Scientific Journal of Impact Factor(SJIF): 3.134
e-ISSN(O): 2348-4470
p-ISSN(P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 2, Issue 4, April -2015

CFD analysis of modified car air distribution system by changing supply and return air vents

Madhubala Singh[1] & Dr. V N Bartaria[2,]

[1]P.G. Scholar, [2]Professor & HOD, Department of Mechanical Engineering, Lakshmi Narain College of Technology, Bhopal, India

Abstract: *A numerical study of a three-dimensional, turbulent, recirculating flow within a passenger car cabin is presented. The study is based on the solution of the partial differential equations representing conservation of mass, momentum, temperature, turbulence energy and its dissipation rate in finite volume form. Different parameters are considered to illustrate their influences on the flow filed and temperature distribution inside car cabin. These parameters include number and location inside car cabin, different air temperatures at the inlets and outlet, different air velocities at the inlets and outlet. This paper focuses on the numerical study of the temperature field and air flow inside a passenger's cabin using computational fluid dynamics (CFD) method.*

Generally, the results indicate some of negative effects such as development of zones of high air circulation. Also it is found that the location of inlets inside car cabin play an important role in determining car air conditioning system efficiency. Moreover, the air temperature and velocity at inlets play an important role in determining cabin climate. The results are used to enhance the understanding of the airflow fields within a road vehicle passenger cabin.

KEYWORDS: *Computational Fluid Dynamics, Air Conditioned Car, Air Flow Analysis, Passenger Cabin.*

1. INTRODUCTION

The main functions of a vehicle cabin are to provide a comfortable environment for its occupants and to protect them from vibrations, noise and other adverse influences. The internal temperature-humidity conditions are an important factor for the comfort and health of passengers, and also for the safety of drivers. Simulation of passenger compartment climatic conditions is becoming increasingly important as a complement to wind-tunnel and field testing to help achieve improved thermal comfort while reducing vehicle development time and cost. Thermal analysis of a passenger compartment involves not only geometric complexity but also strong interactions between airflow and the three modes of heat transfer, namely, heat conduction, convection, and thermal radiation.

Numerical simulations of a two-dimensional, and a three-dimensional airflow in a passenger compartment were performed by Hara *et al.* [5]. Also, the effect of four HVAC design parameters on passenger thermal comfort was analyzed in a simplified passenger compartment using CFD [Lin *et al.*, 10]. 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. A study by Ishihara *et al.* [6] examined the airflow inside a one-fourth scale three-dimensional model. Lee *et al.* [8, 9] stressed on temperature distribution characteristics of automobile interior both numerically and experimentally when operating the HVAC system in the summer. In the present study, the flow field and temperature distribution within a 2D full scale model of a vehicle compartment are investigated using CFD to determine the capability of the method. Thermo-graphy and temperature measurements by K-type probe are presented and evaluated.

2. NUMERICAL ANALYSIS

The CFD, Computational Fluid Dynamics, software identifies the method which, through numerical algorithms, leads to the solution of the equations which modelizes the laminar, or a fluid's turbulent motion and of the related thermo-dynamic processes.In this work, FLUENT 6.3 version software package was used for numerical analysis of air flow and heat transfer in the automobile cabin. Fluent software solves continuum, energy and transport equations numerically with natural convection effects. In numerical solution, second-order discretization method was used for convection terms and SIMPLE algorithm was chosen for pressure velocity coupling. In the numerical analysis, a realizable k-ε model for modelling the turbulent flow is used. This turbulence model is generally used for such calculations due to stability and precision of numerical results. The realizable k-ε turbulence model is derived from the instantaneous Navier–Stokes equations, using a mathematical technique. This model is different from standard k-ε model, additional terms and functions in the transport equations for k and ε. In the computational domain, 3-D tetrahedral mesh was generated which contained triangular elements at the surfaces of the cabin parts and tetrahedral elements in the central-volume region.

2.1. Modelling Geometry:

The simplified two dimensional geometry of an automobile compartment was generated in SOLID WORK. The numerical model is shown in Figure 1. The simplified geometry represents the actual dimensions of a car are in mm (manufactured by Skoda suparb) compartment and seating arrangements. The tetrahedral volume mesh of the model is illustrated in Figure 2.

