Celal Bayar University Journal of Science
Volume 18, Issue 4, 2022, p 409-416
Doi: 10.18466/cbayarfbe.1073131

Celal Bayar University Journal of Science

Numerical Simulation of Different Ventilation Systems in an Airplane Cabin

Levent Bilir[1]*, Hasan Çelik[2], Barış Özerdem[3]
[1] Yaşar University, Energy Systems Engineering Department, İzmir/Turkey
[2] İzmir University of Economics, Mechanical Engineering Department, İzmir/Turkey
[2] İzmir University of Economics, Aerospace Engineering Department, İzmir/Turkey

* levent.bilir@yasar.edu.tr
* Orcid: 0000-0002-8227-6267

Received: 16 February 2022
Accepted: 17 October 2022
DOI: 10.18466/cbayarfbe.1073131

Abstract

Airplanes are the most popular way of transportation worldwide, especially for long haul. It facilitates the growth of global trade as well, besides promoting tourism and other employment developments. Passenger comfort and hygiene inside an airplane cabin became main concern for aircraft manufacturers. The possibility for a potential spread of infectious virus or bacteria even maximized this concern. Therefore, supplying sterile and particle-free air inside the aircraft cabin became extremely crucial more than ever. In order to ensure comfort and hygiene, regardless of the environment conditions inside the aircraft cabin, paved the way for researchers to focus on this topic, recently. It is obvious that, an important precaution for the spread of micro-organisms can be selecting an adequate air ventilation system inside the airplane cabin. In this study, a part of an airplane passenger cabin is modelled for four different scenarios. The streamlines of air, which is sent to the cabin from air ducts, are obtained and air flow path is observed for the investigated cases. The results of the numerical simulations are presented as the outcomes of this study. It is observed that the air mixing between different seat rows occur slightly only for sidewall supply and bottom return mixing ventilation and displacement ventilation systems, whereas the air mixing for the same seat row is seen for all ventilation systems. In conclusion, sidewall supply and bottom return mixing ventilation system is found the most appropriate one, even though it causes air recirculation at the same row seats.

Keywords: Air flow, Ansys-Fluent analysis, numerical analysis, airplane passenger cabin.

1. Introduction

Using of airplanes is the fastest transportation mode which is suitable for traveling people and carrying of goods over a long distance. Therefore, the air breathing in the cabin by passengers and crew must be clean and micro-organism free, as much as possible. The air quality along with thermal comfort are directly related to the ventilation system used in aircraft cabins. Recent studies show that airborne diseases spread rapidly in non-ventilated environments. Because of that, researchers and design engineers try to discover new methodologies and techniques to enhance air quality. Some of the aforementioned methodologies can be found in Conceicao et al. [1] and Khalil [2], in detail. The literature survey shows that the passenger cabins of airplanes and trains are studied frequently by many researchers, experimentally and numerically. Some experimental studies to give insight into airflow distribution in the cabin are summarized as follows. Melikov and Dzhartov [3] investigated the performance of a personal ventilation combined with a local exhaust at each seat, experimentally. They also studied the distribution of airflow to minimize airborne cross-infection in a cabin environment. Cao et al. [4] made 2-dimensional Particle Image Velocimetry (PIV) measurements for the air distribution inside an airplane cabin. They provided air velocity distribution and turbulence intensity contours using the obtained measurement data. Zhang et al. [5] conducted an experimental study and compared different ventilation systems. Their results revealed advantages and disadvantages of the considered systems. Li et al. [6] studied the dynamic behavior of the airflow inside the cabin by 2D-PIV. They measured the characteristics of airflow from several different points in the mock-up cabin and concluded the factors that affect dynamic thermal comfort. Wu et al. [7] investigated criteria required for personal air supply in aircraft cabins. They measured indoor air quality by using a monitoring station. They concluded the impacts needed to be considered for the thermal comfort of the passengers.

