DESIGN AND ANALYSIS OF HVAC SYSTEM FOR VEHICLE CABIN

Md Waquar Uddin[1]*, Mr. Inder Singh Nagar, Mr. Pushkar Dwivedi[2]
[1]M. Tech Scholar, Mechanical Engineering, School of Research and Technology, People's University, Bhopal, India.
[2]Assistant Professor, Mechanical Engineering, School of Research and Technology, People's University Bhopal, India.

Abstract: HVAC (Heating, Ventilation and Air-Conditioning) is a system of achieving thermal comfort in any closed space. This comfort level of heat is needed in every kind of workspace and offices. Nowadays, to attain this comfort level, even vehicle compartments are designed in such a way to improve the thermal comfort. In this work, new design of vehicle compartment is proposed to achieve thermal comfort and also different cases of designs are being compared to find the best among them. Four cases consist of different locations of window and inlet ducts. Modeling of compartment is done on CAD software CATIA V5 and CFD simulation is carried out on ANSYS 19 to analyze and observe temperature range and distribution in a compartment. Factors that affect the flow of heat in compartment are studied and designs are improved on the basis of that. Identification of the design in the fourth case has been done as the best among all for achieving thermal comfort. Fourth case shows the lowest temperature among all four cases when observed on different locations of the compartment.

Keywords: HVAC, Air conditioner, Vehicle Cabin, CFD, ANSYS.

I. INTRODUCTION

The internal environment is controlled by the HVAC (Heating, Ventilation, and Air-Conditioning) systems for the whole year so that the comfortable facilities can be ensured under offices, homes along with the commercial services. The HVAC systems are creating a very healthy and wealthy life for humans also more prolific. However, in a suitably controllable environment, a various number of products can be created keeping in mind about the economy, superiority also quickly for the use. The developed countries from all over the world, there is an year-round that is taking place for controlling the environment in residential place, commercial areas, institutions, and industrial sectors.

A. *Components*

The components that are including in the HVAC systems are performing a smooth working in these systems and with the help of this the fresh air can be consumed for the further procedures. The components that are involved in the working of the HVAC systems are mentioned below:

- Air conditioners
- AHUs
- Dehumidifiers/ Heater
- Filters (Pre & HEPA)
- Dust extractor
- Ducting (for delivery of controlled air)
- Supply fans
- Smoke detectors
- Damper
- Humidity/Temperature/ Pressure sensor
- Heating & Cooling coils

B. *Air conditioners*

To change the temperature of air along with the humidity level in a particular area, the air conditioners are designed that are utilized for cooling also heating which are totally based on the properties of the air in that specific time. A simple refrigeration cycle is utilized for making the room temperature cool but sometimes, the technique of evaporation is also used.

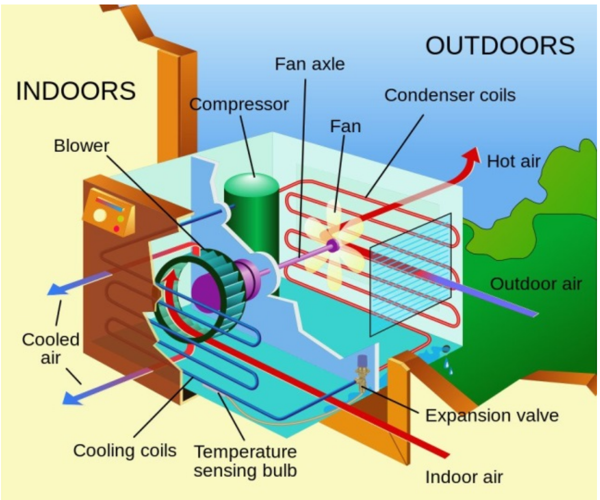

Fig. 1. Air conditioner

C. Classification of HVAC systems

Two main categories, the HVAC systems can be categorized that are intended as conventional and innovative or alternative systems. The convectional systems are usually determined by electrical working that are depending on the HCFCs along with CFCs refrigerants. Taking into consideration for the enhancement of the demands of the energy as well as problems in the environment that are linked with the conventional systems, innovative or alternative systems are managing or controlling the renewable along with low graded sources of energy that are demonstrated as the apparent alternative that can acquire conditions of comfort easily. In a basic sense, the HVAC systems can be categorized under so many ways.

