat accumulation, Thermal comfort

1. Introduction

The temperature inside the compartment of a car parked in open or un-shaded parking area in summer season gets increased very high after some time which is uncomfortable to the passengers and driver in initial part of journey. In general, passengers are used to wait for some time before getting into the car so that the compartment gets cool either by rolling down the windows or opening the doors. Cooling of the cabin could also maintained by operating air-conditioning system at high speed but there is extra energy consumption in removing this excessive accumulated heat of cabin. The excessive heating of compartment will reduce the aesthetic look and life of interiors. At the same time it may be fatal to life of toddlers/ infants, pets, etc. when locked inside due to any reason.

During summer, when ambient temperature is in between 30^0C – 45^0C, the difference of temperature between interior cabin and ambient temperature goes up to 45^0C[1]. Worldwide there are number of deaths occurring particularly of young children /infants every year. In USA itself there are average 37 deaths of children per year since 1998 due to the locking inside the cabin [2]. In India deaths of children are 17 during 2014-16 due to heat / suffocation with same reason as reported in news. So, it is important to reduce the unwanted rise of temperature in the cabin of car parked in open. This can be achieved by continuous ventilation of hot air. In this paper numerical study is carried out for temperature distribution and flow field for ventilation of heat to achieve better thermal comfort.

2. Literature Review

Saidur et al.[3] studied the existing commercial ventilation system with some modification for better air flow rate and to reduce temperature of the compartment of car. They found that modified ventilator reduced the inside temperature of cabin of car by at least 11% in comparison to the existing ventilator. The flow rate also got improved by 5.5 times higher than the existing ventilator. Al-Kayiem et al.[4] studied the thermal accumulation and distribution inside the cabin of car parked under the direct sunshine and found that the highest temperatures rise to 75°C occurs near the windshields when all windows were closed. There was drastic reduction in thermal accumulation by installation of sun shade underneath the front windshield and reduction in the temperature of the cabin was about 27%.

Vishweshwara and Dhali [5] installed a solar powered cabin air ventilator behind the back seat and investigated the air temperature in the cabin. They observed that there is maximum 65 ^{0}C temperature in the cabin at outside ambient temperature 43 ^{0}C without ventilator. There was reduction of 10 ^{0}C in the cabin temperature with ventilator. Singh et al.[6] investigated the effect of dynamic vents on the cabin temperature in initial cooling condition of Air-conditioning system with computational fluid dynamics (CFD). The experimental and simulation data shows that improved cooling of the cabin as well as maintaining a uniform temperature distribution in the cabin can be achieved at a particular vent angle. They also suggested that additional air-inlets on the roof top can improve air circulation for the rear passengers. Some more investigations were found but still there is problem of heating and suffocation in parked car. Most of the study is 3D but in this study we had conducted 2D analysis of the car cabin to investigate the heat ventilation and air circulation to improve thermal comfort and safety of passengers.

3. Problem Formulation

The present investigation is carried out for numerical prediction of flow field and temperature distribution within the cabin of car to investigate thermal comfort and ventilation with specific position of inlet and outlet vents. A two dimensional, incompressible, laminar steady flow with Boussinesq's approximation is considered due to low inlet velocity for the numerical analysis. Navier- Stokes equations and energy equation are solved with CFD software (Ansys Fluent V.14.0) and SIMPLEC algorithm is used for velocity-pressure coupling assuming that all physical properties are constant.

3.1 Governing Equations

The following governing equations are used for the study of various flow parameters within the compartment. Continuity equation (Conservation of mass):

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

Conservation of x-momentum:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \qquad (2)$$

Conservation of y-momentum:

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+v\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)-g \qquad (3)$$

Energy conservation equation:

$$\rho C_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)+\Phi \qquad (4)$$

Where, $\Phi = dissipation\ function\ of\ energy$

$$=\mu\left\{2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial x}\right)^2\right]+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2\right\}$$

3.2 Geometry and Boundary Conditions

The geometry for analysis is actual size of passenger compartment of a small ALTO™ car as shown in fig.1. Inlet vent is taken at just below the roof of car in front while outlet vent is taken below the rear windshield in goods cabin. The vent size for inlet as well as outlet is o.15 m.