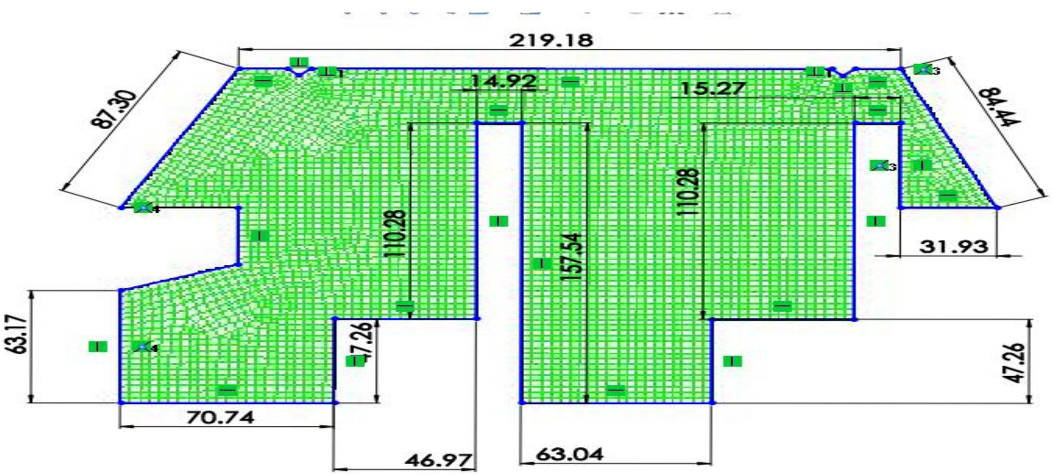

Figure.1 2D car cabin model

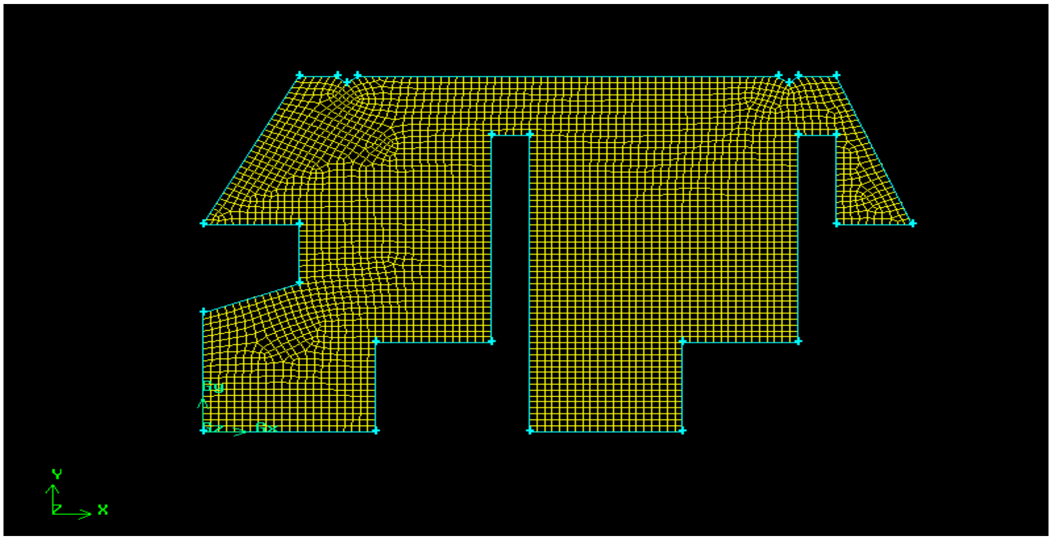

Figure.2 the section view of the computational grid

2.2. Meshing Structure:

In numerical calculations, mesh structure of the computational domain is very important for getting predicted results in good accuracy and reducing computing time. The volume of the seats are excluded from the meshing process since they were treated as solid bodies. Car model was meshed by using Gambit CFD software. In this study, 3-D tetrahedral mesh was used in the present computations. This mesh structure contains triangular elements on the surfaces, tetrahedral elements in the volume region. The computational grid used in this study consists of about 3818volume cells. The cross sectional view of volume cell is shown in Figure 3.