Mainly, based on the fluid media, the HVAC systems are classified which are utilized under thermal distribution system. There are three kinds of HVAC systems which are mentioned below:

- All-air HVAC systems
- Air-water HVAC systems
- All water HVAC systems
- Refrigerant-based HVAC systems (Unitary systems)

D. Thermal Comfort

The expression “thermal comfort” came into life along with the development of HVAC systems. Around six factors were defined as effective parameters of thermal comfort. They are air temperature, air velocity, relative humidity, mean radiant temperature, clothing insulation, and activity level i.e. metabolic rate. “Thermal comfort” is defined by the American Society of Heating Refrigeration and Air Conditioning Engineers (ASHRAE) as “the state of mind that expresses satisfaction with the surrounding environment”. Thermal comfort not only affects the energy consumption of the HVAC system, but is also a key parameter for passenger health. It contributes to safe driving by reducing the driver stress, avoiding windshield fogging, and guaranteeing good visibility. In addition, it is believed that achieving an improved thermal comfort system will lead to substantial cost reductions.

II. LITERATURE REVIEW

(Mohamad Kanaan, 2019)[1]Several strategies and techniques have been recently developed to decrease energy consumption in heating, ventilation, and air-conditioning (HVAC) systems. One method is to recirculate a fraction of return air and make use of the remaining part (exhaust) to preheat fresh air in order to reduce the heating load. However, limitations must be imposed to the return fraction to maintain acceptable indoor air quality (IAQ).

(Lu Zhang, 2019)[2]The change of the transient indoor temperature is crucial to evaluate electrical power consumption of a heat pump and thermal comfort of the room. Computational Fluid Dynamics (CFD) model can provide the most comprehensive information in transient indoor temperature simulations. In CFD models, the transient flow and temperature fields are fully coupled. This coupling leads to high computation cost and time, which limits the CFD application in practice. A segregated CFD model was then proposed to speed up the computation. However, such model is not very accurate during the transient analysis. In this paper, a semi-coupled CFD model combining advantages of the fully-coupled CFD model and the segregated CFD model was proposed. In this CFD model, the flow and the temperature fields are firstly solved by fully coupled simulation during the transient start up period and then the segregated CFD is used for the transient temperature rising simulation.

(Xiaofang Shan, 2019)[3]HVAC systems are utilized to construct a thermally comfortable environment for occupants. As people spend more than 90% of time indoors, thermal conditions of indoor environment constructed by HVAC systems demand precise assessment. Predicted mean vote (PMV), a synthesized index, can reveal thermal conditions by evaluating occupants' thermal sensations. Four environmental parameters affecting PMV: air temperature, air speed, radiant temperature and relative humidity. This study integrates CFD simulations and wireless-sensor measurements to assess distributions of PMV considering radiation models. The distributions of environmental parameters: velocity, temperature, radiant temperature, inside an office room with fan coil unit (FCU) are firstly presented. Based on these distributions, spatial profiles of PMV are obtained to intuitively illustrate thermal conditions. Combined with experimental database collected by thermal-flow wireless-sensors, CFD simulations offer detailed predictions of indoor airflow and thermal parameters.

(Chen Ren, Shi-Jie Cao, 2019)[4]It has been of great importance to develop better control techniques for heating, ventilation and air conditioning (HVAC) systems to provide occupancy driven energy and comfort management due to the significance of building energy conservation. Following with our previous work, where low-dimensional linear ventilation model (LLVM) and artificial neural network (ANN) were incorporated to realize online control of indoor air quality (IAQ), we continued to expand the work on indoor thermal comfort (ITC) with low-dimensional linear temperature model (LLTM) and contribution ratio of indoor climate (CRI) to provide dependable support for HVAC online control. Two steps were required to be implemented, respectively considering pollutant and temperature responses. As a premise for control, the database was constructed by CFD, which were verified by the corresponding experiments. Linear ventilation model (LVM) and linear temperature model (LTM) could be well employed to expand the CFD database.

