International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 4, Issue 6, June-2018

CFD Analysis of Airflow inside a Car Compartment

[1]Megha Shukla, [2]Lakshminarasimha N
[1,2]Assistant Professor, Mechanical Engineering Department,
New Horizon College of Engineering, Bengaluru

Abstract-
In the present study a finite volume based computational procedure is used to investigate the air flow analysis inside a car compartment. The compartment used for flow and heat diffusion analysis has been bounded by convective boundary conditions for the bottom wall, the top surface and the front and rear wind shields open to heat flux radiating from the sun, the air entering the car compartment with certain velocity and temperature through the inlet vent, the outlet vent to be at gauge pressure, humans and seats are considered to be a solid part and thus considering no heat radiating from humans. The flow is assumed to be turbulent and the air to be incompressible. The analysis has been carried out for 2D car compartment.

1. Introduction

As and when the human needs have increased, technology has had a parallel evolution in order to fulfill the same. Thermal comfort is an important concern for occupants in an enclosed environment such as the passenger compartment of a vehicle. Hence the need for development of air conditioning inside a car without affecting the car's performance is of top priority. In order to optimize the car's performance, evaluation of the airflow inside the cabin is required.

2. Literature Survey

Conducted a survey to find if any analysis on an internal flow of a car has been conducted on similar ground as ours. During survey, came across analysis conducted by few individuals. Their analysis was on internal flow of a car but was dependent on other factors and parameters. The summary of their reports are given below.

[1] Haslinda Mohamed Kamar et.al,

- This paper presents a numerical study on the temperature field inside a passenger's compartment of a Proton Wira saloon car using computational fluid dynamics (CFD) method.
- The main goal is to investigate air-flow condition in the passenger's compartment.
- Results of the CFD analysis show that the temperature of the air around the driver is slightly lower than the air around the rear passenger.

[2] Jalal M. Jalil et.al,

- A numerical study of a two-dimensional, turbulent, recirculating flow within a passenger car cabin is presented.
- The study is based on the solution of the elliptic partial differential equations representing conservation of mass, momentum, temperature, turbulence energy and its dissipation rate in finite volume form. Algebraic expressions for the turbulent viscosity and diffusion coefficients are calculated using the two-equation model ($k-\varepsilon$).
- The results indicate some of negative effects such as development of zones of low air circulation. Also it is found that the number 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 understating of the airflow fields within a road vehicle passenger cabin.

[3] Alex Alexandrov et.al,

- Paper describe physical problem definition and present work-in-progress results for computer simulations of coupled internal and external flows in the compartment of a generic passenger four-seat vehicle (Chrysler benchmark configuration). The vehicle speed, percentage of open window area and its distribution between two sides determine the overall impact of the external flow. The impact of the ventilation system is accounted by changes in the inlet flow velocities of the HVAC (Heating-Ventilation-Air-Conditioning) system.

[4] Amit K. Ahirrao et.al,

- Effect of different vent shapes for same duct geometry is studied for thermal comfort of passenger.
- Effect of vent shapes on velocity at vent outlet, pressure drop across ducts and overall flow distribution from ducts also part of study of this thesis. Changes in vent shape can affect major results in velocities over manikin and hence the temperature distribution inside cabin.
- For the same duct geometry, occupant comfort levels could be improved by changing only vent. Change in vent shapes at outlet of ducts louvers improves flow velocity and directivity, helping the overall flow management inside a car cabin.

[5] A. Arrousi et.al,

- This paper compares the performance of side window defrosting & demisting mechanism of several current model vehicles.
- The study highlights the drawbacks of current designs & points the way to improved passive defrosting mechanism.
- The result shows that the current designs of a side-defroster nozzle give maximum airflow rates in the vicinity of the lower part of the window, which yields to unsatisfactory visibility.

3. Methodology

The various steps involved in an analysis is as described below.
Step 1: Build a CAD model using modeling software's.
Step 2: Mesh the CAD model using meshing software's by applying the appropriate meshing parameters.
Step 3: Apply the Boundary conditions to the meshed model.
Step 4: Set for Computational analysis
Step 5: No Error: Visualization
Error: Remesh/refine/Improve the model, and carry out Step 3

3.1 Geometry

Fig. 1 shows the 2D geometry of car compartment with passenger.

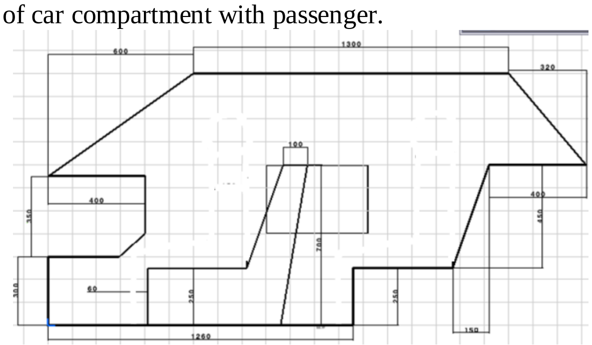

Fig. 1: 2D car compartment

3.2 Meshing

Fig. 2 and Fig. 3, shows the coarse mesh and fine mesh of 2D car compartment. The model is meshed with Quad elements.

