IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

CFD ANALYSIS OF CAR COMPARTMENT TO OPTIMIZE THERMAL CONTROL

1SHUBHAM RATHORE, 2AKHLESH KUMAR SHAKYA
1STUDENT , 2HOD ME DEPARTMENT
1RGPV Bhopal,
2RGPV Bhopal

ABSTRACT
In the present work three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) for air conditioner system have been used to perform computational fluid dynamic analysis to investigate best thermal comfort condition with even temperature distribution. There are following conclusion have been drawn from the computational fluid dynamics analysis. After performing Computational fluid dynamic analysis on all three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) the temperature distribution on various segment of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger. It has been observed that the average temperature inside the first and second model of car cabin is higher than third model hencecar cabin with front and top AC inlet is suggested for the future implementation.

Keyword - Car cabin,

I.INTRODUCTION

The HVAC system is an air conditioning technology in buildings and vehicles. It is designed to provide fresh air at all times and to control the internal temperature through cooling or heating. It also contributes to internal security by removing fog, fog and moisture from the windshield and window (Parrino and Carnino, 2004). It includes three functions: heating, ventilation and air conditioning. These three functions are connected and offer occupants comfortable temperature and good air quality, summer and winter. Thermal comfort can be achieved thanks to the perfect execution of an HVAC system.

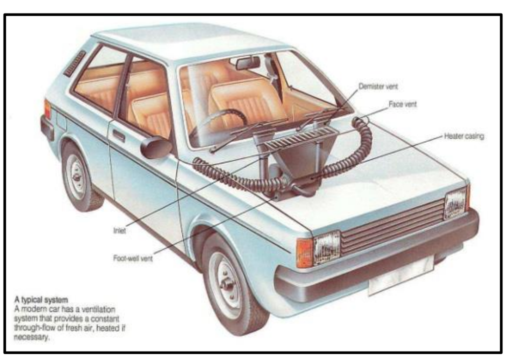

Figure 1: Modern HVAC system

Figure 1.1 shows a 3D model of a typical modern car with an HVAC system. In the figure, openings and ventilation channels are visible in vehicles. HVAC system, consisting of front-end and back-end. The front part consists of mechanical electronic switches in the dashboard. The back of the system includes one or more ventilation motors, actuators (for the control of air circulation, air flow and temperature) and an air conditioning unit coupled with numerous ducts through which the air is directed towards the cabin

II. LITERATURE REVIEW

Tong-Bou Chang at el. [1] A CFD simulation model was developed in this study to explore the effects of outdoor air ventilation rate on vehicle cabin indoor air quality and the amount of outdoor air required for each person in a vehicle. The results show that using the outdoor air supply rate recommendations of ASHRAE Standard 62.1 (i.e. 2.5 l/s per person) the mean CO2 concentrations in the cabin are around 2850 ppm. The results also show that using the outdoor air supply rate recommendations of 4.0 l/s per person for improved human wellbeing, the corresponding mean CO2 concentrations in the cabin are around 1810 ppm. Moreover, the present study found that an outdoor fresh air flow rate of 9.2 l/s per passenger was sufficient to reduce the carbon dioxide concentration within the cabin to a safe value of 1000 ppm.

Tobias Dehne at el. [2] Compared three vertical ventilation concepts to dashboard ventilation in a generic car cabin with the aim to improve thermal passenger comfort and energy efficiency of future cars. Temperatures were analyzed with an infrared camera and local temperature sensors. Omni-directional velocity probes were used to capture the fluid velocities and temperatures in the vicinity of thermal passenger dummies, which were used to simulate the thermal impact of the passengers.

Chunling Qi, Yaxin Helian, Jiying Liu & Linhua Zhang [3] This paper mainly introduced the influence of car passenger compartment's temperature variation on thermal comfort by conducting a field experiment. Two different cases were analyzed during parking and driving stage. One consisted of opening window gaps and adding sunshade during parking, the other mainly compared the difference between ventilation and air conditioning when driving. Measured data showed the temperature difference between the inside and outside of the car passenger compartment was relatively small when car opened window gap.

III. OBJECTIVE

The main objectives of the present work are as follows:

1. To study about thermal comfort in a passenger compartment by considering the spectral solar radiation.
2. To perform the Computational Fluid Dynamics analysis by using solar radiation at Bhopal location.
3. To perform the Computational Fluid Dynamics analysis by changing the location of AC inlet for improving the performance of Thermal comfort in car cabin.

IV. METHODOLOGY

A) Computational fluid dynamics analysis:

The computational fluid dynamics analysis is carried out using Ansys fluent for car cabin. The input parameters have been taken from the experimental data's. The governing equations such as continuity equation, momentum equation, energy equations, K equation and ε equations are used to perform this computational analysis.

B) Algorithm used for Computational fluid dynamics analysis:

Figure:2 Algorithm used for Computational fluid dynamics analysis

In the present work three CDA model of car cabin is designed using the CATIA designing software. For creating the model approximate dimension of Tata Nano were considered. The three models created were