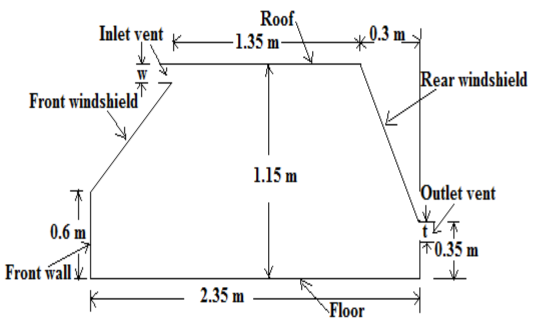


Fig.1 Geometry of car

The boundary conditions for roof, front wall and floor are constant heat fluxes of 202 W /m^2 , 112 W /m^2 and 35 W/m^2 respectively Chien et al. [7].Mixed (convection and radiation) boundary condition is taken for front and rear windshield with free stream temperature as 400C, external emissivity as 0.88 and external radiation temperature as ambient temperature (400C). Convective heat transfer co-efficient [8];

$h = 1.163\ (4+12\sqrt{v})$ W / (m^2 ×0C)

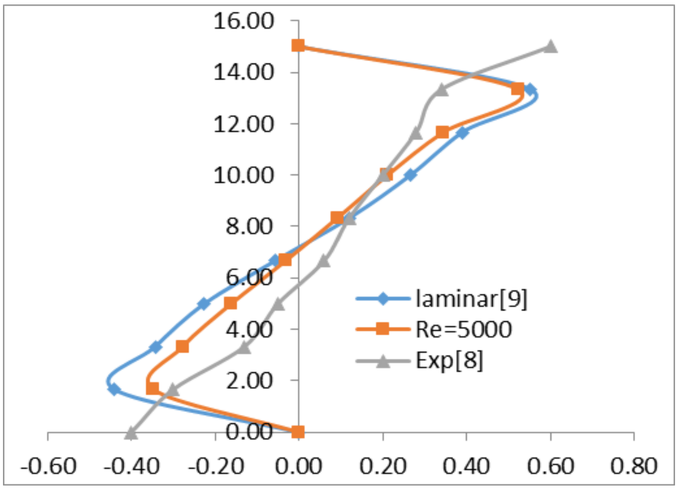
Where, v = wind speed relative to car, m/s

Inlet: velocity inlet, temperature 40^{0}C

Outlet: pressure outlet, gauge pressure = 0

3.3 Numerical Solution Procedure

The computational domain is meshed into triangular cells of 21532 nodes and 20984 elements and grid refinement done at inlet and outlet vents. Grid-independence test has also been performed in order to ensure accuracy of the results. Scheme for pressure-velocity coupling is taken as SIMPLEC and for discretization of momentum and energy equation is adopted as first order upwind. Convergence criteria for residual of x-velocity and y-velocity are taken as 1e-3 while for continuity and energy as1e-4 and 1e-6 respectively.



X-axis: u/U (non-dim velocity), Y-axis: y/w

Fig. 2 Comparison of velocity profile at rear seat position (x = 2/3 L)

The numerical results have been validated with experimental results reported by Neilsen [9] and numerical solution of Tripathi and Moulic [10] and comparative result is shown in fig.2. For validation of the result inlet velocity is taken as 0.455 m/s and Re = 5000. It shows the comparison of x-velocity (u) at vertical cross-section of rear cabin (x=2/3 L). Results are in good agreement with the standard solutions.

4. Results and Discussion

An investigation for airflow and temperature field has been performed in a car compartment with different inlet velocities at summer condition of 40^{0}C. An inlet vent at just below the roof in front windshield and an outlet just below the rear windshield in goods cabin of the car have been considered for the analysis. The circulation of air and temperature difference between the cabin and ambient environment (T - T_a) has been studied to find ventilation of heated air from the cabin for thermal comfort of the passenger. Computed numerical results are shown in figs. 3 to 12.

