ISSN 2395-1621

Numerical Simulation and Analysis of HVAC Duct

#1 Mr. Shivanand Doddaganiger, #2 Dr. Narendra Deore

1 sudhi.08@gmail.com
2 naren.deore@gmail.com

#1 PG Student, Department of mechanical Engineering, Savitribai Phule Pune University.
Pimpri Chinchwad College of engineering, Nigdi, Pune.
#2 Professor, Department of mechanical Engineering, Savitribai Phule Pune University.
Pimpri Chinchwad College of engineering, Nigdi, Pune.

ABSTRACT

Simulation of passenger compartment climatic conditions is becoming increasingly important as a complement to wind-tunnel and field testing to achieve improved airflow comfort while reducing vehicle development time and cost. Computational Fluid Dynamics (CFD) analysis of a passenger compartment involves not only geometric complexity but also strong interactions of airflow. Temperature and velocities are major factors responsible for cabin temperature. Primary focus of the study is to assess existing airflow and thermal comfort performance and propose improvement in its duct shape and vent orientation for passenger comfort. Air-flow management inside a vehicle cabin because of airflow distribution over manikin is also part of the study.Investigation of fluid flow through HVAC duct form different outlets of an automotive heating ventilation and air conditioning (HVAC) system will carried out in the present work. The CAD model was developed and analysis will be done. To analyze the air flow, a simulation is performed using Computational Fluid Dynamics, and with the help of this simulation we can roughly estimate the behavior of air. The performance of the HVAC system is judged by parameters like air discharge rate at cabin level, pressure drop through the system, uniformity of the air flow at the outlet faces and distribution between different duct outlets. Pressure loss is another aspect which will require a lot of attention at this stage of development. It is one of the major characteristic which will ensure a smooth flow of air inside the HVAC system. All these parameters are predicted by computational fluid dynamics (CFD) analysis. should meet the standards. Comparison between CFD and testing results will be made in the HVAC system development by incorporating CFD as a design tool.

Keywords— Air flow Analysis, Numerical simulation, HVAC, Computational Fluid Dynamics.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

CFD analysis plays an important role of late as a design tool and is being used by many automobile companies. This has been brought about by the rapid development in computing powers as well as numerical techniques[1]. To Improve air conditioning performance and occupant thermal comfort requires an thorough understanding of the fluid motion prevailing in the HVAC system. HVAC system dynamic airflow should be based on achieving uniform passenger comfort inside the passenger compartment at the specified maximum cooling and heating conditions. However care needs to be exercised in air flow selection because increasing
airflow leads to greater noise levels, blower motor power consumption[4].

Recent advancement in CFD are able to simulate actual heating, cooling performance of the HVAC module. This allows less number of prototypes build up, and being able to make component selection, internal design, and positioning early in the design process, thus shortening the design cycle[2].

To accomplish CFD simulation, reduce computational efforts, and to develop a optimum HVAC system, HVAC ducts should be designed and analyzed perfectly. CFD methodology to study the airflow field development of new developing HVAC duct. Subsequently all the CFD results have been validated with test data[4].

II. METHODOLOGY

The geometry of the HVAC ducts are modelled in CATIA V5 where the complete wireframe and face data are generated. The data is then translated to IGES format and read into Hyper mesh -12.0 , where the fluid surface model is extracted followed by tri surface mesh generation. T-Grid is used for tetrahedral volume mesh generation. Fig 1 shows a typical tetrahedral mesh of the HVAC duct and nomenclature.

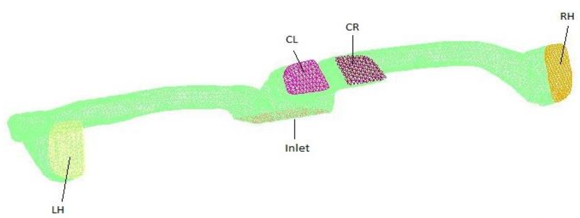

Fig. 1 Meshed Air Distribution Duct

III. NUMERICAL ANALYSIS AND BOUDARY CONDITIONS

The code used throughout the study is the finite volume CFD software Fluent version 14.5. This commercial CFD tool is widely used in the numerical simulation of different flow conditions with various complexities. It is chosen in this study because of its proven capability in flows similar to those investigated here [1]. In present simulation the flow fields are calculated by solving the Realizable K-ε turbulence model. This involves splitting the geometry into many sub-volumes and then integrating the differential equations over these volumes to produce a set of coupled algebraic equations for the velocity components, and pressure at the centroid of each volume. The solver guesses the pressure field and then solves the discretized form of momentum equations to find new values of the pressure and velocity components. This process carries on, in an iterative manner, until the convergence criterion is satisfied.