1. Simple car cabin
2. Car cabin with AC inlet at its sides
3. Car cabin with AC inlet at its roof

The steps for designing the car model are as follow –

– Open CATIAV5R20 and select a plane
– Draw the sketch of the car model

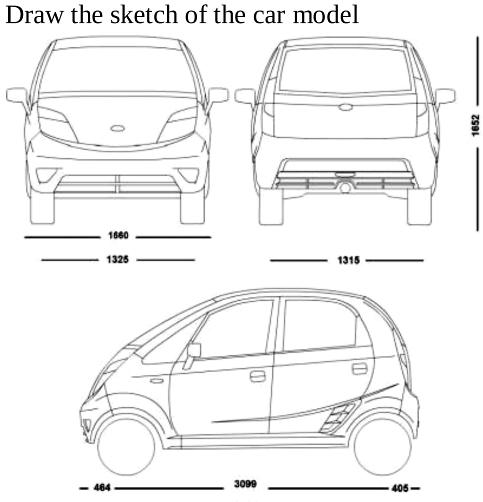

Figure:3 dimensions of TATA NANO

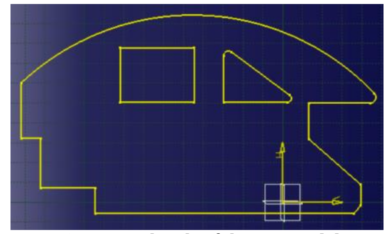

Figure: 4 sketch of the car model

– Pad the sketch with mirrored extant

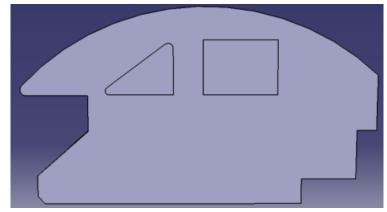

Figure: 53D CAD model of car model

– Draw the sketch for inlet, outlet, and mirrors at appropriate places
 ▪ Use pocket command to cut them out

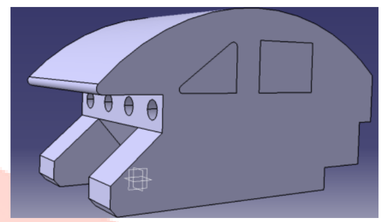

Figure: 6 3D CAD model of car with inlet & outlet

– Draw the sketch for the seats at appropriate place and pad them

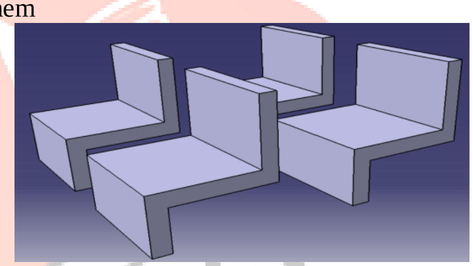

Figure: 7 3D CAD model of car with seats

-Save the model in .stp format

Computational fluid dynamics analysis has been carried out using ANSYS FLUENT tool. The steps for the analysis are shown below:

- ❖ Import the STEP file of the car cabin in the ANSYS FLUENT module.
- ❖ After importing the step file in ANSYS open DESIGN modular of the ANSYS FLUENT and created the named selection of the parts of the car model
- ❖ After giving the proper named selection meshing of the geometry was done. Meshing is the process of breaking the model into number of nodes and elements

C) Selecting the Location for Solar Study:

For the solar study on car cabin Bhopal (M.P.) location is selected. Longitudinal location of Bhopal is 77.4126 degree and latitude location is 23.2599 degree. Time zone is 5.5 (+-GMT). For the solar intensity summer season is considered at maximum temperature.

D) Governing Equations

1. Conservation of mass or continuity equation:

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\,\vec{v}) = S_m$$

Where S_m= mass added to the continuous phase or any user defined sources.

For 2D axisymmetric geometries, the continuity equation is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\rho v_r}{r} = S_m$$

Where x is the axial coordinate, r is the radial coordinate, v_x is the axial velocity, and v_r is the radial velocity.

2. Momentum Conservation Equations

Conservation of momentum in an inertial reference frame is described by

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F}$$

Where p= static pressure

$\bar{\bar{\tau}}$ = stress tensor,

$\rho\vec{g}$ = gravitational body force and

$\vec{F}$ = external body forces

The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}) - \frac{2}{3}\nabla.\vec{v}I\right]$$

where μ = molecular viscosity

I = unit tensor,

For 2D axisymmetric geometries, the axial and radial momentum conservation equations are given by

$$\frac{\partial}{\partial x}(\rho v_x) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_x) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_r) = -\frac{\partial p}{\partial x} + \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(2\frac{\partial v_x}{\partial x} - \frac{2}{3}(\nabla.\vec{v})\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial v_x}{\partial r} + \frac{\partial v_r}{\partial x}\right)\right] + F_x$$

And

$$\frac{\partial}{\partial t}(\rho v_r) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_r) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_r) = -\frac{\partial p}{\partial r} + \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(\frac{\partial v_r}{\partial x} + \frac{\partial v_x}{\partial r}\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(2\frac{\partial v_r}{\partial r} - \frac{2}{3}(\nabla.\vec{v})\right)\right] - 2\mu\frac{v_r}{r^2} + \frac{2\mu}{3r}(\nabla.\vec{v}) + \rho\frac{v_z^2}{r} + F_r$$

Where

$$\nabla.\vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$$

Where v_x= Axial velocity

v_r= Radial velocity

$\boldsymbol{v_z}$=swirlvelocity

3. Energy Equation:

The energy equation for the mixture takes the following form:

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}(\alpha_k\rho_k E_k) + \nabla.\sum_{k=1}^{n}\left(\alpha_k\vec{v}_k(\rho_k E_k + p)\right) = \nabla.\left(k_{eff}\nabla T\right) + S_E$$

where k_{eff} = effective conductivity

S_E= volumetric heat sources

$$E_k = h_k - \frac{p}{\rho k} + \frac{v_k^2}{2}$$

Where

$E_k = h_k$ for an incompressible phase and h_k= sensible enthalpy for phase k

4.$\boldsymbol{k-\in}$ model:

The turbulence kinetic energy, k, and its rate of dissipation, $\in$, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\in - Y_M + S_k$$

and

$$\frac{\partial}{\partial t}(\rho\in) + \frac{\partial}{\partial x_i}(\rho\in v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\in}\right)\frac{\partial\in}{\partial x_j}\right] + C_{1\in}\frac{\in}{k}(G_k + C_{3\in}G_b) - C_{2\in}\rho\frac{\in^2}{k} + S_\in$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients,

G_b is the generation of turbulence kinetic energy due to buoyancy,

Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate,

$C_{1\in}$, $C_{2\in}$, and $C_{3\in}$ are constant. σ_k and $\sigma_\in$ are turbulent Prandtl numbers for k and$\in$,

S_k And $S_\in$ are user-defined source terms.

