International Refereed Journal of Engineering and Science (IRJES)
ISSN (Online) 2319-183X, (Print) 2319-1821
Volume 2, Issue 4(April 2013), PP.01-06
www.irjes.com

Cfd Analysis Of Air Flow And Temperature Distribution In An Air Conditioned Car

Rameshkumar.A[1], Jayabal.S[2], and Thirumal.P[3].
[1]P.G. Scholar, [2]Assistant Professor, Department of Mechanical Engineering.
A.C.College of Engineering and Technology. Karaikudi-630004.
[3]Assistant Professor, Department of Mechanical Engineering.
Government College of Engineering. Bargur-635104.

Abstract: *Air conditioning is widely applied for the improvement of standard of living in human life. The human comfort conditions were affected by various Indoor Air Quality parameters in an air conditioned space. The prediction methods for an air conditioned system evaluates the indoor environmental conditions of a specific location and selects the most appropriate actions so as to reach the set points and contribute to the indoor environmental quality by minimizing energy costs. This paper focuses on the numerical study of the temperature field and air flow inside a passenger's cabin with different human load using computational fluid dynamics (CFD) method. The main goal is to investigate the distribution of temperature and air flow with various human loads inside the passenger compartment in the steady-state conditions.*

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

I. INTRODUCTION:

Air flow and temperature will be varied due to various human load inside the cabin, so it essential to study the human thermal comfort. More numerical and experimental studies should performed under different human loads to get how air flow and temperature distribution inside a car cabin. In previous literatures, numerical and experimental studies for the air flow and temperature was carried out for limited human load. ASHRAE standard 55 defines thermal comfort as "that state of mind which expresses satisfaction with the thermal environment" [1]. The principal factors affecting human thermal comfort are the air temperature, its relative humidity, the mean radiant temperature and the relative air velocity.The thermal comfort is a complicated problem because it is related to both psychological and physiological factors, the air-flow and temperature fields are identified as the most important factors. This makes it necessary to investigate the air-flow field and the temperature distributions inside the passenger compartment in the design process or to improve the thermal comfort conditions for the passengers. Comfortable vehicular climate control in many cases not only help to reduce the driver stress but also guarantee good visibility by avoiding the fogging phenomenon, hence contributing to safer driving experience. The need to reduce the heat loads that enter passenger compartments has become an important issue in the early stages of vehicle design. An improved thermal comfort system will lead to substantial cost reductions. Han [2] used a three-dimensional model of a passenger car compartment and numerically simulated the thermal environment inside the compartment. Kilic and Sevilgen [3] 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 [4] measured the air-flow velocity distributions in the car compartment by visualization and then compared them with their numerical simulation results. Aroussi and Aghil [5] 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 [6] 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 [7] 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 [8] 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. [9] 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 [10] developed a numerical model that integrates solar load when computing the temperature distribution in a passenger cabin. This present study is focused on comparison of numerical and experimental values for different human loads in distribution of air-temperature and velocity inside the passenger car cabin in the steady-state conditions.

II. EXPERIMENTAL SETUP

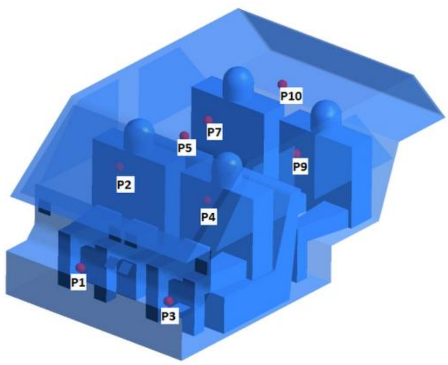
The Experimental tests were performed in passenger car at steady state conditions. During the experiments, both internal and external temperatures and air velocity of inlets measurements were taken for different human loads. The console vents were fully opened and all measurements were taken in a parked automobile with varying human load inside. The experimental setup is shown in Figure 1.

Figure.1 Photographic image of Experimental set up

The interior air temperature distribution inside the cabin was recorded by IAQ probe (measuring range: -55ºC to +150ºC). It has the following sensors like temperature, relative humidity, carbon dioxide, carbon monoxide and oxygen. The measurement points are given in Table 1. The test conditions were achieved after the automobile was kept idle for two hours in the outer environment which had high temperature value.