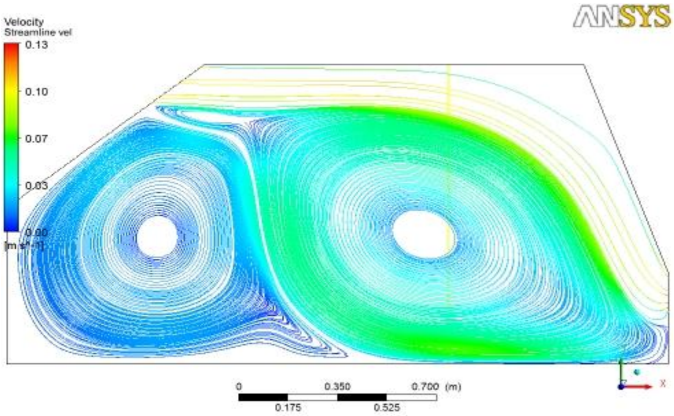


Fig.3 Streamlines for inlet velocity, U = 0.1 m/s

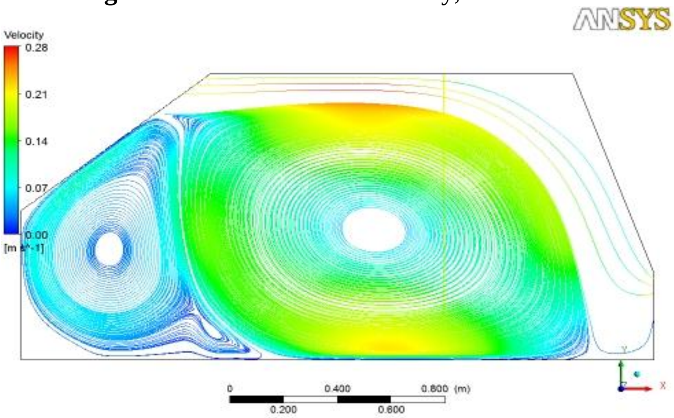


Fig.4 Streamlines for inlet velocity, U = 0.2 m/s

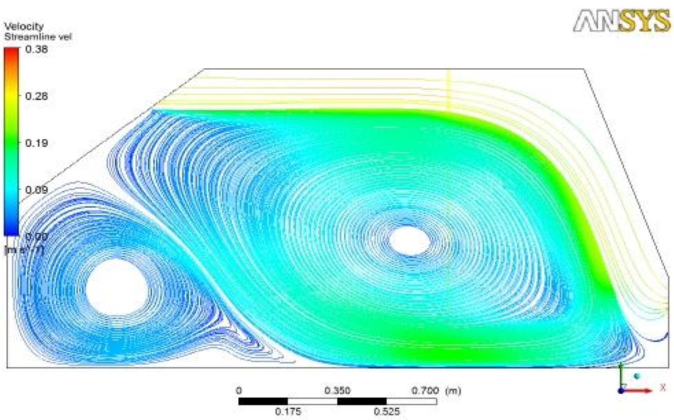


Fig.5 Streamlines for inlet velocity, U = 0.3 m/s

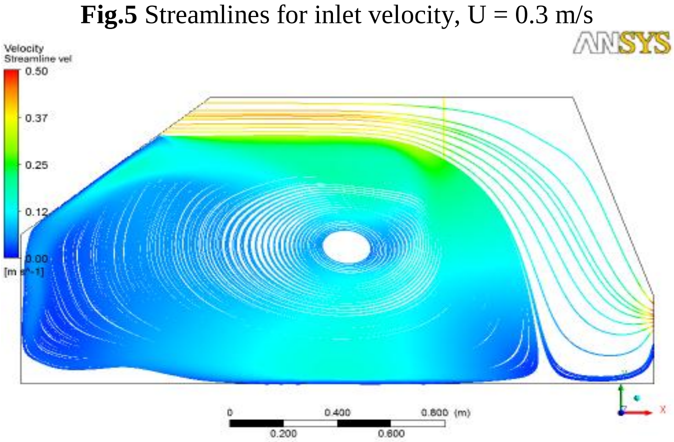


Fig.6 Streamlines for inlet velocity, U = 0.4 m/s

The streamlines of airflow in the compartment of car is shown in fig.3- 6. It shows that circulation of air is more uniform in rear compartment in comparison to front compartment and it improves with increase in inlet-velocity of air. It also shows that air circulation loop in the rear compartment moves to front compartment with increase in inlet - velocity.

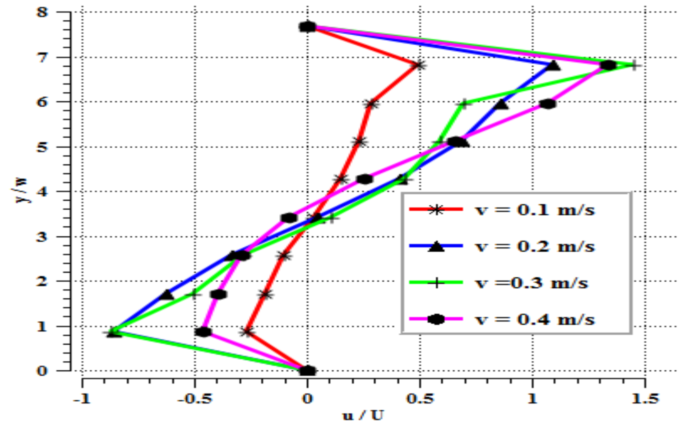


Fig.7 Comparison of velocity profiles at rear seat position (x = 2/3 L)

