

A Review on CFD Analysis in Air-Conditioning System

Sumesh Pillai*, Saurabh Bhand†, Vikas Shinde†, Siddharth Pawar† and Pravin Katare†

†Mechanical Engineering Department, Marathwada Mitra Mandal's Institute of Technology, SPPU, Pune-47, India

Accepted 01 Oct 2016, Available online 05 Oct 2016, Special Issue-6 (Oct 2016)

Abstract

In today's world air conditioning is almost used everywhere for the sake of human comfort but this comfort was affected by various other parameters like airflow and temperature distribution. This paper focuses on the review of numerical and experimental study of the temperature distribution and air flow in air-conditioning system with varying human load using the simulation software named computational fluid dynamics (CFD).

Keywords: Airflow characteristics, CFD, HVAC, EPA, TDV

1. Introduction

Air conditioning nowadays is being used everywhere these systems basically operate on varying load i.e. human load which brings changes in temperature and air flow. Therefore, study of human comfort with these changes in air flow and temperature parameters is very necessary thus many numerical calculations and experimental setup has been performed on different load conditions to obtain results. In this paper we have reviewed about various air conditioning systems and we have also reviewed experimental setups and numerical simulation done by people. For human comfort and safety of driver air temperature and air flow is very necessary. CFD tool helps in solving these type of problems and helps in estimating the results with the numerical calculations. CFD tool is used in solving various problems in car aerodynamics, HVAC systems, Ventilation systems in hospitals, Offices etc. the results obtained by CFD are almost similar to those performed by numerical method therefore this software is very reliable and widely used in industries in solving problems (Rameshkumar, *et al*, 2013).

1.1 Air-conditioning in car-cabin

Rameshkumar *et al* (2013), has suggested that comfort of human was greatly affected by air quality in air conditioned space. The prediction included evaluation of environmental conditions inside the cabin at specific location thus selecting suitable methods to reach the set points thus contributing the indoor atmosphere by reducing energy costs. The study focused on the numerical aspects of the temperature and air

distribution inside a car cabin with varying human load by using computational fluid dynamics (CFD). Main goal was to investigate the distribution of air and temperature with varying human loads inside the car compartment in the steady-state conditions. The obtained results by using CFD were closed to numerical calculations. The CFD model completely predicted the temperature and air distribution in cabin.

1.2. CFD Analysis of Airflow Characteristics in Office Room

Eunsu Lim *et al* (2011), investigated the performance of airflow inside the office room by hybrid air-conditioning system has been carried out by CFD analysis. Hybrid air-conditioning is composed of supply air outlets required task zone and the natural air supply openings at ambient zone. The estimated data at office room in existence was taken as the boundary condition for natural ventilation opening and required task flow outlets for CFD. For estimating the impact of position of the natural ventilation opening, two models with non-identical position of openings for natural ventilation was created. First case is with the opening of natural ventilation under ceiling and the second case is with the opening of natural ventilation close to the window side. It was seen that the ceiling-model supplied more fresh air more to the occupied region than counter-model. However, Analysis of ceiling-model using CFD tool was carried out to find the influence of outdoor air condition. The distribution of fresh air was more uniform under north wind as compared with south wind. Thus various parameters were examined such as location of the natural ventilation opening, velocity, wind direction and outside temperature on the inside air distribution

*Corresponding author: Sumesh Pillai

pattern was determined by using CFD analysis. Air distribution patterns by natural ventilation were analyzed.

1.3. Computational fluid dynamics analysis of HVAC system in auditorium

M Raman ME (2016), Suggested that CFD simulation had a key role in displacement ventilation system in auditorium by re-arranging the position of diffusers and exhaust thus avoiding wastage of air-flow. In this displacement air system the supply of air is located at floor rather than at ceiling and are nowadays gaining popularity because these systems consume less energy and are more efficient thus maintaining comfort level. These systems are difficult to design due to temperature gradient and draft supply of air. A school auditorium was simulated by using CFD tool so as to evaluate the air flow, temperature and pressure, the simulation showed existence of cold spots and hot spots and well as stagnant areas inside the auditorium. CFD simulation helped in improving the air flow by changing the position of diffusers and exhaust. The results the obtained improved the air supply in auditorium. Change in exhaust and diffuser positions improved the air supply the circulation patterns are better. Both the new and old model had stagnation of air and high temperature issues at back stalls this can be improved by achieving feedback from people in auditorium.