Table.1 Location of measuring points inside the cabin

Points	Locations
P1	Knee level (right front)
P2	Chest level (right front)
P3	Knee level (left front)
P4	Chest level (left front)
P5	Head level (between two front seats)
P6	Knee level (right rear)
P7	Chest level (right rear)
P8	Knee level (left rear)
P9	Chest level (left rear)
P10	Head level (between two rear seats)



III. NUMERICAL SIMULATION:

In this work, FLUENT 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.

3.1 Modelling Geometry: The 3D car cabin was modelled by using design modeller (ANSYS) software. Isometric view of the passenger car cabin with four human loads is shown in Figure 2.

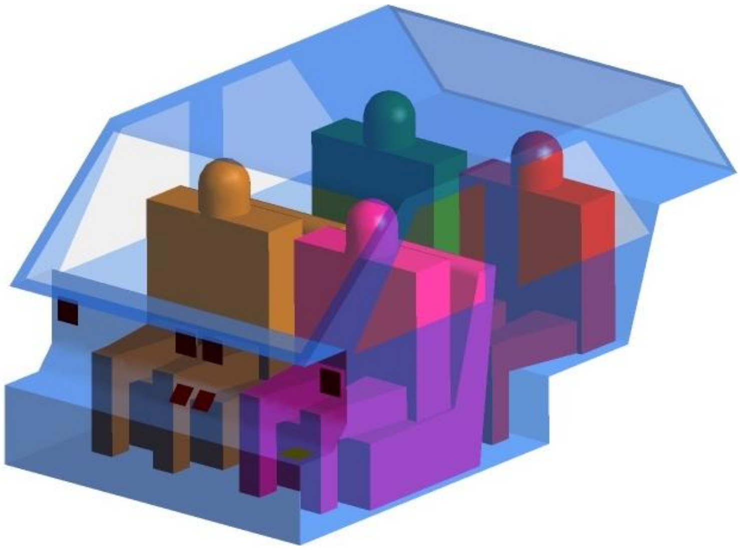


Figure.2 3D car cabin model

A datum vertex is located at the bottom right of the compartment and the width of the model is 1.45 m in the positive z-axis direction. The construction of all curve surfaces of the human body was assumed as flat surfaces except for the head and neck. The dimensions of the human models were based on a male person with body weight of 65 kg and medium height. The human models were set at a 90^0 posture.

3.2 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 human models and seats were excluded from the meshing process since they were treated as solid bodies. Car model was meshed by using ICEM 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 surface mesh had a maximum skewness of 0.80 and the volume mesh had 0.92. The computational grid used in this study consists of about 15,00,000 volume cells. The cross sectional view of volume cell is shown in Figure 3.

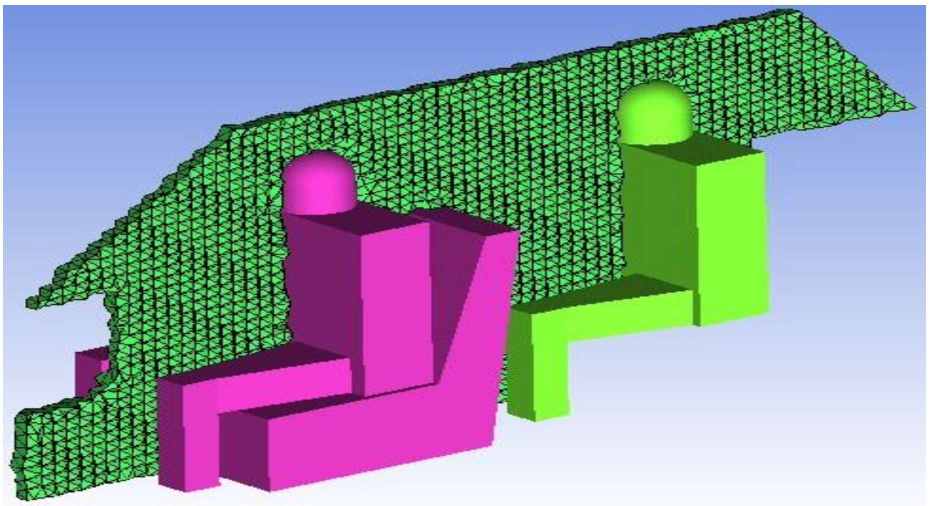


Figure.3 Cross sectional view of volume mesh cell

3.3 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 288 K for all human loads. Convective boundary condition was considered on the glass surfaces and outer surfaces of the cabin. Convective heat transfer coefficient at outside of the cabin was set as 30 w/m²ºC.

