Research Article | Submit Manuscript

Turkish Journal of Materials

Investigation of Air Flow Inside an Airplane Passenger Cabin

Levent Bilir[1,*], Hasan Çelik[2], Mehmet Barış Özerdem[3]

[1] Department of Energy Systems Engineering, Yaşar University, İzmir,Turkey
[2] Department of Mechanical Engineering, İzmir University of Economics, İzmir, Turkey
[3] Department of Aerospace Engineering, İzmir University of Economics, İzmir, Turkey

Received: 22.04.2021; Accepted: 28.06.2021; Published: 30.12.2021
Turk. J. Mater. Vol: 6 No: 2 Page: 19-24 (2021) ISSN: 2636-8668
SLOI: http://www.sloi.org/sloi-name-of-this-article
*Correspondence E-mail: levent.bilir@yasar.edu.tr

ABSTRACT Air flow characteristics are important in passenger cabins of airplane in terms of thermal comfort and health issues. Many parameters such as temperature, air velocity, carbon dioxide level should be controlled during the flight. In the present study, a generic model represents the air flow in a commercial airplane's passenger cabin has been investigated, numerically. The generic model was considered a half aircraft cabin due to symmetrical conditions. The air flow inside the cabin was analysed for ceiling supply bottom return mixing ventilation system by using Ansys-Fluent software. As a conclusion, the streamlines of air released from air ducts was obtained and the flow path of air inside the passenger cabin was determined. The flow path of air gives information about the contaminant spread inside the cabin, as well. The results revealed that for the investigated air ventilation system, there was no significant air recirculation for different seat rows. But the air recirculation on seats at the same row was at higher level.

Keywords: Airplane cabin, air flow path, ANSYS-Fluent analysis, numerical analysis, contaminant spread.

Cite this article: L. Bilir, H. Çelik, M.B. Özerdem. Investigation of Air Flow Inside an Airplane Passenger Cabin. Turk. J. Mater. 6(2) (2021) 19-24.

1. INTRODUCTION

Air travel is getting more and more inevitable due to long distances and time limitation. Hence, the passenger number using airplanes is steadily increasing. Therefore, aircraft manufacturers work hard to provide the occupants with safe air by meeting safety guidelines, standards, and codes. As the cabin of airplanes are confined and pressurized places, the air supplied by the air distribution system can reach some remote places inside the cabin. Air containing viruses and bacteria have been detected on flights. Some very dangerous, even fatal illnesses, such as SARS, H1N1, Ebola, and Covid-19 show this type of transmissivity. If the contaminant, especially in the form of viruses or bacteria reaches other seats, this causes a very serious health problems for passengers and crew members. Hence, novel design of ventilation and air conditioning of aircraft cabins is important in order to increase health and safety of occupants.

The studies presented in the literature are mainly composed of particle image velocimetry (PIV), particle dispersion, computational fluid dynamics (CFD), tracer gas, bleed air or combination of them. Hosni and Jones [1, 2] studied velocity and turbulence intensities by using PIV technique. Lebbin et al. [3] compared CFD and PIV results for a generic cabin model. Another study comparing CFD and PIV techniques was conducted by Lin et al. [4]. Mazumdar and Chen [5] considered a seat row inside an airplane cabin and established a one-dimensional analytical model for longitudinal transport of a contaminant. Fiser and Jicha [6] took a small airplane into account and modeled its passenger cabin using a CFD software. They considered three different air distribution systems and investigated these systems under cold, mild, and hot ambient conditions. They determined which system provides the most stable air distribution during the cabin ventilation.

Cao et al. [7] took two-dimensional PIV measurement of air distribution inside an airplane cabin. They presented the air velocity distribution and turbulence intensity contours using the measured data. Yang et al. [8] numerically investigated smoke spread inside the cabin of an airplane. They determined the smoke distribution and velocity vectors at different cross sections. They also presented carbon dioxide concentrations at various cross sections inside the cabin. Maier et al. [9] evaluated thermal comfort inside the airplane cabin for different ventilation systems. They compared mixing ventilation system with displacement system and hybrid (50:50) mixing and displacement system. Zhang et al. [10] also compared different ventilation systems, experimentally. They revealed advantages and disadvantages of the considered systems. Kotb and Khalil [11] evaluated the spread of a cough droplet from a moving passenger inside an aircraft cabin. They investigated the path of the cough droplet from the passenger moving with different velocities.