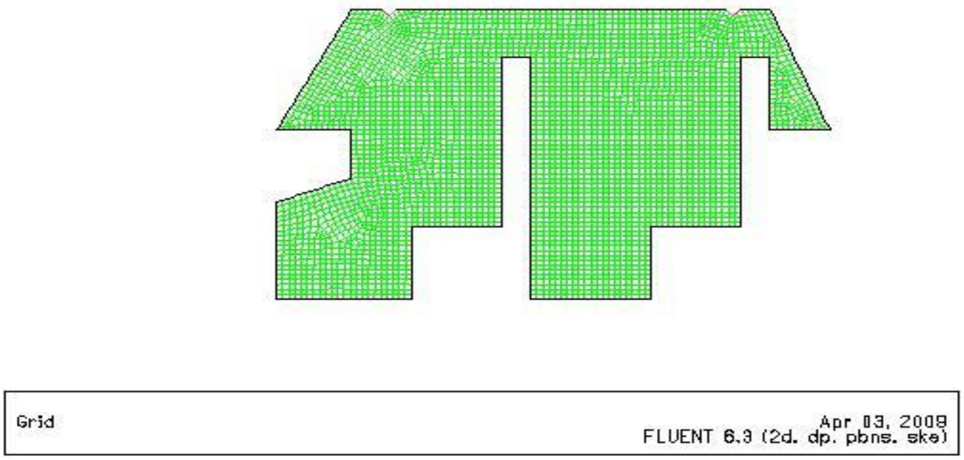

Fig. 3 Overall computational mesh (3818cells) of the model.

2.3. Governing Equation:

i- Continuity Equation (Mass Conservation)

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

ii- Momentum Equation (Navier-Stokes Equation)

U-momentum (x-direction)

$$\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial x}(\rho uv)=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial x}\right) \tag{2}$$

V-momentum (y-direction)

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial y}\right) \tag{3}$$

iii-Temperature Equation

$$\frac{\partial}{\partial x}(\rho uT) + \frac{\partial}{\partial y}(\rho vT) = \frac{\partial}{\partial x}\Gamma_{eff}\left(\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\Gamma_{eff}\left(\frac{\partial T}{\partial y}\right) + S_T \qquad (4)$$

2.4. Boundary Conditions:

The CFD analysis on the virtual model of the car's passenger compartment was performed in order to investigate the conditions of the air-velocity and temperature distributions in the compartment under a given set of conditions. To do this, certain boundary conditions were prescribed on the passenger compartment model involving four parameters namely air pressure, air temperature, heat gain through the compartment's walls and the air velocity at the air-conditioning inlet vents. The air velocity of inlet vents were specified as 8 m/s and temperature was set as 293 K. Convective boundary condition was considered on the glass surfaces and outer surfaces of the cabin. The most of the region of occupied is within the comfort range. The glasses are considered to increase the heat load in the spaces through solar heating with heat flux of 300W/m2.

Velocity at inlet	Temperatures				
	Inlet	Roof	Glass(Heat Flux)/Temp	Seating	Other wall
8 m/s	293K	300K	300W/m2/315K	305K	305K

The reported mass flow at inlets is 0.69 kg/s and heat transfer rate from the passenger sitting space is 33.94W.

3. RESULTS AND DISCUSSIONS

Vertical XY plane and horizontal XZ plane of air flow distribution and temperature distribution results were predicted for all human loads. High temperature values were computed near the ceiling surfaces which were more affected from the solar radiation. In the front region of the vertical plane, a computed temperature values changed between 289 K and 294 K and on the rear region the values ranged from 291 K to 297 K.