(Popovici, Hudișteanu and Cherecheș, 2018)[5]intended to highlight the role of the ventilation and air conditioning system for a theatre. It was chosen as a case study the "VasileAlecsandri" National Theatre of Jassy. The paper also sought to make a comparison in three distinct scenarios for HVAC Main Hall system - ventilation and air conditioning system of the Main Hall doesn't work; only the ventilation system of the Main Hall works and ventilation and air conditioning system of the Main Hall works. For analysing the comfort parameters, the ANSYS-Fluent software was used to build a 2D model of the building and simulation of HVAC system functionality during winter season, in all three scenarios. For the studied scenarios, the external conditions of Jassy and the indoor conditions of the theatre, when the entire spectacle hall is occupied were considered. The main aspects evaluated for each case were the air temperature, air velocity and relative humidity. The results are presented comparatively as plots and spectra of the interest parameters.

(Zhijian Liu, 2018)[6] With the outbreaks of SARS, MERS, H7N9 and many other epidemic diseases, microbial contamination has become a significant focus in both HVAC and epidemiology fields. HVAC systems are the "respiratory system" of modern buildings, regulating indoor temperature, humidity, airflow and cleanness. However, they will be the sources of microbial contamination worsening indoor environment and threatening occupants' health if they are improperly designed or operated. To prevent and reduce microbial contamination, it is essential to have an overall understanding of microbial characteristics in HVAC systems. Therefore, this paper provides a comprehensive review of the distribution characteristics, growth and transmission modes of microorganisms in HVAC systems, as well as the microbial control strategies.

(Wei Tian, 2017)[7]Multizone models are widely used in building airflow and energy performance simulations due to their fast computing speed. However, multizone models assume that the air in a room is well mixed, consequently limiting their application. In specific rooms where this assumption fails, the use of computational fluid dynamics (CFD) models may be an alternative option. Previous research has mainly focused on coupling CFD models and multizone models to study airflow in large spaces. While significant, most of these analyses did not consider the coupled simulation of the building airflow with the building's Heating, Ventilation, and Air-Conditioning (HVAC) systems. This paper tries to fill the gap by integrating the models for HVAC systems with coupled multizone and CFD simulations for airflows, using the Modelica simulation platform. To improve the computational efficiency, we incorporated a simplified CFD model named fast fluid dynamics (FFD). We first introduce the data synchronization strategy and implementation in Modelica. Then, we verify the implementation using two case studies involving an isothermal and a non-isothermal flow by comparing model simulations to experiment data. Afterward, we study another three cases that are deemed more realistic. This is done by attaching a variable air volume (VAV) terminal box and a VAV system to previous flows to assess the capability of the models in studying the dynamic control of HVAC systems. Finally, we discuss further research needs on the coupled simulation using the models.

(Ibrahim, Lal and Metha, 2016)[8]study and analyze the air distribution and thermal effects of the conditioned air that is supplied to the cabin. This is because all the passengers should be comforted with the ac system. The ac system is one of the most popular features of the automobile industry. In this work, a numerical investigation is carried out using the commercial software ANSYS about the flow and thermal analysis of the cabin air circulation. We had created a 3d model of the maruti 800 car using design modeler and it is analyzed using the FLUENT software. Contours of velocity and temperature distribution are studied in detail. Four different combinations like front vent, front and rear vent, front and rear side vent, front and rear top vent are created and analyzed. The best configuration is find out from the temperature and velocity analysis.