In the present study, a section of a commercial airplane cabin, consisting of three seat rows, was modeled using Ansys-Fluent CFD software. The ceiling supply bottom return mixing ventilation (CMV) system is chosen for air distribution. The air stream graphs are presented to illustrate how the air moves inside the cabin. The presented graphs also give information about the contaminant move inside the cabin.

2. METHOD

Ansys-Fluent program is used in order to obtain a numerical solution for the air distribution inside the evaluated cabin. The Navier-Stokes equations presented in Equations (1-4) are solved with turbulence equations in order to get the flow field.

$Continuity$:

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

$X-Momentum$:

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

$Y-Momentum$:

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

$Z-Momentum$:

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

2.1 Considered Airplane Cabin Model

The section including three seat rows of a commercial airplane cabin is taken into consideration due to symmetrical conditions. In order to obtain a numerical solution, the area of interest is divided into 8,199,032 polyhedral meshes. The view of the generated meshes is given in Figure 1.

Figure 1. Polyhedral meshes generated for the model of passenger cabin.

The needed supply airflow is taken as 9.4 L/s per person [11]. As there are nine passengers in the selected cabin section the airflow is determined as 84.6 L/s and the corresponding mass flow rate is calculated as 0.11 kg/s. The air is supplied from the top side section of the cabin (blue section) and it is exhausted form the bottom side section (red section) of the model. The front, back and side (yellow section) surfaces of the cabin are defined as symmetry. Bottom surface and the outer side (gray sections) surfaces of the cabin are taken as wall.

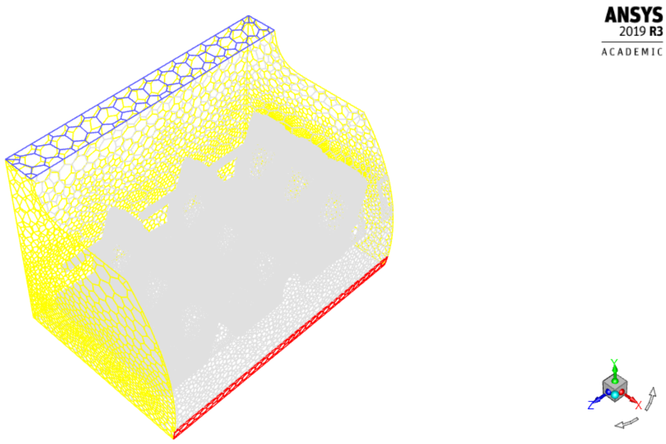
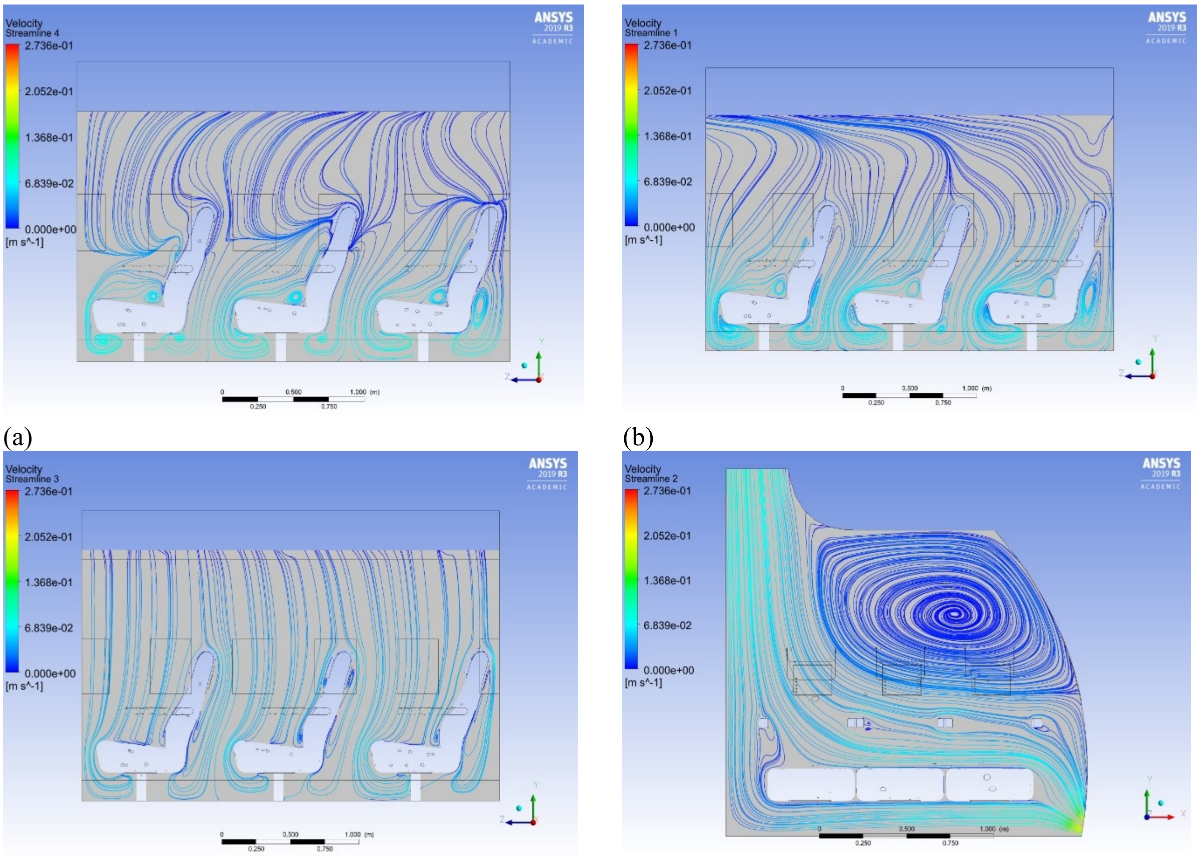


