IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
CETCME-2017
"Cutting Edge Technological Challenges in Mechanical Engineering"
Noida Institute of Engineering & Technology (NIET), Greater Noida
Vol. 4, Special Issue 3, February 2017

Investigation of Air Circulation and Temperature Distribution in Car Compartment

Ajay Giri[1], Brajesh Tripathi[2], H.C. Thakur[3]

Research Scholar, Department of Mechanical Engineering, School of Engineering, GBU, Greater Noida,India [1]

Associate Professor, Department of Mechanical Engineering, Rajasthan Technical University Kota, India[2]

Assistant Professor, Department of Mechanical Engineering, School of Engineering, GBU, Greater Noida, India[3]

Abstract: Temperature inside the cabin of a car in summer season gets very high in parking condition which is uncomfortable to driver as well as to passengers boarding in it. In this paper an investigation has been done for air-circulation and temperature distribution in compartment of a car with different inlet velocities considering an inlet vent in front and an outlet in rear of the car. It is observed that air circulation is more efficient in rear compartment than front compartment of the car with increase in inlet velocity of air. Temperature distribution gets better with increase in the inlet velocity and overall cabin temperature is reduced with providing vent in the cabin.

Keywords: thermal comfort, CFD, passenger cabin.

I. INTRODUCTION

Passenger thermal comfort and reduction in energy consumption because of increasing price of fuels is main criterion in design stage of an automotive vehicle. In major parts of India ambient temperature in summer rises in between 30-40^{0}C, this high temperature causes rise in cabin temperature of car up to 60-70^{0}C in parking condition. This creates discomfort to passengers and driver in early part of the journey after they enter in the cabin. On the other hand cooling load of the air conditioning system gets increased.

This high rise in temperature of cabin with no ventilation can fatal to life of a child also when left inside due any reason because all openings are closed in parking condition and heat is absorbed by air and materials of interior of the compartment. In this study air circulation and temperature distribution in compartment is investigated by considering an inlet and outlet vent within the compartment.

II. LITERATURE REVIEW

Wakashima [1] found that increase in temperature inside the cabin gets high with time, particularly in the summer season during parking a car for long duration. Levinson et al. [2] observed that vehicle thermal loads for air conditioning system are greatly affected by radiation of solar energy on the car and showed that there would be decrease in the ''soak'' temperature of the air in the car compartment on application of solar reflective coatings on the body of the vehicle shell. Giri et al. [3] observed that heat accumulation and ventilation in compartment of a car during parking is very prominent problem and they suggested that there must be an active or passive ventilation system in an automotive compartment to save the lives and reduce fuel consumption in vehicle.

Although interior of a car has very complex geometry and three dimensional, but with a two-dimensional numerical analysis we could find many important characteristics of air flow. This investigation has been performed considering 2-D air flow for fixed inlet and outlet with varying inlet velocities.

III. PROBLEM FORMULATION

It is assumed that flow inside the compartment is laminar due to low inlet velocities. A two dimensional, incompressible, steady flow is considered for the numerical analysis. Boussinesq's approximation has been also considered. Assuming all physical properties constant, 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.

A. Governing Equation

Governing equations used for the study of air circulation and temperature distribution in car compartment are as given below.

Continuity equation (Conservation of mass):

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

Conservation of x-momentum:

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

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
CETCME-2017
"Cutting Edge Technological Challenges in Mechanical Engineering"
Noida Institute of Engineering & Technology (NIET), Greater Noida
Vol. 4, Special Issue 3, February 2017

Conservation of y-momentum:

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

Energy conservation equation:

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

Where, ϕ = dissipation function of energy
υ = kinematic viscocity, m^2/s
ρ = density, kg / m^3
C_p = sp. heat at constant pressure, kJ/ (kg×K)

B. Geometry and Boundary Conditions
Analysis of air circulation and temperature distribution is performed on the interior of a small ALTO™ car as shown in fig.1.The size of inlet vent (in front of car)is 'w' and that of outlet vent (in rear of car) is't' and size of vents are 0.15m for both inlet and outlet.