(Sinha and Majumdar, 2016)[9]investigation the distribution of mean velocity are experimentally studied on three constant area rectangular curved ducts with an aspect ratio of 2.4. First one is C-shape, second one is S-shape and third one is a DS-shape duct. The experiment is carried out at mass averaged mean velocity of 40m/s for all the ducts. The velocity distribution shows for C-duct, the bulk flow shifting from outer wall to the inner wall along the flow passage and for S-duct, the bulk flow shifting from outer wall to the inner wall in the first half and from inner wall to the outer wall in the second half along the flow passage of curved ducts are very instinct. Due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the forms of counter rotating vortices have been generated within both the curved duct. For DS-duct the velocity distributions shows the Bulk of flow shifting from inner watt to outer wall in the first bend and third bend of the duct and outer wall to inner wall in the second bend and forth bend of the duct along the flow passage is very instinct. Flow at end of the DS-duct is purely uniform in nature due to nonexistence of secondary motion. The experimental results then were numerically validated with the help of Fluent, which shows a good agreement between the experimental and predicted results for all the ducts.

(Calautit *et al.*, 2015)[10] Mechanical Heating Ventilation and Air-Conditioning (HVAC) systems account for 60% of the total energy consumption of buildings. As a sector, buildings contributes about 40% of the total global energy demand. By using passive technology coupled with natural ventilation from wind towers, significant amounts of energy can be saved, reducing the emissions of greenhouse gases. In this study, the development of Computational Fluid Dynamics (CFD) analysis in aiding the development of wind towers was explored. Initial concepts of simple wind tower mechanics to detailed design of wind towers which integrate modifications specifically to improve the efficiency of wind towers were detailed. From this, using CFD analysis, heat transfer devices were integrated into a wind tower to provide cooling for incoming air, thus negating the reliance on mechanical HVAC systems. A commercial CFD code Fluent was used in this study to simulate the airflow inside the wind tower model with the heat transfer devices.

(Shojaeefard *et al.*, 2015)[11]improving HVAC system performance leads to more energy saving of the vehicle which is a critical factor for nowadays automotive. Besides, one crucial task of HVAC system is defrosting/defogging of windshield which is considered as a mandatory requirement in most countries. In the current study, the defrosting/defogging performance of HVAC system in the main product of national vehicle platform is numerically evaluated based on the ECE-78-715 legal requirement. For this purpose, after validation and mesh independency study, the transient air flow in three-dimensional cabin geometry is simulated by SSTk-ω turbulence model via ANSYS Fluent software and the windshield thermal condition is reported during defrosting/defogging. Besides, two national HVAC standards of AERC-10-01 and AERC-10-02 are also checked.

(Hongming Zhou, 2015)[12] Maintaining a desired comfort level while minimizing the total energy consumed is an interesting optimization problem in Heating, ventilating and air conditioning (HVAC) system control. This paper proposes a localized control strategy that uses Computational Fluid Dynamics (CFD) simulation results and K-means clustering algorithm to optimally partition an air-conditioned room into different zones. The temperature and air velocity results from CFD simulation are combined in two ways: 1) based on the relationship indicated in predicted mean vote (PMV) formula; 2) based on the relationship extracted from ASHRAE RP-884 database using extreme learning machine (ELM). Localized control can then be effected in which each of the zones can be treated individually and an optimal control strategy can be developed based on the partitioning result.

(Banjac, 2014)[13]paper illustrates the basic methodology and principal problems of CFD (Computational Fluid Dynamics) approach in designing a proper cooling system (capacity of cooling, determination of optimal position, number and size of openings for cooling, etc.) in special purpose objects. Taking as an example the design of the cooling for electrical equipment placed in a container of crane MK-46 - "GOSA FOM", the problem of the selection of proper cooling system was demonstrated. It has been shown that the numerical fluid dynamics represents a superior approach to the design of these systems., with its ability to, provide precise three-dimensional images of velocity and temperature field in the fluid even before the construction of the facility, as well as to predict the changes of those fields that would appear with modification of geometry, boundary and spatial conditions (opening for injection and exhausting of air, position of the walls and obstacles)

(Mohan and Govindarajan, 2011)[22] Experimental and theoretical investigations of the thermal performance of a variety of heat sinks have been made. The heat sinks investigated were: straight finned, elliptical finned, small pin finned, circular disc finned, elliptical disc finned, frustum finned and double base straight finned. Realistic, manufactural geometries are considered for minimizing thermal resistance at low velocity. The experimental results of several of the simple geometry heat sinks have been compared to those predicted by a commercially available computational fluid dynamics code fluent.