E) Computational fluid dynamics analysis of design-1for car cabin:

For the present study TATA NANO is considered for CFD analysis for that dimensions are used to create CAD model is as given below.

1. CAD geometry: In the present work three CDA model of car cabin is designed using the CATIA designing software. For creating the model approximate dimension of Tata Nano were considered and a three dimensional view is shown in figure No.

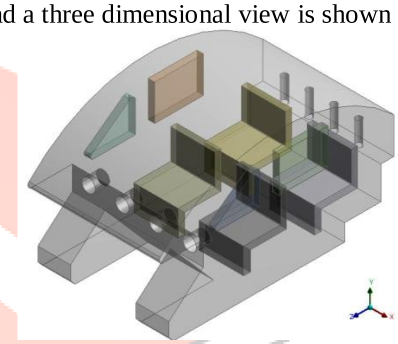

Figure:8 CDA model of car cabin as design-1

2. Meshing: After completing the CDA model of car cabin is imported in ANSYS workbench for further computational fluid dynamics analysis where next step is meshing. Meshing is a critical operation in finite element analysis in this process CAD geometry is divided into large numbers of small pieces called mesh. The size of element is set as 5 mm to generate mesh and the total no of nodes generated in the present work is 43036 and total No. of Elements is 189818. Types of elements used are rectangular and triangular which is a rectangular/triangular in shape with four and three nodes on each element.

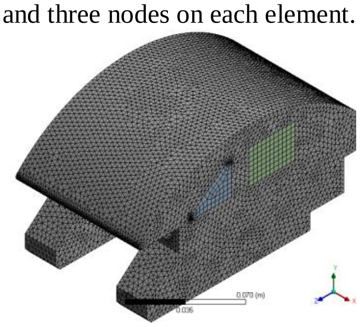

Figure: 9 Meshing of design-1 of car cabin (Total no. of nodes 43036, total element 189818)

V. RESULT

After performing Computational fluid dynamic analysis on all three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) the temperature distribution and velocity distribution on various segment of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger. The results of CFD on all three design of car model are discussed in this chapter by tabulated data and graphical representation.

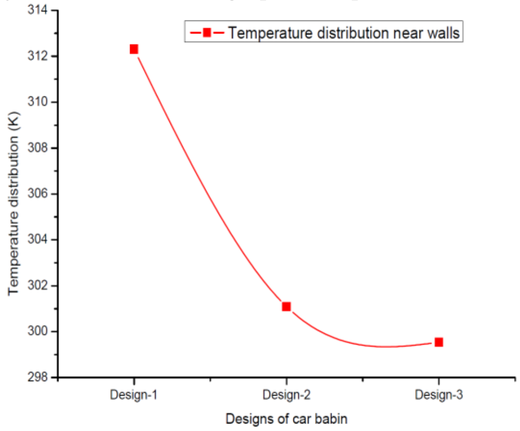

Figure: 10 Temperature distribution on a plane created near the wall of the car cabin on yz plane

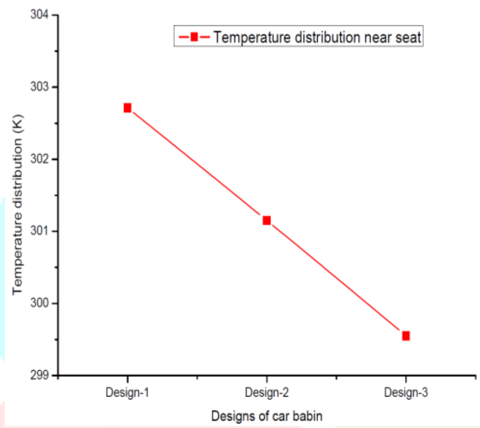

Figure:11 Temperature distribution on a plane created near the seat of the car cabin on yz plane

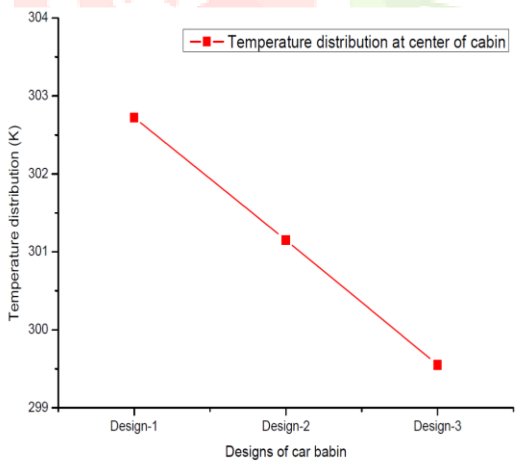

Figure: 12 Temperature distribution on a plane created at center of the car cabin on yz plane

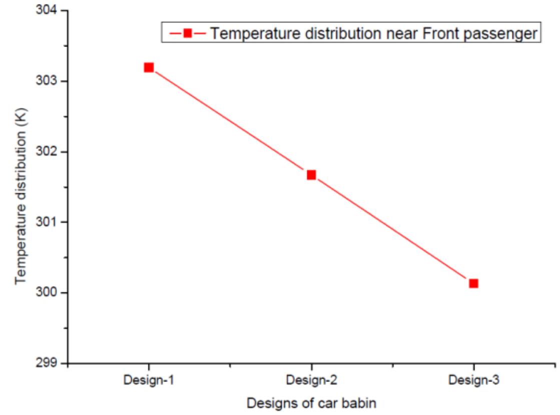

Figure: 13 Temperature distribution on a plane created at front seat of the car cabin on zx plane

