

e-ISSN: 2319-8753 | p-ISSN: 2320-6710

# IJIRSET

International Journal of Innovative Research in  
**SCIENCE | ENGINEERING | TECHNOLOGY**



# INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 10, Issue 4, April 2021

**ISSN** INTERNATIONAL  
STANDARD  
SERIAL  
NUMBER  
INDIA

**Impact Factor: 7.512**

# Experimental and Numerical Investigation of Air Flow inside a Vehicle Cabin Using CFD

Anoshka A<sup>1</sup>, Kingson Samuel R<sup>2</sup>, Nova Aloysius Elton X<sup>3</sup>, Praylin Jerish J<sup>4</sup>  
Ramkumar P<sup>5</sup>.

U.G. Student, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli,  
Tamil Nadu, India<sup>1,2,3,4</sup>

Assistant Professor, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli,  
Tamil Nadu, India<sup>5</sup>

**ABSTRACT:** Nowadays people use cars in huge level in their day-to-day activities. Thermal comfort is an important factor that increases the embarrassment to the passenger; hence, prediction of thermal comfort plays an important role in the current situation. In olden days, conventional physical testing methods were employed for developing the air conditioning system of an automotive vehicle, which is a time-consuming technique. In order to overcome this problem rapid development of Computational Fluid Dynamics (CFD) technique was introduced to develop the air conditioning system of a vehicle and this technique significantly reduces the time required for testing. This present work was mainly focused on the experimental and numerical study of temperature distribution inside the Maruti Suzuki Alto Lxi vehicle cabin by varying the position of air vents namely the Dashboard and Top vent positions respectively. The model of this car was developed by CREO and the temperature distribution is determined using FLUENT software. The results of CFD analysis shows that 1) when air vent is being fixed in dashboard position the temperature of air in the front region is slightly lower than the rear region and 2) when the air vent is fixed at top position both front and rear region gets nearly even distribution of temperature. A comparative study is also made to compare experimental and numerical results. From the comparative study, it is concluded that the thermal comfort is better when the air vent is fixed at top position.

**KEYWORDS:** CFD, Thermal comfort, Temperature distribution, HVAC.

## I. INTRODUCTION

Due to Modern technology, we found lot of changes in cars which we look around in the traffic. The cars are not developed by its outer structure, but also developed in their interiors. The luxury cars are primarily developed for the inner comfort. Their intention is to improve the experience of the passenger within the car.

When we enter into car cabin, initially we feel high temperature, this was the major problem encountered in cars today. This observable fact of Thermal growth is clearly noticeable when the car has been parked in the sun for a few hours. The car consumer is forced to wait for a period of approximately 2-5 minutes to allow the cabin environment to be like the ambient.

When Engine is ideal in a parked car, we can also aware of a temperature rise inside the car because of the heat leakages from the Engine division. Hence comfort pays a major role in the field of improvement and demand in the Automobile Industry. A wonderful HVAC (Heating Ventilation -Air Conditioning) system has to be adapted by the research specialists.

To realize this, information about the thermal distribution inside the car is fundamental. In this research, an automobile cabin model of a five-seated car has been used to analyse the various factors that affect the cabin temperatures and its improvisations to attain thermal comfort level using CFD Fluent Software.

Parameters like Solar Radiations, Engine Temperature, Number of Air Inlets, Number of Air Outlets, location of Air Conditioning Vent and Number of Passengers sitting inside the car will also affect the thermal comfort of a vehicle cabin. In order to simplify the study this investigation is mainly focused on the Locations of AC vents.

The theme of this investigation is to suggest a better cabin model by relocating the AC vent in order to improve the thermal comfort inside a car cabin.

#### Application

- High attentions of vehicle comfort are progressively more demanded by consumers. Hence developing the Heat Ventilation and Aero conditioning system for a new vehicle is a difficult task for climate control engineers, this investigation simplifies the process of those engineers.
- While the car was parked under the sun, more heat is accumulated on the interior and exterior surface of the vehicle. This causes damage to the vehicle parts. Hence if the AC vent is kept near the glass window it affects the efficiency of cooling phenomenon. This investigation helps in selecting the optimum location of AC vents in an automobile.
- This investigation will also afford a fairly accurate explanation for the causes of temperature rise within the car.
- This investigation can be further used in any closed atmosphere where temperature distributions play a very important role.