Table.2 The computed values of temperature

Points	Locations	Human load 1	Human load 2	Human load 3	Human load 4
P1	Knee level (right front)	292.84	292.08	294.09	293.85
P2	Chest level (right front)	292.81	292.08	293.99	292.61
P3	Knee level (left front)	292.38	293.64	292.75	294.50
P4	Chest level (left front)	291.84	292.60	291.94	293.00
P5	Head level (between two front seats)	294.44	293.76	294.88	294.45
P6	Knee level (right rear)	293.72	293.92	293.93	294.99
P7	Chest level (right rear)	292.34	292.72	293.00	293.47
P8	Knee level (left rear)	292.49	292.77	292.82	293.27
P9	Chest level (left rear)	292.45	292.83	292.20	292.87
P10	Head level (between two rear seats)	293.05	293.34	292.91	293.54

The free stream temperature is the outside temperature was set to 318 K for outside surfaces and glass surfaces. In this study, heat gain from the occupants was not considered because the calculation is quite complex. The humans and seats were simply assumed as solid bodies. After prescribing the above boundary conditions on the meshed model of the passenger compartment, the model was submitted for the CFD analysis. Computed values of temperature and air velocity are shown in Table 2 and Table 3 respectively.

Table.3 The computed values of velocity

Points	Locations	Human load 1	Human load 2	Human load 3	Human load 4
P1	Knee level (right front)	1.08	0.98	1.08	1.11
P2	Chest level (right front)	0.36	0.84	0.46	0.33
P3	Knee level (left front)	2.31	2.44	2.25	2.54
P4	Chest level (left front)	0.44	0.73	0.40	0.35
P5	Head level (between two front seats)	0.94	0.75	0.86	0.67
P6	Knee level (right rear)	0.28	0.24	0.11	0.08
P7	Chest level (right rear)	0.28	0.33	0.54	0.25
P8	Knee level (left rear)	0.34	0.16	0.29	0.18
P9	Chest level (left rear)	0.30	0.37	0.35	0.29
P10	Head level (between two rear seats)	0.45	0.46	0.33	0.41

IV. RESULTS AND DISCUSSION:

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 glazing or ceiling surfaces which were more affected from the solar radiation for all human loads.

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. The simulation result of temperature distribution for the human load of 4 is shown in Figure 4. In the front region of the vertical plane, computed velocity values changed between 3.14 m/s and 0.65 m/s. On the rear region the values ranged from 1.34 m/s to 0.18 m/s for the human load of 4. The simulation results of air flow distribution for the human load of 4 are shown in Figure 5. The above procedure was followed for the human load of 3, 2 and 1 and the simulation results for temperature and air flow were obtained. The comparison of predicted and experimental values of temperature distribution and air flow velocity for varying human loads are shown in Figures 6 and 7 respectively.