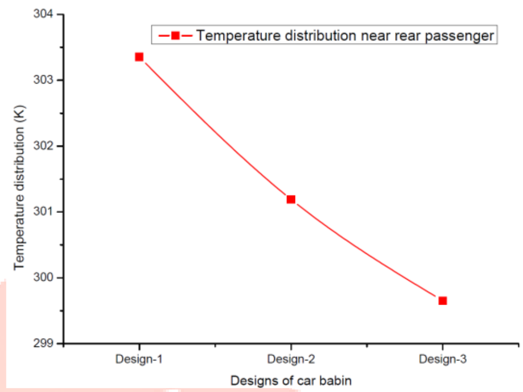

Figure: 14 Temperature distribution on a plane created at rear seat of the car cabin on zx plane

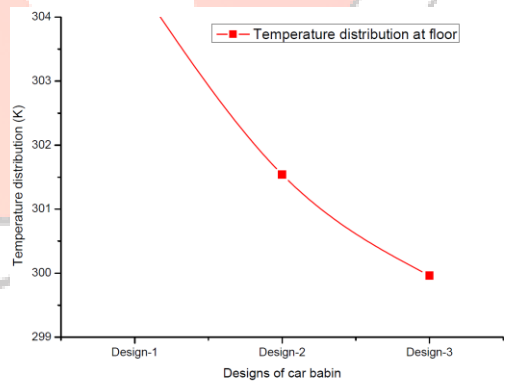

Figure: 15 Temperature distribution on a plane created at floor of the car cabin on xy plane

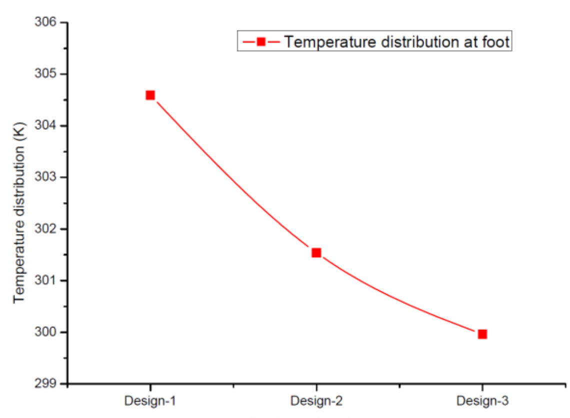

Figure: 16 Temperature distribution on a plane created at foot of the car cabin on xy plane

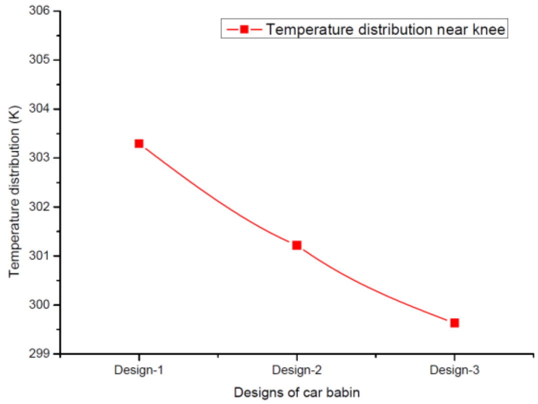

Figure: 17 Temperature distribution on a plane created near knee of the passenger on xy plane

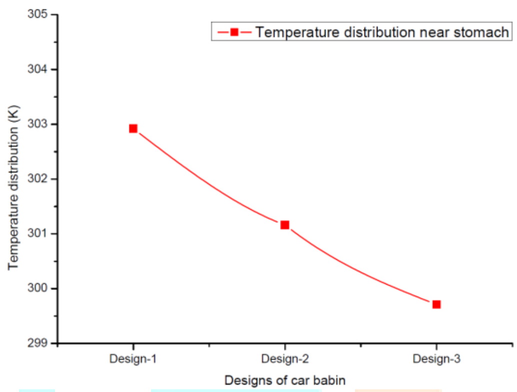

Figure: 18 Temperature distribution on a plane created near stomach of the passenger on xy plane

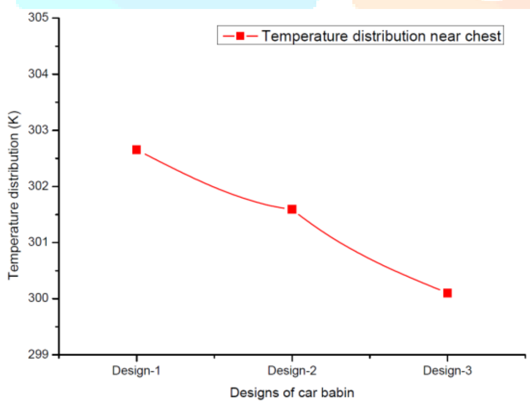

Figure: 19 Temperature distribution on a plane created near chest of the passenger on xy plane

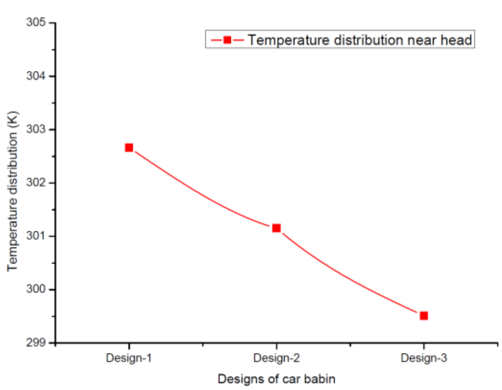

Figure: 20 Temperature distribution on a plane created near head of the passenger on xy plane