## II. METHODOLOGY

- Interior dimensions of a car model are measured using conventional techniques.
- According to the measured dimensions, a car model was created by varying vent location using CREO-Modelling software.
- Later the created model is subjected to fine mesh using Hyper mesh software.
- Then the temperature distribution inside the car cabin was analyzed using FLUENT CFD.
- Simultaneously experimental measurement of temperature at various positions inside the car cabin was recorded manually using Infrared thermometer.
- Finally, both the numerical and experimental results were compared and the investigation is concluded.

## III. EXPERIMENTATION

An infrared thermometer was used in this current investigation to analyse the thermal comfort ability inside a car cabin. Experimental setup and treatment were carried out at CMS College of engineering and technology, Coimbatore. The tests were performed on Maruti Suzuki Alto Lxi. Before starting the experiment, inside controlled environment was allowed to attain equilibrium with outdoor conditions. During the experiment all the three vents were fully opened to determine the interior temperature distribution inside the compartment and the measurements were obtained at various position of the cabin. Initially the experiment was conducted with five loads and the results are tabulated. The tests were conducted when the outside environment attains 32°C.

#### Modelling of the Passenger Compartment Region

- ❖ The total space of car consists of computational kingdom and the internal surfaces of the compartment are considered as the boundary conditions. The 3D form of car was equipped through CREO software. The entire volume of cabin model was 6.93 cubic meters. This investigation concentrates in the thermal distribution inside a five seater car.
- ❖ There are two types of vent present in the dashboard of a car, namely outlet air vent and recirculation inlet vent. The recirculation vent was provided to circulate the air inside the passenger compartment. The location of these air vents in a 3D model is shown in Fig.1 & 2. In this investigation both the inlet and outlet vents were included into the model for CFD analysis.
- ❖ Representative models of driver and the passenger were also included into the compartment's model as shown in Fig.3

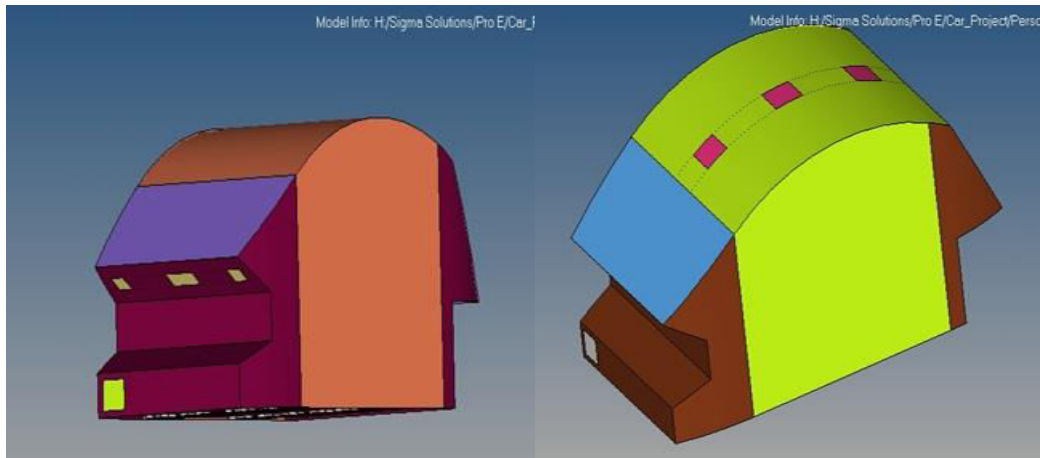


Fig.1: Vents located at Dashboard Fig.2: Vents located at top position

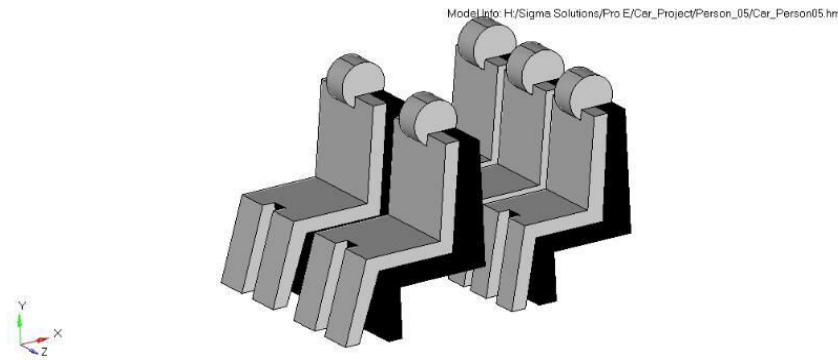


Fig.3: Representative models of human being

The following assumptions were made in this present investigation.