III. METHODOLOGY

Expected Procedure to be followed during the complete study:

1. Collect the raw data of HVAC System
2. Design the Model
3. Import the geometry in ANSYS
4. Assigning the name selection to the different parts of HVAC System.
5. Meshing of Model for performing the simulation process
6. Providing the suitable boundary conditions according to the selected base paper.
7. Assigning the material properties.
8. Setting the proper setup for CFD analysis procedure.
9. Evaluating the results after the finish of simulation work.

A. Design of Model

For achieving thermal comfort, different cases are taken. In each case, geometry according to different locations of window and duct is changed and best one is observed. The geometry in case 1 involves a window, an inlet duct just beneath the window on a side and one present below the sitting area. Three outlet ducts can be seen on the opposite side of window.

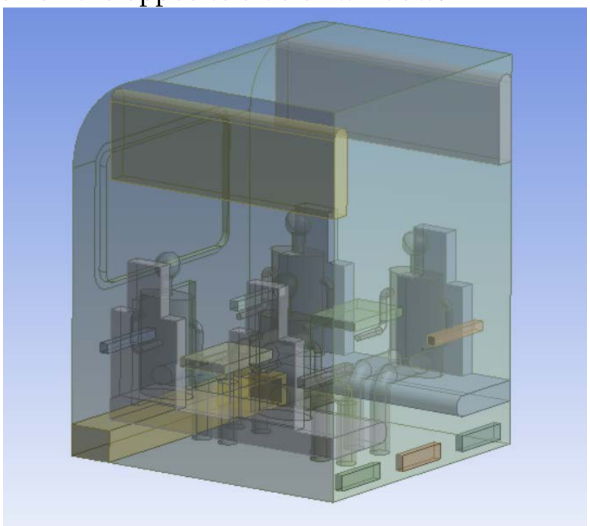

Fig. 2. Geometry of vehicle cabin

B. Meshing

- Meshing is defined as the process of dividing the whole component into number of elements so that whenever the load is applied on the component it distributes the load uniformly called as meshing.
- After Meshing, the entire structure is

divided into number of elements and each element having its own stiffness while loading.

Meshing is done for the model of compartment with tetrahedron meshing element. Nodes and elements are given below, i.e. 292772 and 1515587 respectively. Meshing is shown below in figure 3.

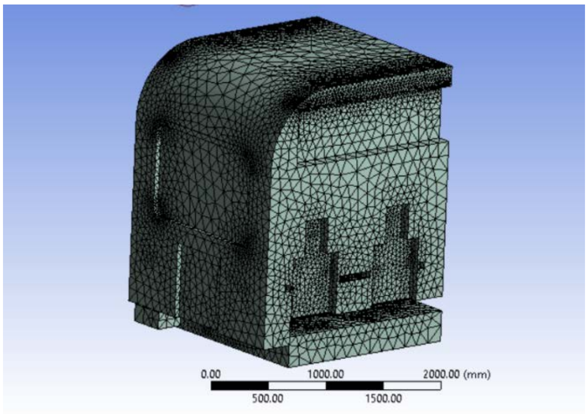

Fig. 3.Meshing of compartment model

Four locations were taken in each case at single plane position that is z = -0.58.Four different locations are chosen on this plane and temperatures are taken at these four locations. Locations are given below in the table. For different cases, the temperature on these locations is compared in figure 4.