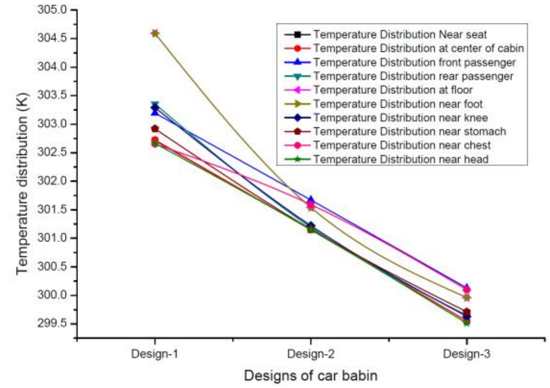

Figure: 21comparative result of temperature distribution at various plane created during CFD analysis

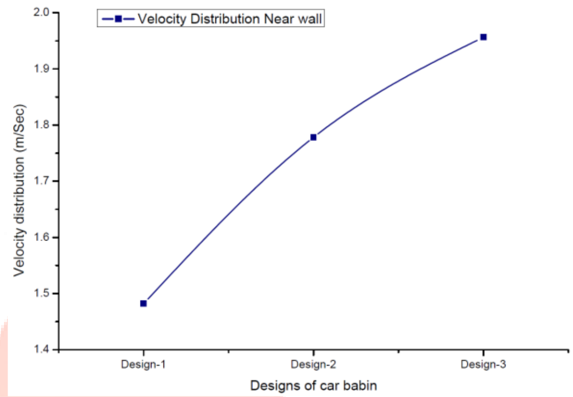

Figure: 22 Velocity distribution on a plane created near the wall of the car cabin on yz plane

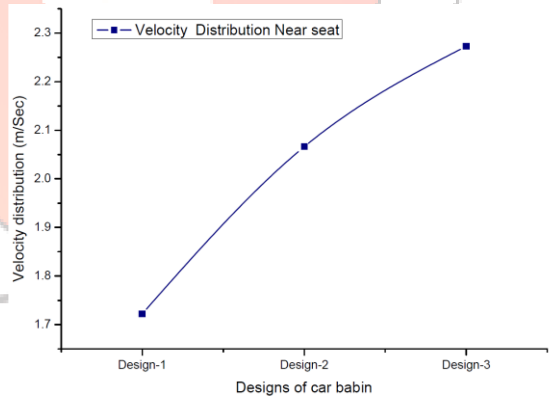

Figure: 23 Velocity distribution on a plane created near the seat of the car cabin on yz plane

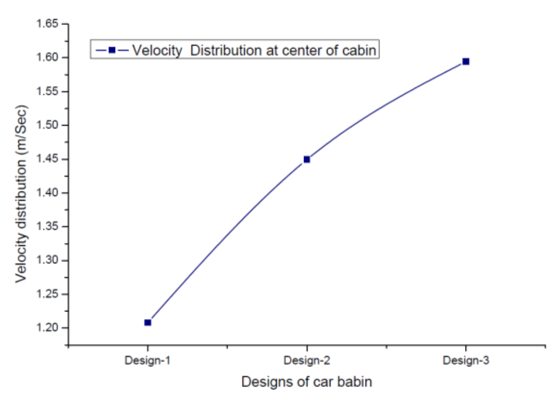

Figure: 24 Velocity distribution on a plane created at center of the car cabin on yz plane

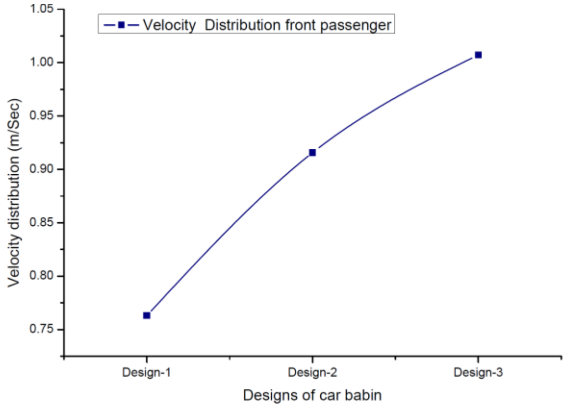

Figure: 25 Velocity distribution on a plane created at front seat of the car cabin on zx plane

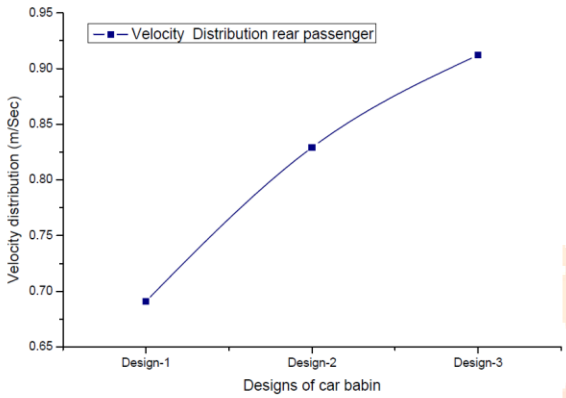

Figure: 26 Velocity distribution on a plane created at rear seat of the car cabin on zx plane

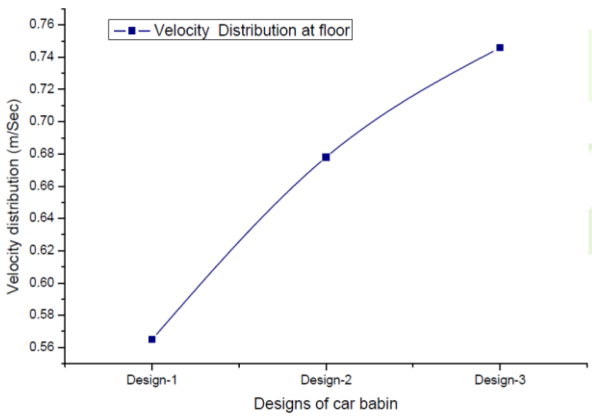

Figure: 27 Velocity distribution on a plane created at floor of the car cabin on xy plane