- ❖ There are no VOC emissions inside the cabin.
- ❖ Pollutants like suspended particles are not considered.
- ❖ The conditioned space is free from cigarette smolder, dirt and other foreign particles.
- ❖ Engine heat and outdoor temperature were not considered during the analysis.
- ❖ When the doors were closed, the leakage is neglected.
- ❖ Heat produced due to Solar radiation and Engine are neglected.

#### Meshing in fluent

The meshing of the structure was done by Hyper Mesh software as show in Fig.4 & 5. Unstructured grids are also adopted, and the majority of grids are tetrahedrons. The mesh was given with a spacing of .15 units for correct and standard meshing. Since the cabin model was adopted with non-uniform geometrical structure, tetrahedral meshing was chosen in order to achieve the accurate results than quadrilateral. The meshing furnishes a totality of 47379 tetrahedral cells and 189516 nodes.



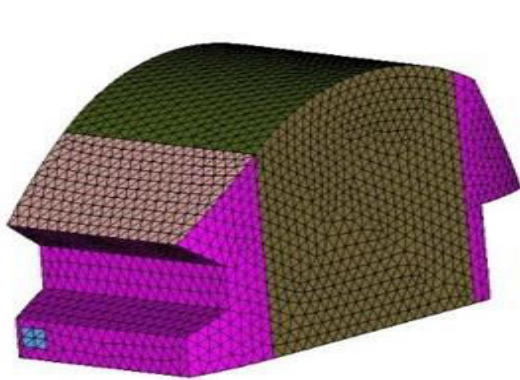


Fig.4: The meshed passenger Car's compartment

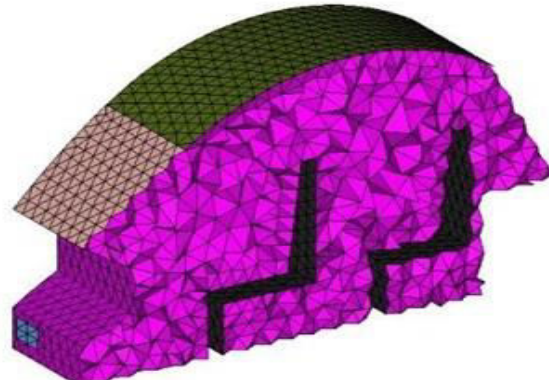


Fig.5: the meshed model of car model

#### Boundary conditions in CFD model

The internal surface of the vehicle was given as boundary conditions of the model. Conditions are mainly applied to eight elements in the car model. They are mentioned as follows:

1) Front glass 2) Back glass 3) Top layer 4) Bottom layer 5) Sidewall 6) Seat 7) Human body 8) car body

Materials are mainly classified in to two main categories. They are solid state and fluid state. Fluid state is mainly of air and water related components, whereas solid state is classified into materials like glass, seat, car body, human body, etc.

#### Glass property:

Density =  $2500 \text{ kg/m}^3$ ,  $C_p = 750 \text{ J/kg-K}$ , Thermal conductivity =  $1.5 \text{ W/m-K}$

#### Seat property

Density =  $1380 \text{ kg/m}^3$ ,  $C_p = 0.21 \text{ J/kg-K}$ , Thermal conductivity =  $0.15-0.4 \text{ W/m-K}$

#### Human body

Density =  $900-1600 \text{ kg/m}^3$ , Thermal conductivity =  $0.29-1.06 \text{ W/m-K}$

Remaining elements were considered as wall property.