The increase in computing power led the researchers to investigate aircraft cabins computationally by using the techniques of Computational Fluid Dynamics (CFD), as well. The passenger cabins of airplanes and trains have been analyzed by many researchers so far. Mazumdar and Chen [8] took a seat row inside an airplane cabin into consideration and made a 1-dimensional analytical investigation for longitudinal transport of a contaminant. Fiser and Jicha [9] modeled the passenger cabin of a small airplane, numerically. They focused on three different air distribution systems under different ambient conditions. As a result, the most stable air distribution and the best quality of the cabin ventilation are determined. Wang et al. [10] investigated air distribution systems of three different high-speed train cabins. They presented air velocity distribution and cough droplets removal capability in the analyzed cabins. Yang et al. [11] performed a numerical study to determine smoke spread inside an airplane cabin. They showed the smoke distribution, air velocity vectors and CO_2 concentration at different cross sections. Maier et al. [12] evaluated different ventilation systems for an airplane cabin in thermal comfort aspect. Kotb and Khalil [13] investigated the path followed inside an aircraft cabin by a cough droplet from a moving passenger. The distribution of ozone in an aircraft cabin in terms of air pollution is studied by Shi et al. [14]. Three different air supply methods are considered in their study and compared with each other in terms of ozone concentration in the cabin. The design methodology of a cabin air supply nozzle is analyzed by Pan et al. [15]. The shape of air supply nozzle is redesigned to reduce particle deposition in their study. Wang et al. [16] investigated evaluation of dynamic airflow structures in a single-aisle aircraft cabin mock-up, both experimentally and numerically. They studied the effect of several parameters such as air supply speed, air supply angle and swing amplitude of the instantaneous airflow structure. Thysen et al. [17] evaluated the air movement inside an empty airplane cabin using a CFD program with different turbulence models. They considered the air released from two opposite wall jets. They also made PIV measurements. They found acceptable similarity for global flow patterns for all simulations with laboratory measurements. Pirouz et al. [18] made CFD simulations of air movement inside a typical car, bus and airplane. They concluded that the minimum health risk due to contaminated respiration aerosols is in airplane cabin.

In the present study, a three-seat row section of a commercial airplane cabin was modeled using Ansys-Fluent CFD software. Ceiling supply bottom return mixing ventilation (CMV), sidewall supply and bottom return mixing ventilation (SMV), ceiling and sidewall supply and bottom return mixing ventilation (CSMV)and displacement ventilation (DV) systems are evaluated for air distribution characteristics inside the cabin. The streamlines of the air are presented to illustrate how air moves inside the aircraft cabin. In order to validate the numerical model used in this study, first, a numerical model which considered experimental study conducted by Zhang et al. [5] was created. And then, the streamlines presented by Zhang et al. [5] were compared with the results found for this created model. It was seen that, there is a very good agreement between experimentally obtained and numerically calculated results. Therefore, it is concluded that the numerical results obtained for different cases in the present study are very much reliable.

2. Methodology

As the computer source is limited for numerical solutions, a section consisting of three row seats is modeled using Ansys-Fluent software. Furthermore, half of the airplane section including three row seats is taken into consideration for the model due to geometrical symmetry. The view of the model is given in Fig. 1a. In order to model the cabin, to generate control volumes, and to get numerical results, an HP Z840 computer is used during this study.

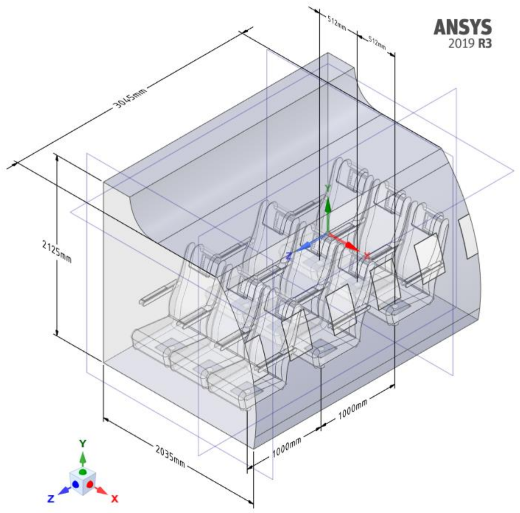
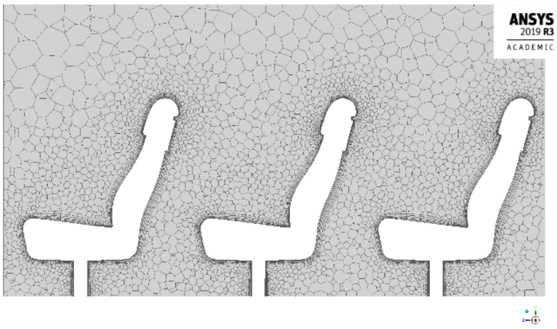


Figure 1. View of the airplane cabin model.