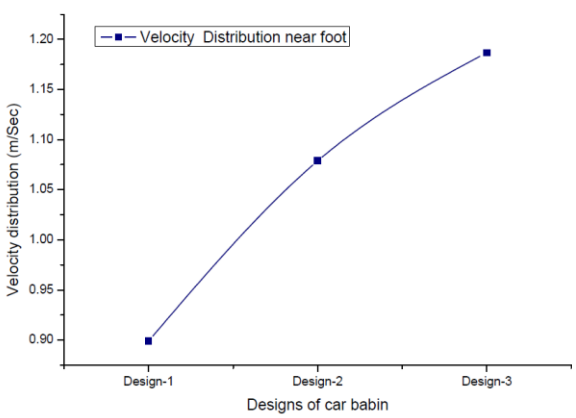

Figure: 28 Velocity distribution on a plane created at foot of the car cabin on xy plane

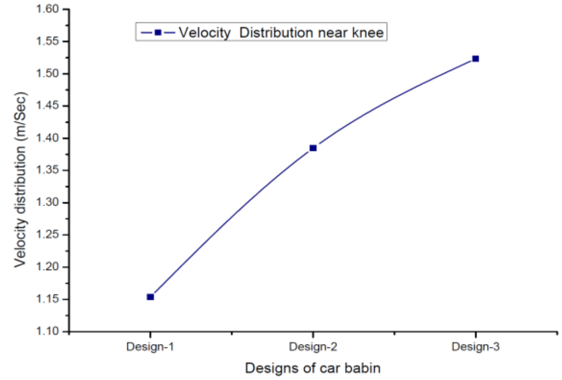

Figure: 29 Velocity distribution on a plane created near knee of the passenger on xy plane

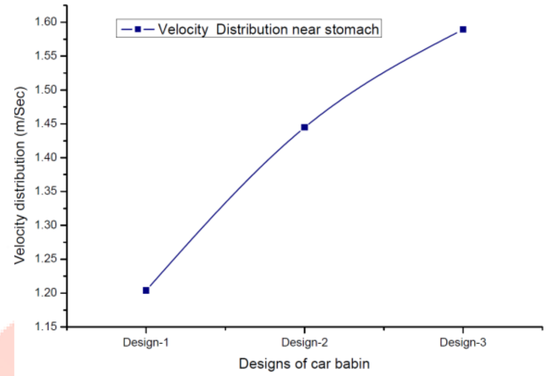

Figure: 30 Velocity distribution on a plane created near stomach of the passenger on xy plane

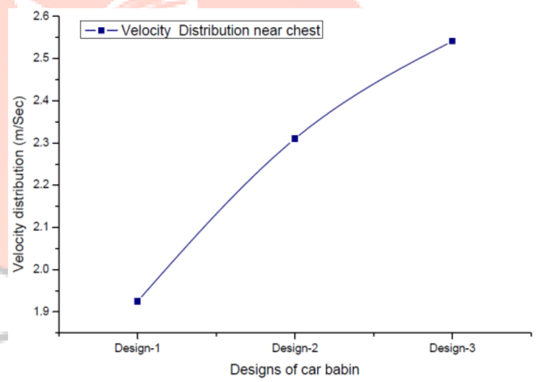

Figure: 31 Velocity distribution on a plane created near chest of the passenger on xy plane

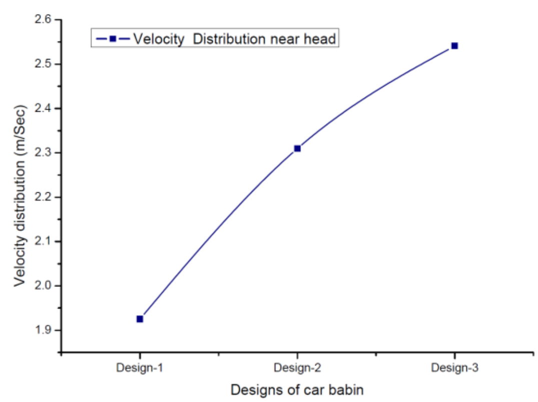

Figure: 32 Velocity distribution on a plane created near head of the passenger on xy plane

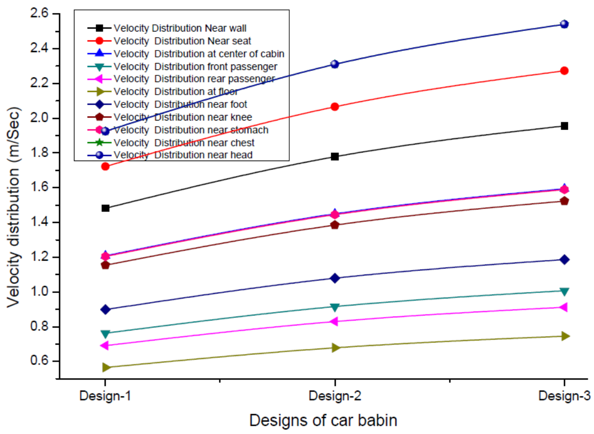

Figure: 33comparative result of velocity distribution on various plane created

VI. Validation

The main objective of the present work to enhance the thermal comfort by changing the position of AC inlets in the car cabin, For that computational fluid dynamic analyses have been performed for three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof). For the validation of computational fluid model used in present work is taken from a research paper of Jin Woon Lee "Influence of the spectral solar radiation on the air flow and temperature distributions in a passenger compartment" International Journal of Thermal Sciences vol. 75 year 2014, page from 36- 44 contents available at science direct.

After performing Computational fluid dynamic analysis on all three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) the temperature distribution on various segment of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger .

After the validation with temperature and velocity distribution some other plane may be taken for all three design of car cabin have been used for computational fluid dynamics analysis to enhance thermal comfort.