Normal temperature of human body was considered as  $37^\circ\text{C}$  and the normal temperature inside the cabin model was considered as  $30^\circ\text{C}$ . Based on these boundary conditions the Simulation was performed and the results are plotted.

### IV. RESULTS AND DISCUSSION

The meshed cabin model was imported to the Fluent software. The solver furnishes the following outcomes:

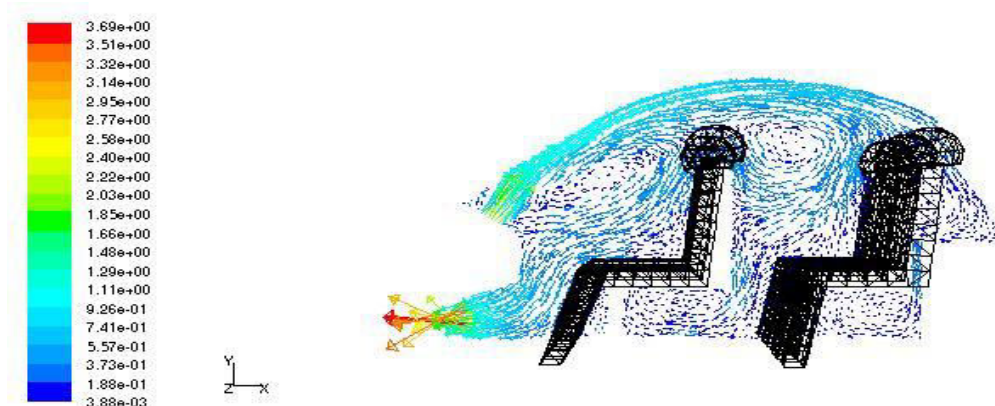


Fig.6: Simulation result of velocity distribution when the vent is located at Dashboard

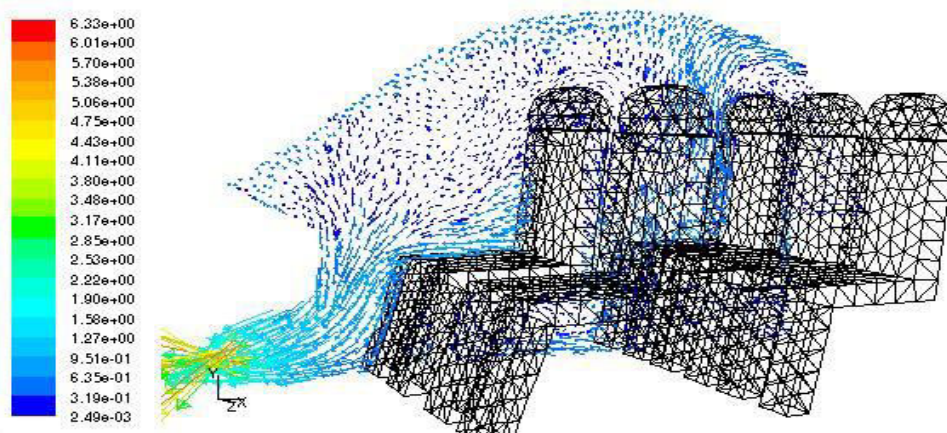


Fig.7: Simulation result of velocity distribution when the vent is located at top position

Fig.6 displays the velocity vector inside the cabin model when the Air conditioning vent was fixed at the dashboard and Fig.7 exhibits the velocity vector inside the cabin model when the air vent is fixed at the Roof Top. From these figures it is concluded that, when the vent is fixed at dashboard the flow was not sufficient at the low level. In order to overcome this problem a trail is made to fix the vent at rooftop and it is analyzed. The obtained results express that, when the vent is kept at rooftop the airflow was more sufficient even at the low level.