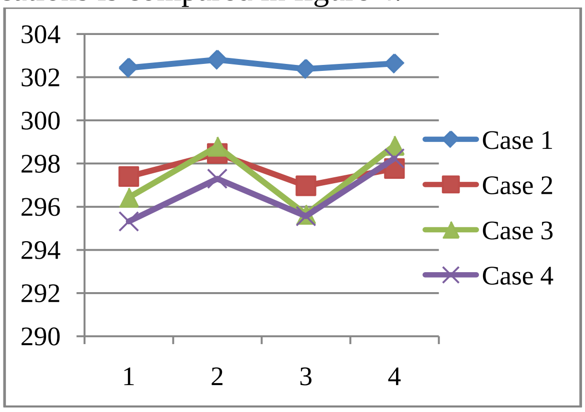

Fig. 4.Comparison of four cases on given location of z = -0.58

At z = 0.58, four locations are taken presented in table below. For different cases, the temperature on these locations is compared in figure 5.

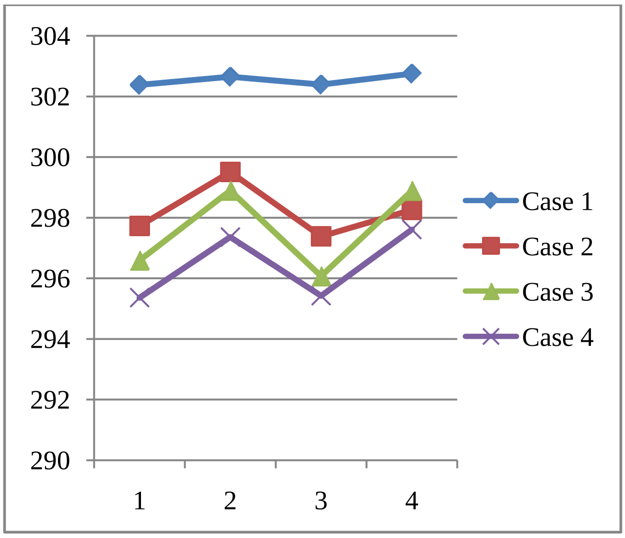

Fig. 5.Comparison of four cases on given location of z = 0.58

At z = 0, four locations are taken that are presented in table below. For different cases, the temperature on these locations is compared in figure 6.

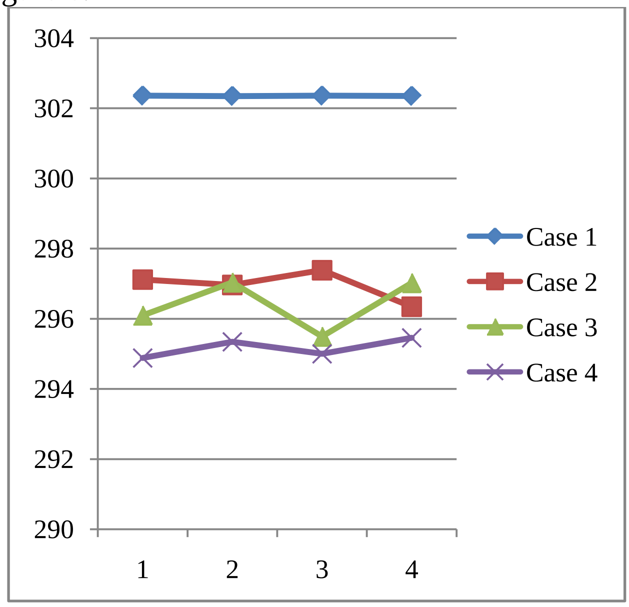

Fig. 6.Comparison of four cases on given location of z = 0

IV. CONCLUSION

A compartment is taken to measure the thermal comfort at different locations. To improve the thermal results, some design changes were made after observing results from previous designs. Changes in results shows the effect of design changes in the compartment.

Case I – This case consist of the basic model of compartment. In this case, it is observed that, due to the placement of window the air from the inlet collides with the edge of window and thus remains unable to flow in the compartment.

Case II – In this case, since the edgy design of window was stopping the air from inlet to rise. This has been countered by giving the fillet to the edges of windows which gives the path for the air to rise. Also the inlet duct has been relocated to the center position of the compartment instead of single sided position. This maintains the temperature of the compartment evenly.