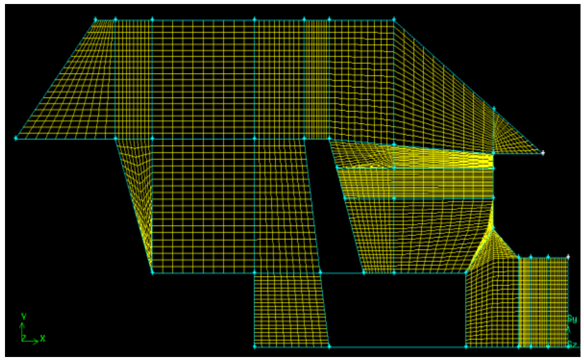

Fig. 2: Coarse Mesh

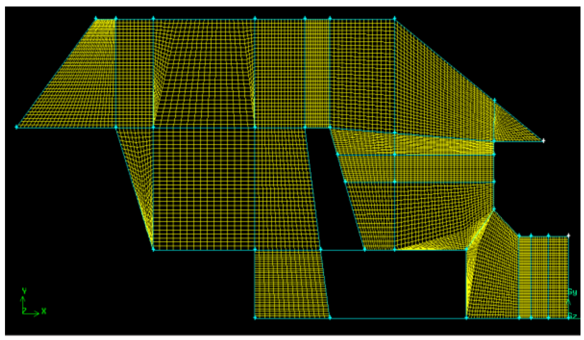

Fig 3: Fine mesh

3.3 Assumptions and Boundary conditions

Assumptions:

- Flow is incompressible
- Flow is steady
- 2D flow

Boundary conditions [1]

- Inlet Velocity-2.77 m/s.
- Outlet Gauge Pressure- 0 atm
- Temperature-290k
- Heat flux – 252 & 260 W/m^2
- No slip wall condition

3.4 Analysis Results

The velocity and Temperature results obtained for 2D car compartment with coarse and fine mesh are shown in this section.