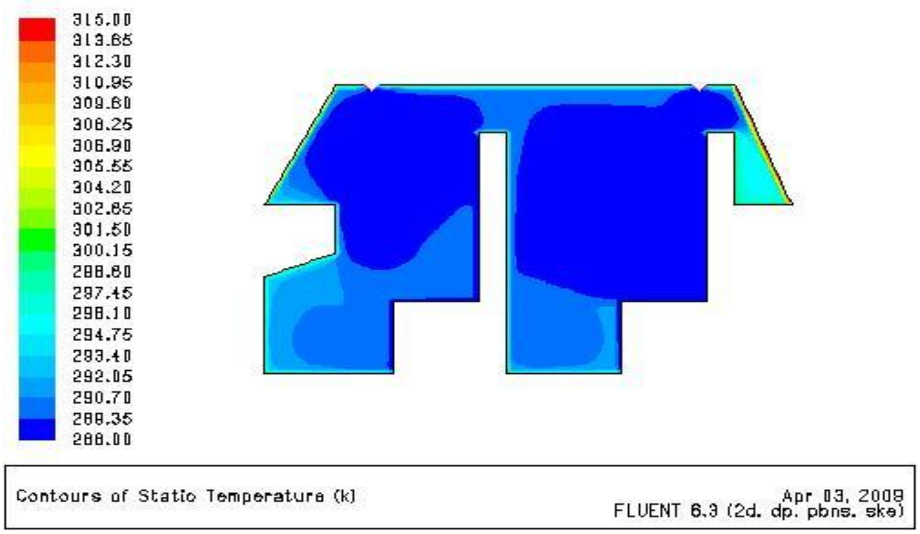

Figure.4 Temperature distributions for front and rear compartment

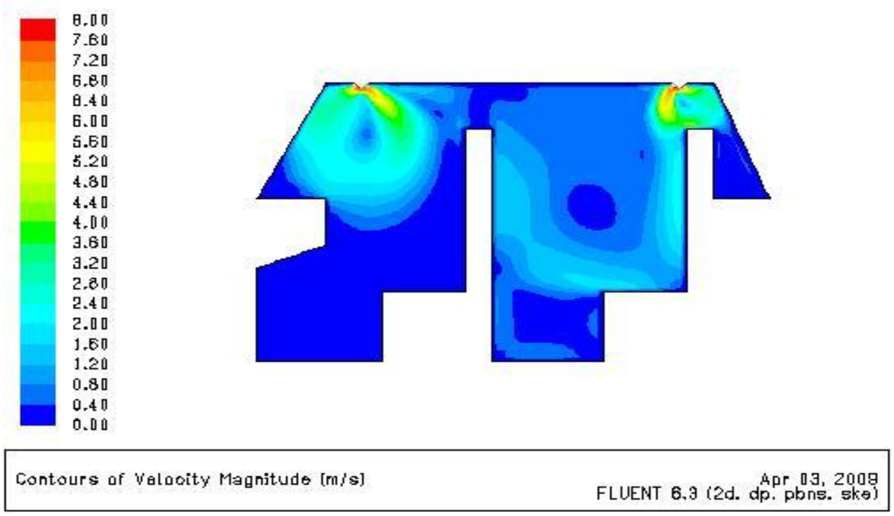

Figure.5 Velocity distributions for front and rear compartment

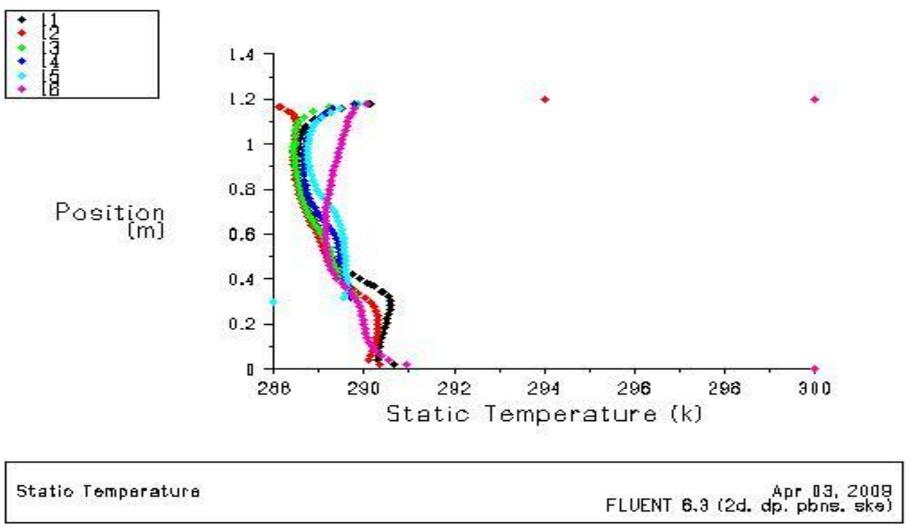

Figure.6 Comparison of temperature distribution values for test location 1-6

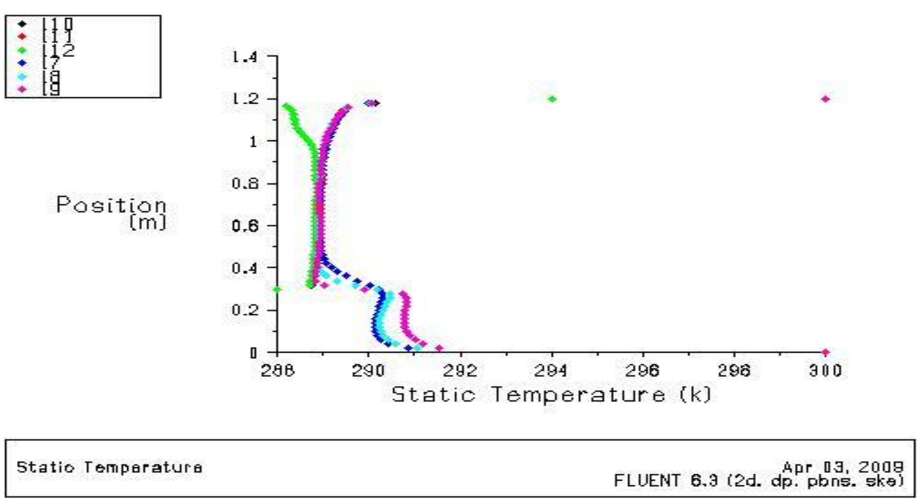

Figure.7 Comparison of temperature distribution values for test location 7-12

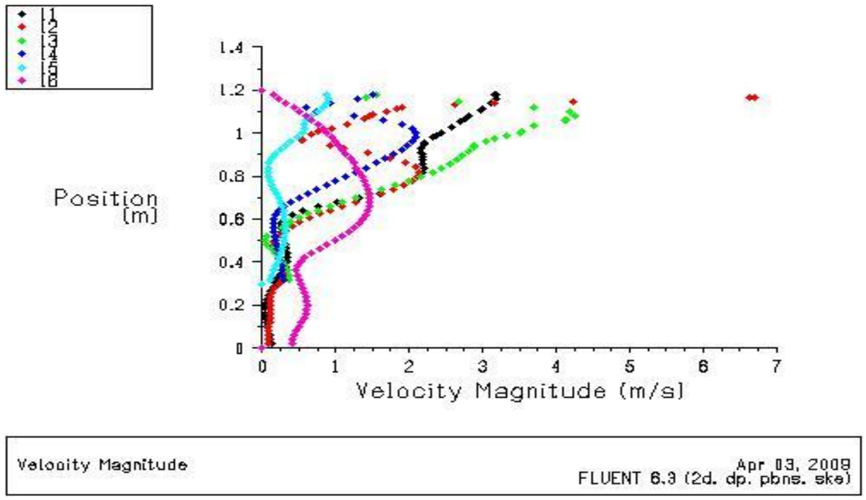

Figure.8 Comparison of velocity distribution values for test location 1-6

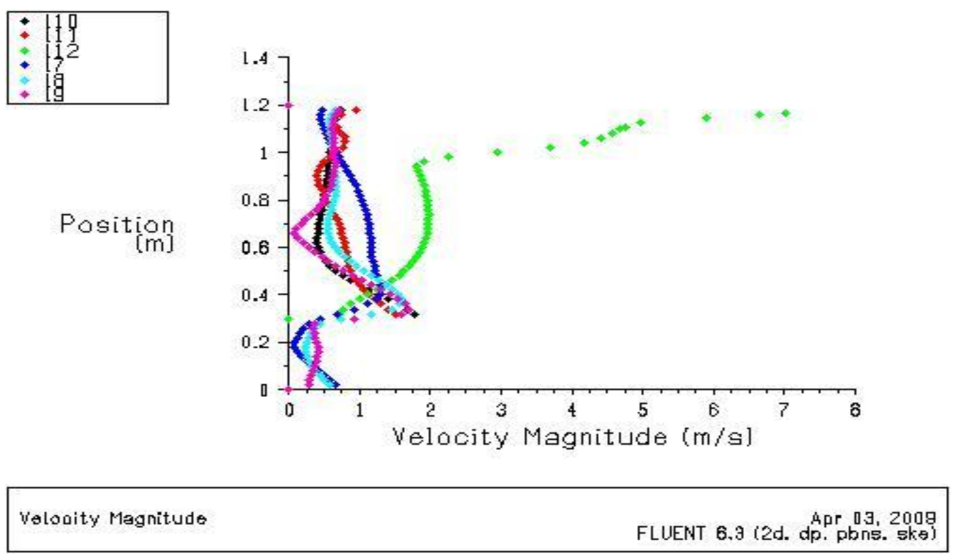

Figure.9 Comparison of velocity distribution values for test location 7-12

4. CONCLUSION

The temperature distribution and air flow distribution were simulated in passenger car cabin using CFD analysis in this current investigation.it was observed that the predicted values are very closer to experimental values It revealed that the developed CFD model accurately predicted the temperature distribution and air flow distribution in passenger car cabin. The systematic CFD method proposed in this present investigation may be used to study temperature distribution and air flow distribution for different car models. The present method may also be used for optimization of supply and return air vent for occupant comfort.