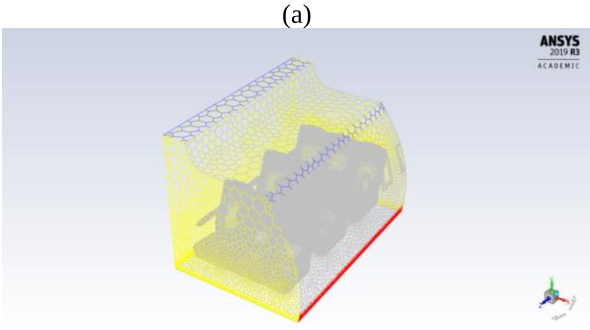
Firstly, the area of interest is divided into tetrahedral meshes and then these tetrahedral meshes are converted into about 8.2 million polyhedral meshes for all investigated cases. The use of polyhedral control volumes enabled us to obtain converged results in a moderate time. The average skewness and the orthogonality values for the used meshes are 0.24769 and 0.74332, respectively. The view of the generated polyhedral meshes is presented in Figure 2a. A boundary layer is created near the surfaces of the seats and walls as can be seen in the same figure. The front and back surfaces of the model (seen in yellow in Figure 2b) are defined as symmetrical surfaces. The blue colored surfaces are determined as air mass flow inlet for CSMV

Celal Bayar University Journal of Science
Volume 18, Issue 4, 2022, p 409-416
Doi: 10.18466/cbayarfbe.1073131

case. However, the air inlet surface changes depending on the investigated case. Only the top blue colored surface is defined as air inlet for CMV case, while only the side blue colored surface is taken as air inlet for SMV case. The red colored surface is defined as outflow for these three cases. For the DV system, the air inlet surface is only the red colored surface while the air outlet surface is only the top blue colored surface. Lastly the top, bottom and side surfaces (gray colored surfaces shown in Figure 2b) are defined as walls.



(a)



(b)

Figure 2. a) View of the generated polyhedral meshes. b) View of the boundary condition surfaces.

The needed supply airflow is determined as 9.4 liters/s per person taken from the study of Zhang et al. (2017). As nine passengers will be travelling in the modeled section of the airplane cabin, total needed airflow is calculated as 84.6 liters/s. By considering this value, the supply air mass flow rate is calculated as 0.11 kg/s.

ANSYS-Fluent software provides the air streamlines inside the cabin, which are the indicator of the contaminants' path as well. The software solves Navier-Stokes equations presented in Equations (1-4) along with turbulence equations in order to obtain the flow field.

$Continuity$:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{1}$$

$X - Momentum$:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial \rho}{\partial x} + \frac{1}{Re_r}\left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z}\right] \tag{2}$$

$Y - Momentum$:

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial \rho}{\partial y} + \frac{1}{Re_r}\left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z}\right] \tag{3}$$

$Z - Momentum$:

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial \rho}{\partial z} + \frac{1}{Re_r}\left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}\right] \tag{4}$$

Realizable k-ε turbulence model with enhanced wall treatment is used as the turbulence model for the numerical solution due to its accuracy and easy convergence. The realizable k-ε model differs from the standard k-ε model in two important ways [19];

- The realizable k-ε model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ε, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG k-ε model is realizable. An immediate benefit of the realizable model is that it more accurately predicts the spreading rate of both planar and round jet flows. It is also likely to provide superior performance for jet flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation [19].

In the present study, as the recirculation of air inside the airplane cabin is encountered frequently, the use of realizable k-ε model for turbulence is considered as adequate.

The modelled transport equations for k and ε in the realizable model are [19];

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{5}$$

and

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\gamma\varepsilon}} + C_{1\varepsilon} \tag{6}$$

Celal Bayar University Journal of Science
Volume 18, Issue 4, 2022, p 409-416
Doi: 10.18466/cbayarfbe.1073131

where,

$$C_1 = max\left[0.43, \frac{\eta}{\eta+5}\right], \eta = S\frac{k}{\varepsilon}, S = \sqrt{2S_{ij}S_{ij}}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients and G_b represents the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. C_2 and C_ε are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms [19].

The assumptions made for the numerical solution are;

- An airplane cabin section of three rows with three seats, each, is considered in the numerical analyses.
- The side, front and back surfaces of the model are considered as symmetry surfaces.
- Thermophysical properties of air are taken constant in the numerical analyses.
- The supplied air is sent to the cabin homogeneously from the inlets.
- The effect of people seating on the cabin is neglected.