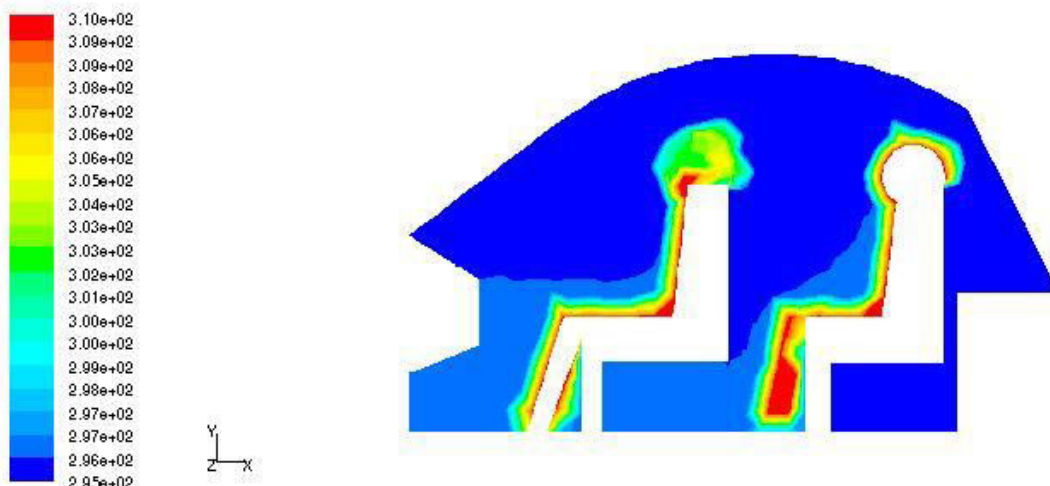


Fig.8: Simulation result of Temperature distribution when the vent is located at Dash board