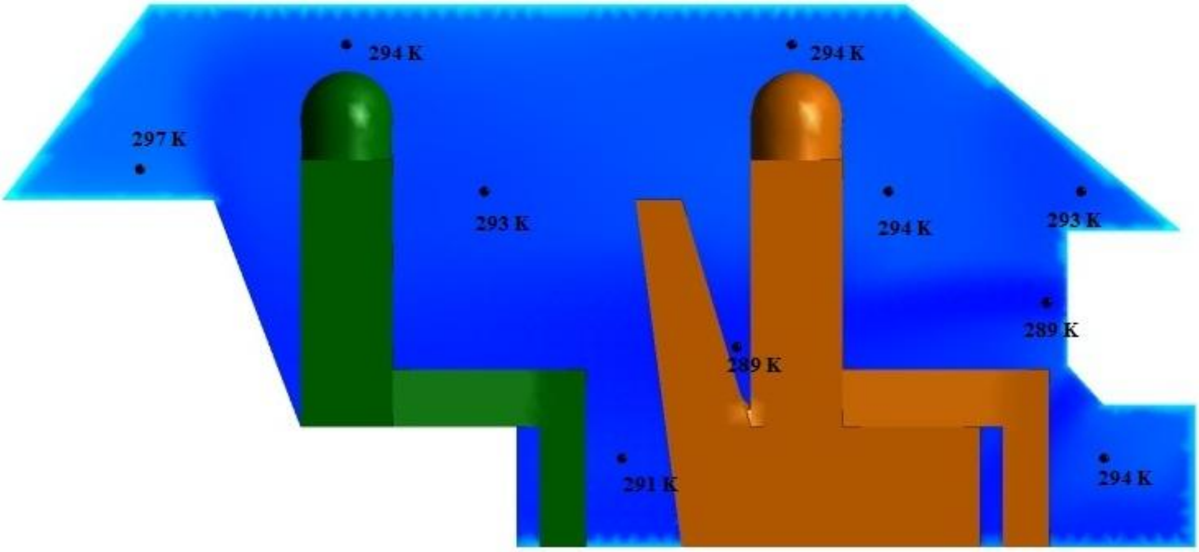


Figure.4 Temperature distributions for human load of 4

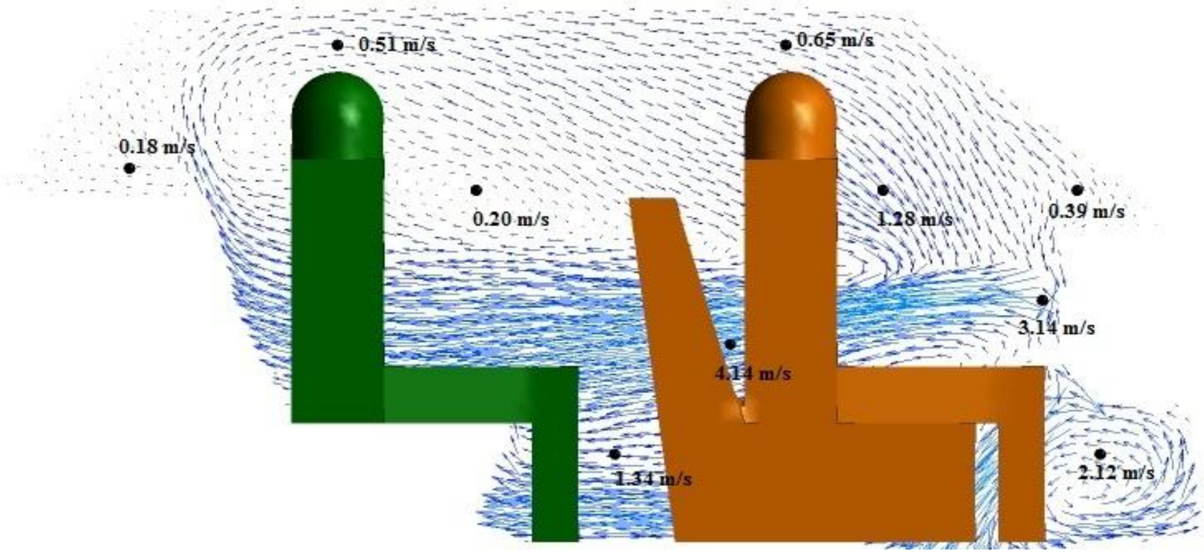


Figure.5 Velocity distributions for human load of 4

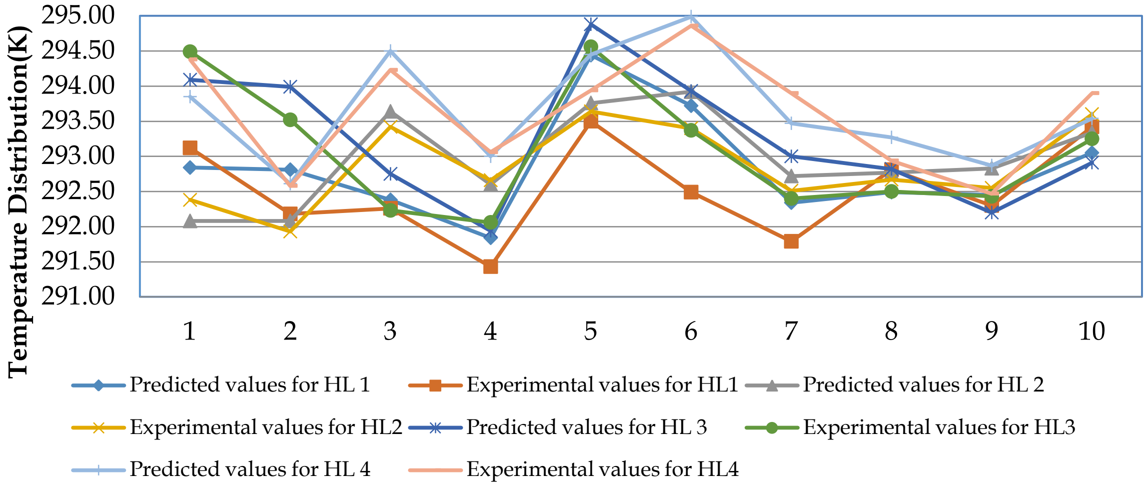


Figure.6 Comparison of temperature distribution values for varying human loads

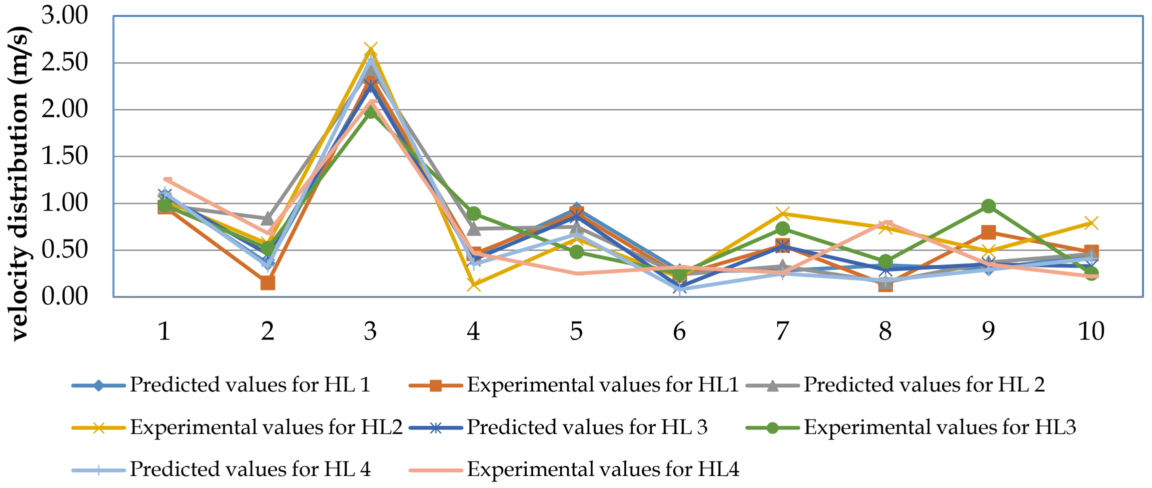


Figure.7 Comparison of velocity distribution values for varying human loads

V. CONCLUSION:

The temperature distribution and air flow distribution were simulated in passenger car cabin using CFD analysis in this current investigation. From the comparison graph 6 and 7, 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 is may be used to study temperature distribution and air flow distribution for different car models.

REFERENCES:

1. Aroussi, A., and Aghil, S., Characterisation of the flow field in a passenger car model, *Optical Diagnostics in Engineering*, **4(1),** 1–15 (2000).
2. Currle, J., Numerical simulation of the flow in a passenger compartment and evaluation of the thermal comfort of the occupants, Proceedings of the *International Congress and Exposition*, ISSN: 1054–6693, SAE, Detroit, MI, USA, 149–159 (1997).
3. Hagino, M., and Hara, J., Development of a method for predicting comfortable airflow in the passenger compartment, *Worldwide Passenger Car Conference and Exposition*, ISSN: 0148-7191, SAE, Dearborn, MI, USA, 1–10 (1992).