5. FUTURE WORKS

The focus of the present study was the optimization of air flow distribution inside vehicle model. The vent arrangements, energy consumption, and thermal comfort level were investigated for the worst case scenario. In the present dissertation work numerical simulation of thermal environment is obtained. In future more work for energy conservation and other design aspects may be studied e.g. better diffuser and better location of supply and return air vent may be studied for energy conservation. Design optimization of supply and return air vents can be done to get better air distribution inside the cabin, especially at front side of cabin.

6. REFERENCES

[1] Rameshkumar.A1, Jayabal.S2, and Thirumal.P3 , "Cfd Analysis Of Air Flow And Temperature Distribution In An Air Conditioned Car"International Refereed Journal of Engineering and Science (IRJES), Volume 2, Issue 4(April 2013).

[2] Mingyu Wang, Edward Wolfe, and Debashis Ghosh, Jeffrey Bozeman, Kuo-huey Chen,and Taeyoung Han, Hui Zhang and Edward Arens, "Localized Cooling for Human Comfort" SAE Int. J. Passeng. Cars - Mech. Syst. / Volume 7, Issue 2 (August 2014).

[3] Gokhan Senilgen & Muhsin Kilic, "Three dimensional numerical analysis of temperature Distribution in an Automobile cabin", Volume -16, 2012.

