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 hence car cabin with front and top AC inlet is suggested for the future implementation.

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...
The 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:

1. Open CATIAV5R20 and select a plane
2. Draw the sketch of the car model
3. Pad the sketch with mirrored extant
4. Draw the sketch for inlet, outlet, and mirrors at appropriate places
   - Use pocket command to cut them out
5. Draw the sketch for the seats at appropriate place and pad them

Figure: 4 sketch of the car model
Figure: 53D CAD model of car model
Figure: 6 3D CAD model of car with inlet & outlet
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:
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m
\]

Where \( S_m \) = mass added to the continuous phase or any user-defined sources.

For 2D axisymmetric geometries, the continuity equation is given by
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \nu_r)}{\partial r} + \frac{\partial (\rho \nu_z)}{\partial z} = S_m
\]

Where \( r \) is the radial coordinate, \( z \) is the axial coordinate, \( \nu_r \) is the axial velocity, and \( \nu_z \) is the radial velocity.

2. Momentum Conservation Equations

Conservation of momentum in an inertial reference frame is described by
\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\tau) + \vec{F}
\]

Where \( p \) = static pressure
\( \vec{t} \) = stress tensor,
\( \rho \vec{g} \) = gravitational body force and
\( \vec{F} \) = external body forces

The stress tensor \( \vec{t} \) is given by
\[
\vec{t} = \mu \left( \nabla \vec{v} + \nabla \vec{v}^T \right) - \frac{2}{3} \nabla \cdot \vec{v} I
\]

where \( \mu \) = molecular viscosity
\( I \) = unit tensor.

For 2D axisymmetric geometries, the axial and radial momentum conservation equations are given by
\[
\frac{\partial}{\partial t} (\rho \nu_z) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho \nu_z \nu_r) = -\frac{\partial p}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu \left( \frac{\partial \nu_z}{\partial r} + \frac{\partial \nu_r}{\partial z} \right) \right)
\]

And
\[
\frac{\partial}{\partial t} (\rho \nu_r) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho \nu_z \nu_r) = -\frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu \left( \frac{\partial \nu_r}{\partial r} + \frac{\partial \nu_r}{\partial z} \right) \right)
\]

Where
\[
\nabla \cdot \vec{v} = \frac{\partial \nu_x}{\partial x} + \frac{\partial \nu_r}{\partial r} + \frac{\partial \nu_z}{\partial z}
\]

\( \nu_r \) = Axial velocity
\( \nu_r \) = Radial velocity
\( \nu_z \) = Swirl velocity

3. Energy Equation:

The energy equation for the mixture takes the following form:
\[
\frac{\partial}{\partial t} \sum_{k=1}^{n} (\alpha_k \nu_k E_k) + \nabla \cdot \sum_{k=1}^{n} (\alpha_k \nu_k \nu_k E_k + p) = \nabla \cdot (k_{eff} \nabla T) + S_E
\]

where \( k_{eff} \) = effective conductivity
\( S_E \) = volumetric heat sources

\[ E_k = h_k - \frac{p}{\rho k} + \frac{v_z^2}{2} \]

Where
\( E_k = h_k \) for an incompressible phase and \( h_k \) = sensible enthalpy for phase \( k \)

4. k-\( \varepsilon \) model:

The turbulence kinetic energy, \( k \), and its rate of dissipation, \( \varepsilon \), are obtained from the following transport equations:
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial \xi_i} (\rho k v_i) = \frac{\partial}{\partial \xi_i} \left[ \mu \left( \frac{\partial^2 v_i}{\partial \xi_i^2} + \frac{\partial^2 v_i}{\partial \xi_j^2} \right) \right] + 6_k + G_k - \rho \varepsilon - Y_M + S_k
\]

And
\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial \xi_i} (\rho \varepsilon v_i) = \frac{\partial}{\partial \xi_i} \left[ \frac{\mu_k}{\sigma_e} \frac{\partial \varepsilon}{\partial \xi_i} \right] + C_{1e} \varepsilon \left( \frac{\partial v_i}{\partial \xi_i} \right)^2 - C_{2e} \frac{\varepsilon^2}{k} + S_e
\]

In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients,
\( Y_M \) is the generation of turbulence kinetic energy due to buoyancy,
\( C_{1e}, C_{2e}, C_{3e} \) are constant. \( \sigma_k \) and \( \sigma_e \) are turbulent Prandtl numbers for \( k \) and \( \varepsilon \).

\( S_k \) and \( S_e \) are user-defined source terms.

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

For the present study TATA NANO is considered for CFD analysis for that dimensions are used to create CAD model 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. 8.

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.

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: 21 Comparative 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 ye plane

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

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

### 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.
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, 2.066 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.
- Velocity distribution on a plane created near seat of the car cabin on zx plane for all three car models are 3.307 m/sec, 3.605 m/sec and 3.899 m/sec respectively.
- Velocity distribution on a plane created at foot of the car cabin on zx plane for all three car models are 4.276 m/sec, 4.674 m/sec and 5.072 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 hence car cabin with front and top AC inlet is suggested for the future implementation.

**REFERENCE**