Temperature distribution at body segment with respect to designs for front passenger

Body segment	Front Passenger		
	Design-1	Design-2	Design-3
Head	301.4	319.6	305.6
Chest	301.1	318.6	305.4
Stomach	300.7	316.9	304.9
Knee	299.8	313	303.5
Foot	293.4	296.5	296.9
Floor	297.6	303.1	301.3

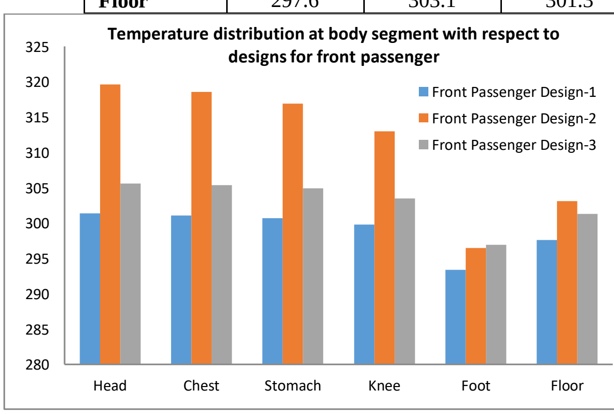

Figure: 34 Temperature distribution at body segment with respect to designs for front passenger

Temperature distribution at body segment with respect to designs for rear passenger

Body segment	Rear Passenger		
	Design-1	Design-2	Design-3
Head	301.3	319.3	305.6
Chest	301	318.3	305.4
Stomach	300.6	316.7	304.9
Knee	299.5	313.3	303.4
Foot	296.4	299.8	296.1
Floor	297.2	301.4	300.4

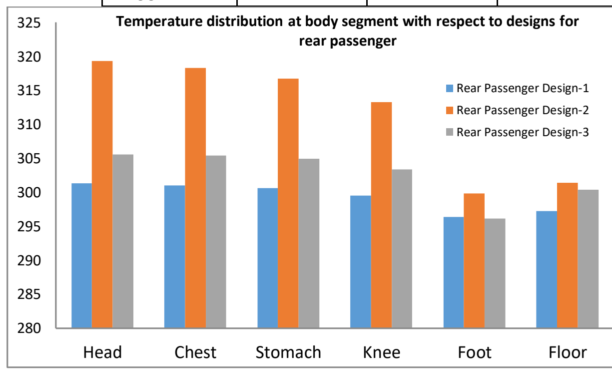

Figure: 35 Temperature distribution at body segment with respect to designs for rear passenger

body segment and places in car cabin	Temperature Distribution		
	Design-1	Design-2	Design-3
Near wall	312.309	301.086	299.540
Near seat	302.713	301.150	299.550
At center of cabin	302.722	301.150	299.550
Front passenger	303.194	301.670	300.130
Rear passenger	303.355	301.190	299.650
At floor	304.593	301.540	299.960
Near foot	304.593	301.540	299.960
Near knee	303.292	301.220	299.630
Near stomach	302.92	301.160	299.710
Near chest	302.655	301.590	300.100
Near head	302.662	301.150	299.510

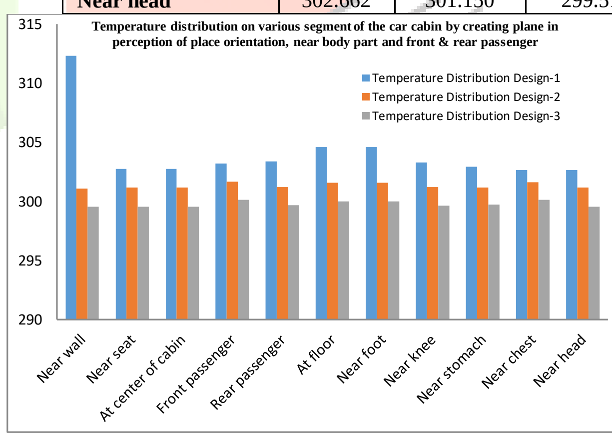

Figure: 36 Temperature distribution on various segments of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger

body segment and places in car cabin	Velocity Distribution		
	Design-1	Design-2	Design-3
Near wall	1.482	1.7784	1.956
Near seat	1.722	2.066	2.273
At center of cabin	1.208	1.450	1.595
Front passenger	0.763	0.916	1.007
Rear passenger	0.691	0.829	0.912
At floor	0.565	0.678	0.746
Near foot	0.899	1.079	1.187
Near knee	1.154	1.385	1.523
Near stomach	1.204	1.445	1.589
Near chest	1.925	2.310	2.541
Near head	1.925	2.310	2.541

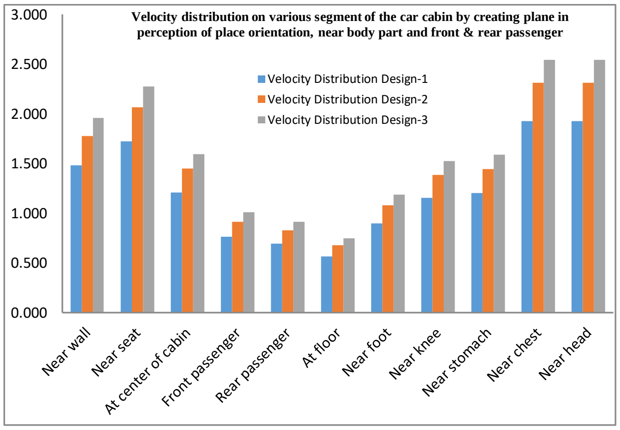

Figure: 37 Velocity distribution on various segments of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger

VII. CONCLUSION

In the present work three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) for air conditioner system have been used to perform computational fluid dynamic analysis to investigate best thermal comfort condition with even temperature distribution. There are following conclusion have been drawn from the computational fluid dynamics analysis.

After performing Computational fluid dynamic analysis on all three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) the temperature distribution on various segment of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger.