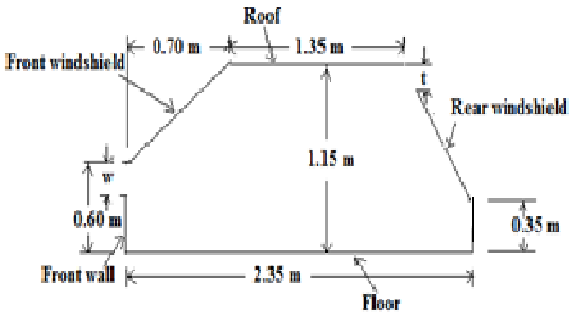


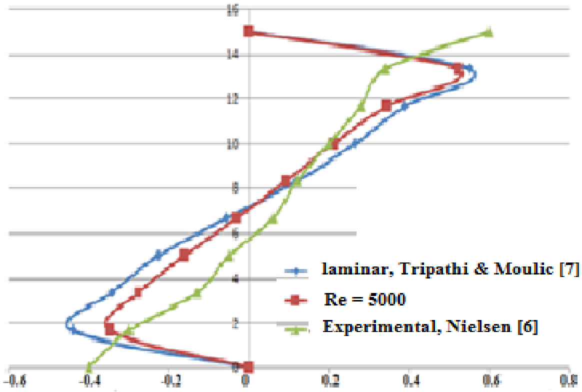
Fig1. Car geometry

Mixed (convection and radiation) boundary condition is taken forfront and rear windshield with free stream temperature as 40^0C, external emissivity as 0.88 and external radiation temperature as ambient temperature 40^0C. The value of convective heat transfer co-efficient is calculated by the formula [4].

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

Where, v = wind speed relative to car, m/s.
The boundary conditionfor the roof, front wall and floor has been taken as constant heat flux of 202 W/ m^2, 112W/ m^2 and 35W/ m^2 respectively as per the experimental data by Chien et al. [5]. Inlet boundary is velocity inlet with temperature 40^0C and Outlet boundary is pressure outlet with zero gauge pressure.

C. Numerical Solution Procedure
The computational domain is meshed by mapped face meshing into triangular cells of 20523 elements and 10530 nodes and refinement of grids has been done at inlet and outlet boundary. A grid-independence test has been conducted to ensure accuracy of the results. Validation of the numerical results has been done with experimental results reported by Neilsen [6] and numerical solution of Tripathi and Moulic [7] and results are shown in fig.2.



X-axis: u/U (non-dim velocity), Y-axis: y/w
Fig 2. Comparison of velocity profile at x = 2/3 L.

The inlet and outlet were located at top of front windshield and bottom wall of rear goods cabin respectively with inlet velocity 0.455 m/s. Comparison of x-component of velocity (u) at vertical cross-section of rear compartment (x=2/3 L) is done with standard solutions and it shows that results are in good agreement.

IV. RESULTS AND DISCUSSION

An investigation has been done for airflow and temperature field in compartment of a car with different inlet velocities. An inlet vent in front and an outlet in rear of the car have been considered for the analysis and computed numerical results are shown in figs. 3 to 10.

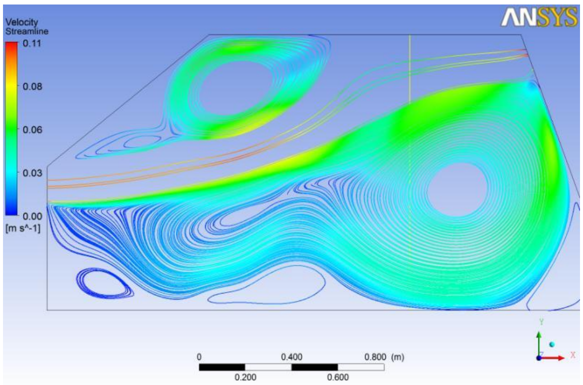


Fig 3. Streamline, v= 0.1 m/s

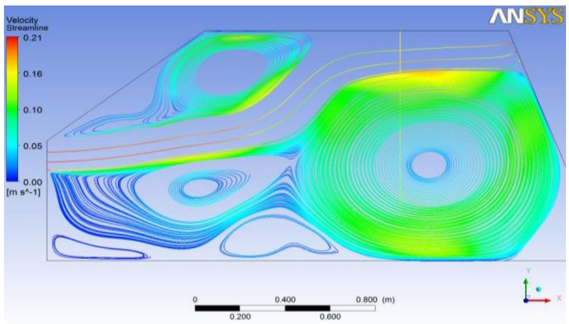


Fig 4. Streamline, v= 0.2 m/s

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
CETCME-2017
"Cutting Edge Technological Challenges in Mechanical Engineering"
Noida Institute of Engineering & Technology (NIET), Greater Noida
Vol. 4, Special Issue 3, February 2017