4. Huajun Zhang., Lan Dai., GuoquanXu., Yong Li., Wei Chen., and Wen-Quan Tao., Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy, *Applied Thermal Engineering* **29,** 2022–2027 (2009).
5. Ishihara,Y., and Hara,J., Determination of flow velocity distribution in a vehicle interior using a visualization and computation techniques , *SAE International Congress and Exposition,* ISSN: 0099-5908, SAE, Detroit, MI, USA, 51–62 (1991).
6. J.W. Wan., and J. van der Kooi., Influence of the position of supply and exhaust openings on comfort in a passenger vehicle, *Int. J. Vehicle Des,* **12,** 588–597 (1991) .
7. Jonas Jonsson., Including solar load in CFD analysis of temperature distribution in a car passenger compartment, M.S Thesis, Lulea University of Technology, (2007).
8. Kilic, M., and Sevilgen, G., Evaluation of heat transfer characteristics in an automobile cabin with a virtual manikin during heating period, *Numerical Heat Transfer, Part A: Applications,* **56,** 515–539 (2009).
9. T. Han., Three-dimensional navier-stokes simulation for passenger compartment cooling, *Int. J. Vehicle De,* **10(2),** 175–186 (1989).
10. *Thermal Environmental Conditions for Human Occupancy,* ASHRAE Standard 55, Atlanta, 1992.