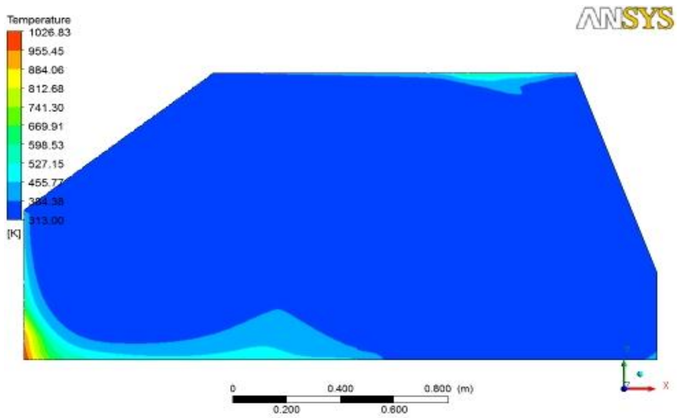


Fig.8 Isotherms for inlet velocity, U = 0.1 m/s

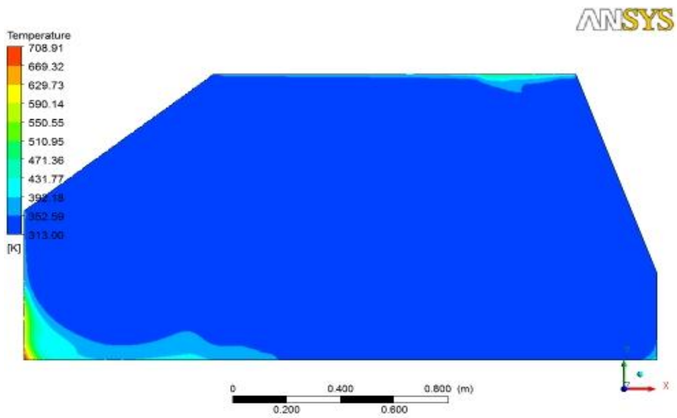


Fig.9 Isotherms for inlet velocity, U = 0.2 m/s

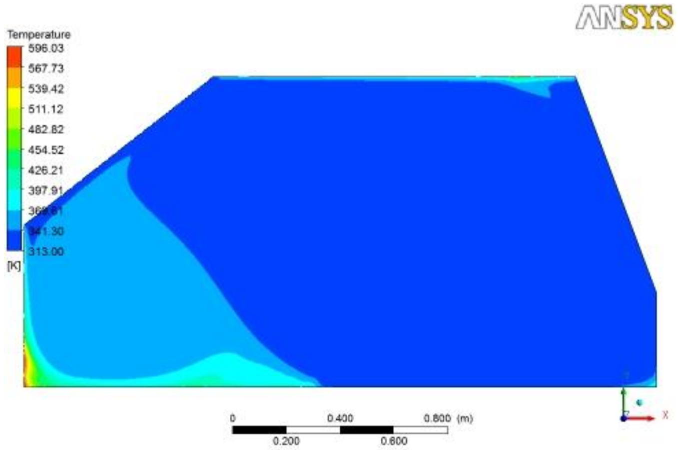


Fig.10 Isotherms for inlet velocity, U = 0.3 m/s

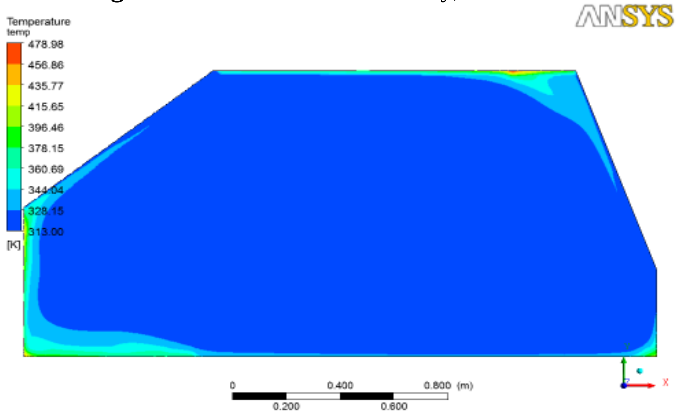


Fig.11 Isotherms for inlet velocity, U = 0.4 m/s

Figs. 8 to 11 show the isotherms for different inlet velocities. It is observed that temperature distribution is almost uniform in cabin except in the region near the roof and floor and uniformity of temperature gets improved with increase in inlet-velocity. There is heat accumulation in the corner region of front wall which is the weak circulation zone in the cabin.