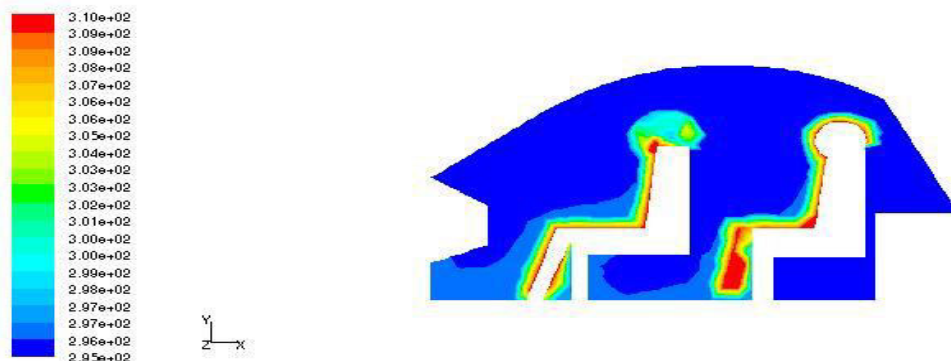
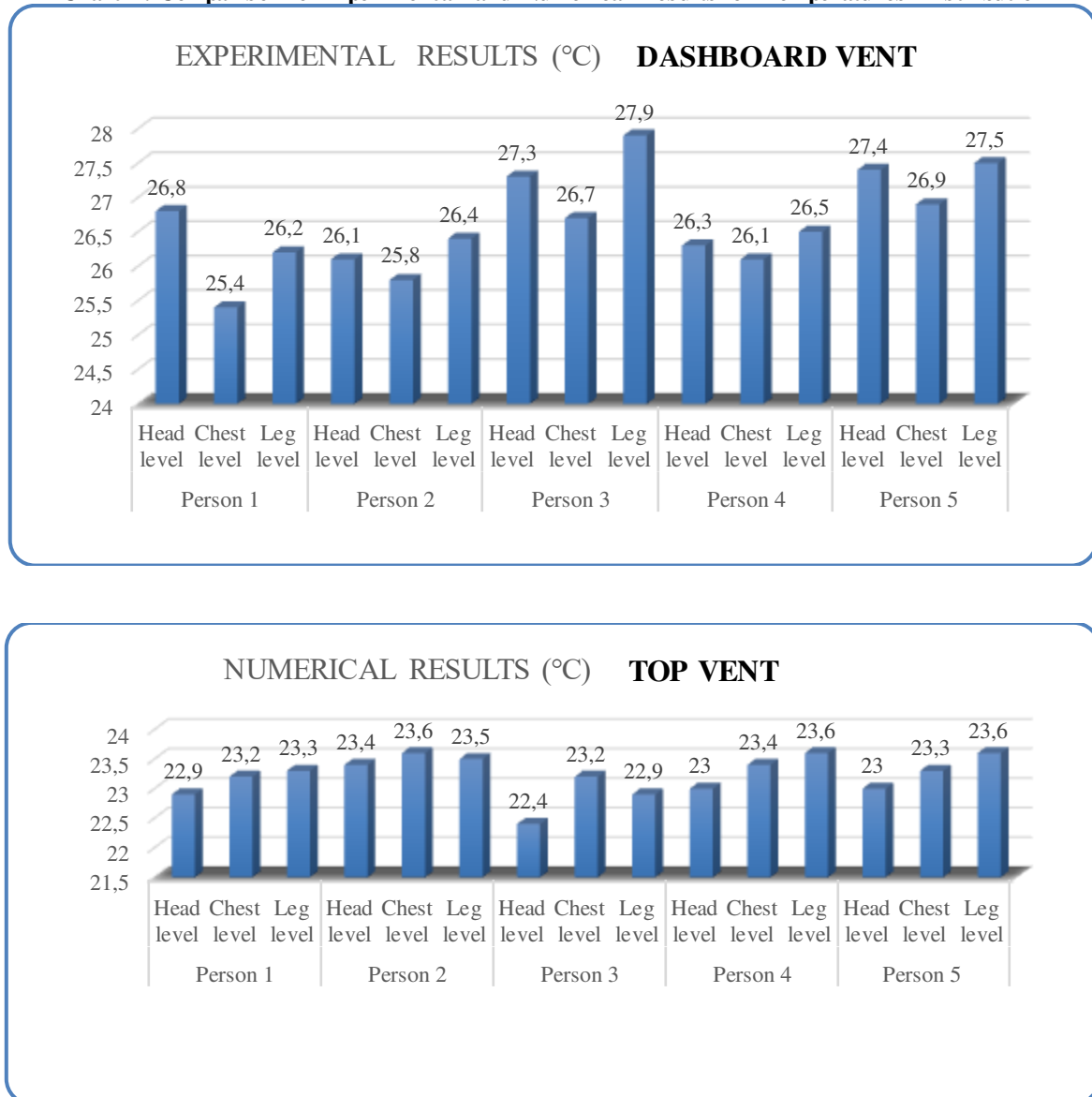
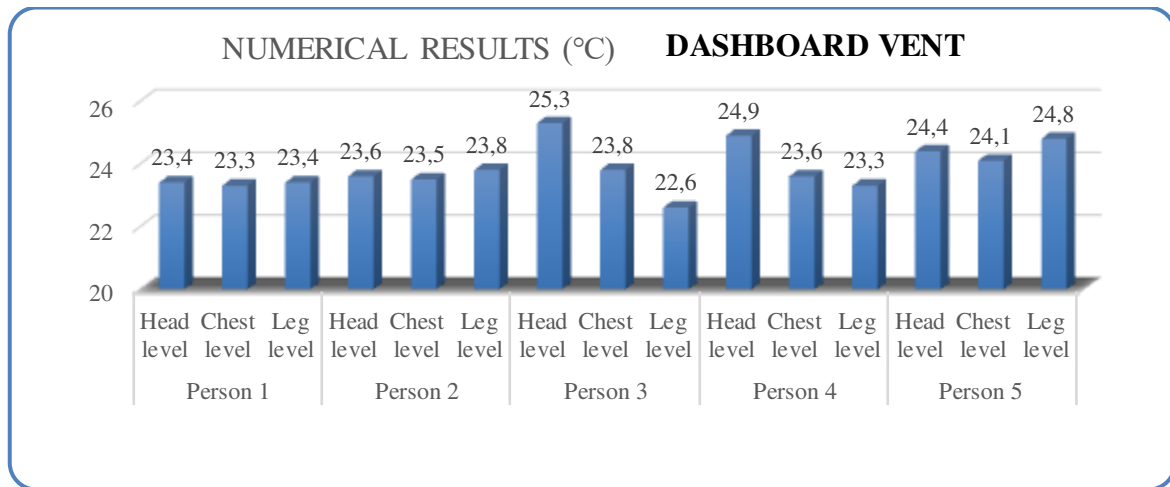


Fig.8: Simulation result of Temperature distribution when the vent is located at top position

Fig. 7& 8 exhibits the contours of static temperature when the vent is kept at dashboard and roof top. The red colour represents the high temperature area which is 310K. The blue areas represent the locations where the temperature is 295K. The openings were given for the inlet and outlet vents and the temperature of stream air from the inlet vent is 289K. The simulation was conducted by neglecting the effect of Engine Temperature and radiations. From fig. 7&8 it is determined that the comfort temperature at below the knee level is accomplished by fixing the air conditioning vent at the roof top.