Considering these assumptions, the following flow diagram in Figure 3 for numerical simulations can be given;

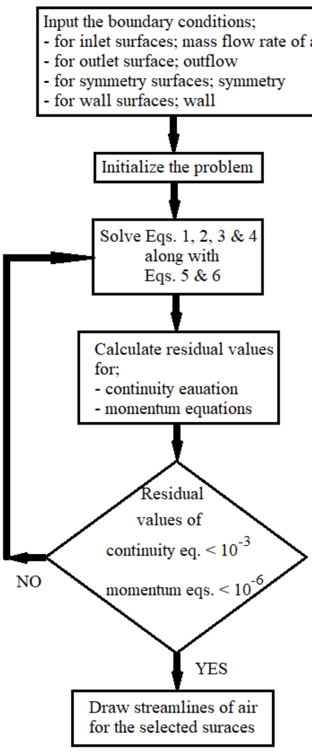


Figure 3. Flowchart for solution algorithm.

3. Results and Discussion

After the modeling and meshing operations, numerical solutions are obtained for four different ventilation systems. The iterations are continued until getting lower residual values of 10^{-3} for continuity equation and 10^{-6} for all other equations. The cross sections, from which the air streamline views are taken, are illustrated in Figure 4.

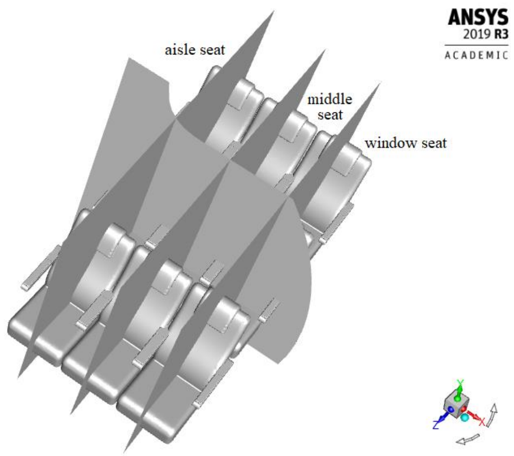
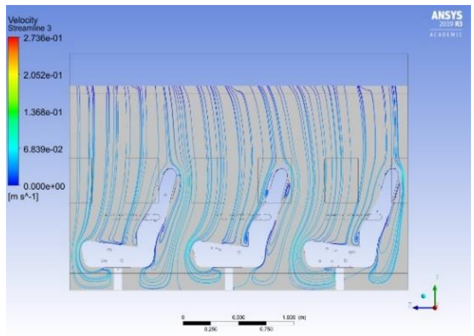
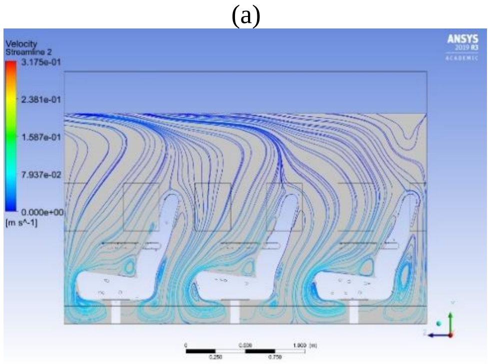


Figure 4. Cross sections from which air streamlines are visualized.

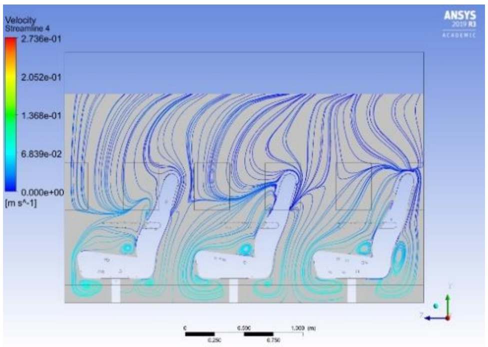
As a result, the air movement inside the airplane passenger cabin is visualized at the cross section which cuts the three aisle, middle and window seats (a, b and c, respectively) and at the cross section of a seat row (d) in Figures 5 (CMV case), 6 (SMV case), 7 (CSMV case) and 8 (DV case).