Figure 2. Boundary condition used for the numerical solution.

The reliable k-ε turbulence model with enhanced wall treatment is used as the turbulence model for the numerical solution. This turbulence model is chosen due to its accuracy and easy convergence.

3. RESULTS and DISCUSSION

The created model is transferred to Fluent solver and iterations for numerical solution are continued until all residual values become lower than 10^{-3} (for continuity) and 10^{-6} (for all others). As a result, the streamlines for different cross sections inside the airplane cabin are presented in Figure 3.



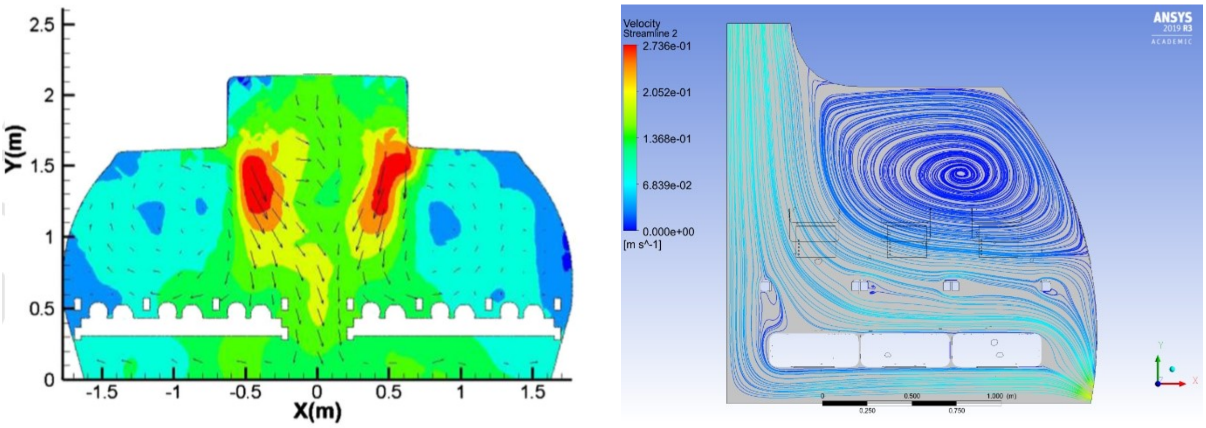
(a) (b)

(c) (d)

Figure 3. Streamline views at different cross sections.

The streamlines in Figure 3(a) are on the cross section which passes through the middle of the window side seats, while the streamlines in Figure 3(b) are on the cross section which passes through the middle of the center seats. Figure 3(c) shows the streamlines on the cross section which passes through the middle of the aisle side seats. Lastly, the streamlines on the vertical cross section which passes through three seats on the middle row shown in Figure 3(d).