At inlet the volumetric flow rate is 450 m^3/h. Air density 1.145 kg/m3 at 35 ºC and 101325 pa. The outlet pressure is considered as 1 bar. Flow medium is air, with properties taken at standard temperature and pressure condition. Assumed that the flow is steady and turbulent.Below figures shows the CFD results obtained.

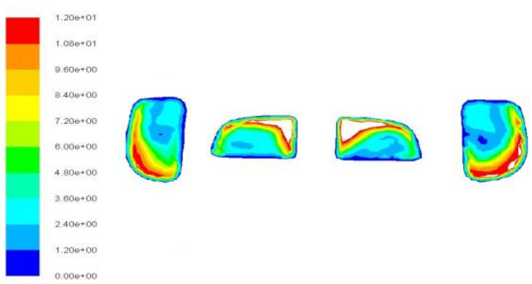

Fig. 2 Velocity Contours at different outlets

In Fig. 2 & Fig. 2 results are shown for the velocity contour and velocity magnitude of duct. It is observed that maximum velocity is 22 m/s in duct and average velocities at duct outlet are shown in Fig .5

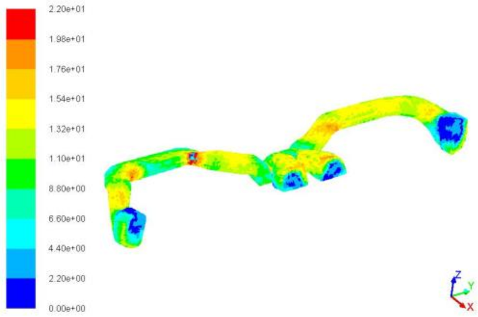

Fig. 3 Contours of Velocity Magnitude (m/s)

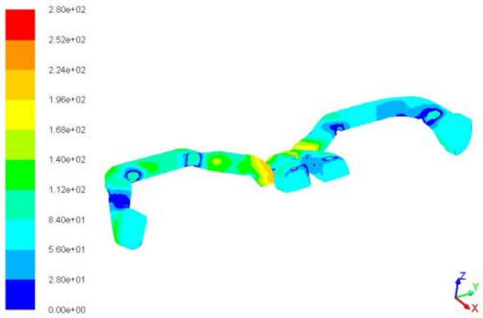

Fig. 4 Contours of Static Pressures (Pascals)

In Fig. 4 results are shown for the pressure contour of duct. In addition to this pressure drop values for given airflow rate, low pressure zone has been shown in figure. Vent distribution box and duct has been designed in such a way that it gives accurate air flow distribution with minimum pressure drop. Further air flow direction at the outlet should be uniform and directing to the passenger as per the comfort conditions. Sudden change of the area and portion of low pressure zone has checked and modified accordingly. Similarly, it has been observed that pressure drop in the module was within acceptable range that has been based on bench mark vehicle and the total air flow requirement.

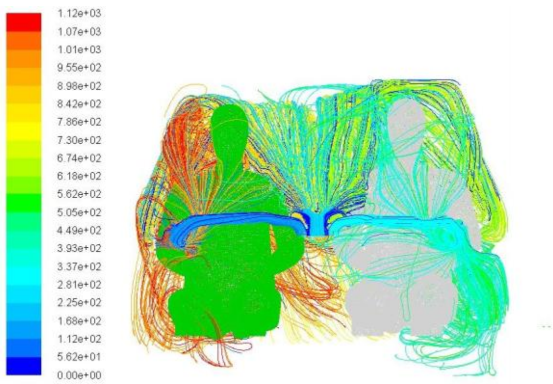

Fig. 5 Pathlines Colored by Particle ID

In Fig. 5 Particle path lines have been presented to show flow field development inside the module with corresponding ducts. It gives total airflow distribution from face. Similarly, complete analysis for all the modes with duct is done to see results at the outlet. This enabled us to understand and predict HVAC system performance which compared with passenger comfort and benchmark vehicle requirements. The results were validated experimentally.

IV. RESULTS AND DISCUSSIONS

CFD obtained results for duct airflow and air distribution has been compared and validated with the experimental test results. In Fig. 5 testing data has been taken from bench level test and same has been compared with CFD simulated data at constant blower speed. It has been observed that CFD result is almost matching with experimental test data. Thus it is proven that CFD methodology can be used accurately for passenger HVAC duct development.