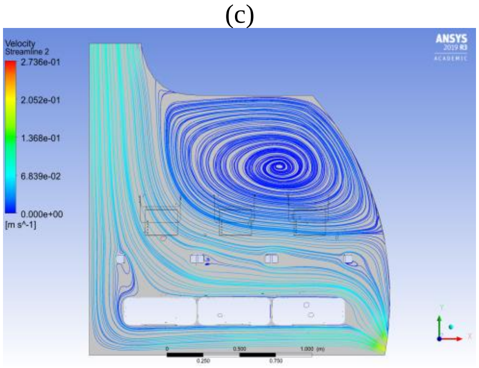
(a)



(b)



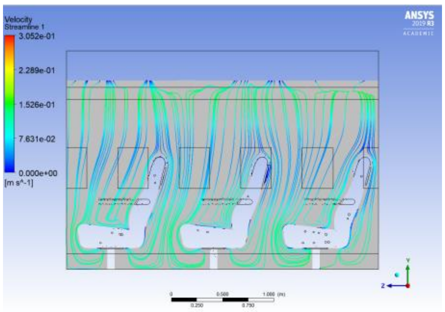
(c)



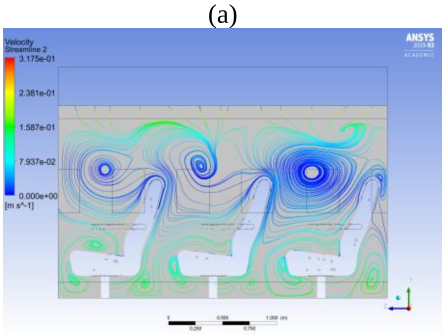
(d)

Figure 5. Air streamlines at different cross sections for CMV case. [20]

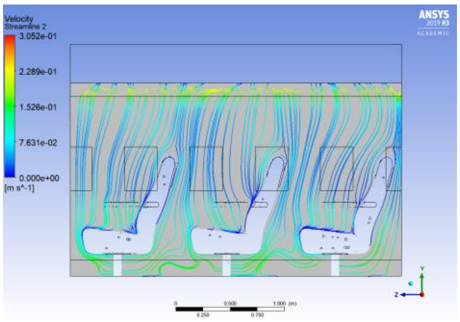
It can be seen from Figure 5 that for the CMV case, the air mixing is not significant for the aisle and middle seats, whereas air stream of window seat shows relatively significant mixing. However, the air still does not reach to the front and back row window seats. Nevertheless, the air mixing for the seats at the same row is significant as can be seen from Figure 4d.



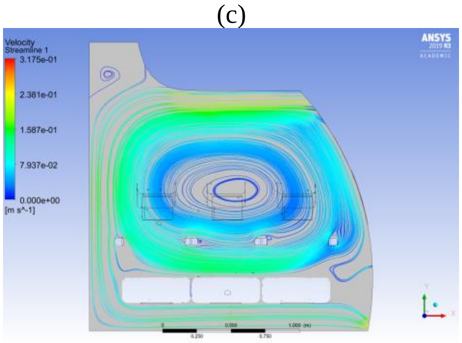
(a)



(b)



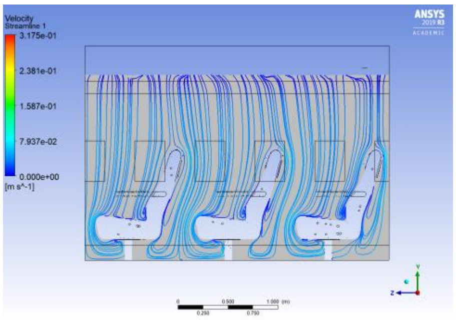
(c)



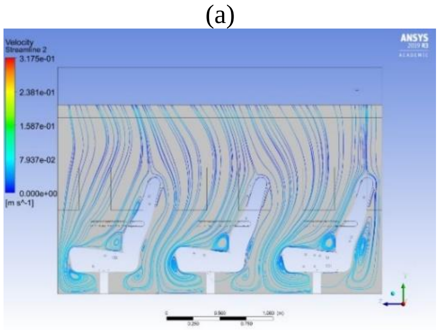
(d)

Figure 6. Air streamlines at different cross sections for SMV case.