1.4. CFD Analysis of Air Conditioning Equipment of CAD Lab for Enhancing its Performance

K Vaseemul Rahaman *et al* (2015), suggested that the Thermal displacement ventilation (TDV) is very reliable concept for all kind of buildings. Results obtained from analysis by using CFD tool which is essential for modeling and study of complex flows which are connected with TDV. CFD is used for significant forecast and fluid flow analysis, In the year 1995, the Government Accounting Office came to conclusion that about 1/4th of their schools faced indoor air quality problems. After some years the EPA outlined even greater percentage of their buildings faced Indoor air quality problems. Most of these IAQ problems were due to extremely poor ventilation system. TDV, which is been nowadays used in United States, proves wrong that the improvement in IAQ in an air-conditioned space used higher energy consumption. The conclusion made was that many parameters such as velocity of air position of air should be accounted in mechanical systems thus project team came with methodology of developing 3-D model thus performing CFD analysis which was used to evaluate thermal comfort of room. After the study and results from CFD it was observed that there was discomfort at feet/ankle region but there was good air flow for rest of the room. Results reported that cooling the ceiling has benefited in same way like low ceiling air gradient

and that the thermal comfort is obtained for greater metabolism rates thus CFD results were validated with measurements. Even after considering the solar radiations it was seen that the room was mostly affected by indoor vertical velocity component. CFD bridges the gap between understanding and application of CFD analysis and field measurement of natural air flow system which is done by guidelines, best-practices and standards. The study was done to enhance forecast for implementing designing radiant ceiling cooling with mechanical systems engaging CFD and thus showed the proposed system operated with the sensible satisfaction.

1.5. CFD Simulation and Analysis of the Combined Evaporative Cooling and Radiant Ceiling Air-conditioning System

Huang Xiang *et al* (2006), propose that due to the disadvantage of big air ducts and high energy consumption due to current evaporative cooling systems used and the advantage of ceiling cooling system providing comfort and also reducing energy usage. Therefore, a combined system comprising of evaporative cooling system and ceiling cooling system in which evaporative system handles the load and also deals with one part of the sensible load while on the other hand ceiling cooling system deals with other part of sensible load in the cooling zone thus avoiding condensation of radiant panel and lack of cooling capacity. The new air-cooling system and already present outdoor air evaporative cooling system are practically applied in a project Lanzhou city. The Energy consumed analysis are carried out by using the energy consumption code and the velocity of air and the distribution of temperature is calculated using CFD and thus results obtained from both the systems were compared. It was clear that the new system requires less energy, confined space, better thermal comfort and less initial cost. Later it was made clear that the combined system comprising of ceiling cooling system and evaporative cooling system has less energy consumption, less occupied space and can be used in public buildings in arid areas and half arid areas.

1.6. CFD Modeling for Ventilation System of a Hospital Room

Alireza Kermani (2015), investigated that good Efficiency of ventilation can offer reduction in cooling energy utilization of buildings, thus reducing risk of infection through air, increasing the level of comfort increasing comfort in hospital cabins. In this paper, study of air ventilation in a hospital room considering natural and forced ventilation was done, and also the flow of bacteria coming out of a sick patient and this study was done using CFD analysis. After the study it was clear that CFD can be used to improve the comfort level and thus enhancing the air ventilation improving efficiency of hospital building. Good air ventilation also helps in minimizing the risk of air borne infection in building with the help of CFD we can study aerosol

contamination flow characteristics thus optimizing the air flow in building.