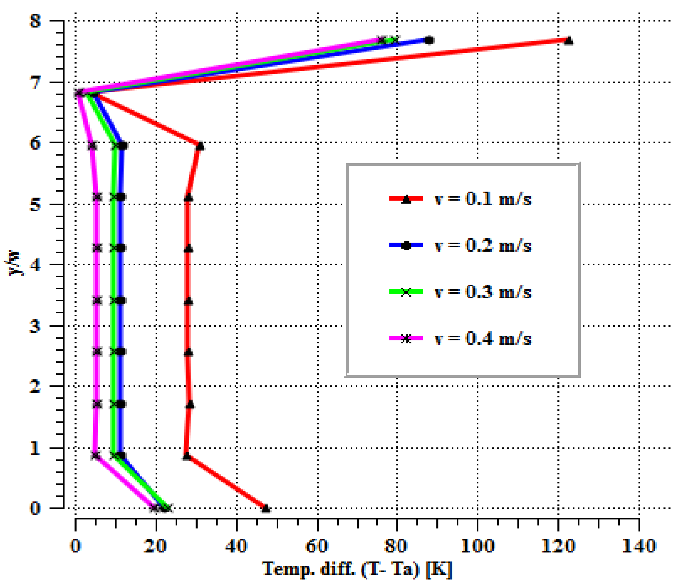


Fig.12 Comparison of temperature profiles at rear seat position (x = 2/3 L)

Fig.12 shows the comparison of cabin temperature difference with ambient temperature at different inlet velocities vertical cross-section at 2/3rd distance from front (i.e. at rear passenger seat in rear compartment). It shows that there is uniformity in cabin temperature except in the region between near the floor and roof. The cabin temperature reduces with increase in inlet velocity because circulation of air gets improved and difference between cabin and ambient temperature reduces from 28^{0}C to 5^{0}C with increase in inlet velocity from 0.1 m/s to 0.4 m/s within the working zone.

5. Conclusions

Numerical analysis of the cabin of car was performed to investigate the air circulation and temperature field inlet velocities as 0.1 m/s, 0.2 m/s, 0.3 m/s and 0.4 m/s with front inlet and rear outlet vents. The simulation results show that air circulation in rear compartment is good in comparison to front compartment. The circulation and temperature distribution gets improved with increase in inlet velocity. There is improvement in reduction of cabin temperature with air circulation from 28 ^{0}C to 5 ^{0}C.

References

[1] Dadour Almanahle I.,Fowkes N., Grant Keady, Vijyan K., Temperature variation in the parked car, Forensic science International, 1(207), 2009, 205-211.

[2] Catherine McLaren, Jan Null and James Quinn, Heat Stress From enclosed Vehicles: Moderate Ambient Temperature Cause Significant Temperature rise in Enclosed Vehicles, Pediatrics, 116, 2005; 109-112.

[3] Saidur, R., H. H. Masjuki, and M. Hasanuzzaman , Performance of an Improved Solar Car Ventilator, International Journal of Mechanical and Materials Engineering (IJMME) 4 (1), 2009, 24 -34.

[4] Al-Kayeim, Hussain H., M.Firdaus Bin M.Sidlik, and Yuganthira R.A.L Munusammy, Study on the Thermal Accumulation and Distribution Inside a Parked Car Cabin. American Journal of Applied Sciences 7(6), 2010, 784-789.

[5] Sudhir Chitrapady Vishweshwara and Jalal Marhoon AL. Dhali, Experimental investigations on the performance of solar powered cabin air ventilator,World Journal of Engineering, 12(6), 2015 ,607-618.

[6] Singh, O.P., Raut, R., Biswas, M. Singh, R., Effect of dynamic vents on the thermal comfort of a passenger car, Journal of Mechanical Engineering, 61 (10), 2015, 561-570.

[7] C.H. Chein, J.Y. Jang, Y.H. Chen, S.C. Wu, 3-D Numerical and Experimental Analysis For Airflow within a Passenger Compartment, International Journal of Automotive Technology, 9(4), 2008, 437−445.

[8] Huajun Zhang, Lan Dai, GuoquanXu, Yong Li, Wei Chen, Wenquan Tao, Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy. Part I: Test/numerical model and validation, Applied Thermal Engineering, 29, 2009, 2022–2027.

[9] P.V. Neilsen, Specification of a two-dimensional test case, International energy agency, Energy conservation in buildings and community systems, 1990 (Annex: 20, air flow patterns within buildings).

[10] B. Tripathi ,and S. G. Moulic, Investigation of the buoyancy affected airflow patterns in the enclosure subjected at the different wall temperatures, Energy and buildings, 39, 2007, 906-912.