- ❖ Temperature distribution on a plane created at center of the car cabin on yz plane for all three car models are 302.713K, 301.150K and 299.550K respectively.
- ❖ Temperature distribution on a plane created near the wall of the car cabin on yz plane for all three car models are 312.309K, 301.086K and 299.54K respectively.
- ❖ Temperature distribution on a plane created near the seat of the car cabin on yz plane for all three car models are 302.713K, 301.150K and 299.550K respectively.
- ❖ Temperature distribution on a plane created at front seat of the car cabin on zx plane for all three car models are 303.194K, 301.670K and 300.130K respectively.
- ❖ Temperature distribution on a plane created at rear seat of the car cabin on zx plane for all three car models are 303.355K, 301.190K and 299.650K respectively.
- ❖ Temperature distribution on a plane created at floor of the car cabin on xy plane for all three car models are 304.593K, 301.540K and 299.960K respectively.
- ❖ Temperature distribution on a plane created at foot of the car cabin on xy plane for all three car models are 304.593K, 301.540K and 299.960K respectively.
- ❖ Temperature distribution on a plane created near knee of the passenger on xy plane for all three car models are 303.292K, 301.220K and 299.630K respectively.
- ❖ Temperature distribution on a plane created near stomach of the passenger on xy plane for all three car models are 302.92K, 301.160K and 299.710K respectively.
- ❖ Temperature distribution on a plane created near chest of the passenger on xy plane for all three car models are 302.655K, 301.590K and 300.100K respectively.
- ❖ Temperature distribution on a plane created near head of the passenger on xy plane for all three car models are 302.622K, 301.150K and 299.510K respectively.

After performing Computational fluid dynamic analysis on all three design of car model (Simple car cabin, Car cabin with AC inlet at its sides and Car cabin with AC inlet at its roof) the velocity distribution on various segment of the car cabin by creating plane in perception of place orientation, near body part and front & rear passenger.

- ❖ Velocity distribution on a plane created at center of the car cabin on yz plane for all three car models are 1.482 m/sec, 1.778 m/sec and 1.956 m/sec respectively.
- ❖ Velocity distribution on a plane created near the wall of the car cabin on yz plane for all three car models are 1.722 m/sec, 0.066 m/sec and 2.273 m/sec respectively.
- ❖ Velocity distribution on a plane created near the seat of the car cabin on yz plane for all three car models are 1.208 m/sec, 1.450 m/sec and 1.595 m/sec respectively.
- ❖ Velocity distribution on a plane created at front seat of the car cabin on zx plane for all three car models are 0.763 m/sec, 0.916 m/sec and 1.007 m/sec respectively.
- ❖ Velocity distribution on a plane created at rear seat of the car cabin on zx plane for all three car models are 0.691 m/sec, 0.829 m/sec and 0.912 m/sec respectively.
- ❖ Velocity distribution on a plane created at floor of the car cabin on xy plane for all three car models are 0.565 m/sec, 0.678 m/sec and 0.746 m/sec respectively.
- ❖ Velocity distribution on a plane created at foot of the car cabin on xy plane for all three car models are 0.899 m/sec, 1.079 m/sec and 1.187 m/sec respectively.
- ❖ Velocity distribution on a plane created near knee of the passenger on xy plane for all three car models are 1.154 m/sec, 1.385 m/sec and 1.523 m/sec respectively.
- ❖ Velocity distribution on a plane created near stomach of the passenger on xy plane for all three car models are 1.204 m/sec, 1.445 m/sec and 1.589 m/sec respectively.
- ❖ Velocity distribution on a plane created near chest of the passenger on xy plane for all three car models are 1.925 m/sec, 2.310 m/sec and 2.541 m/sec respectively.
- ❖ Velocity distribution on a plane created near head of the passenger on xy plane for all three car models are 1.925 m/sec, 2.310 m/sec and 2.541 m/sec respectively.

It has been observed that the average temperature inside the first and second model of car cabin is higher than third model hencecar cabin with front and top AC inlet is suggested for the future implementation.

REFERANCE

[1] Tong-Bou Chang at el. "Development of a CFD model for simulating vehicle cabin indoor air quality" available on Science direct, Elsevier, Transportation Research Part D 62 (2018) 433–440.

[2] Tobias Dehne at el. "Vertical ventilation concepts for future passenger cars" Science direct, Elsevier, Building and Environment, Received Date: 24 August 2017, Accepted Date: 16 November 2017.

[3] Chunling Qi, Yaxin Helian, Jiying Liu & Linhua Zhang "Experiment Study on the Thermal Comfort inside a Car Passenger Compartment" available on Science direct, Elsevier, 10th International Symposium on Heating, Ventilation and Air Conditioning, ISHVAC2017, 19- 22 October 2017, Jinan, China, Procedia Engineering 205 (2017) 3607–3614.

[4] Daniel kristanto and thananchai leephakpreeda "Sensitivity Analysis of Energy Conversion for Effective Energy Consumption, Thermal Comfort, and Air Quality within Car Cabin" 2017 International Conference on Alternative Energy in Developing Countries and Emerging Economies 2017

AEDCEE, 25-26 May 2017, Bangkok, Thailand. Energy Procedia 138 (2017) 552–557.

[5] James Brusey, Diana Hintea, Elena Gaura & Neil Beloe "Reinforcement learning-based thermal comfort control for vehicle cabins" available on Science direct, Elsevier, Mechatronics, (2017) 1–9.

[6] Renmei Zhang, Haiquan Bi & Xiaoliang Wang " Study on the operation mode of air conditioning system for plateau passenger train" 10th International Symposium on Heating, Ventilation and Air Conditioning, ISHVAC2017, 19- 22 October 2017, Jinan, China, Procedia Engineering 205 (2017) 2349–2356.