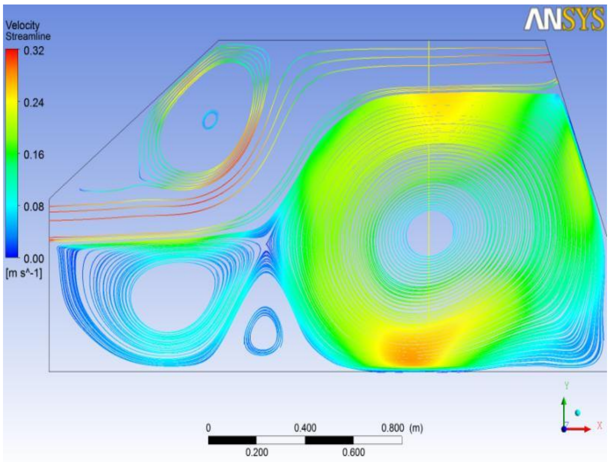


Fig 5. Streamline, v= 0.3 m/s

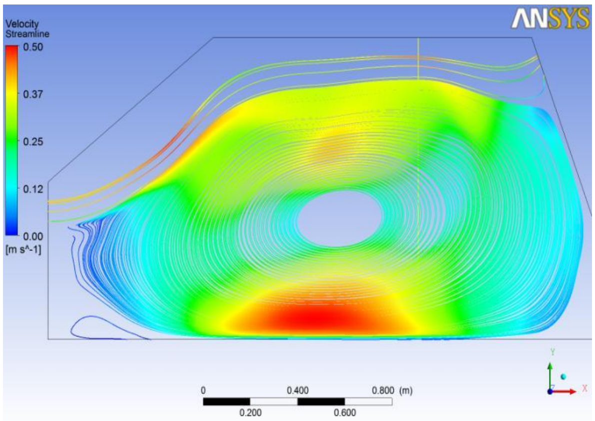


Fig 6. Streamline, v= 0.4 m/s

Figs. 3 to 6 show the streamlines of airflow in the cabin. It is observed that circulation of air is more in rear compartment in comparison to front compartment and circulation of air improves with increase in velocity of inlet air. It is also observed that loop of air movement shifts from rear to front compartment with increase in inlet velocity.