For the SMV case (Figure 6), a similar airstream pattern is observed for the aisle seats as the CMV case, nevertheless for the middle seats, strong air recirculation is encountered. For the window seats, smooth air streamlines are seen, as well. For the SMV case, there is a stronger air recirculation for the seats at the same row than the one for the CMV case. Hence, it can be noted that SMV air distribution strategy is not appropriate when it is compared with the CMV air distribution strategy.

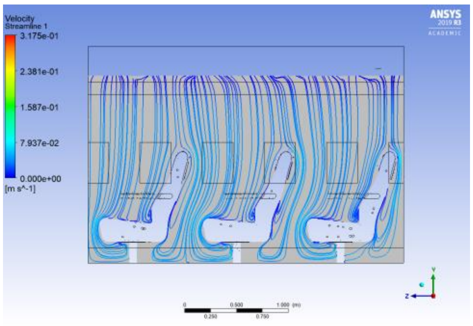


(a)

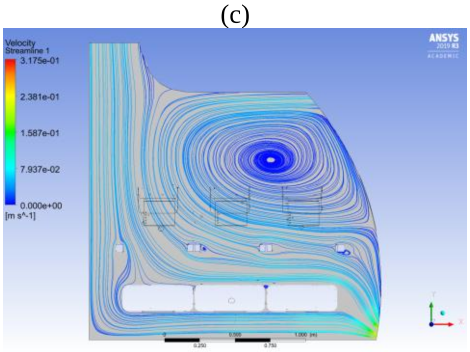


(b)

Celal Bayar University Journal of Science
Volume 18, Issue 4, 2022, p 409-416
Doi: 10.18466/cbayarfbe.1073131



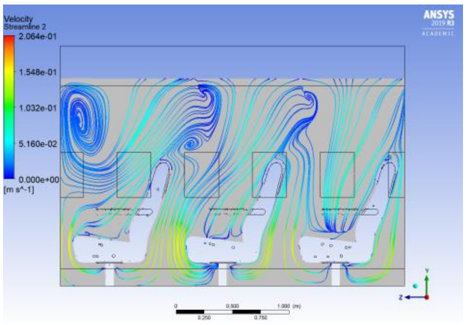
(c)



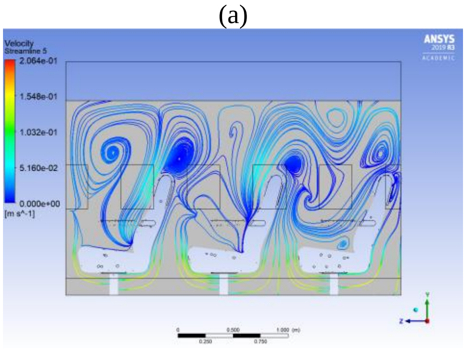
(d)

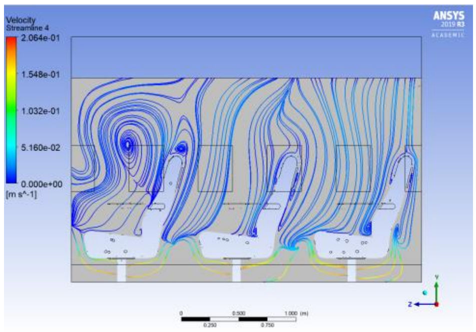
Figure 7. Air streamlines at different cross sections for CSMV case.

It can be seen from Figure 7 that, the airstreams for the aisle, middle, and window seats are very smooth and there is no air mix between front and back row seats for the CSMV case. Even though the streamlines for the same row seats show recirculation pattern, this pattern is not as strong as the CMV and SMV cases. So, it can be concluded that CSMV air distribution strategy is more preferable against CMV and SMV strategies.

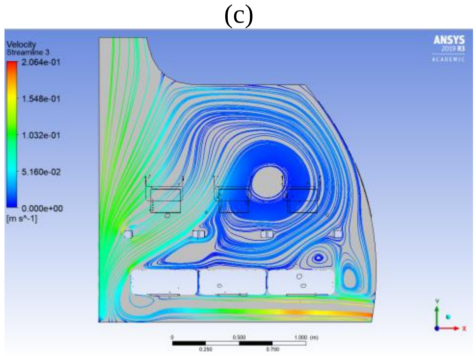


(a)





(c)



(d)

Figure 8. Air streamlines at different cross sections for DV case.

Lastly, DV case shows recirculation pattern for all seat rows (Figure 8). Due to this outcome, DV air distribution system can be considered inappropriate when comparing with CSMV strategy.