Case III – To improve the results, further the placement of window was done from outside. Thus there is no interference of any edge of the window on inside.

Case IV – The last case consist of the changes of the location of the inlet duct along with the position of the window. Window is shifted towards outside and thus do not hinders the air flow.

Out of these four cases, four different designs of compartment are made and analyzed. Fourth case among all, stands best in terms of thermal comfort. In this case, temperature obtained is the lowest and in thermal comfort zone and also temperature distribution is great as observed in contour.

REFERENCES

[1] M. Kanaan, "CFD optimization of return air ratio and use of upper room UVGI in combined HVAC and heat recovery system," *Case Stud. Therm. Eng.*, vol. 15, no. February, p. 100535, 2019.

[2] L. Zhang, X. Yu, Q. Lv, F. Cao, and X. Wang, "Study of transient indoor temperature for a HVAC room using a modified CFD method," *Energy Procedia*, vol. 160, no. 2018, pp. 420–427, 2019.

[3] X. Shan, W. Xu, Y. K. Lee, and W. Z. Lu, "Evaluation of thermal environment by coupling CFD analysis and wireless-sensor measurements of a full-scale room with cooling system," *Sustain. Cities Soc.*, vol. 45, no. December 2018, pp. 395–405, 2019.

[4] C. Ren and S. J. Cao, "Development and application of linear ventilation and temperature models for indoor environmental prediction and HVAC systems control," *Sustain. Cities Soc.*, vol. 51, p. 101673, 2019.

[5] C. G. Popovici, S. V. Hudişteanu, and N. Cherecheş, "The necessity of HVAC system for the registered architectural cultural heritage building," *E3S Web Conf. 32, EENVIRO 2017*, vol. 01008, pp. 1–5, 2018.

[6] Z. Liu, S. Ma, G. Cao, C. Meng, and B. J. He, "Distribution characteristics, growth, reproduction and transmission modes and control strategies for microbial contamination in HVAC systems: A literature review," *Energy Build.*, vol. 177, pp. 77–95, 2018.

[7] W. Tian, T. A. Sevilla, W. Zuo, and M. D. Sohn, "Coupling fast fluid dynamics and multizone airflow models in Modelica Buildings library to simulate the dynamics of HVAC systems," *Build. Environ.*, vol. 122, pp. 269–286, 2017.

[8] S. Ibrahim, A. S. Lal, and R. C. Metha, "CFD INVESTIGATION OF CABIN AIR FLOW AND FINDING OUT THE MOST EFFICIENT DUCT COMBINATIONS," *IJRET Int. J. Res. Eng. Technol.*, pp. 2319–2321, 2016.

[9] P. K. Sinha and B. Majumdar, "A Computational Investigation of Flow Structure Within a Sinuous Duct," *Int. J. Eng. Res. Appl.*, vol. 6, no. 3, pp. 115–124, 2016.

[10] J. Calautit, D. O'Connor, P. Sofotasiou, and B. Hughes, "CFD Simulation and Optimisation of a Low Energy Ventilation and Cooling System," *Computation*, vol. 3, no. 2, pp. 128–149, 2015.

[11] M. H. Shojaeefard, G. R. Molaeimanesh, N. Aghamirzaei, S. Ghezelbiglo, and B. Zeinolabedini, "Numerical evaluation of the defrosting / defogging performance of HVAC system in the main product of the national vehicle platform," *Int. J. Automot. Technol.*, vol. 5, no. 4, 2015.

[12] H. Zhou, Y. C. Soh, and X. Wu, "Integrated analysis of CFD data with K-means clustering algorithm and extreme learning machine for localized HVAC control," *Appl. Therm. Eng.*, vol. 76, pp. 98–104, 2015.

[13] M. J. Banjac, "Application of computational fluid dynamics in cooling systems design for special purpose objects," *FME Trans.*, vol. 42, no. 1, pp. 26–33, 2014.