1.7. CFD Simulation of Flow Behavior in Evaporating unit of a Window Air-Conditioning System

Ikramuddin Sohail Md (2015), suggested that CFD is essential in engineering analysis. The objective was to see the classic air conditioning system so as to analyze fluid flow properties at different positions by preparing model of window air conditioner and thus found that mass of air flowing through the system is $0.27\text{kg}\cdot\text{s}^2$ on basis of the model geometry the fluid velocity ranges from $1.329\text{m}\cdot\text{s}^{-1}$ to $14.083\text{m}\cdot\text{s}^{-1}$ and variation in static pressure ranges from -1.356 Pa to $+0.348\text{ Pa}$ at inlet and outlet sections of designed air-conditioner. A numerical model of window air conditioned was designed and the analysis was performed using CFD tool. The continuity equations were satisfied and the flow was physically efficient, different contours were constructed at different position for different variables. The velocity over cooling coils should be between 1.5m/s to 3 m/s for air conditioning applications therefore velocity in this case was kept between 1.6m/s to 1.7m/s so that the capacity of cooling of coils remains efficient and thus vapor quantity of refrigerant in tubes is expected to be in vapor state so that there is no liquid at suction and compressor operation continues. The developed model can be used in prediction of changes in design and thus more accurate results can be obtained by using exact material properties and positions where temperature is known.

1.8. Studying Thermal Performance of Split-Type Air-Conditioners at Building Re-Entrant Via Computer Simulation

T.T. Chow *et al* (2011), investigated that the use of split air conditioners has nowadays become very popular in Hong Kong, since it has very effective heat rejection at outdoor condensing units. When these units working together at outdoors the heat rejected by one unit may affect the rejection rate of heat of other unit and hence reduce their performance. At the designing phase an HVAC engineer finds it difficult to forecast such a thermal effect when hundreds of these units are to be placed in a single building. The simulation method is used to tackle the problem and by providing proper spacing between condensing units. The simulation software used is CFD simulation technique is nowadays very useful in studying condenser and heat dissipation from buildings. The approach of simulation was with the $k-\epsilon$ model with the energy evaluation model in analysis. The numerical simulation was supported by experimental approval based on identical thermal studies. In practical it was demonstrated through application example.

The derivation of Condenser Group Performance Indicator grants a portrayal of the total performance whereby single parameter monitors multiple effects. The required expenses and total time in this approach is supported by its application in real life building projects. HVAC engineers can use the technique mentioned for excellent design result.

Conclusions

Now a day's better comfort of air and better air quality has become mandatory thus improvement in design and structure has become important. To experimentally test these parameters it's very difficult and comment on its solution is very time consuming and also wastage of money. Therefore CFD simulation is used for the analysis of various systems mentioned above to get appropriate results and thus comparing with the numerical values obtained. Better air flow characteristics and thermal comfort can be obtained by placing the air inlet vents and exhaust ports at proper positions CFD tool can be used in designing and thus getting the required optimum results.

References

- Rameshkumar *et al*, (2013), CFD Analysis of Air Flow and Temperature Distribution in an Air Conditioned Car, *International Refereed Journal of Engineering and Science*, 2(4), pp. 01-06.
- Eunsu Lim, Kazunobu Sagara, Toshio Yamanaka, Hisashi Kotani, Noriaki Mishima, Susumu Horikawa, and Tomoaki Ushio ,(2011), CFD Analysis of Airflow Characteristics in Office Room with Task Air-Conditioning and Natural Ventilation. *The Kansai Electric Power CO. INC., Japan 3 M Raman ME*, (2016), Computational fluid dynamics Analysis of HVAC system in auditorium, *International Journal of Advanced Research and Development*, vol. 1, pp. 68-72.
- K Vaseemul Rahaman, Prof. Shankar Kumar, (2015), CFD Analysis of Air Conditioning equipment of CAD Lab for Enhancing its Performance, *International Journal of Recent Development in Engineering and Technology*, 1(5) , pp. 68-72
- Huang Xiang, Li Yinming, Wu Junmei, (2006), CFD Simulation and Analysis of the Combined Evaporative Cooling and Radiant Ceiling Air-conditioning System, *HVAC Technologies for Energy Efficiency*, 4(3) , pp. 03-03.
- Alireza Kermani, (2015), CFD modeling for ventilation system of a hospital Room.
- Ikramuddin Sohail Md, (2015), CFD Simulation of Flow Behavior in Evaporating Unit of a Window Air-Conditioning System, *International Journal of Mechanical Engineering and Technology (IJMET)*, 6(10), pp. 243-251.
- T.T. Chow, Z. Lin, Q.W.Wang and J.W.Z. Lu, (2011), Studying Thermal Performance of Split-Type Air-Conditioners at Building Re-Entrant via Computer Simulation, *City University of Hong Kong Tat Chee Avenue, Kowloon Tong, Hong Kong*, 31(2).