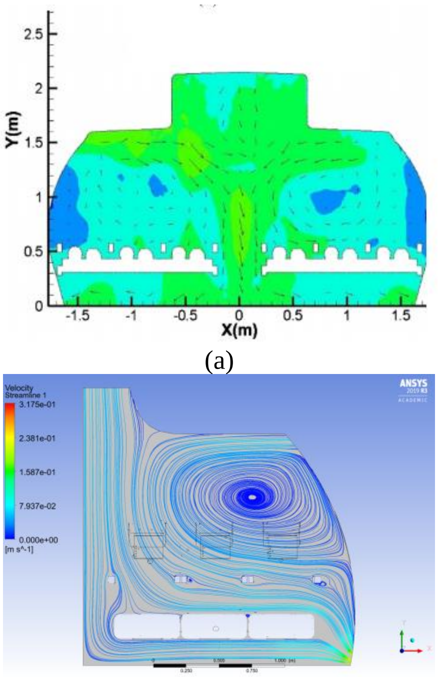
In conclusion, according to the numerical analyses made for four different air distribution strategies, for all cases, air recirculation occurs for the same row seats. However, the strongest and complex air recirculation is observed for SMV system, which makes this system the most inappropriate. The smoothest air movement for the same row seats is observed for CSMV system as seen in Figure 7. It is also observed that, the air mixing between seat rows does not occur for CMV and CSMV systems, whereas small air mixing can occur, especially for the middle seats of different rows, for SMV and DV systems. According to these observations, it is concluded that the CSMV system is the most appropriate ventilation system among the investigated cases.

3.1 Validation of the used numerical model

In order to test the accuracy of the numerical model used in this study, a comparison of air streamlines given in the experimental study conducted by Zhang et al. [5] and in this study is done for CSMV case. In Figure 8a, the velocity vectors of air obtained from the experimental work of Zhang et al. [5] are seen, while the air streamlines obtained from the numerical analyses of the present study are given. It can be said that there is a very good agreement with the present study and the experimental outcome of Zhang et al. [5] as the air movement is in the

Celal Bayar University Journal of Science
Volume 18, Issue 4, 2022, p 409-416
Doi: 10.18466/cbayarfbe.1073131

similar trend in two studies. The results show that the numerical findings of the present study are quite reliable.



(a)

(b)

Figure 8. Comparison of streamlines of present study with Zhang et al. (2017) study for CSMV case.
a) Experimental results given by Zhang et al. [5] for CSMV case.
b) Streamlines of the present study for CSMV case.

4. Conclusion

The airplane passenger numbers are increasing day by day, as it is the fastest and the most comfortable way of travel, especially for long hauls. However, since the airplane passenger cabin is a confined space, there is a risk of contaminant spread inside the cabin, and this spread can cause very serious health problems. In the current study, a part of the passenger cabin of an airplane is modeled, considering the symmetrical conditions. Four different ventilation systems are evaluated, numerically, and the air streamlines are obtained by using Ansys-Fluent software.

The streamlines are plotted for the cross section which passes through the window, aisle and middle seats of different rows and through the cross section which passes through the three seats at the same row. According to the numerical analyses made for four different air distribution strategies, air recirculation occurs for the same row seats, for all cases.

However, the strongest and the most complex air recirculation is observed for SMV system, which makes this system the most inappropriate case. The smoothest air movement for the same row seats is observed for CSMV system. It is also observed that, the air mixing between rows does not occur for CMV and CSMV systems, whereas small air mixing can occur, especially for the middle seats of different rows, for SMV and DV systems. According to these observations, it is concluded that CSMV system is the most appropriate ventilation system among the investigated cases.

Author's Contributions *(mandatory)*

Levent Bilir: Dr. Levent Bilir has made the meshing operation for the airplane cabin model and performed the numerical analyses for all cases by using Ansys-Fluent software. He has written the entire manuscript by getting feedback from other authors. He also, has contributed to interpretation of the results.

Hasan Çelik: Dr. Hasan Çelik has created the solid model of the airplane cabin. He also, has made contribution, mainly to the literature survey.

Barış Özerdem: The original idea of this study belongs to Dr. Barış Özerdem. He has made contribution, mainly to literature survey, results and discussion along with conclusion parts. He also, has revised the entire manuscript.

Ethics (mandatory)

There are no ethical issues after the publication of this manuscript.