In Table-I result has been shown for air flow distribution in percentage from all vent outlet. Test has been conducted at bench level and air flow measures at the respective ducts outlet. CFD results have been compared with experimental test results and showing almost similar distribution. From Fig. 5 it can also be easily described that centre duct outlets are having more air flow as compare to side duct outlets, meeting exactly the requirement of passenger comfort.

TABLE I
AIR FLOW DISTRIBUTION AT DIFFERENT OUTLETS

Parameter	Inlet	Outlet-1 (LH)	Outlet-2 (CL)	Outlet-3 (CR)	Outlet-4 (RH)
Volumetric Flow(m^3/h)	450	108.65	110.80	118.21	111.61
Distribut-ion (%)	100	24.14	24.62	26.27	24.80
Target Value (%)	100	25	25	25	25

V. CONCLUSIONS

The ability to evaluate and improve the design of the HVAC duct at the very early stage of the development cycle greatly reduces the need for design changes late in the process, which are expensive and time-consuming. Besides reducing the design cycle, CFD tool has also reduced the proto shop and testing expenses for each development project with fewer prototypes. In this paper CFD simulation results and benefits have been discussed with new HVAC system development. All the simulated CFD results have been validated closely with obtained test results. Thus it is concluded that proper use of CFD tool can be used to reduce the overall product development process resulting in optimum cost saving.

ACKNOWLEDGMENT

Thanks to Mr. Swatantra Singh and Mr. Rohan Gulvani for their valuable guidance in helping for the CFD simulations. Also thanks to all who are supported for undergoing the project.

REFERENCES

[1] Ashok Patider, Shankar Nataranjan & Manoj Pande “CFD Analysis & Validation of Automotive HVAC System” SAE International 2009.

[2] Fred Z.Shen, Gerald P Backer, Duon Swanson “HVAC Plenum Design Analysis” SAE International 1995.

[3] Moulay Bel-Hasan,Asad Sardar & Reza Ghias “CFD Simulations of an Automotive HVAC Blower; Operating under Stable & Unstable Flow Conditions” SAE International 2008.

[4] Sumit Tiwari,Roopak Agarwal,Puneet Saxena and James Acre “CFD based design enhancements in passenger vehical HVAC module”.SAE International 2009.

[5] C.Hipp-Kaltoff, A.Eilemann & J.Kilian“Acoustic Optimization of HVAC Systems” SAE International 1997.

[6] Amit K. Ahirrao, H. K. Narahari, S.Umesh, Vivek Kumar “Effect of Vent Shape on Thermal Comfort of passengers in a car”, sastech volume 10- 2011.

[7] Huajun Zhang,Lan Dai,Guoquan Xu,Yong Li,Wei Chen,Wen-Quan Tao,“Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy. Part I: Test/numericalmodel and validation” 2008.

[8] Debashis Ghosh,Mingyu Wang,Edward Wolfe,Kuo-Huey Chen,Shailendra Kaushik & taeyoung, “Energy Efficient HVAC system with spot cooling in an Automobile-Design and CFD”, SAE International 2012.

[9] L.Bennett,C.W.S.Dixon & S.Watkins, “Modelling & Testing of Air Flow in a HVAC Module”,SAE International 2002.

[10] Amit B.Shah,Cralg M.Cless, John S.Curlee,Jr and Jeremy Edmondson, “Thermal Analysis and Simulations for Optimizing HVAC Load on Heavy Trucks” SAE International 2008.

[11] Gurunathan varun Kumar,Meer Reshma Sheerin,Vedachalam Saravana Prabu,Kallikadan Jean,Chaitnya Rajguru,Murugesan Dinesh & Andrew Croft, “Comparison of Different Approaches for temperature Analysis in an Automotive HVAC Sytem.SAE International 2014.

[12] Somnath Sen, “Performance Improvement of HVAC by optimizing the Design of Blower Involute & Casing.SAE International 2010.

[13] Dr.-Ing. Markus Stephan, Dr.-Ing. Dipl.-Phys. Pascal Haubler, Dipl.-Math. Michael Bohm , “CFD Topology Optimizationof Automotive Components” FE-DESIGN GmbH, Karlsruhe, Germany-2009

[14] Huajun Zhang,Lan Dai,Guoquan Xu,Yong Li,Wei Chen,Wen-Quan Tao,“Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy. Part II: Test/numericalmodel and validation” 2008.