**Chart 1: Comparison of Experimental and Numerical Results for Temperatures Distribution**



The above chart illustrates the temperature level at 15 points inside the cabin model with a comparison of numerical and simulation results. It also describes that the temperature inside the cabin model gradually decreases from head level to knee level for condition 2, and the comfort temperature is achieved at chest level for condition 1 (Condition 1: Air conditioning vent located at Dashboard, Condition 2: Air conditioning vent located at Roof Top)

## V. CONCLUSION

A simulation of 3D automobile cabin model was executed with the aid of simulation software such as Fluent. The car was subjected to a choice of exterior and interior constraints. The automobile cabin environment was investigated when it was kept in Coimbatore at 12pm on 25<sup>th</sup> march. To represent the inlet and outlet duct in a car, slit with definite size were cut from the model and chill air was permitted to come into the cabin. A Computational Fluid Dynamic approach was used to investigate temperature field inside the traveller compartment under a steady-state circumstances for varying positions of air vents.

When the air conditioning vent is kept at dashboard the cabin area experienced the high temperature at knee level while comparing to air conditioning vents located at roof top. From the simulation, it was concluded that the position of vents plays a dynamic role in defining the cabin environment. So, the Heat Ventilation and air Conditioning system in an automotive vehicle plays a critical part in shaping the cabin atmosphere. From the present investigation, it is suggested that the systematic CFD method used to study the temperature distribution can also be used for different car models and upcoming research can take the benefit of simulation software to design the ventilation system more effectively.

## REFERENCES

- [1] Yan Wang, Qing Gao, Tianshi Zhang, Guohua Wang, Zhipeng Jiang and Yunxia Li (2017) "Advances in Integrated Vehicle Thermal Management and Numerical Simulation", 2017
- [2] Paul Danca, Florin Bode, Ilinca Nastase, Amina Meslem "On the possibility of CFD modeling of the indoor environment in a vehicle", 2016
- [3] Mihaela Simion, Lavinia Socaciu, Paula Unguresan "Factors which influence the thermal comfort inside of vehicles", 2015
- [4] P. Thirumal, K.S. Amirthagadeswaran & S. Jayabal "Experimental and Statistical Analysis of Indoor Air Quality Characteristics in an Air Conditioned Car" Volume 13, 2014
- [5] P. Thirumal, K.S. Amirthagadeswaran and S. Jayabal "Optimization of Indoor Air Quality Characteristics in an Air-Conditioned Car Using Multi-Objective Genetic Algorithm", 2013
- [6] Ramesh Kumar A., Jayabal S., and Thirumal P. "CFD Analysis of Air Flow and Temperature Distribution in an Air Conditioned Car", 2013
- [7] M. Kilic, S.M. Akyol, "Experimental Investigation of Thermal Comfort and Air Quality in an Automobile Cabin during the Cooling Period", 2012
- [8] Amit K Ahirrao, H.K. Narahari, S. Umesh, Vivek Kumar "Effect of Vent Shape on Thermal Comfort of Passengers in a Car" Volume 10, Issue 2, 2011





- [9] Catalin-Adrian and Muhsin Kilic "Three-dimensional Numerical Analysis of Temperature Distribution in an Automobile Cabin", Department of Mechanical Engineering, Faculty of Engineering and Architecture, Uludag University, Bursa, Turkey, 2011
- [10] J. P. Holman, "Heat Transfer", 9th ed., pages 1-13, 2008
- [11] "ASHRAE Handbook of Heating and Ventilation", 1997



**Impact Factor:  
7.512**



# **INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH**

**IN SCIENCE | ENGINEERING | TECHNOLOGY**

**9940 572 462 6381 907 438 ijirset@gmail.com**



**www.ijirset.com**

Scan to save the contact details