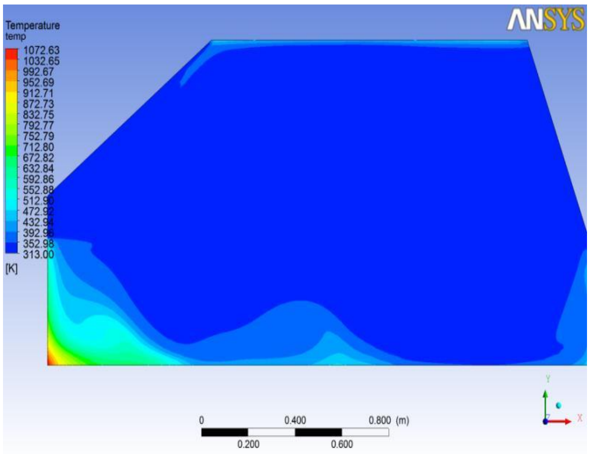


Figure 7. Isotherm, v= 0.1 m/s

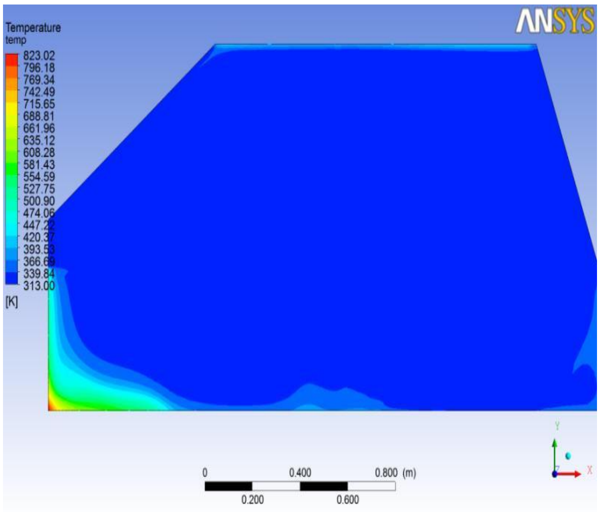


Fig 8. Isotherm, v= 0.2 m/s

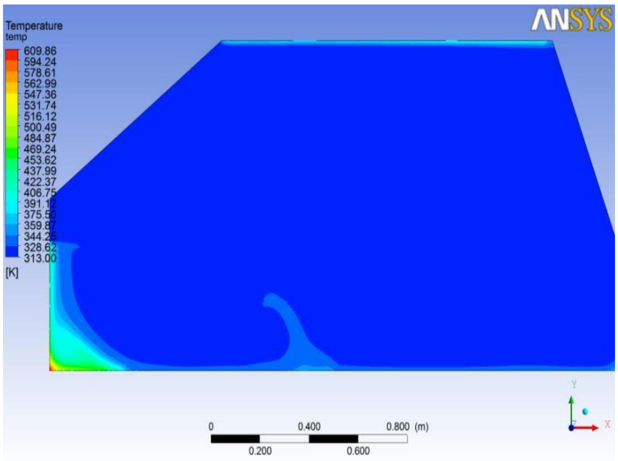


Fig 9. Isotherm, v= 0.3 m/s

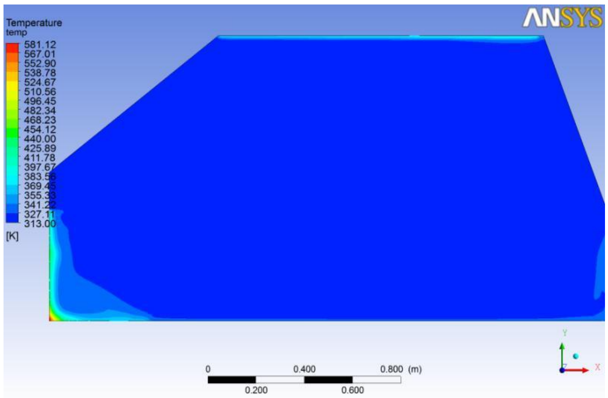


Fig 10. Isotherm, v= 0.4 m/s

Figs. 7 to 10 show the isotherms in the cabin of car. It shows that temperature distribution is uniform in most ofthe parts of cabin except in region of roof and floor. There is also heat accumulation in region of front wall. Uniformity of temperature gets improved with increase in inlet velocity.

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
CETCME-2017
"Cutting Edge Technological Challenges in Mechanical Engineering"
Noida Institute of Engineering & Technology (NIET), Greater Noida
Vol. 4, Special Issue 3, February 2017

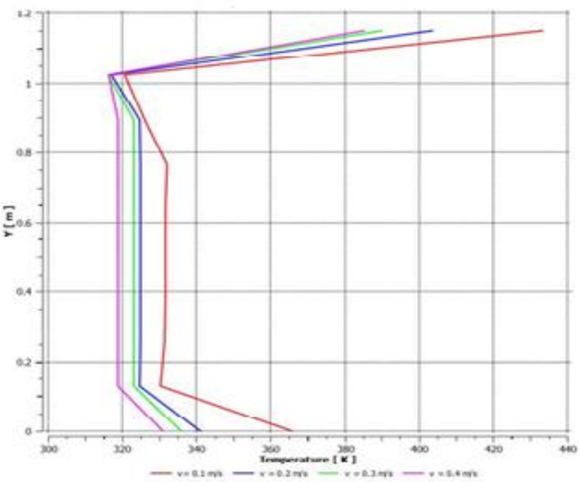


Fig11. Comparison temperature profile in rear compartment (at x = 2/3 L)

Fig.11 shows the comparison of temperature profile at vertical cross-section of rear passenger seat in rear compartment. It shows that there is uniformity in cabin temperature in between 12 cm above the floor and 12 cm below the roof. In region of Floor and roof it is not satisfactory. There is overall reduction in cabin temperature with increase in inlet velocity and in working zone temperature reduces from 58^{0}C to 46^{0}Cas inlet velocity increases from 0.1 m/s to 0.4 m/s.

V. CONCLUSION

Numerical analysis was performed with considering inlet velocities as 0.1 m/s, 0.2 m/s, 0.3 m/s and 0.4 m/s. It is found that air circulation in rear of the compartment is good in comparison to front part and it gets improved with increase in inlet velocity. Temperature distribution is also improved with increase in inlet velocity. Results shows that air circulation and temperature reduction in the compartment could be more efficient with proper design of vents.

REFERENCES

[1] Shinichiro Wakashima, Developemnt of the Thermal Environment Simulator of an Automobile, 47th Japan Heat Transfer Symposium proceedings (2010).

[2] Ronnen Levinson , Heng Pan, George Ban-Weiss, Pablo Rosado, Riccardo Paolini , Hashem Akbari, Potential benefits of solar reflective car shells: Cooler cabins, fuel savings and emission reductions, Applied Energy 88, 2011, 4343–4357.

[3] Ajay Giri, Brajesh Tripathi, and H.C. Thakur, "Passenger Thermal Comfort in a car: a Review,"JOBARI, 8(1), 2015, 1-10.

[4] Huajun Zhang, Lan Dai, Guoquan Xu, 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.

[5] 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.

[6] 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).

[7] 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.