References

[1]. Conceiçao ST, Pereira, ML, Tribess, A. 2011, A review of methods applied to study airborne bio contaminants inside aircraft cabins. *International Journal of Aerospace Engineering*; vol. 2011: Article ID 824591, doi:10.1155/2011/824591.

[2]. Khalil, EE. The comfort in commercial aircrafts cabins: A review. AIAA Scientific Forum, Orlando, FL, 6-10 January 2020.

[3]. Melikov, AK, Dzhartov, V. 2013. Advanced air distribution for minimizing airborne cross infection in aircraft cabin. *HVAC&R Research*; 19: 926-933.

[4]. Cao, X, Liu, J, Pei, J, Zhang, Y, Li, J, Zhu, X. 2014. 2D-PIV measurement of aircraft cabin air distribution with a high spatial resolution. *Building and Environment*; 82: 9-19.

[5]. Zhang, Y, Liu, J, Pei, J, Li, J, Wang, C. 2017. Performance evaluation of different air distribution systems in an aircraft cabin mockup. *Aerospace Science and Technology*; 70: 359-366.

[6]. Li, J, Liu, J, Wang, C., Wesseling, M, Müller, D. 2017. PIV experimental study of the large-scale dynamic airflow structures in an aircraft cabin: Swing and oscillation. *Building and Environment*; 125: 180-191.

[7]. Wu, Y, Liu, H, Li, B, Yong, C, Tan, D, Fang, Z. 2017. Thermal comfort criteria for personal air supply in aircraft cabins in winter. *Building and Environment*; 125: 373-382.

[8]. Mazumdar, S, Chen, QA. 2009. One-dimensional analytical model for airborne contaminant transport in airliner cabins. *Indoor Air*; 19: 3-13.

[9]. Fiser, J, Jicha, M. 2013. Impact of air distribution system on quality of ventilation in small aircraft cabin. *Building and Environment*; 69: 171-182.

[10]. Wang, H, Lin, M, Chen, Y. 2014. Performance evaluation of air distribution systems in three different China railway high-speed train cabins using numerical simulation, *Building Simulation*; 7: 629-638.

[11]. Yang, S, Sun, X, Yu, T, Chen, X. 2015. Research on the numerical simulation of aircraft cabin smoke. *Procedia Engineering*; 121: 357-364.

[12]. Maier, J, Marggraf-Micheel, C, Dehne, T, Bosbach, J. 2017. Thermal comfort of different displacement ventilation systems in an aircraft passenger cabin. *Building and Environment*; 111: 256-264.

[13]. Kotb, H, Khalil, EE. 2020. Numerical simulation of airflow and airborne pathogen transport in aircraft cabins: Dynamic mesh analyses. AIAA Scientific Forum, 6-10 January 2020, Orlando, FL.

[14]. Shi, Z, Bai, J, Han, Y. 2020. Distribution of ozone and its volatiles in indoor environment: a numerical simulation with CFD for the aircraft cabin. *Environmental Technology*; 24: 3146-3156.

[15]. Pan Y, Lin, CH, Wei, D, C. Chen, 2020. Influence of surface roughness on particle deposition distribution around multi-slot cabin supply air nozzles of commercial airplanes. *Building and Environment*; 176: 106870.

[16]. Wang, C, Zhang, J, Chao, J, Yang, C, Chen, H. 2021. Evaluation of dynamic air flow structures in a single-aisle aircraft cabin mockup based on numerical simulation. *Indoor and Built Environment*; https://doi.org/10.1177/1420326X21992094.

[17]. Thysen JH, van Hooff, T, Blocken, B, van Heijst, GJF. 2021. CFD simulations of two opposing plane wall jets in a generic empty airplane cabin: Comparison of RANS and LES. *Building and Environment*; 205: 108174.

[18]. Pirouz, B, Mazzeo, D, Palermo, SA, Naghib, SN, Turco, M, Piro, P. 2021. CFD investigations of vehicle's ventilation systems and analysis of ACH in typical airplanes, cars and busses. *Sustainability*; 13: 6799.

[19]. Ansys - Fluent Theory Guide, 2012, Realizable k-ε model, https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node60.htm (accessed at 03.02.2022)

[20] Bilir, L, Çelik, H, Özerdem, B. 2021. Investigation of Air Flow Inside an Airplane Passenger Cabin. *Turkish Journal of Materials;* 6(2): 19-24.