[4] Debashis Ghosh and Mingyu Wang, Edward Wolfe, Kuo-huey Chen, Shailendra Kaushik and Taeyoung Han " Energy Efficient HVAC System with Spot Cooling in an Automobile "Ghosh et al / SAE Int. J. Passeng. Cars - Mech. Syst. / Volume 5, Issue 2(June 2012).

[5] Acta Technica Corviniensis Bulletin of engineering "Improvement of thermal comfort in a passenger car by localized air distribution" ISSN. 20673809 , 2011 (January-March)

[6] Haslinda Mohamed Kamar, Nazri Kamsah,and Ahmad Miski Mohammad Nor,"Numerical Analysis of Air-Flow and Temperature Field in a Passenger Car Compartment", April 2008.

[7] Zeya Ahmad Quadri & Jomon Jose, "Computional analysis of thermal distribution within passenger car cabin", ISSN. 2319-3182, Volume-2, Issue-2, 2013.

[8] M.A. Jasni and F.M. Nasir, "Experimental Comparison Study of the Passive Methods in Reducing Car Cabin Interior Temperature", International Conference on Mechanical, Automobile and Robotics Engineering (ICMAR'2012).

[9] Jalal M.Jalil,Haider Qassim Alwan,*CFD Simulation for a Road Vehicle Cabin*,JKAU:Eng Sci,Vol.18 no.2,123-142,2007.

[10] Ishihara Y., Hara J., Sakamoto H., Kamemoto K. and Okamoto H.: Determination of flow velocity distribution in a vehicle interior using a visualization and computation techniques, SAE paper No. 910310(SP-855), pp 51-62, and 1991.