3.4.1 Coarse Mesh: The velocity and Temperature contours for coarse mesh is as shown in Fig. 4 and Fig. 5.

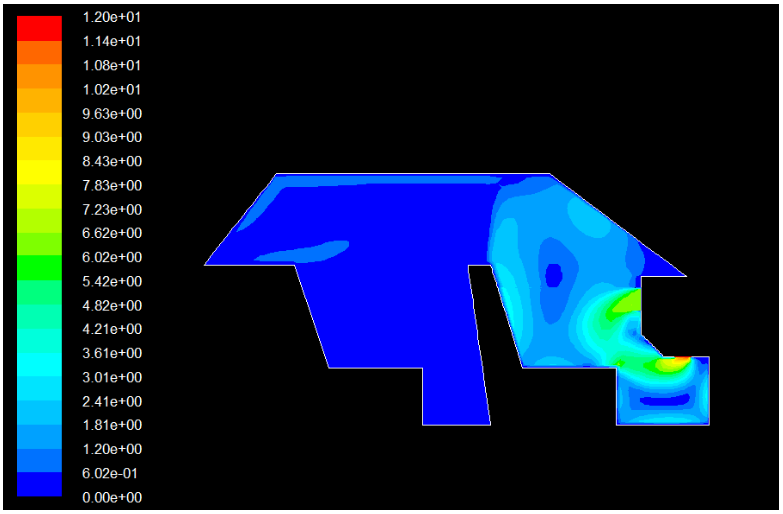

Fig. 4: Velocity contour- coarse mesh

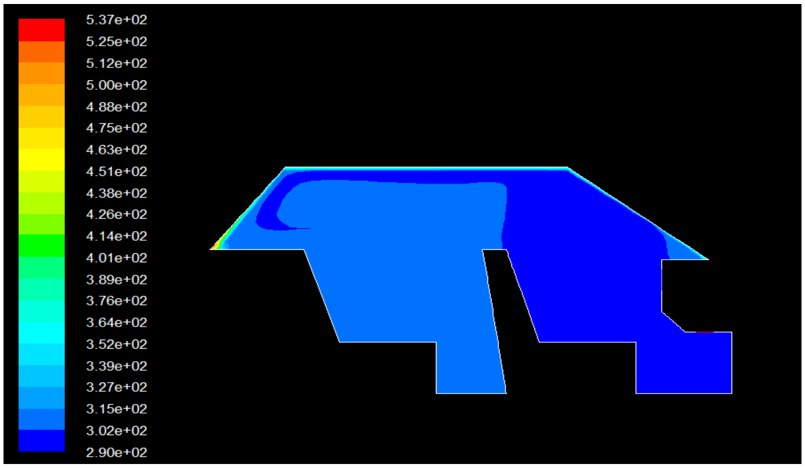

Fig. 5: Temperature contour- coarse mesh

3.4.2 Fine Mesh: The velocity and Temperature contours for coarse mesh is as shown in Fig. 6 and Fig. 7.

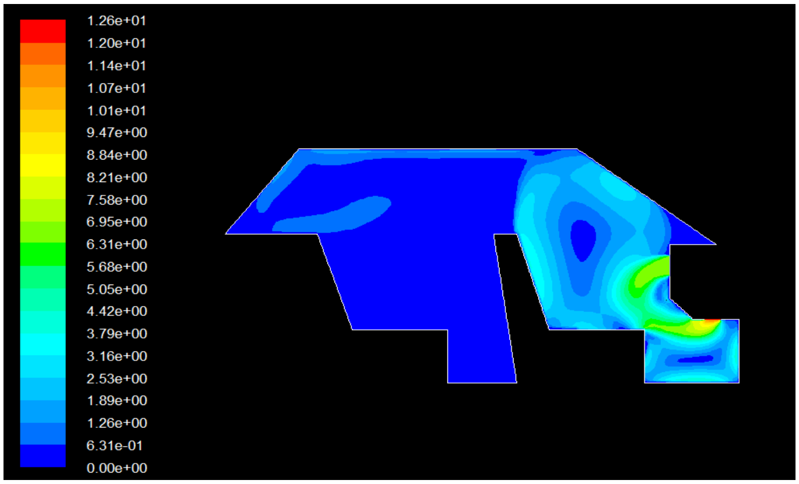

Fig. 6: Velocity contour- Fine mesh

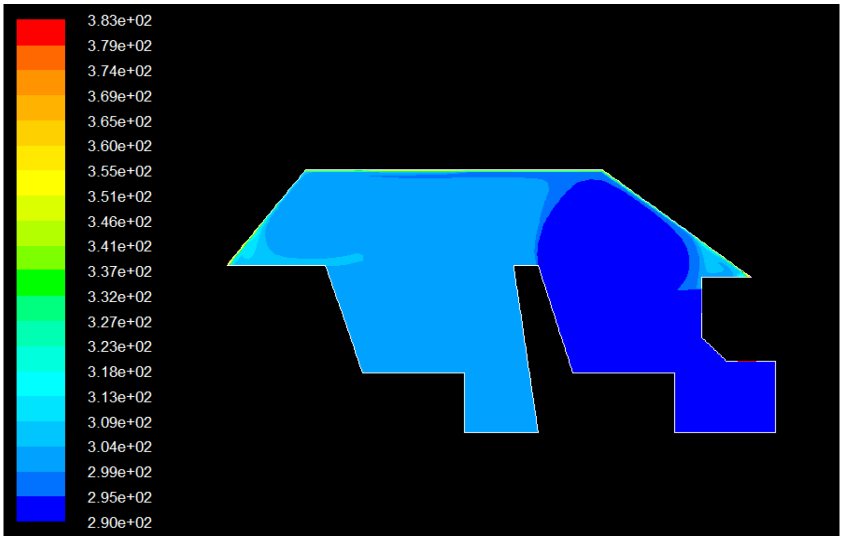

Fig. 7: Temperature contour- Fine mesh

Table below, shows the summary of Analysis results for velocity and Temperatures.

Velocity (m/s)			
Coarse mesh		Fine mesh	
Driver	Passenger	Driver	Passenger
2.43	0.608	2.53	0.636

Temp (k)			
Coarse mesh		**Fine mesh**	
Driver	Passenger	Driver	Passenger
290	301	290	290

3.5 Conclusion

After doing the analysis of 2D air flow and heat transfer inside a passenger cabin of the automobile, the results obtained from the analysis leads to the following conclusion.

(a) CFD is the effective tool in determining the velocity and temperature in a car compartment

(b) For the inlet air velocity2.77m/s, near the Driver and Passenger the velocity found to be approximately 2.5 m/s and 0.6 m/s and the temperature found to be approximately 290K

(c) Analysis becomes a ready reckoner for beginning Engineers to carry further 3D analysis for different inlet velocity and unsteady conditions
Current work is no more exhaustive, it can be extended for 3D analysis, considering heat generation by passengers and car compartment subjected to different unsteady state conditions.

References

[1] Numerical Analysis of Air-Flow and Temperature Field in a Passenger Car Compartment, Haslinda Mohamed Kamar, Nazri Kamsah, and Ahmad Miski Mohammad Nor (e-journal)

[2] CFD Simulation for a Road Vehicle Cabin, Jalal M. Jalil and Haider Qassim Alwan, JKAU: Eng. Sci., Vol. 18 No. 2, pp: 123-142 (2007 A.D. /1428 A.H.)

[3] Analysis of coupled internal-external flow and ventilation in a generic passenger car, Alex Alexandrov,Vladimir kudriavtsev,Marcelo reggio, e-journal

[4] Effect of vent shape on thermal comfort of passengers in a car, Amit K. Ahirrao, H. K. Narahari, S.Umesh, Vivek Kumar, SAS TECH Volume 10, Issue 2, Sep 2011

[5] Vehicle side window desrosting and demisting process, A.Aroussi,A.Hassan,EACC 2003,first Europien conference, Bingen,Germany 25-26 june2003