According to the numerical results for the air distribution inside the passenger cabin, it can be seen that the air between the seat rows is not mixed considerably for the evaluated air ventilation system. However, there is an air recirculation for the air on the same row, seen in Figure 3 (d). This recirculation can cause a health problem. In order to overcome this problem some air can also be send into the passenger cabin over the window side seat.



(a) (b)

Figure 4. Comparison of the streamline with the experimental findings of [10].

When the velocity distribution taken from the experimental study performed by Zhang et al. [10] (Figure 4a) is compared with the streamlines found in Figure 3 (d) (Figure 4b), it can be said that there is a very good agreement with the present study and the experimental outcome of [10]. The result shows that the numerical findings of the present study are reliable.

4. CONCLUSIONS

As the passenger number using airlines is increasing with a high trend, the air quality inside the passenger cabins is gaining more importance. Moreover, the danger of contaminant spread inside the cabin can cause serious health problems. In the present study, the air distribution inside a commercial airplane cabin is investigated numerically for ceiling supply bottom return mixing ventilation system. A section of the cabin is modeled using a CFD program called Ansys-Fluent and meshed with polyhedral volume elements. A total of approximately 8.2 million volume elements are generated and the numerical solution is obtained by defining appropriate boundary conditions. The results are compared with the experimental findings in the literature, and it is seen that there is a good agreement between the studies. According to the air streamlines, it is found that air mixing between the rows of the seats is not significant. However, it is observed that the air recirculation is encountered for the same row seats. This problem can be solved by applying another air ventilation strategy, which is the subject of a further study.

REFERENCES

[1] Hosni M, Jones BW. Measurement of Ain an Aircraft Cabin using Particle Imaging Velocimetry Technique, *4th International Conference on Indoor Air Quality, Ventilation and Energy Conservation in Buildings*; 2-5 October 2001, Changsha, China.

[2] Hosni M, Jones BW. Application of Particle Image Velocimetry for the Measurement of the Airflow Characteristics in an Aircraft Cabin, *ASHRAE Transactions,* **109***,*101-110, 2003.

[3] Lebbin PA, Hosni M, Jones BW, Beck BT. Feasibility Study of Using Various Instrumentation for Measurement of Air Velocity and Turbulence Intensity in a Test Room, *ASHRAE Transactions,* **111**, 54-59, 2005.

[4] Lin C, Wu T, Horstman R, Lebbin P, Hosni M, et al. Comparison of LED Simulation Predictions with PIV Data for the Airflow in a Generic Cabin Model, *HVAC&R Research,* **12**, 935-951, 2006.

[5] Mazumdar, S., Chen, Q. A One-Dimensional Analytical Model for Airborne Contaminant Transport in Airliner Cabins, *Indoor Air,* **19**, 3-13, 2009.

[6] Fiser, J., Jicha, M. Impact of Air Distribution System on Quality of Ventilation in Small Aircraft Cabin , *Building and Environment*, **69**, 171-182, 2013.

[7] Cao, X., Liu, J., Pei, J., Zhang, Y., Li, J., Zhu, X. 2D-PIV Measurement of Aircraft Cabin Air Distribution with a High Spatial Resolution, *Building and Environment***, 82**, 9-19, 2014.

[8] Yang, S., Sun, X., Yu, T, Chen, X. Research on the Numerical Simulation of Aircraft Cabin Smoke, *Procedia Engineering,***, 121**, 357-364, 2015.

[9] Maier, J., Marggraf-Micheel, C., Dehne, T, Bosbach, J. Thermal Comfort of Different Displacement Ventilation Systems in an Aircraft Passenger Cabin, *Building and Environment***, 111**, 256-264, 2017.

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

[11] Kotb, H., Khalil, E.E. Numerical Simulation of Airflow and Airborne Pathogen Transport in Aircraft Cabins: Dynamic Mesh Analyses, *AIAA SCientific Forum,* Orlando, FL, 6-10 January 2020.

Symbol List

Re	Reynolds number
t	time [s]
u	velocity in x direction [m s^{-1}]
v	velocity in y direction [m s^{-1}]
w	velocity in z direction [m s^{-1}]

Greek Letters

ρ	Density [kg m^{-3}]
τ	Shear stress [N m^{-2}]

Acknowledgement: 5th International Anatolian Energy Symposium
24-26 March 2021, Karadeniz Technical University, Trabzon/Turkey